Commands Reference, Volume 3, i - m
Note
Before using this information and the product it supports, read the information in "Notices," on page 687.

This edition applies to AIX 5L Version 5.2 and to all subsequent releases of this product until otherwise indicated in new editions.

A reader's comment form is provided at the back of this publication. If the form has been removed, address comments to Information Development, Department H6DS-905-6C006, 11501 Burnet Road, Austin, Texas 78758-3493. To send comments electronically, use this commercial Internet address: aix6kpub@austin.ibm.com. Any information that you supply may be used without incurring any obligation to you.

© Copyright International Business Machines Corporation 1997, 2004. All rights reserved.
US Government Users Restricted Rights – Use, duplication or disclosure restricted by GSA ADP Schedule Contract with IBM Corp.
Contents

About This Book .................................. xi
Who Should Use This Book ....................... xi
How to Use This Book .............................. xi
Highlighting ....................................... xi
ISO 9000 ........................................... xiv
32-Bit and 64-Bit Support for the UNIX98 Specification .......................... xiv
Related Information ................................ xiv

Alphabetical Listing of Commands.
ibm3812 Command ................................... 1
ibm3816 Command ................................... 2
ibm5585H-T Command ................................. 3
ibm5587G Command ................................... 4
iconv Command ...................................... 5
id Command ......................................... 6
ifconfig Command ................................... 8
ike Command ...................................... 16
ikedb Command .................................... 21
imake Command .................................... 23
imapd Daemon ...................................... 25
impfill Command .................................... 26
importvg Command .................................. 26
imptun Command .................................... 29
inc Command ....................................... 30
indent Command ..................................... 32
inkxibib Command .................................. 35
inetd Daemon ....................................... 37
infocmp Command .................................. 39
infocenter Command ................................. 42
install Command ................................... 42
install_all_updates Command .................... 44
install_assist Command ............................. 46
install_mh Command ................................ 47
install_wizard Command ............................ 48
installbsd Command ................................ 48
instalp Command ................................... 49
installm Command ................................... 60
instfix Command .................................... 63
inucp Command ..................................... 65
inudcm Command .................................... 66
inulag Command .................................... 67
inuxcv Command .................................... 68
inuxrest Command .................................. 70
inuxr Command ..................................... 71
inuxsave Command .................................. 72
inuxc Command ..................................... 75
inuxmsg Command ................................... 76
invscout Command .................................. 77
invscoutd Command ................................ 79
iow Command ...................................... 85
iostat Command .................................... 97
ipcrm Command .................................... 102
ipcs Command ...................................... 103
<table>
<thead>
<tr>
<th>Command</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ipfilter Command</td>
<td>106</td>
</tr>
<tr>
<td>ipreport Command</td>
<td>107</td>
</tr>
<tr>
<td>ipsec_convert Command</td>
<td>107</td>
</tr>
<tr>
<td>ipsecstat Command</td>
<td>108</td>
</tr>
<tr>
<td>ipsectrcbuf Command</td>
<td>108</td>
</tr>
<tr>
<td>iptrace Daemon</td>
<td>109</td>
</tr>
<tr>
<td>isC2host Command</td>
<td>111</td>
</tr>
<tr>
<td>isCCHost Command</td>
<td>112</td>
</tr>
<tr>
<td>isstat Command</td>
<td>113</td>
</tr>
<tr>
<td>jobs Command</td>
<td>114</td>
</tr>
<tr>
<td>join Command</td>
<td>116</td>
</tr>
<tr>
<td>joinvg Command</td>
<td>119</td>
</tr>
<tr>
<td>kdb Command</td>
<td>120</td>
</tr>
<tr>
<td>kdestroy Command</td>
<td>122</td>
</tr>
<tr>
<td>keyadd Command</td>
<td>123</td>
</tr>
<tr>
<td>keycomp Command</td>
<td>124</td>
</tr>
<tr>
<td>keydelete Command</td>
<td>126</td>
</tr>
<tr>
<td>keyenvoy Command</td>
<td>127</td>
</tr>
<tr>
<td>keylist Command</td>
<td>128</td>
</tr>
<tr>
<td>keylogin Command</td>
<td>130</td>
</tr>
<tr>
<td>keylogout Command</td>
<td>130</td>
</tr>
<tr>
<td>keypasswd Command</td>
<td>131</td>
</tr>
<tr>
<td>keyserv Daemon</td>
<td>132</td>
</tr>
<tr>
<td>kill Command</td>
<td>133</td>
</tr>
<tr>
<td>killall Command</td>
<td>135</td>
</tr>
<tr>
<td>kinet Command</td>
<td>136</td>
</tr>
<tr>
<td>klist Command</td>
<td>138</td>
</tr>
<tr>
<td>kmodctrl Command</td>
<td>139</td>
</tr>
<tr>
<td>kppasswd Command</td>
<td>140</td>
</tr>
<tr>
<td>krllogind Daemon</td>
<td>140</td>
</tr>
<tr>
<td>krshd Daemon</td>
<td>142</td>
</tr>
<tr>
<td>ksh Command</td>
<td>144</td>
</tr>
<tr>
<td>kvno Command</td>
<td>146</td>
</tr>
<tr>
<td>last Command</td>
<td>147</td>
</tr>
<tr>
<td>lastcomm Command</td>
<td>148</td>
</tr>
<tr>
<td>lastlogin Command</td>
<td>150</td>
</tr>
<tr>
<td>lb_admin Command</td>
<td>150</td>
</tr>
<tr>
<td>lb_find Command</td>
<td>153</td>
</tr>
<tr>
<td>lbxproxy Command</td>
<td>154</td>
</tr>
<tr>
<td>ld Command</td>
<td>156</td>
</tr>
<tr>
<td>ldd Command</td>
<td>174</td>
</tr>
<tr>
<td>ledit Command</td>
<td>175</td>
</tr>
<tr>
<td>learn Command</td>
<td>176</td>
</tr>
<tr>
<td>leave Command</td>
<td>177</td>
</tr>
<tr>
<td>lecstat Command</td>
<td>178</td>
</tr>
<tr>
<td>lex Command</td>
<td>180</td>
</tr>
<tr>
<td>libxrx Command</td>
<td>186</td>
</tr>
<tr>
<td>line Command</td>
<td>187</td>
</tr>
<tr>
<td>link Command</td>
<td>188</td>
</tr>
<tr>
<td>lint Command</td>
<td>189</td>
</tr>
<tr>
<td>listgrp Command</td>
<td>194</td>
</tr>
<tr>
<td>listvgbackup Command</td>
<td>195</td>
</tr>
<tr>
<td>listX11input Command</td>
<td>197</td>
</tr>
<tr>
<td>lbdb Daemon</td>
<td>197</td>
</tr>
<tr>
<td>ln Command</td>
<td>198</td>
</tr>
<tr>
<td>locale Command</td>
<td>200</td>
</tr>
</tbody>
</table>
localedef Command ................................................................. 202
lock Command ........................................................................ 204
lockd Daemon .......................................................................... 205
locktrace Command ................................................................ 207
elogevent Command, logevent Command ............................... 208
logform Command ................................................................. 209
logger Command .................................................................. 211
login Command .................................................................... 212
logins Command .................................................................. 214
logname Command ................................................................ 217
logout Command .................................................................. 218
look Command ...................................................................... 218
lookbib Command .................................................................. 219
lorder Command .................................................................... 220
lp Command ......................................................................... 221
lp.cat, lp.set, lp.tell Command ............................................. 227
lpadmin Command .................................................................. 229
lp Command .......................................................................... 236
lpd Command ....................................................................... 238
lpfilter Command .................................................................. 240
lpforms Command .................................................................. 243
lpmove Command .................................................................... 247
lpchk Command .................................................................... 248
lpmgr Command ..................................................................... 250
lpq Command .......................................................................... 252
lpr Command ......................................................................... 255
lprm Command ........................................................................ 260
lpsched Command .................................................................... 262
lpstat Command ..................................................................... 263
lpsystem Command .................................................................. 267
lptest Command ....................................................................... 269
lpusers Command .................................................................... 269
ls Command ........................................................................... 271
ls-secldapclntd Command ..................................................... 275
lsactdef Command .................................................................. 276
lsallq Command ..................................................................... 280
lsallqdev Command .................................................................. 281
lsarm command ....................................................................... 282
lsattr Command ....................................................................... 283
lsaudrec Command .................................................................... 287
lsauthent Command .................................................................. 291
lsC2admin Command .................................................................. 292
lsCCadmin Command .................................................................. 292
lscfg Command ........................................................................ 293
lsclass Command ...................................................................... 295
lslong Command ..................................................................... 297
lscondition Command ................................................................ 299
lscondresp Command .................................................................. 303
lsconn Command ...................................................................... 308
lscons Command ..................................................................... 310
lsdev Command ........................................................................ 311
lsdisp Command ....................................................................... 317
lsfiit Command ........................................................................ 318
lsfont Command ....................................................................... 319
lsfs Command .......................................................................... 320
lsgroup Command .................................................................... 321
About This Book

This book is Volume 3 of the six-volume AIX 5L Version 5.2 Commands Reference, which contains reference information on operating system commands. It describes the tasks each command performs, how commands can be modified, how they handle input and output, who can run them, and provides a master index for all six volumes.

For a quick reference list of commands arranged in functional groups, see Volume 6.

This edition supports the release of AIX 5L Version 5.2 with the 5200-04 Recommended Maintenance package. Any specific references to this maintenance package are indicated as AIX 5.2 with 5200-04.

Who Should Use This Book

This book is intended for users of operating system commands.

How to Use This Book

A command is a request to perform an operation or run a program. You use commands to tell the operating system what task you want it to perform. When commands are entered, they are deciphered by a command interpreter (also known as a shell) and that task is processed.

Some commands can be entered simply by typing one word. It is also possible to combine commands so that the output from one command becomes the input for another command. This is known as pipelining.

Flags further define the actions of commands. A flag is a modifier used with the command name on the command line, usually preceded by a dash.

Commands can also be grouped together and stored in a file. These are known as shell procedures or shell scripts. Instead of executing the commands individually, you execute the file that contains the commands.

Some commands can be constructed using Web-based System Manager applications or the System Management Interface Tool (SMIT).

Highlighting

The following highlighting conventions are used in this book:

| **Bold** | Identifies commands, subroutines, keywords, files, structures, directories, and other items whose names are predefined by the system. Also identifies graphical objects such as buttons, labels, and icons that the user selects. |
| **Italic** | Identifies parameters whose actual names or values are to be supplied by the user. |
| **Monospace** | Identifies examples of specific data values, examples of text similar to what you might see displayed, examples of portions of program code similar to what you might write as a programmer, messages from the system, or information you should actually type. |

Format

Each command may include any of the following sections:

- **Purpose** A description of the major function of each command.
- **Syntax** A syntax statement showing command line options.
- **Description** A discussion of the command describing in detail its function and use.
### Flags
A list of command line flags and associated variables with an explanation of how the flags modify the action of the command.

### Parameters
A list of command line parameters and their descriptions.

### Subcommands
A list of subcommands (for interactive commands) that explains their use.

### Exit Status
A description of the exit values the command returns.

### Security
Specifies any permissions needed to run the command.

### Examples
Specific examples of how you can use the command.

### Files
A list of files used by the command.

### Related Information
A list of related commands in this book and related discussions in other books.

## Reading Syntax Statements
Syntax statements are a way to represent command syntax and consist of symbols such as brackets ([ ]), braces ({ }), and vertical bars (|). The following is a sample of a syntax statement for the `unget` command:

```
unget [ -rSID ] [ -s ] [ -n ] File ...
```

The following conventions are used in the command syntax statements:
- Items that must be entered literally on the command line are in **bold**. These items include the command name, flags, and literal characters.
- Items representing variables that must be replaced by a name are in *italics*. These items include parameters that follow flags and parameters that the command reads, such as `Files` and `Directories`.
- Parameters enclosed in brackets are optional.
- Parameters enclosed in braces are required.
- Parameters not enclosed in either brackets or braces are required.
- A vertical bar signifies that you choose only one parameter. For example, [ a | b ] indicates that you can choose a, b, or nothing. Similarly, { a | b } indicates that you must choose either a or b.
- Ellipses (...) signify the parameter can be repeated on the command line.
- The dash (-) represents standard input.

## Listing of Installable Software Packages
To list the installable software package (fileset) of an individual command use the `lslpp` command with the `-w` flag. For example, to list the fileset that owns the `installp` command, enter:

```
lslpp -w /usr/sbin/installp
```

Output similar to the following displays:

<table>
<thead>
<tr>
<th>File</th>
<th>Fileset</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/sbin/installp</td>
<td>bos.rte.install</td>
<td>File</td>
</tr>
</tbody>
</table>

To list the fileset that owns all file names that contain `installp`, enter:

```
lslpp -w "*installp*"
```

Output similar to the following displays:

<table>
<thead>
<tr>
<th>File</th>
<th>Fileset</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/sbin/installp</td>
<td>bos.rte.install</td>
<td>File</td>
</tr>
<tr>
<td>/usr/clvm/sbin/installp</td>
<td>prpq.clvm</td>
<td>File</td>
</tr>
<tr>
<td>/usr/lpp/bos.sysmgmt/nim/methods/c_installp</td>
<td>bos.sysmgmt.nim.client</td>
<td>File</td>
</tr>
</tbody>
</table>
Running Commands in the Background

If you are going to run a command that takes a long time to process, you can specify that the command run in the background. Background processing is a useful way to run programs that process slowly. To run a command in the background, you use the & operator at the end of the command:

```
Command &
```

Once the process is running in the background, you can continue to work and enter other commands on your system.

At times, you might want to run a command at a specified time or on a specific date. Using the `cron` daemon, you can schedule commands to run automatically. Or, using the `at` and `batch` commands, you can run commands at a later time or when the system load level permits.

Entering Commands

You typically enter commands following the shell prompt on the command line. The shell prompt can vary. In the following examples, `$` is the prompt.

To display a list of the contents of your current directory, you would type `ls` and press the Enter key:

```
$ ls
```

When you enter a command and it is running, the operating system does not display the shell prompt. When the command completes its action, the system displays the prompt again. This indicates that you can enter another command.

The general format for entering commands is:

```
Command Flag(s) Parameter
```

The flag alters the way a command works. Many commands have several flags. For example, if you type the `-l` (long) flag following the `ls` command, the system provides additional information about the contents of the current directory. The following example shows how to use the `-l` flag with the `ls` command:

```
$ ls -l
```

A parameter consists of a string of characters that follows a command or a flag. It specifies data, such as the name of a file or directory, or values. In the following example, the directory named `/usr/bin` is a parameter:

```
$ ls -l /usr/bin
```

When entering commands, it is important to remember the following:

- Commands are usually entered in lowercase.
- Flags are usually prefixed with a `-` (minus sign).
- More than one command can be typed on the command line if the commands are separated by a `;` (semicolon).
- Long sequences of commands can be continued on the next line by using the `\` (backslash). The backslash is placed at the end of the first line. The following example shows the placement of the backslash:

```
$ cat /usr/ust/mydir/mydata > \n/usr/usts/yourdir/yourdata
```

When certain commands are entered, the shell prompt changes. Because some commands are actually programs (such as the `telnet` command), the prompt changes when you are operating within the command. Any command that you issue within a program is known as a subcommand. When you exit the program, the prompt returns to your shell prompt.
The operating system can operate with different shells (for example, Bourne, C, or Korn) and the commands that you enter are interpreted by the shell. Therefore, you must know what shell you are using so that you can enter the commands in the correct format.

**Stopping Commands**

If you enter a command and then decide to stop that command from running, you can halt the command from processing any further. To stop a command from processing, press the Interrupt key sequence (usually Ctrl-C or Alt-Pause). When the process is stopped, your shell prompt returns and you can then enter another command.

**ISO 9000**

ISO 9000 registered quality systems were used in the development and manufacturing of this product.

**32-Bit and 64-Bit Support for the UNIX98 Specification**

Beginning with Version 4.3, the operating system is designed to support The Open Group’s UNIX98 Specification for portability of UNIX-based operating systems. Many new interfaces, and some current ones, have been added or enhanced to meet this specification, making Version 4.3 even more open and portable for applications.

At the same time, compatibility with previous releases of the operating system is preserved. This is accomplished by the creation of a new environment variable, which can be used to set the system environment on a per-system, per-user, or per-process basis.

To determine the proper way to develop a UNIX98-portable application, you may need to refer to The Open Group’s UNIX98 Specification, which can be obtained on a CD-ROM by ordering *Go Solo: The Authorized Guide to Version 2 of the Single UNIX Specification*, a book which includes The Open Group’s UNIX98 Specification on a CD-ROM.

**Related Information**

The following books contain information about or related to commands:

- AIX 5L Version 5.2 Commands Reference, Volume 1
- AIX 5L Version 5.2 Commands Reference, Volume 2
- AIX 5L Version 5.2 Commands Reference, Volume 3
- AIX 5L Version 5.2 Commands Reference, Volume 4
- AIX 5L Version 5.2 Commands Reference, Volume 5
- AIX 5L Version 5.2 Commands Reference, Volume 6
- AIX 5L Version 5.2 Files Reference
- AIX 5L Version 5.2 Glossary
- AIX 5L Version 5.2 Guide to Printers and Printing
- AIX 5L Version 5.2 Installation Guide and Reference
- AIX 5L Version 5.2 AIX Installation in a Partitioned Environment
- AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide
- AIX 5L Version 5.2 Operating System Installation: Getting Started
- AIX 5L Version 5.2 Performance Management Guide
- AIX 5L Version 5.2 Performance Tools Guide and Reference
- AIX 5L Version 5.2 Security Guide
- AIX 5L Version 5.2 System Management Concepts: Operating System and Devices
- AIX 5L Version 5.2 System Management Guide: Communications and Networks
- AIX 5L Version 5.2 System Management Guide: Operating System and Devices
• AIX 5L Version 5.2 System User’s Guide: Communications and Networks
• AIX 5L Version 5.2 System User’s Guide: Operating System and Devices
• AIX 5L Version 5.2 Technical Reference: Base Operating System and Extensions Volume 1
• AIX 5L Version 5.2 Technical Reference: Base Operating System and Extensions Volume 2
• AIX 5L Version 5.2 Technical Reference: Communications Volume 1
• AIX 5L Version 5.2 Technical Reference: Communications Volume 2
• AIX 5L Version 5.2 Technical Reference: Kernel and Subsystems Volume 1
• AIX 5L Version 5.2 Technical Reference: Kernel and Subsystems Volume 2
• AIX 5L Version 5.2 Web-based System Manager Administration Guide
• Performance Toolbox Version 2 and 3 for AIX: Guide and Reference
Alphabetical Listing of Commands

ibm3812 Command

Purpose
Postprocesses the troff command output for the IBM 3812 Model 2 Pageprinter.

Syntax
ibm3812 [-altpaper] [-landscape] [-quietly] [-FDirectory] [-i] [File...]

Description
The ibm3812 command is a postprocessor that can be used on intermediate output produced by the troff command.

Note: An entire page is placed in memory before it is printed.

If given one or more file names as options, the ibm3812 command processes those files. If no file names are specified, this command acts as a filter interpreting standard input.

The ibm3812 command's font files allow the postprocessor to send characters of more than one byte to the printer. These can be characters that require multiple bytes to represent them, such as code page and point; or, they can be characters that are composed of two or more concatenated glyphs.

For example, the character code for the \ib (improper subset) special character is:
"\001\125\xe2\xff\xe8\xe3\%00\x16\001\074\xe3\xff\xea"

The printer is in Page Map Primitive (PMP) mode when these bytes are sent, so you must use the 001 directive to introduce a character. For single-byte codes, this Generic Font Patterns command is automatically handled by the postprocessor. The % (percent sign) characters escape the bytes containing 0, which would otherwise terminate the code sequence. To obtain a literal % character, escape it with another % character so that a percent sign is displayed as %. A single-byte % code is assumed to be a literal percent sign, so that the single-byte % character needs no special handling in the font file.

Notes:
1. The ibm3812 command depends on the files with names ending in .out in the /usr/lib/font/devibm3812 directory. It does not produce usable output unless these files have been properly set up.
2. The postprocessor requires additional font information to be stored in the /usr/lib/font/devibm3812/fonts file. If new fonts are added to this file, make sure that the DESC file is also updated to reflect the additional fonts and special characters.

The format of the file must be preserved. The file contains the following four fields:
- The one- or two-letter name of the font
- The full name of the font on the printer-font diskette
- The one- or two-letter name of the substitute font
- An array of five available sizes.

Flags
- -altpaper Specifies that the file should be printed from the alternate paper drawer. By default, the ibm3812 command prints from the primary paper drawer.

© Copyright IBM Corp. 1997, 2004
-landscape Specifies that the file should be printed in landscape orientation, so that the wider part of the paper is horizontally oriented. This flag rotates the page to the right by 90 degrees. By default, the ibm3812 command prints in portrait orientation.

-quietly Suppresses all non-fatal error messages.

-F Directory Specifies the directory holding the font files. The default file is devibm3812. The command looks for font files in the /usr/lib/font directory by default.

-i Suppresses initialization of the printer that runs the PMP.init macro, after the job has printed.

Example
Following is an example of the troff command used with the ibm3812 command:
troff file|ibm3812|qprt-dp

Files
/usr/lib/font/devibm3812/*.out Contains font files for the ibm3812 command.
/usr/lib/font/devibm3812/fonts Contains information about the available fonts for the ibm3812 command.

Related Information
The [ibm3816](#) command, [troff](#) command.

The [troff](#) font file format specifies description files for the troff command.

**ibm3816 Command**

**Purpose**
Postprocesses the troff command output for the IBM 3816 Pageprinter.

**Syntax**
ibm3816 [-altpaper] [-landscape] [-quietly] [-F Directory] [ -i ] [File...]

**Description**
The ibm3816 command is a postprocessor that can be used on intermediate output produced by the troff command.

**Note:** An entire page is placed in memory before it is printed.

If given one or more file names as options, the ibm3816 command processes those files. If no file names are specified, this command acts as a filter interpreting standard input.

The ibm3816 command’s font files allow the postprocessor to send characters of more than one byte to the printer. These can be characters that require multiple bytes to represent them, such as code page and point; or, they can be characters that are composed of two or more concatenated glyphs.

For example, the character code for the \{ib (improper subset) special character is:

"\001\125\xe2\xff\xe8\xe3\%\x00\1\107\xe3\xff\xe8\xe3\%\x00\1\107\xe3\xff\xe8"

The printer is in Page Map Primitive (PMP) mode when these bytes are sent, so you must use the 001 directive to introduce a character. For single-byte codes, this Generic Font Patterns command is automatically handled by the postprocessor. The % (percent sign) characters escape the bytes containing 0, which would otherwise terminate the code sequence. To obtain a literal % character, escape it with
another % character so that a percent sign is displayed as %%. A single-byte % code is assumed to be a literal percent sign, so that the single-byte % character needs no special handling in the font file.

Notes:
1. The ibm3816 command depends on the files with names ending in .out in the /usr/lib/font/devibm3816 directory. It does not produce usable output unless these files have been properly set up.
2. The postprocessor requires additional font information to be stored in the /usr/lib/font/devibm3816/fonts file. If new fonts are added to this file, make sure that the DESC file is also updated to reflect the additional fonts and special characters.

The format of the file must be preserved. The file contains the following four fields:
- The one- or two-letter name of the font
- The full name of the font on the printer-font diskette
- The one- or two-letter name of the substitute font
- An array of five available sizes.

Flags
- -altpaper Specifies that the file should be printed from the alternate paper drawer. By default, the ibm3816 command prints from the primary paper drawer.
- -landscape Specifies that the file should be printed in landscape orientation, so that the wider part of the paper is horizontally oriented. This flag rotates the page to the right by 90 degrees. By default, the ibm3816 command prints in portrait orientation.
- -quietly Suppresses all non-fatal error messages.
- -FDirectory Specifies the directory holding the font files. The default file is devibm3816. The command looks for font files in the /usr/lib/font directory by default.
- -i Suppresses initialization of the printer that runs the PMP.init macro, after the job has printed.

Example
Following is an example of the troff command used with the ibm3816 command:
troff file|ibm3816|qprt-dp

Files
/usr/lib/font/devibm3816/*.out Contains font files for the ibm3816 command.
/usr/lib/font/devibm3816/fonts Contains information about the available fonts for the ibm3816 command.

Related Information
The ibm3812 command, troff command.

The troff font file format specifies description files for the troff command.

ibm5585H-T Command

Purpose
Processes troff command output for the IBM 5585H-T printer.
Syntax

ibm5585H-T [-F Directory] [ File ]

Description

The ibm5585H-T command processes the output of the troff command for output to the IBM 5585H-T printer for traditional Chinese language. This command is provided exclusively for traditional Chinese language support.

The ibm5585H-T command processes one or more files specified by the File parameter. If no file is specified, the ibm5585H-T command reads from standard input.

The ibm5585H-T command uses font files in the /usr/lib/font/devibm5585H-T directory that have command names ending with .out. The ibm5585H-T command does not produce correct output unless these files are provided.

Flag

-F Directory Specifies a directory name as the place to find font files. By default, the ibm5585H-T command looks for font files in the /usr/lib/font/devibm5585H-T directory.

Example

To process the reports file for the IBM 5585H-T printer, enter:

troff reports | ibm5585H-T | qprt -dp

The ibm5585H-T command first processes the output of the troff command, then sends the file to a print queue.

File

/usr/lib/font/devibm5585H-T/*.out Contains font files.

Related Information

The troff command.

The troff font file format.

ibm5587G Command

Purpose

Postprocesses troff command output for the IBM 5587-G01, 5584-H02, 5585-H01, 5587-H01, and 5589-H01 printers with the (32x32/24x24) cartridge installed. This command is used exclusively for Japanese Language Support.

Syntax

ibm5587G [-F Directory] [-quietly] [ File ]

Description

The ibm5587G command processes the output of the troff command for output to the 5587-G01, 5584-H02, 5585-H01, 5587-H01, and 5589-H01 printers.
If given one or more files as options, the **ibm5587G** command processes those files. If no files are specified, it acts as a filter interpreting standard input.

**Note:** The **ibm5587G** command assumes that the (32x32/24x24) cartridge is installed in the printer. Incorrect output from the printer may result if the wrong cartridge is installed in the printer.

The **ibm5587G** command depends on the files with names ending in `.out` in the `/usr/lib/font/devibm5587G` directory. It does not produce reasonable output unless these files have been properly set up.

**Flags**

- `-F Directory` Specifies a directory name as the place to find the font files. By default, the **ibm5587G** command looks for font files in the `/usr/lib/font/devibm5587G` directory.

- `-quietly` Suppresses all nonfatal error messages.

**Files**

```
/usr/lib/font/devibm5587G/*.out
```

Contains font files.

**Related Information**

The **troff** command formats text for printing on typesetting devices.

The **troff** font file format specifies description files for the **troff** command.

**iconv Command**

**Purpose**

Converts the encoding of characters from one code page encoding scheme to another.

**Syntax**

```
iconv [-f FromCode] [-t ToCode] [ FileName... ]
```

**Description**

The **iconv** command converts the encoding of characters read from either standard input or the specified file from one coded character set to another and then writes the results to standard output. The input and output coded character sets are identified by the **FromCode** and **ToCode** parameters. The input data should consist of characters in the code set specified by the **FromCode** parameter. If the **FileName** parameter is not specified on the command line, the **iconv** command reads from standard input.

You can use a Web-based System Manager System application (**wsm system** fast path) to run this command. You could also use the System Management Interface Tool (SMIT) **smit iconv** fast path to run this command.

**Flags**

- `-f FromCode` Specifies the code set in which the input data is encoded. The space between the `-f` flag and the **FromCode** parameter is optional.

- `-t ToCode` Specifies the code set to which the output data is to be converted. The space between the `-t` flag and the **ToCode** parameter is optional.

**FileName** Specifies a file to be converted.
The list of supported code set converters is provided in "List of Converters" in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

**Exit Status**

This command returns the following exit values:

0  Input data was successfully converted.
1  The specified conversions are not supported; the given input file cannot be opened for read; or there is a usage-syntax error.
2  An unusable character was encountered in the input stream.

**Examples**

1. To convert the contents of the mail.x400 file from code set IBM-850 and store the results in the mail.local file, enter:
   ```bash
   iconv -f IBM-850 -t ISO8859-1 mail.x400 > mail.local
   ```
2. To convert the contents of the mail.japan file from the 7-bit interchange (ISO2022) encoding to the Japanese EUC code set (IBM-eucJP), enter:
   ```bash
   iconv -f fold7 -t IBM-eucJP mail.japan > mail.local
   ```
3. To convert the contents of a local file to the mail-interchange format and send mail, enter:
   ```bash
   iconv -f IBM-943 -t fold7 mail.local | mail fxrojas
   ```

**Related Information**

The `genxlt` command describes how to define a conversion table.

The `iconv` subroutine, `iconv_close` subroutine, and `iconv_open` subroutines provide a method to use the conversion service from within a program.

Converters Overview in AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide.

For information on installing the Web-based System Manager, see Chapter 2: Installation and System Requirements in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

Converters Overview for Programming in AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide.

**id Command**

**Purpose**

Displays the system identifications of a specified user.

**Syntax**

```
id [ {-G | -g} [ -r ] | -u [ -r ] ] [ -n ] [ User ]
```

**Description**

The id command writes to standard output a message containing the system identifications (ID) for a specified user. The system IDs are numbers which identify users and user groups to the system. The id command writes the following information, when applicable:

- User name and real user ID
- Name of the user’s group and real group ID
- Name of user’s supplementary groups and supplementary group IDs
Supplementary group information is written only for systems supporting multiple-user groups and only if the specified user belongs to a supplementary group.

The `id` command also writes effective user and group IDs, but only for the user that invoked the `id` command. (If the `User` parameter is specified with the `id` command, the effective IDs are assumed to be identical to real IDs.) If the effective and real IDs for the invoking user are different, the `id` command writes the following effective ID information, when applicable:

- Effective user name and effective user ID
- Name of effective user’s group and effective group ID

The `id` command will fail if the specified user does not exist or if the command cannot read the user or group information.

**Flags**

The contents and format of the message written by the `id` command can be altered with the following flags:

- `-G` Specifies that the `id` command write the effective, real, and supplementary group IDs only. If there are multiple entries for the effective, real, or supplementary IDs, they are separated by a space and placed on the same line.
- `-g` Specifies that the `id` command write only the effective group ID.
- `-u` Specifies that the `id` command write only the effective user ID.
- `-r` Specifies that the `id` command write the real ID instead of the effective ID. This flag can be invoked with either the `-g` flag to write the real group ID, or the `-u` flag to write the real user ID.
- `-n` Specifies that the `id` command outputs the name, instead of the ID number, when it is specified with the `-G`, `-g`, and `-u` flags.
- `User` Specifies the login name of a user for the `id` command. If no user is specified, the user invoking the `id` command is the default.

**Security**

Access Control: This program should be installed as a normal user program in the Trusted Computing Base.

**Exit Status**

This command returns the following exit values:

- `0` Successful completion.
- `>0` An error occurred.

**Examples**

1. To display all system identifications for the current user, enter:

```bash
id
```

Output for the `id` command is displayed in the following format:

```
uid=1544(sah) gid=300(build) euid=0(root) egid=9(printq) groups=0(system),10(audit)
```

In this example, the user has user name sah with an ID number of 1544; a primary group name of `build` with an ID number of 300; an effective user name of `root` with an ID number of 0; an effective group name of `printq` with an ID number of 9; and two supplementary group names of `system` and `audit`, with ID numbers 0 and 10, respectively.

2. To display all group ID numbers for the current user, enter:

```bash
id -G
```
Output is displayed in the following format:
0 10 300 9

The -G flag writes only the group IDs for a user. In this example, user sah is a member of the system (0), audit (10), build (300), and printq (9) groups.

3. To display all group names for the current user, enter:
   id -Gn

Output is displayed in the following format:
system audit build printq

The -n flag writes only the names instead of the ID numbers.

4. To display the real group name for the current user, enter:
   id -gnr

Output is displayed in the following format:
build

Files
/usr/bin/id Contains the id command.

Related Information
The getty command, login command, setgroups command, su command, tsm command.

**ifconfig Command**

**Purpose**
Configures or displays network interface parameters for a network using TCP/IP.

**Syntax**

```plaintext
ifconfig Interface [ AddressFamily ] [ Address [ DestinationAddress ] ] [ Parameters... ]
ifconfig Interface [ ProtocolFamily ] Interface ProtocolFamily
ifconfig -a [ -I ] [ -d ] [ -u ] [ ProtocolFamily ]
```

**Description**

You can use the ifconfig command to assign an address to a network interface and to configure or display the current network interface configuration information. The ifconfig command must be used at system startup to define the network address of each interface present on a machine. After system startup, it can also be used to redefine an interfaces address and its other operating parameters. The network interface configuration is held on the running system and must be reset at each system restart. The ifconfig command interprets the IFF_MULTICAST flag and prints its value if it is set.

An interface can receive transmissions in differing protocols, each of which may require separate naming schemes. It is necessary to specify the AddressFamily parameter, which may change the interpretation of the remaining parameters. The address families currently supported are inet, inet6, and ns.
For the DARPA-Internet family, **inet**, the address is either a host name present in the host name database, that is, the `/etc/hosts` file, or a DARPA-Internet address expressed in the Internet standard dotted decimal notation.

For the Xerox Network Systems (XNS) family, **ns**, addresses are `net:a.b.c.d.e.f`, where `net` is the assigned network number (in decimals), and each of the 6 bytes of the host number, a through f, are specified in hexadecimal. The host number may be omitted on 10-Mbps Ethernet interfaces, which use the hardware physical address, and on interfaces other than the first interface.

While any user can query the status of a network interface, only a user who has administrative authority can modify the configuration of those interfaces.

The **ifconfig** function displays the current configuration for a network interface when no optional parameters are supplied.

If a protocol family is specified, **ifconfig** will report only the details specific to that protocol family.

Only a super user may modify the configuration of a network interface.

Gratuitous ARP is supported for Ethernet, token-ring, and FDDI interfaces. This means when an IP address is assigned, the host sends an ARP request for its own address (the new address) to inform other machines of its address so that they can update their ARP entry immediately. It also lets hosts detect duplicate IP address. If you get a response to the ARP request, an error is logged in `/var/adm/ras/errlog` which can be viewed using **errpt** command (or using SMIT interface) for the error ID **AIXIF_ARP_DUP_ADDR**.

**Flags**

- **-a**
  Optionally, the `-a` flag may be used instead of an interface name. This flag instructs **ifconfig** to display information about all interfaces in the system.

- **-d**
  The `-d` flag displays interfaces that are down.

- **-i**
  This flag may be used to list all available interfaces on the system, with no other additional information. Use of this flag is mutually exclusive with all other flags and commands, except for `-d` and `-u`.

- **-u**
  The `-u` flag displays interfaces that are up.

**ProtocolFamily**

This flag specifies protocols such as **tcp**, **udp**, **tcp6**, **udp6**, **icmp**, and **icmp6**.

**Parameters**

**Address**

Specifies the network address for the network interface. For the **inet** family, the `Address` parameter is either a host name or an Internet address in the standard dotted decimal notation.

**AddressFamily**

Specifies which network address family to change. The **inet**, **inet6**, and **ns** address families are currently supported. This parameter defaults to the **inet** address family.

**DestinationAddress**

Specifies the address of the correspondent on the remote end of a point-to-point link.
**Interface**

Specifies the network interface configuration values to show or change. You must specify an interface with the `Interface` parameter when you use the `ifconfig` command. Abbreviations for the interfaces include:

- `at` for ATM (Asynchronous Transfer Mode)
- `en` for Standard Ethernet (inet, xns)
- `et` for IEEE 802.3 Ethernet (inet, xns)
- `tr` for Token-Ring (inet, xns)
- `xt` for X.25 (inet)
- `sl` for serial line IP (inet)
- `lo` for loopback (inet)
- `op` for serial (inet)
- `vi` for Virtual IP Address (inet)

Include a numeral after the abbreviation to identify the specific interface (for example, `tr0`).

If `Interface` is not yet loaded, `ifconfig Interface` loads that interface and `netstat -in` lists it. In processing a status query for `Interface`, that interface is loaded (if not already loaded) to complete the query processing.

**Parameter**

Allows the following parameter values:

- **alias**
  Establishes an additional network address for the interface. When changing network numbers, this parameter is useful for accepting packets addressed to the old interface.

- **allcast**
  Sets the Token-Ring interface to broadcast to all rings on the network.

- **-allcast**
  Confines the Token-Ring interface to broadcast only to the local ring.

- **arp**
  Enables the `ifconfig` command to use the Address Resolution Protocol (ARP) in mapping between network-level addresses and link-level addresses. The `arp` value is the default.

- **-arp**
  Disables the use of the Address Resolution Protocol.

- **authority**
  Reserved for future use.

- **bridge**
  Reserved for future use.

- **-bridge**
  Reserved for future use.

- **broadcast Address**
  (inet only) Specifies the address to use to broadcast to the network. The default broadcast address has a host part of all 1s.

- **-dad**
  (inet6 only) Does not perform duplicate IPv6 address detection.

- **-debug**
  Disables driver-dependent debug code.

- **delete**
  Removes the specified network address. This is used when an alias is incorrectly specified or when it is no longer needed. Incorrectly setting an `ns` address has the side effect of specifying the host portion of the network address. Removing all `ns` addresses allows you to re-specify the host portion.

- **device dev_name**
  This parameter applies to ATM Network interface only. It specifies the device name this interface is associated with. Unlike Token Ring or Ethernet, in case of ATM, there is not a one-to-one correspondence between interface and device. In the case of ATM, there can be more than one interface for every device.
detach  Removes an interface from the network interface list. If the last interface is detached, the network interface driver code is unloaded. In order for the interface route of an attached interface to be changed, that interface must be detached and added again with `ifconfig`.

down  Marks an interface as inactive (down), which keeps the system from trying to transmit messages through that interface. If possible, the `ifconfig` command also resets the interface to disable reception of messages. Routes that use the interface, however, are not automatically disabled.

eui64 (inet6 only)  The real IPv6 address is computed by replacing the last 64 bytes of the given address with the Interface Identifier.

first  Puts an IPv6 address at the first place on an interface in order to select it as the source for unbound sockets. The syntax for using this parameter is,
`ifconfig interface inet6 first address`

firstalias (inet6 only)  Same as alias, but sets the address in front of the interface address list in order to select it as the source for unbound sockets.

group  ID  Adds a group ID to the group ID list for the interface. This list is used in determining the route to use when forwarding packets that arrived on the interface.

-group  ID  Removes a group ID from the group ID list for the interface. This list is used in determining the route to use when forwarding packets that arrived on the interface.

hwloop  Enables hardware loopback. The hardware loopback specifies that locally addressed packets handled by an interface should be sent out using the associated adapter.

-hwloop  Disables hardware loop-back. The hardware loop-back specifies that locally addressed packets handled by an interface should be sent out using the associated adapter.

ipdst  Specifies an Internet host willing to receive IP packets encapsulating ns packets bound for a remote network. An apparent point-to-point link is constructed, and the specified address is taken as the ns address and network of the destination.

ipv6dst  Used to specify an IPv6 node that is willing to receive IPv6 packets encapsulating IPv6 or IPv4 packets through a tunnel. The apparent destination of the point to point tunnel interface may not be the real destination of the packets. At the tunnel endpoint, the decapsulated packets may then be forwarded to their final destination.
link [0-2]
Enables special processing of the link level of the interface. These three options are interface-specific. In actual effect, however, they are generally used to select special modes of operation. An example of this is to enable SLIP compression, or to select the connector type for some Ethernet cards. Refer to the manual page for the specific driver for more information.

-li nk [0-2]
Disables special processing at the link level with the specified interface.

metric Number
Sets the routing metric of the interface to the value specified by the Number variable. The default is 0 (zero). The routing metric is used by the routing protocol (the routed daemon). Higher metrics have the effect of making a route less favorable. Metrics are counted as addition hops to the destination network or host.

mtu Value
Sets the maximum IP packet size for this system. The Value variable can be any number from 60 through 65535, but is media dependent. See [“Automatic Configuration of Network Interfaces”] in AIX 5L Version 5.2 System Management Guide: Communications and Networks for maximum transmission unit (MTU) values by interface.

netmask Mask
Specifies how much of the address to reserve for subdividing networks into subnetworks. This parameter can be used only with an address family of inet.

The Mask variable includes both the network part of the local address and the subnet part, which is taken from the host field of the address. The mask can be specified as a single hexadecimal number beginning with 0x, in standard Internet dotted decimal notation, or beginning with a name or alias that is listed in the [etc/networks] file.

In the 32-bit address, the mask contains 1s (ones) for the bit positions reserved for the network and subnet parts and 0s for the bit positions that specify the host. The mask should contain at least the standard network portion, and the subnet segment should be contiguous with the network segment.

pvc
This parameters applies to ATM Network interface only. It specifies that this interface will support PVC (Permanent Virtual Circuit) types of virtual connections only.

pktchain
Enables the flag to indicate that this interface can handle multiple packets chained together on the output path.

-pktchain
Disables the flag that indicates that this interface can handle multiple packets chained together on the output path.

cvc_s server_addr
This parameter applies to ATM Network interface only. It specifies that this interface will support both SVC (Switched Virtual Circuit) and PVC types of virtual connections. It further specifies that this interface will be an ARP client. The server_addr is the list of 20 byte ATM address of the ARP servers that this client will use. The addresses are specified in the form of xx.xx....xx. The first entry is considered the Primary ARP server and the rest are considered Secondary ARP servers. The list of 20 byte ARP server addresses must separated by a comma.
site6  Sets the IPv6 site number (default is zero). This should be used only with site-local addresses on a multi-sited node.

svc_s  This parameter applies to ATM Network interface only. It specifies that this interface will support both SVC and PVC types of virtual connections. It further specifies that this interface will be the ARP server for this Logical IP Subnetwork (LIS).

security
Reservded for future use.

snap   Reserved for future use.

-snap  Reserved for future use.

tcp_nocksum
Disables verification of the checksum of TCP data for local traffic to the subnet attached to the interface. Checksum verification of TCP, UDP and IP headers continues. Checksum verification of TCP data read or written from this interface from or to remote networks also continues.

-tcp_nocksum
Enables verification of the checksum of TCP data for local traffic to the subnet attached to the interface. This is the default.

thread  (inet only) Configures dedicated kernel threads for an interface. This parameter can only be used on SMP machines that have multiple CPU's. This parameter causes input packets to be queued to a kernel thread after processing by the device driver and input demuxer. The input packet is processed in IP and TCP or UDP by the thread instead of directly on the interrupt level. Setting this parameter can improve throughput when high speed adapters bottleneck on a single CPU during interrupt processing by allowing the input packets to be processed on other CPU's running the kernel threads (improved pipelining). For some work loads, this parameter increases the per packet overhead, due to thread scheduling overhead, resulting in higher CPU utilization an possibly lower throughput. This parameter only applies to AIX 4.3.3 or later.

-thread (inet only) Configures Disables kernel thread support that has been configured with the thread parameter. This parameter only applies to AIX 4.3.3 or later.

up     Marks an interface as active (up). This parameter is used automatically when setting the first address for an interface. It can also be used to enable an interface after an ifconfig down command.

vipa_iflist
Add the interfaces to the list of interfaces that should use this vipa as the source address in the outgoing packets.

-vipa_iflist
Removes the interfaces from the list of interfaces that are configured to use this vipa as the source address in the outgoing packets.

In AIX 4.3.3 and later versions, the following network options, commonly known as ISNO (Interface Specific Network Options), can be configured on a per interface basis:

rfc1323 [0 | 1]
Enables or disables TCP enhancements as specified by RFC 1323, TCP Extensions for High Performance. A value of 1 specifies that all TCP connections using this interface will attempt to
negotiate the RFC enhancements. A value of 0 disables rfc1323 for all connections using this interface. The SOCKETS application can override this ISNO and global behavior on individual TCP connections with the setsockopt subroutine.

-rfc1323
Removes the use of ISNO for rfc1323 for this network. A SOCKETS application can override the global behavior on individual TCP connections using the setsockopt subroutine.

tcp_mssdflt Number
Sets the default maximum segment size used in communicating with remote networks. If communicating over this interface, a socket uses Number as the value of the default maximum segment size.

tcp_mssdflt
Removes the use of ISNO for tcp_mssdflt. The global value, manipulated via /usr/sbin/no, is used instead.

tcp_recvspace Size
Specifies the default socket buffer Size for interface sockets receiving data. The buffer size affects the window size used by TCP. (See the no command for more information.)

tcp_recvspace
Removes the use of ISNO for tcp_recvspace. The global value is used instead.

tcp_sendspace Size
Specifies the default socket buffer Size for interface sockets sending data. The buffer size affects the window size used by TCP. (See the no command for more information.)

tcp_sendspace
Removes the use of ISNO for tcp_sendspace. The global value is used instead.

tcp_nodelay [0 | 1]
Specifies that sockets using TCP over this interface follow the Nagle algorithm when sending data. By default, TCP follows the Nagle algorithm.

tcp_nodelay
Removes the use of ISNO for the tcp_nodelay option.

Note: ISNO parameters set by ifconfig are lost at the next reboot. Use the chdev command to change the ODM database for each interface if you wish to make the ISNOs permanent. Use lsdev -E -l [interface] to see the interface attributes and chdev -l -a [attribute=value] to change the desired attribute. For example:

lsattr -E -l en0
chdev -l en0 -a tcp_sendspace=65536

Examples
The following are examples using the ifconfig command on a TCP/IP network and an XNS network:

Inet Examples
1. To query the status of a serial line IP interface, enter the command in the following format:

   ifconfig si1

   In this example, the interface to be queried is si1. The result of the command looks similar to the following:

   si1: flags=S1<UP,POINTOPOINT,RUNNING>
   inet 192.9.201.3 --> 192.9.354.7 netmask ffffff00

2. To configure the local loop-back interface, enter the command in the following format:

   ifconfig lo0 inet 127.0.0.1 up

3. To mark the local Token-Ring interface as down, enter the command in the following format:
ifconfig tr0 inet down

In this example, the interface to be marked is token0.

Note: Only a user with root user authority can modify the configuration of a network interface.

4. To turn rfc1323 off for all connections over en5 (assuming that the global value is 1), enter:
   ifconfig en0 rfc1323 0

5. To configure a list of interfaces to use a vipa, enter:
   ifconfig vi0 vipa_iflist en0,en1,tr0

6. To remove interface(s) that are configured to use vipa, enter:
   ifconfig vi0 -vipa_iflist en1,tr0

7. To find out which interfaces are configured to use a vipa, say vi0, enter:
   ifconfig vi0

XNS Examples

1. To configure a Standard Ethernet type interface for XNS, enter the command in the following format:
   ifconfig en0 ns 110:02.60.8c.2c.a4.98 up

   In this example, ns is the XNS address family, 110 is the network number and 02.60.8c.2c.a4.98 is the
   host number, which is the Ethernet address unique to each individual interface. Specify the host
   number when there are multiple Ethernet hardware interfaces, as the default may not correspond to
   the proper interface. The Ethernet address can be obtained by the commands:
   ifconfig en0
   netstat -v

   The XNS address can be represented several ways, as shown in the following examples:
   • 123#9.89.3c.90.45.56
   • 5-124#123-456-900-455-749
   • 0x45:0x9893c9045569:90
   • 0456:9893c9045569H

   The XNS address in the first example is in decimal format. The address in the second example is
   broken into groups of three digits, with each group separated by a - (minus sign). The 0x and H
   addresses are in hex format. Finally, the 0 in front of the last address indicates that the number is in
   octal format.

2. To configure an IEEE Ethernet 802.3 type interface for XNS, enter the command in the following
   format:
   ifconfig et0 ns 120:02.60.8c.2c.a4.98 up

   The et0 and en0 interfaces are considered as separate interfaces even though the same Ethernet
   adapter is used. Two separate networks can be defined and used at the same time as long as they
   have separate network numbers. Multiple Ethernet adapters are supported.

   Note: The host number should correspond to the Ethernet address on the hardware adapter. A
   system can have multiple host numbers.

3. To configure an Internet encapsulation XNS interface, enter the command in the following format:
   ifconfig en0 inet 11.0.0.1 up
   ifconfig en0 ns 110:02.60.8c.2c.a4.98 up
   ifconfig en0 ns 130:02.60.8c.34.56.78 ipdst 11.0.0.10

   The first command brings up the Internet with the inet address 11.0.0.1. The second command
   configures the en0 interface to be network 110 and host number 02.60.8c.2c.a4.98 in the ns address
   family. This defines the host number for use when the XNS packet is encapsulated within the Internet
packet. The last command defines network 130, host number 02.60.8c.34.56.78, and destination Internet address 11.0.0.10. This last entry creates a new network interface, nsip. Use the netstat -i command for information about this interface.

Files

/etc/host Contains the host-name database.
/etc/networks Contains network names.

Related Information

The netstat command.
The hosts file format, networks file format.


ike Command

Purpose

Starts, stops, and monitors IP Security dynamic tunnels which use the Internet Key Exchange Protocol (ISAKMP/Oakley).

Syntax

ike cmd:Subcommand [ parameter ... ]

Description

The ike is used to start, stop, and monitor IP Security dynamic tunnels using the Internet Key Exchange (IKE) protocol. IP Security tunnels protect IP traffic by authenticating and/or encrypting IP data. The ike command performs several functions. It can activate, remove, or list IKE and IP Security tunnels. For an overview of IP Security and IKE tunnels, see IP Security in the AIX 5L Version 5.2 Security Guide.

Note: You must have root access to use the ike command.

The IKE negotiation occurs in two phases. The first phase authenticates the two parties and sets up a Key Management (also known as phase 1) Security Association for protecting the data that is passed during the negotiation. In this phase the key management policy is used to secure the negotiation messages. The second phase negotiates Data Management (also known as the phase 2) Security Association, which uses the data management policy to set up IP Security tunnels in the kernel for encapsulating and decapsulating data packets. The secure channel established in phase 1 can be used to protect multiple data management negotiations between 2 hosts.

The ike command is used to activate tunnels with identification and policy information which has already been entered using the kedb command or the Web-based System Manager Graphical User Interface (GUI) under Virtual Private Networks (IP Security) in the Network application. The parameters to be used during the negotiation are entered by the user and stored in a database. The ike command allows the activation, removal and listing of tunnels that have been started using the security parameters stored in the database.
In most uses of the `ike` command, activation and deletion occurs for both phases, however the command allows these operations to be done separately.

Subcommands

activate

Purpose
Start the negotiation of an IKE tunnel. If phase is not specified, both a phase 1 and phase 2 tunnel are started. If IP addresses are supplied, the tunnel is setup using those IP addresses. If the IDs used during the negotiation are not IP addresses, the local and remote host IDs must be entered using the Virtual Private Networks Web-based System Manager Graphical User Interface (GUI) panels. A unique tunnel number is created. The tunnel can then be referenced by the tunnel number in the `ike` command to indicate the particular tunnel to be started.

Syntax

```
ike cmd=activate [ phase=1|2 ] [ numlist=tunnel_num_list ] [ namelist=tunnel_name_list ] [ remid=remote_id ] [ ipaddr=src_addr,dst_addr ] [ autostart]
```

Description
The `activate` subcommand works using a two phase paradigm. A phase 1 tunnel must be established before a phase 2 tunnel can be started. If a phase 1 tunnel is specified, then only the phase 1 tunnel negotiation takes place. If a phase 2 tunnel is specified, the system checks for the existence of the corresponding phase 1 tunnel before creating the phase 2 tunnel. If the phase 1 negotiation has not been started, it is started automatically.

Upon successful completion of a phase 2 tunnel, the tunnel definition and corresponding filter rules are inserted into the IP Security kernel, and the new tunnel is activated. Traffic described by the tunnel definition passing between the designated endpoints is protected by the encryption and authentication algorithms indicated by the associated IKE security policy.

Multiple phase 2 tunnels can be started under the same phase 1 tunnel. A situation where this may be desired is if different types of traffic between two endpoints need different levels of security protection. The Security Association used for the phase 1 tunnel can be shared by multiple phase 2 tunnels. The phase 2 tunnels would specify the type of traffic (by protocol and port, or subnet mask, for instance) and could have different security policies protecting them.

The `ike` command returns if either a negotiation has been initiated, an error returns, or the tunnel already exists. Since the remote host must be contacted during the negotiation and the amount of time needed to complete the negotiation is uncertain, the `list` subcommand should be used to determine if the negotiation was successful.

Errors that are detected during the negotiation process can be captured by using `syslog`. 
Flags

**phase**  Specifies the type of negotiation desired. If omitted, the **activate** subcommand activates both a phase 1 and phase 2 tunnel. The **phase** flag is an optional flag.

**numlist**  Initiates the **ike** tunnel number which corresponds to the desired phase 1 or phase 2 tunnel(s) to be started. The , (comma) and - (dash) characters can be used to delimit values and indicate ranges. The **list** subcommand with the database option **db** can be used to determine the tunnel number for a particular tunnel. An example using tunnel numbers is shown below:

```bash
ike cmd=activate numlist=1,3,5-7
```

This would start tunnels 1, 3, 5, 6 and 7.

**remid**  Starts phase 1 or phase 2 tunnel(s) from the local ID to the specified remote ID. **remid** could be a phase 1 ID (such as IP address, FQDN, user FQDN and X500DN), a phase 2 ID (such as IP address, subnet and IP address range) or a group ID. The , (comma) is used to delimit the subnet id and subnet mask, and the starting and ending IP address. If **remid** is a group name, a tunnel is initiated for each group member. **remid** is an optional flag and can only be used with the **activate** subcommand. It cannot be used in conjunction with the **ipaddr**, **numlist** or **namelist** flags.

1. To activate a phase 1 tunnel to remote IP address 9.3.97.100, type:

   ```bash
   ike cmd=activate phase=1 remid=9.3.97.100
   ```

2. To activate a phase 2 tunnel to remote subnet ID 9.3.97.100,255.255.255.0, type:

   ```bash
   ike cmd=activate phase=2 remid=9.3.97.100,255.255.255.0
   ```

**ip_addr**  Starts a phase 1 or phase 2 tunnel between the specified IP Addresses.

**autostart**  Causes the activation of all phase 1 and phase 2 tunnel database entries which were created with the **autostart** parameter set. The **autostart** flag does not work in conjunction with any other flags pertaining to the **activate** subcommand.

**namelist**  Specifies a tunnel name or comma-separated list of tunnel names to be activated. This flag requires the use of the **phase** flag.

Examples

1. To activate a phase 2 tunnel between source IP address x.x.x.x and destination IP address y.y.y.y, enter:

   ```bash
   ike cmd=activate phase=2 ipaddr=x.x.x.x,y.y.y.y
   ```

   The security policy indicated in the database for the IP addresses x.x.x.x and y.y.y.y is used for activating the tunnel.

2. To activate phase 1 tunnels for tunnels 1 and 2, enter:

   ```bash
   ike cmd=activate phase=1 numlist=1,2
   ```

3. To activate phase 2 tunnels for inactive tunnels named AIXFW1_DM and remote_office in the database enter:

   ```bash
   ike cmd=activate phase=2 namelist=AIXFW1_DM,remote_office
   ```

   **Note:** Because each phase 2 tunnel must have an associated phase 1 tunnel, a phase 1 tunnel is automatically activated before the phase 2 tunnel is activated.

**list**

**Purpose**  Monitors the status of IP Security tunnels by phase. It is also used to view tunnel entries defined in the IKE database.

**Syntax**  

```bash
ike cmd=list [phase=1|1+|2] [numlist= tunnel_num_list] [db | role=iir] [verbose]
```
The list subcommand queries the Tunnel Manager and lists phase 1 and phase 2 tunnel status and information according to the result of the query. This command can also be used to view information in the Tunnel Definition database. The default behavior is to list the tunnels currently active. To list the tunnels in the database, the db option must be used.

**Flags**

<table>
<thead>
<tr>
<th>Flag</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>phase</td>
<td>Indicates the type and order of the tunnel(s) to be listed. A phase value of 1 results in only the requested phase 1 tunnel information being displayed. A phase value of 2 results in the information for the requested phase 2 tunnel(s) and their associated phase 1 tunnel(s) should be displayed. A phase value of 1+ means that the requested phase 1 tunnel and all associated phase 2 tunnels should be displayed. The default phase value is 1+.</td>
</tr>
<tr>
<td>numlist</td>
<td>Lists of the tunnel numbers which you would like to view. If omitted, the information from all tunnels is displayed. The, (comma) and - (dash) characters can be used to delimit values and indicate ranges. For example: ike cmd=list numlist=1,3,5-7</td>
</tr>
<tr>
<td>db</td>
<td>Shows the entries in the database. If this flag is omitted, only active tunnels are displayed. This cannot be used in conjunction with role. Supply the list of tunnel numbers which you would like to view.</td>
</tr>
<tr>
<td>role</td>
<td>Allows the display of tunnels by the point of initiation. If i is specified, then the tunnels that were initiated by the local host are displayed. If r is specified, then the tunnels where the local host acted as a responder are displayed. If this flag is omitted, both initiator and responder tunnels are shown. This flag cannot be used in conjunction with db.</td>
</tr>
<tr>
<td>verbose</td>
<td>Shows extended information about the specified tunnels. If this flag is not specified, then only a concise entry for each tunnel is shown.</td>
</tr>
</tbody>
</table>

**Examples**

**Note:** Tunnel numbers from the database and tunnel numbers from the tunnel manager do not necessarily reflect the same tunnel.

1. To perform a concise (short form) listing of phase 1 tunnels with entries in the tunnel manager, enter:

```bash
db ike cmd=list phase=1 numlist=1,2,3
```

These tunnels are either being negotiated, in the active state, or have expired. Only tunnels 1, 2, and 3 are listed. Tunnels can be either initiator or responder role.

2. To perform a concise (short form) listing of all the specified phase 2 tunnels in the database with each preceded by the associated phase 1 tunnel, enter:

```bash
db ike cmd=list phase=2 numlist=1-3 db
```

These are tunnels defined in the database which may or may not be currently active in the tunnel manager. All tunnels in the database are used in the initiator role only.

3. To perform a verbose (long form) listing of a phase 1 tunnel followed by all of its associated phase 2 tunnels from the tunnel manager, enter:

```bash
db ike cmd=list phase=1+ role=r verbose
```

Only tunnels which were activated in the responder role are listed. All available tunnel numbers are listed since no numlist was specified.

The remove subcommand deactivates the specified phase 1 or phase 2 tunnel(s).

**Syntax**

```bash
ike cmd=remove [phase=1|2] [numlist= tunnel_num_list] [all]
```
The `remove` subcommand requests the deactivation of phase 1 or phase 2 tunnel(s). Because phase 2 tunnels are associated with a phase 1 tunnel, if a phase 1 tunnel is deactivated, all phase 2 tunnels under the phase 1 tunnel are not refreshed when the phase 2 tunnel lifetime expires.

**Flags**

- **phase** Indicates the phase of the tunnel to be deactivated and must be specified. A phase value of 1 refers to a phase 1 tunnel and a phase value of 2 refers to a phase 2 tunnel.

- **numlist** Lists the tunnel numbers you would like to deactivate. The , (comma) and - (dash) characters can be used to delimit values and indicate ranges. For example:

  ```
  ike cmd=remove numlist=1,3,5-7
  ```

  When **numlist** is omitted, all tunnels are deactivated.

**Examples**

1. To deactivate phase 1 tunnels numbered 1, 2, and 3, enter:
   ```
   ike cmd=remove phase=1 numlist=1-3
   ```
2. To deactivate all phase 1 and phase 2 tunnels, enter:
   ```
   ike cmd=remove all
   ```
3. To deactivate all phase 2 tunnels but keep all phase 1 tunnels active, enter:
   ```
   ike cmd=remove phase=2 all
   ```
4. To deactivate all phase 1 tunnels (corresponding phase 2 tunnels will not be refreshed), enter:
   ```
   ike cmd=remove phase=1 all
   ```

**log**

**Purpose**

Read the ISAKMP daemon log level from `/etc/isamkpdp.conf` and start logging at that level.

**Syntax**

```
ike cmd=log
```

**Note:** If the log level or the output file name in `/etc/syslog.conf` are changed, the `refresh -s syslogd` command must also be run.

**Description**

The `log` subcommand causes the ISAKMP daemon to read the log level from `/etc/isamkpdp.conf`, and a filename from `/etc/syslog.conf`. The logging level specified is set and the log output, along with other `syslog` output, is placed in the file specified.

**Note:** There are four valid logging levels for the ISAKMP daemon. They are **none**, **errors**, **events**, and **information**. **none** means no logging, **errors** means logging of only ISAKMP daemon errors will occur, **events** means errors and other ISAKMP daemon events will be logged, and **information** is the highest level of logging which is all inclusive.

**Files**

- `/usr/sbin/ike` Location of the `ike` admin commands.
- `/etc/isamkpdp.conf` Configuration file for the `iksakmpd` daemon.
- `/etc/syslog.conf` Provides configuration information for the `syslogd` daemon.

**Related Information**

The `syslog` subroutine.

The `syslog.conf` file.

The `syslogd` daemon.
The **ikedb** command.

# ikedb Command

## Purpose
Retrieves, updates, deletes, imports, and exports information in the IKE database.

## Syntax

```bash
ikedb [-p F s] [ -e entity-file ] [ XML-file ]
```

```bash
ikedb [-g r] [ -t type ] [ -n name ] [ -i ID -y ID-type ] ]
```

```bash
ikedb [-d] [ -t type ] [ -n name ] [ -i ID -y ID-type ] ]
```

```bash
ikedb [-c F] [ -l linux-file ] [ -k secrets-file ] [ -f XML-file ]
```

```bash
ikedb -x
```

```bash
ikedb -o
```

## Description
The **ikedb** command allows the user to write to (put) or read from (get) the IKE database. The input and output format is an Extensible Markup Language (XML) file. The format of an XML file is specified by its Document Type Definition (DTD). The **ikedb** command allows the user to see the DTD that is used to validate the XML file when doing a put. While entity declarations can be added to the DTD using the -e flag, this is the only modification to the DTD that can be made.

Any external DOCTYPE declaration in the input XML file will be ignored and any internal DOCTYPE declaration might result in an error. The rules followed to parse the XML file using the DTD are specified in the XML standard. **/usr/samples/ipsec** has a sample of what a typical XML file that defines common tunnel scenarios looks like.

## Flags

- **-p** Performs a put, which writes to the database, based on the given **XML-file**.

- **-F** Forces a put, even if a specified tunnel, protection, proposal, group, or pre-shared key would overwrite one that already exists in the database. The default is for such put attempts to fail.

- **-s** Swaps the local and remote IDs of all tunnels. This facilitates importing a tunnel generated by a peer machine. This flag only affects tunnels. This option is illegal if the remote ID of any tunnel is a group.

- **-e entity-file**
  Specifies the name of the file containing the `<ENTITY ...>` lines as defined by **entity-file**. These lines are added to the internal DTD and allow the user to include XML files in other XML files.

- **XML-file**
  Specifies the XML-file to be used and must be the last argument to be displayed in the command line. The XML-file determines whether the write is to a tunnel, protection, proposal, group, pre-shared key, or all of these. If no XML-file is specified, input is read from stdin. A - (hyphen) can also be used to specify stdin.
-g Performs a get, which displays what is stored in the IKE database. Output is sent to stdout and is in XML format, which is suitable for processing with ikedb -p.

-r Recursive. If this flag is specified for a phase 1 tunnel, information is also returned for all associated phase 2 tunnels and all protections and proposals associated with both sets of tunnels.

-t type Specifies the type of output requested. Type can have the value of any of the XML elements under AIX_VPN, such as IKETunnel, IPSecProtection, and so on. If omitted, the entire database is output.

-n name Specifies the name of the requested object. Name can be the name of a proposal, protection, tunnel, or group, depending on the value of the -t flag. The -n flag is valid with all values specified by the -t flag, except IKEPresharedKey. If omitted, all objects of the specified type will be output.

-i ID Specifies the ID associated with a pre-shared key. The -i flag is only valid with the IKEPresharedKey value of the -t flag. If omitted, all objects of the specified type will be output. The -i flag must be used in conjunction with the -y flag.

-y ID-type Specifies the ID-type defined by the -i flag. ID-type can be any of the legal types allowed in the XML file, such as User_FQDN, IPV4_Address, and so on. The -y flag must be used in conjunction with the -i flag.

-d Performs a delete on the specified item from the database. The flags are the same as for the -g flag, except that -r is not supported.

-c Performs a conversion from a Linux IPSec configuration file to an AIX IPSec configuration file in XML format. It requires as input one or two files from Linux, a configuration file, and possibly a secrets file with pre-shared keys.

-F Forces a put, even if a specified tunnel, protection, proposal, group, or pre-shared key would overwrite one that already exists in the database. The default is for such put attempts to fail. The -F flag has no effect if the -f flag is also used.

-l linux-file Specifies the Linux configuration file as defined by linux-file. If no file is specified, the system looks for the ipsec.conf file in the current directory.

-k secrets-file Specifies the Linux pre-shared keys file as defined by the secrets-file parameter. If no file is specified, the system looks for the ipsec.secrets file in the current directory.

-f XML-file Specifies the XML configuration file to which the Linux configuration files are converted. The default behavior is to do a put directly to the IKE database. If the filename given is a hyphen (-), the results are sent to stdout.

-x Performs an expunge on the database. This empties out the database.

-o Performs an output of the DTD that specifies all elements and attributes for an XML file that is used by the ikedb command. The DTD is sent to stdout.

Files

/usr(samples/ipsec) Examples of an XML file that sets up various tunnel configurations.

Examples

1. To put definitions to the IKE database from an XML file that has been generated on a peer machine and overwrite any existing objects in the database with the same name, type:

   ikedb -pS peer_tunnel_conf.xml

   peer_tunnel_conf.xml is the XML file generated on a peer machine.
2. To get the definition of the phase 1 tunnel named tunnel_sys1_and_sys2 and all dependent phase 2 tunnels with respective proposals and protections, type:
   
   \texttt{ikedb -gr -t IKETunnel -n tunnel_sys1_and_sys2}

3. To delete all preshared keys from the database, type:

   \texttt{ikedb -d -t IKEPresharedKey}

**imake Command**

**Purpose**
C preprocessor interface to the \texttt{make} command.

**Syntax**


**Description**

The \texttt{imake} command generates \texttt{Makefiles} from a template, a set of cpp macro functions, and a per-directory input file called \texttt{Imakefile}. This command keeps machine dependencies (such as compiler options, alternate command names, and special \texttt{make} command rules) separate from the descriptions of the items to build.

\texttt{imake} invokes \texttt{cpp} with any [-I] or [-D] flags passed on the command line and passes to it the following three lines:

```
#define IMAKE_TEMPLATE "Imake.tmpl"
#define INCLUDE_MAKEFILE "Imakefile"
#include IMAKE_TEMPLATE
```

Override \texttt{Imake.tmpl} and \texttt{Imakefile} by using the [-T] and [-f] flags, respectively.

The IMAKE TEMPLATE typically reads the following files:

- A machine-dependent parameters file in which the parameters are specified as cpp symbols
- A site-specific parameters file
- A file that defines variables
- A file containing cpp macro functions for generating \texttt{make} command rules
- The \texttt{Imakefile} (specified by INCLUDE\_MAKEFILE) in the current directory.

The \texttt{Imakefile} file uses the macro functions to indicate what targets to build and the \texttt{imake} command generates the appropriate rules.

Imake configuration files contain two types of variables, imake variables and make variables. The imake variables are interpreted by \texttt{cpp} when the \texttt{imake} command is run. By convention, they are not case-sensitive. The make variables are written into the \texttt{Makefile} for later interpretation by the \texttt{make} command. By convention, make variables are uppercase.

The rules file (usually named \texttt{Imake.rules} in the configuration directory) contains a variety of cpp macro functions that are configured according to the current platform. The \texttt{imake} command replaces any occurrences of the string \texttt{``@@''} with a newline (carriage return) to support macros that generate more than one line of make rules. For example, the macro:

```
#define program_target(program, objlist) @@
program: objlist
    $(CC) -o $@ objlist $(LDFLAGS)
```

when called with \texttt{program_target(foo,foo1.o foo2.o)} will expand to:
foo: foo1.o foo2.o
   $(CC) -o $@ foo1.o foo2.o $(LDFLAGS)

On systems whose cpp reduces multiple tabs and spaces to a single space, the imake command attempts
to put back any necessary tabs (the make command distinguishes between tabs and spaces). For this
reason, precede all colons (:) in command lines by a backslash (\).

Use with AIXwindows

AIXwindows uses the imake command extensively for both full builds within the source tree and builds of
external software. Two special variables, TOPDIR and CURDIR, are set to make referencing files using
relative path names easier. For example, the following command is generated automatically to build the
Makefile in the lib/X directory (relative to the top of the sources):
%  ../../../config/imake -I ../../../config
   -DTOPDIR=../../../. -DCURDIR=./lib/X

To build AIXwindows programs outside the source tree, a special symbol, UseInstalled, is defined and the
TOPDIR and CURDIR variables are omitted. If the configuration files are properly installed, you can use the
xmkmf command.

The imake command reads the following files as used by AIXwindows.

Note: The indented format indicates files that include other files.

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Imake.tmpl</td>
<td>generic variables</td>
</tr>
<tr>
<td>site.def</td>
<td>site-specific, BeforeVendorCF defined</td>
</tr>
<tr>
<td>*.cf</td>
<td>machine-specific</td>
</tr>
<tr>
<td>*Lib.rules</td>
<td>shared library</td>
</tr>
<tr>
<td>Imake.rules</td>
<td>rules</td>
</tr>
<tr>
<td>Project.tmpl</td>
<td>X-specific variables</td>
</tr>
<tr>
<td>*Lib.tmpl</td>
<td>shared library variables</td>
</tr>
<tr>
<td>Imakefile</td>
<td>library rules</td>
</tr>
<tr>
<td>Library.tmpl</td>
<td>server rules</td>
</tr>
<tr>
<td>Server.tmpl</td>
<td>multi-thread rules</td>
</tr>
</tbody>
</table>

Note: The site.def file is included twice, both before and after the *.cf file. Although most site
customizations are specified after the *.cf file, some, such as the choice of compiler, need to be
specified before, because other variable settings may depend on them.

The first time the site.def file is included, the BeforeVendorCF variable is defined, and the second
time, the AfterVendorCF variable is defined. All code in the site.def file should be placed inside a
ifdef macro for one of these symbols.

Flags

-DDefine Passed directly to cpp to set directory-specific variables. For example, X-windows uses this flag
to set the TOPDIR variable to the name of the directory containing the top of the core
distribution, and the CURDIR variable to the name of the current directory, relative to the top.

-e Indicates that the imake command should execute the generated Makefile. The default is to
leave this to the user.

-I FileName Specifies the name of the per-directory input file. The default is the Imakefile file.

-IDirectory (Uppercase i) Passed directly to cpp to indicate the directory in which the imake template and
configuration files are located.

-C FileName Specifies the name of the .c file that is constructed in the current directory. The default is
imakefile.c.

-s FileName Specifies the name of the make description file to be generated, without invoking the make
command. If the FileName variable is a - (dash), the output is written to stdout. The default is
to generate, but not execute, a Makefile.
-TTemplate  Specifies the name of the master template file (which is usually located in the directory specified with -I) used by the cpp command. The default is the Imake.tmpl.

-v  Indicates that make should print the cpp command line that it is using to generate the Makefile.

Environment Variables

Note: The following environment variables may be set, but their use is not recommended because they introduce dependencies that are not readily apparent when the make command is run.

IMAKEINCLUDE  If defined, specifies an include argument for the C preprocessor. For example:
- I /usr/include/local

IMAKECPP  If defined, specifies a valid path to a preprocessor program. For example:
/usr/local/cpp

The default is the /lib/cpp program.

IMAKEMAKE  Specifies a valid path to a make program such as /usr/local/make. By default, make uses whatever make program is found using the execvp subroutine. This variable is only used if the -e flag is specified.

Example

imake - I /usr/lib/X11/config - DTOPDIR=/usr/lpp/X11/Xamples

Files

/usr/tmp/tmp-imake.nnnnnn  Specifies the temporary input file for the cpp preprocessor.
/usr/tmp/tmp-make.nnnnnn  Specifies the temporary input file for make.
/lib/cpp  The default C preprocessor.

Related Information

The make command, xmkmf command.

imapd Daemon

Purpose

Starts the Internet Message Access Protocol (IMAP) server process.

Syntax

imapd [-c]

Description

The imapd command is an IMAP4 server. It supports the IMAP4 remote mail access protocol. Also, it accepts commands on its standard input and responds on its standard output. You normally invoke the imapd command with the inetd daemon with those descriptors attached to a remote client connection.

The imapd command works with the existing mail infrastructure consisting of sendmail and bellmail.

Flags

-c  Suppresses the reverse host name lookup.
Exit Status
All error and status information is written to a logfile if syslogd is configured for logging.

Security
Access Control: You must have root authority to run this command.

Auditing Events: N/A

Files
/usr/sbin/imapd
/etc/services

Contains the imapd command.

Specifies the file with port assignments for required services. The following entry must be in this file:
imap 143/tcp # Internet Mail Access Protocol

Related Information
The pop3d daemon.

impfilt Command

Purpose
Imports filter rules from an export file.

Syntax
impfilt [-v46] [-f directory] [-l filt_id_list]

Description
Use the impfilt command to import filter rules from text export file(s) that are generated by the expfilt command.

Flags
-v IP version of the rules to be imported. The value of 4 specifies IP version 4 and the value of 6 specifies IP version 6. When this flag is not used, both IP version 4 and IP version 6 are imported.
-f Specifies the directory where the imported text files are to be read.
-l Lists the IDs of the filter rules to be imported. The filter rule IDs can be separated by ",". If this flag is not used, all filter rules for the applicable IP version(s) in the text export files will be imported.

Related Information
The expfilt command.

importvg Command

Purpose
Imports a new volume group definition from a set of physical volumes.
Syntax

importvg [\texttt{V MajorNumber }] [\texttt{V VolumeGroup }] [\texttt{f}] [\texttt{c}] [\texttt{x}] [\texttt{l}] [\texttt{L VolumeGroup }] [\texttt{-n}] [\texttt{-f}] [\texttt{-R}] [\texttt{-}]

PhysicalVolume

Description

\textbf{Attention:} When you issue the \texttt{importvg} command to a previously defined volume group, the QUORUM and AUTO ON values will be reset to volume group default values. You should verify the parameters of the newly imported volume group with the \texttt{lsvg} command and change any values with the \texttt{chvg} command.

The \texttt{importvg} command makes the previously exported volume group known to the system. The \texttt{PhysicalVolume} parameter specifies only one physical volume to identify the volume group; any remaining physical volumes (those belonging to the same volume group) are found by the \texttt{importvg} command and included in the import. An imported volume group is automatically varied unless the volume group is Concurrent Capable. You must use the \texttt{varyonvg} command to activate Concurrent Capable volume groups before you access them.

When a volume group with file systems is imported, the \texttt{/etc/filesystems} file is updated with values for the new logical volumes and mount points. After importing the volume group and activating it with the \texttt{varyonvg} command, you must run the \texttt{fsck} command before the file systems can be mounted. However, the mount point information would be missing from the LVCB (logical volume control block) if it is longer than 128 characters. In this case, the \texttt{importvg} command will not be able to update the \texttt{/etc/filesystems} file with the stanza for the newly imported logical volume. You should manually edit the \texttt{/etc/filesystems} file to add a new stanza for this logical volume.

The \texttt{importvg} command changes the name of a logical volume if the name already exists in the system. It prints a message and the new name to standard error, and updates the \texttt{/etc/filesystems} file to include the new logical volume name.

Notes:

1. To use this command, you must either have root user authority or be a member of the \texttt{system} group.
2. AIX Version 4 changed the behavior of \texttt{importvg} so that as part of the \texttt{importvg} process, the volume group is automatically varied on by the system after it is imported. However, if the volume group is Concurrent Capable then the \texttt{importvg} command prompts you to \texttt{varyonvg} the imported volume group manually.
3. A volume group with a mirrored striped logical volume cannot be back ported into a version older than AIX 4.3.3.

You can use the Volumes application in Web-based System Manager (wsm) to change volume characteristics. You could also use the System Management Interface Tool (SMIT) \texttt{smit importvg} fast path to run this command.

Flags

- \texttt{-c} \hspace{1cm} This flag is ignored. On AIX 5.2 and higher only Enhanced Concurrent Capable volume groups will be created.
- \texttt{-f} \hspace{1cm} Forces the volume group to be varied online.
-L VolumeGroup Takes a volume group and learns about possible changes performed to that volume group. Any new logical volumes created as a result of this command emulate the  

owner, group identification, and permissions of the /dev special file for the volume group listed in the -y flag. The -L flag performs the functional equivalent of the -F and -n flags during execution.

Restrictions:
1. The volume group must not be in an active state on the system executing the -L flag.
2. The volume group’s disks must be unlocked on all systems that have the volume group varied on and operational. Volume groups and their disks may be unlocked, remain active and used via the varyonvg -b -u command.
3. The physical volume name provided must be of a good and known state, the disk named may not be in the missing or removed state.
4. If an active node has both added AND deleted logical volumes on the volume group, the -L flag may produce inconsistent results. The -L flag should be used after each addition or deletion, rather than being deferred until after a sequence of changes.
5. If a logical volume name clash is detected, the command will fail. Unlike the basic importvg actions, clashing logical volume names will not be renamed.

-F Provides a fast version of importvg that checks the Volume Group Descriptor Areas of only the disks that are members of the same volume group. As a result, if a user exercises this flag, they must ensure that all physical volumes in the volume group are in a good and known state. If this flag is used on a volume group where a disk may be in missing or removed state, the command may fail or the results may be inconsistent.

-I Causes the importvg command to fail if imfs fails.

-n Causes the volume not to be varied at the completion of the volume group import into the system.

-R Restores the ownership, group ID, and permissions of the logical volume special device files. These values will be restored only if they were set using U, G and P flags of mklv and chlv commands. This flag is applicable only for big vg format volume groups only.

-V MajorNumber Specifies the major number of the imported volume group.

-x This flag is ignored. On AIX 5.2 and higher only Enhanced Concurrent Capable volume groups will be created.

Attention: This entry must be added after the entry used to initiate srcmstr.

-y VolumeGroup Specifies the name to use for the new volume group. If this flag is not used, the system automatically generates a new name.

The volume group name can only contain the following characters: “A” through “Z,” “a” through “z,” “0” through “9,” or “_” (the underscore), “.” (the minus sign), or “.” (the period). All other characters are considered invalid.

Examples
1. To import the volume group bkvg from physical volume hdisk07, enter:

   importvg -y bkvg hdisk07

   The volume group bkvg is made known to the system.

2. To use the -L on a multi-tailed system:

   Node A has the volume group datavg varied on.
   Node B is aware of datavg, but it is not varied on.
   Node A: varyonvg -b -u datavg
   Node B: importvg -L datavg hdisk07
   Node A: varyonvg datavg
Files

/usr/sbin Directory where the `importvg` command resides.
/tmp Directory where the temporary files are stored while the command is running.

Related Information

The `exportvg` command, `varyonvg` command.

The **Logical Volume Storage Overview** in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices* explains the Logical Volume Manager, physical volumes, logical volumes, volume groups, organization, ensuring data integrity, and allocation characteristics.

For information on installing the Web-based System Manager, see *Chapter 2: Installation and System Requirements* in *AIX 5L Version 5.2 Web-based System Manager Administration Guide*.

The **System Management Interface Tool** in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices* explains the structure, main menus, and tasks that are done with SMIT.

imptun Command

Purpose

Adds the exported tunnel definitions and optional user-defined filter rules associated with the tunnels to the local host.

Syntax

`imptun [ directory [ tunnel_id_list ] ] [-v 4 | 6 ] [-n ] [-g ] [-t ] [ -l ] [ -m ]`

Description

Use the `imptun` command to add exported tunnel definitions and optional user-defined filter rules associated with the exported tunnels (files generated by the tunnel owner by using the `exptun` command) to the local host. This command can also import tunnel definitions from the exported files generated by the IBM firewall (SNG) product export command.

A new tunnel ID is generated by the local host when a tunnel is imported to the local tunnel table. The auto-generated filter rules associated with the tunnel also is generated automatically. Importing the exported user-defined filter rules is optional.

If the exported files are transmitted by diskette, it is assumed they will be loaded to a local file directory using a command such as `tar`, depending on the tunnel owner’s instructions.

Flags

- `-f` Specifies the directory from where the exported files will be read.
- `-g` The suppress system auto-generated filter rule flag. If the `-g` flag is not used, the `imptun` command generates two filter rules for each imported tunnel automatically. The auto-generated filter rules allow all traffic between the two end points of the tunnel to go through the tunnel. If the `-g` flag is specified, the command only imports the tunnel ibm definitions, and the user must add user-defined filter rules to use the tunnel.
- `-l` Specifies the type of the tunnel(s) you want to import. If `manual` is specified, only manual tunnel(s) are imported. `-n` and `-l` flags are mutually exclusive.
- `-n` Specifies that the export files were generated by the IBM firewall (version 2.2) tunnel export command. This flag cannot be specified with the `-v` flag. The `-n` flag is also mutually exclusive with the `-r` flag.
-r          Imports the user-defined filter rules associated with the tunnels that are being imported. To use the -r flag, it must have been specified with the `exptun` command when the exported files were generated. The -r flag is mutually exclusive with the -n flag.
-\t        Lists the set of tunnel IDs to be imported from the export files. The tunnel definitions identified by these tunnel IDs are added to the local host. If this flag is not used, all the tunnel definitions in the export files are added to the local host.
-\v        Specifies the IP version of the tunnel definitions from the exported files that you wish to import. If the -v flag is not given, then all IP version 4 and IP version 6 tunnel definitions that exist in the export files are imported.

Related Information
The `gentun`, `chtun`, `mktun`, `exptun`, command, and `lstun` command.

inc Command

Purpose
Files new mail in a folder.

Syntax
\[inc [ +Folder ] [ -noaudit ] [ -audit File ] [ -changelcur ] [ -nochangelcur ] [ -form FormFile ] [ -format String ] [ -help ] [ -file File ] [ -truncate ] [ -notruncate ] [ -nosilent ] [ -silent ] [ -width Number ]\]

Description
The inc command files incoming mail in a specified folder and outputs a list of the messages filed. A folder is a system directory. By default, the inc command removes the new messages from your mail drop and places them in the specified folder. To file new mail without deleting the mail drop, use the -notruncate flag.

If the specified folder does not exist, the inc command prompts you for permission to create it. The system creates the folder as a subdirectory of the user’s Message Handler (MH) directory. The default folder is inbox.

Note: If you do not have a Path: entry specified in your .mh_profile file, the inc command creates the folder as a subdirectory of the current directory.

Filed messages are assigned consecutive message numbers starting with the next highest number in the folder. Each new message receives the protection code specified in the Msg-Protect: entry in your .mh_profile file. If the Msg-Protect: entry does not exist, a protection code of 644 is assigned. If the Unseen-Sequence: entry exists, new messages are added to each sequence specified by the entry.

Flags
-\audit File          Copies the current date to the specified file and appends the output of the inc command to the file.
-\changelcur          Sets the first new message as the current message for the specified folder. This flag is the default.
-\file File           Files messages from the specified file instead of the user’s maildrop.
+Folder               Specifies the folder in which to place new messages. By default, the system creates a subdirectory called inbox in the user’s MH directory.
-\form FormFile        Identifies a file that contains an alternate output format for the inc command.
-\format String        Specifies a string that defines an alternate output format for the inc command.
-help

Lists the command syntax, available switches (toggles), and version information.

Note: For MH, the name of this flag must be fully spelled out.

-noaudit

Suppresses recording of information about any new messages filed. This is the default.

-nochangecur

Prevents alteration of the current message for the specified folder.

-nosilent

Prompts the user for any necessary information. This flag is the default.

-notruncate

Prevents clearing of the mailbox or file from which the inc command is taking new messages. If the -file flag is specified, the -notruncate flag is the default.

-silent

Prevents prompting by the inc command for information. This flag is useful when running the inc command in the background.

-truncate

Clears the mailbox or file from which the inc command is taking new messages. If the -file flag is not specified, the -truncate flag is the default.

-width Number

Sets the number of columns in the command output. The default is the width of the display.

Profile Entries

The following entries are entered in the UserMhDirectory/.mh_profile file:

Alternate-Mailboxes: Specifies alternate mailboxes.
Folder-Protect: Sets the protection level for new folder directories.
Msg-Protect: Sets the protection level for new message files.
Path: Specifies the user’s MH directory.
Unseen-Sequence: Specifies the sequences used to keep track of unseen messages.

Examples

1. To incorporate new mail into the default mail folder, inbox, enter:

   inc

   If the inbox folder exists, the system displays a message similar to the following:

   Incorporating new mail into inbox...
   65+ 04/08 jim@athena.a  Meeting  <<The meeting will
   66 04/08 jim@athena.a  Schedule  <<Schedule change

   In this example, two messages are filed in the inbox folder. The subject of the first message is Meeting, and the first line starts with the words The meeting will. The subject of the second message is Schedule, and the first line starts with the words Schedule change.

2. To incorporate new mail into a new folder called testcases, enter:

   inc testcases

   The system prompts you as follows:

   Create folder "/home/mary/testcases"?

   If you wish to create the folder, enter:

   yes

   A message similar to the following is displayed:

   Incorporating new mail into testcases...
   67+ 04/08 jim@athena.a  Meeting  <<We will begin
   68 04/08 jim@athena.a  Schedule  <<Schedule change
Files

$HOME/.mh_profile
/etc/mh/mstailor
/var/spool/mail/$USER
/usr/bin/inc

Customizes the MH user profile.
Tailors the MH environment to the local environment.
Specifies the location of the mail drop.
Contains the inc command.

Related Information
The mmail command, post command, scan command.
The mh_alias file format, mh_profile file format.


indent Command

Purpose
Reformats a C language program.

Syntax

Description
The indent command reformats a C program as specified by flags entered with the command.

If you only specify the InputFile parameter, the reformatted file is written back into the InputFile parameter and a backup copy of the InputFile parameter is written in the current directory with a .BAK filename suffix.

If you specify the OutputFile parameter, the indent command checks to make sure its name is different from the InputFile parameter.

To set up your own profile of defaults for the indent command, create a file called .indent.pro in your login directory or the current directory. In this file, include as many flags as desired, separated by spaces, tabs, or new lines.

Flags in the .indent.pro file in the current directory override those in your login directory (with the exception of -TType flags, which accumulate). If the indent command is run and a profile file exists, the profile file is read to set up the defaults of the program. Flags on the command line, however, override profile flags.

Comment Handling
The indent command assumes that any comment with a - (dash) or * (asterisk) immediately after the start of a comment marker (/*- or /**/) is a comment surrounded by asterisks. Each line of the comment is left unchanged, except for its indentation. This indentation can be adjusted to account for the change in indentation of the first line of the comment.

All other comments are treated as text. The indent command fits as many words (separated by blanks, tabs, or new-lines) on a line as possible. Blank lines break paragraphs.
A block comment is a comment that is not to the right of the code, and extends for more than one line.

If a comment is on a line with code, it is started in the comment column set by the -cn flag. Otherwise, the comment is started at n indentation levels less than where code is currently being placed, where n is specified by the -dn flag. If the code on a line extends past the comment column, the comment starts further to the right. The right margin can be extended automatically in extreme cases.

Preprocessor Lines Handling
In general, the indent command leaves preprocessor lines alone. The only reformatting it does is to straighten up trailing comments. It leaves embedded comments alone. Conditional compilation (code between #ifdef and #endif lines) is recognized and the indent command attempts to compensate correctly for the syntactic peculiarities introduced.

C Syntax Handling
The parser built into the indent command attempts to cope with incomplete and misformed syntax. In particular, the use of macros like:

```c
#define forever for(;;)
```

is handled properly. For best results, use the indent command on source that is syntactically correct.

Flags

**Note:** Flags can appear before or after file names.

- **-bad** Forces a blank line after every block of declarations.
- **-nbad** Suppresses a blank line after every block of declarations; active unless turned off with the -bad flag.
- **-bap** Forces a blank line after every procedure body.
- **-nbap** Suppresses a blank line after every procedure body; active unless turned off with the -bap flag.
- **-bbb** Forces a blank line before every block comment.
- **-nbbb** Suppresses a blank line before every block comment; active unless turned off with the -bbb flag.
- **-bc** Forces a new line after each comma in a declaration.
- **-nbc** Suppresses a new line after each comma in a declaration; active unless turned off with the -bc flag.
- **-bl** Formats compound statements, structure initializations, and enum initializations, as follows:
  ```c
  if (...) {
    code
  }
  ```
- **-br** Formats compound statements, structure initializations, and enum initializations, as follows:
  ```c
  if (...) {
    code
  }
  ```
  This flag is active unless turned off with the -bl flag.
- **-cn** Sets the initial tab position for comments on code to the n variable. The default value is 33.
- **-cdn** Sets the initial tab position for comments on declarations to the n variable. By default, this flag uses the value defined with the -c flag.
- **-cdb** Enables placing comment delimiters on blank lines; active unless turned off with the -ncdb flag. The -cdb flag affects only block comments, not comments to the right of code. Resulting comments look like the following:
  ```c
  /*
   * this is a comment
   */
  ```
- **-ncdb** Disables placing comment delimiters on blank lines. The -ncdb flag affects only block comments, not comments to the right of code. Resulting comments look like the following:
  ```c
  /* this is a comment */
  ```
- **-ce** Enables forcing else statements to follow the immediately preceding } (left bracket); active unless turned off with the -nce flag.
- **-nce** Disables forcing else statements to follow the immediately preceding } (left bracket).
-c
Indents the continuation lines \( n \) positions from the beginning of the first line of the statement. Expressions in parentheses have extra indentation added to indicate the nesting, unless the \(-lp\) flag is in effect. By default, this flag uses the value defined by the \(-i\) flag.

-cl\( n \)
Indents the case labels \( n \) positions to the right of the containing flag statement. Entering \(-cli0.5\) causes case labels to be indented half a tab stop. This option is the only one that takes a fractional argument. By default, the value is \(-cli10\).

-d\( n \)
Controls the placement of comments that are not to the right of code with the \( n \) variable. Specifying the \(-d1\) flag causes such comments to appear one indentation level to the left of code. By default, this flag uses \(-d0\) and comments are aligned with code. The location of comment lines relative to program code affects the comment indentation.

-di\( n \)
Specifies the number of positions to indent an identifier from a preceding declaration keyword with the \( n \) variable. By default, this flag uses \(-di16\).

-dj
Left-justifies declarations.

-ndj
Indents declarations; active unless turned off with the \(-dj\) flag.

-ei
Enables special else-if processing; active unless turned off with the \(-nei\) flag. The \(-ei\) flag causes if statements following else statements to have the same indentation as the preceding if statement.

-nei
Disables special else-if processing.

-fa
Flips assign operators from old style C code to the ANSI format. This flag remains active unless turned off with the \(-nfa\) flag.

Attention: The possibility of changing the meaning of the code exists if the code was meant for the ANSI compiler. For example, \( A=-B \) becomes \( A==B \).

Note: Use no spaces between operators. If the user means subtraction, then the flipping is necessary; on the other hand, if the user means \( A \) equals the negative of \( B \), the flipping alters the meaning.

-nfa
Suppresses flipping the operators. Use this flag if the code is written for an ANSI compiler.

-fc1
Enables formatting comments that start in column 1; active unless turned off with the \(-nfc1\) flag.

-nfc1
Disables formatting comments that start in column 1.

-ln
Sets the indentation level size. By default, the level size is 8 positions.

- lp
Enables indenting parameter declarations; active unless turned off with the \(-nip\) flag.

-nip
Disables indenting parameter declarations.

-Iln
Sets the maximum column position of comments that are to the right of the code. If the comment does not fit on a line, a maximum of 25 characters are printed.

- lc\( n \)
Sets the maximum line length for block comments to the \( n \) variable. By default, this flag uses the length specified with the \(-i\) flag.

-Ip
Aligns code surrounded by parentheses in continuation lines; active unless turned off with the \(-nlp\) flag. If a line has a left parenthesis with no matching right parenthesis on that line, continuation lines start at the position following the left parenthesis.

With the \(-lp\) flag in effect, such lines appear as follows:
\[
pl = \text{first\_procedure}(\text{second\_procedure}(p2,p3), \\text{third\_procedure}(p4,p5));
\]

Inserting two more new lines yields the following:
\[
pl = \text{first\_procedure}(\text{second\_procedure}(p2, p3), \\text{third\_procedure}(p4, p5));
\]

-nlp
Leaves code surrounded by parentheses in continuation lines unaligned. With the \(-nlp\) flag in effect, such lines appear as follows:
\[
pl = \text{first\_procedure}(\text{second\_procedure}(p2,p3), \\text{third\_procedure}(p4, p5));
\]

-npro
Causes the profile files \(.indent.pro\) and \$HOME/indent.pro\) to be ignored.

-pcs
Inserts a space between each procedure call name and the following \((\text{left parenthesis})\).

-npcs
Suppresses a space between each procedure call name and the following \((\text{left parenthesis})\); active unless turned off with the \(-pcs\) flag.
-ps Inserts spaces on both sides of the pointer following the -> operator.
-nps Suppresses spaces on both sides of the pointer following the -> operator; active unless turned off with the -ps flag.
-psl Left-justifies the names of procedures being defined; active unless turned off with the -npsl flag. The procedure types, if any, remain on the previous lines.
-npsl Disables left-justification of names of defined procedures.
-sc Enables the placement of * (asterisks) to the left of comments; active unless turned off with the -nsc flag.
-nsc Disables the placement of * (asterisks) to the left of comments.
-slb Treats any single-line comment that is not to the right of the code as a block comment.
-nslb Disables treating any single-line comment that is not to the right of the code as a block comment; active unless turned off with the -slb flag.
-sob Removes optional blank lines. Works in combination with any of the following flags: -nbad, -nbap, or -nbbb. Removes only blank lines that were inserted by the -bad, -bap, or -bbb flags.
-nsob Retains optional blank lines; active unless turned off with the -sob flag.
-st Causes the indent command to take its input from stdin and output to stdout.
-
 changes the _Type_ variable to the list of type keywords. Names accumulate so -T can be specified more than once. You should specify all the types appearing in your program defined by _typedef_ statements to produce the best output from the indent command.
-troff Formats the C program for processing by troff. Produces a listing similar to listings produced by the vgrind command. If no output file is specified, the default is standard output, rather than formatting in place.
-v Turns on verbose mode, which reports when one line of input is split into two or more lines of output and gives size statistics at completion.
-nv Turns off verbose mode; active unless turned off with the -v flag.

Examples

1. To format the test.c file using the default settings of the indent command and place the output into the newtest.c file, enter:

   indent test.c newtest.c

2. To format the test.c file so that a blank line is forced after every block of declarations and procedure body, use all other default settings, and store the output in the newtest.c file, enter:

   indent test.c newtest.c -bad -bap

3. To format the test.c file using the default settings of the indent command and to define uint as a type keyword recognizable to the indent command, enter:

   indent test.c newtest.c -TUint

Files

사회/indent.pro   Contains the profile file.
SHOME/indent.pro  Contains the profile file.
/usr/ccs/bin/indent Contains the indent command.

Related Information

The [cb](#) command.


indxbib Command

Purpose

Builds an inverted index for a bibliography.
Syntax

`indexbib Database ...`

Description

The `indexbib` command makes an inverted index to the named database (or files) for use by the `lookbib` and `refer` commands. These files contain bibliographic references (or other kinds of information) separated by blank lines.

**Note:** The `indexbib` command expects the database to exist in the current working directory.

A bibliographic reference is a set of lines, constituting fields of bibliographic information. Each field starts on a line beginning with a % (percent sign), followed by a key letter, then a space character, and finally the contents of the field, which can continue until the next line starting with a % (percent sign). All key letters are ASCII characters.

The `indexbib` command is a shell script that calls the `/usr/lib/refer/mkey` and `/usr/lib/refer/inv` files. The first program, `mkey`, performs the following operations:

1. Truncates words (delimited by blanks or tabs) to six characters.
2. Maps uppercase to lowercase characters.
3. Discards words shorter than three characters.
4. Discards the most commonly used words according to an existing `ign` file. An English language file, `/usr/lib/eign`, has been provided with a list of common English words. It is suggested, but not necessary, that users create their own files, named `ign`, consisting of language-specific common words. This file, if created, should exist in the `/usr/lib/nls/msg/$LANG` directory.
5. Discards numbers (dates) less than 1900 or greater than 2099.

**Note:** All dates should be indexed because many disciplines refer to literature written in the 1800s or earlier.

The second program, `inv`, creates in the working directory an entry file (.ia), a posting file (.ib), and a tag file (.ic).

Files

- `/usr/lib/eign` Contains the default list of common words the `indexbib` command discards while processing.
- `Database.ia` Contains the entry file.
- `Database.ib` Contains the posting file.
- `Database.ic` Contains the tag file.

Environment Variables

- `NLSPATH` Refers to a list of directory names where the message catalog files can be found.

Related Information

The `addbib` command, `lookbib` command, `refer` command, `roffbib` command, `sortbib` command.
inetd Daemon

Purpose
Provides Internet service management for a network.

Syntax
Note: Use SRC commands to control the inetd daemon from the command line. Use the \texttt{rc.tcpip} file to start the daemon with each system restart.

\texttt{/usr/sbin/inetd [-d] [-t SecondsToWait] [ConfigurationFile]}

Description
The \texttt{/usr/sbin/inetd} daemon provides Internet service management for a network. This daemon reduces system load by invoking other daemons only when they are needed and by providing several simple Internet services internally without invoking other daemons.

The \texttt{inetd} daemon starts by default each time you start your system. When the daemon starts, it reads its configuration information from the file specified in the \texttt{ConfigurationFile} parameter. If the parameter is not specified, the \texttt{inetd} daemon reads its configuration information from the \texttt{/etc/inetd.conf} file.

Once started, the \texttt{inetd} daemon listens for connections on certain Internet sockets in the \texttt{/etc/inetd.conf}. The \texttt{/etc/inetd.conf} file describes to the \texttt{inetd} daemon how Internet service requests on Internet sockets should be handled. When the \texttt{inetd} daemon receives a request on one of these sockets, it determines which service corresponds to that socket and then either handles the service request itself or invokes the appropriate server.

Subservers of the inetd Daemon
The \texttt{inetd} daemon (a subsystem) controls the following daemons (subservers):

- \texttt{comsat} daemon
- \texttt{ftpd} daemon
- \texttt{fingerd} daemon
- \texttt{rlogind} daemon
- \texttt{rexecd} daemon
- \texttt{rshd} daemon
- \texttt{talkd} daemon
- \texttt{telnetd} daemon
- \texttt{tftpd} daemon
- \texttt{uucpd} daemon.

The \texttt{ftpd}, \texttt{rlogind}, \texttt{rexecd}, \texttt{rshd}, \texttt{talkd}, \texttt{telnetd}, and \texttt{uucpd} daemons are started by default. The \texttt{tftpd}, \texttt{fingerd}, and \texttt{comsat} daemons are not started by default unless they are uncommented in the \texttt{/etc/inetd.conf} file.

Inetd Configuration File
The \texttt{/etc/inetd.conf} file can be updated by using the System Management Interface Tool (SMIT), the System Resource Controller (SRC), or by editing the \texttt{/etc/inetd.conf}.

If you change the \texttt{/etc/inetd.conf}, using SMIT, then the \texttt{inetd} daemon will be refreshed automatically and will read the new \texttt{/etc/inetd.conf} file. If you change the file using your favorite editor, run the \texttt{refresh -s inetd} or \texttt{kill -1 InetdPID} command to inform the \texttt{inetd} daemon of the changes to its configuration file.
The entries in the /etc/inetd.conf file include the following information:

<table>
<thead>
<tr>
<th>Service Name</th>
<th>Specifies the name of a valid Internet service.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Socket Type</td>
<td>Specifies the type of Internet socket used for the Internet service. (Only stream and datagram sockets are implemented.) Valid values are:</td>
</tr>
<tr>
<td>Protocol</td>
<td>Specifies the Internet Protocol used for the Internet service. Valid values are:</td>
</tr>
<tr>
<td>Wait/Nowait</td>
<td>Specifies whether the inetd daemon should wait for the service to complete before continuing to listen for this type of service request.</td>
</tr>
<tr>
<td>User</td>
<td>Specifies the user name that inetd should use to start the subserver.</td>
</tr>
<tr>
<td>Path</td>
<td>Specifies the fully qualified path name that inetd should execute to provide the service. For services that inetd provides internally, this entry should be internal.</td>
</tr>
<tr>
<td>Command</td>
<td>Specifies the name of the service to start and its parameters. This field is empty for internal services.</td>
</tr>
</tbody>
</table>

The `inetd` daemon can be run with or without the SRC. In addition, the inetd daemon can be controlled by issuing signals using the kill command.

### Flags

- **-d** Sends debugging messages to the `syslogd` daemon.
- **-t SecondsToWait** Specifies the number of seconds to wait in the select() system call before looping. The `SecondsToWait` can be a number from 1 to 999999. Without this flag the `inetd` daemon will block until one of the active services is requested by a network connection. This flag should only be used when a machine is servicing many wait services like `tftp` and is not being used for other services. Since timing out the select() system call will cause the `inetd` daemon to use more CPU cycles, this flag is not recommended for most situations.

### Service Requests

The Internet service requests that are supported internally by the `inetd` daemon are generally used for debugging. They include the following internal services:

- **ECHO** Returns data packets to a client host.
- **DISCARD** Discards received data packets.
- **CHARGEN** Discards received data packets and sends predefined or random data.
- **DAYTIME** Sends the current date and time in user-readable form.
- **TIME** Sends the current date and time in machine-readable form.
Related Information

The `fingerd` daemon, `ftpd` daemon, `rexecd` daemon, `rlogin` daemon, `rshd` daemon, `syslogd` daemon, `talkd` daemon, `telnetd` daemon, `ttfd` daemon.

The `inetd.conf` file format, protocols file format, services file format.

TCP/IP Daemons in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

infocmp Command

Purpose

Manages `terminfo` descriptions.

Syntax

```

Description

The `infocmp` command manages `terminfo` descriptions. You can use this command to:

- Compare a binary `terminfo` entry with other `terminfo` entries.
- Print a `terminfo` description from the binary file.
- Rewrite a `terminfo` description to take advantage of the `use` attribute.

The `infocmp` command prints the Boolean attributes first, the numeric attributes second, and the string attributes last.

Comparing Entries

Use the `-d`, `-c`, and `-n` flags to compare entries. The `-d` flag returns the differences between entries. The `-c` flag produces a list of the capabilities that are set in common between two entries. The `-n` flag returns a list of the capabilities that neither entry has.

To compare `terminfo` entries, you specify two or more `TermName` parameters. The `infocmp` command compares the `terminfo` description of the first `TermName` parameter with each of the descriptions for the subsequent `TermNames` specified. If a capability is defined for only one of the terminal descriptions, the value returned will depend on the type of capability. For Boolean capabilities the `infocmp` command returns an F, the command returns a -1 for integer capabilities, and null for string capabilities.

Producing a Source Listing

Use the `-I` (uppercase i), `-L`, `-C`, and `-r` flags to produce a source listing for one or more terminals. If you do not specify a `TermName` parameter, the system uses the `TERM` environment variable. You can use these source options to produce a source file for a `terminfo` binary when one is not available.

The I (uppercase i) flag produces a listing with the `terminfo` names. The `-L` flag produces a listing using the long `C` variable names listed in `/usr/include/term.h`.

The `-C` flag uses `termcap` names instead of `terminfo` capability names when producing the source listing. The `infocmp` commands translates and outputs only those `terminfo` capabilities that have a corresponding `termcap` code name. To remove this restriction, specifying the `-r` flag. This flag causes the command to output `terminfo` capabilities that cannot be translated into `termcap` format.

Alphabetical Listing of Commands 39
When using the -C and -r flags, the `infocmp` command notes any string parameters it was unable to convert to the `termcap` format. You must edit these parameters manually. The command collects all padding information for strings together and places it at the beginning of the string where `termcap` expects it. Mandatory padding is optional after translation. Mandatory padding is padding information with a trailing / (slash).

**Note:** The -C and -r flags cannot always convert a `terminfo` string into its equivalent `termcap` form. Similarly, a conversion from the `termcap` file format back into the `terminfo` file format does not necessarily reproduce the original source.

### Definitions with the use Attribute

Given a list of terminal menus and the -u flag, the `infocmp` command compares the first terminal’s description against the other terminal descriptions. The `infocmp` command then creates a new description for the first terminal using as much of the subsequent terminal descriptions as possible.

When you specify the -u flag and a list of terminal names, the `infocmp` command does the following:

- Compares subsequent terminal descriptions against the first.
- Creates a description of the first terminal you specified relative to the description of the other terminals.

The new description for the first terminal will have the following:

- Capabilities that exist in the subsequent terminals but do not exist for the first terminal will appear with an @ in the resulting description.

**Note:** The @ implies that the capability does not exist.

- Capabilities defined in a subsequent terminal with the same value are replaced with `use=<subsequent terminal>`.
- Any capabilities in the first terminal not found in any of the other terminals are printed along with the corresponding values.
- If the first terminal has a capability whose value differs from the value found in at least one of the other terminals, the capability is printed.

You can change a description and specify a capability after the `use` attribute. If this capability is also found in the terminal referenced by the `use` attribute, the second capability takes precedence over the one referenced by the `use` attribute.

### Changing Databases

By default, terminal descriptions appear in the system `terminfo` database directory, `/usr/share/lib/terminfo`. You can specify a different database location with the `TERMININFO` environment variable. The `infocmp` command first checks to see if this variable exists. If the variable does not exist, the command uses the system `terminfo` database.

You can use the -A and -B flag with the `infocmp` command to override the system database. The -A flag identifies the `terminfo` database for the first `TermName` parameter. The -B flag identifies the database to use for any subsequent terminals you name. Together, these flags make it possible to compare descriptions for two terminals with the same name located in two different databases.

### Flags

- **-A Directory**  
  Identifies the `terminfo` database for the first `TermName` parameter.

- **-B Directory**  
  Identifies the `terminfo` database for every `TermName` parameter except the first.

- **-C**  
  Uses the `termcap` code names to produce the source listing. Will not list `terminfo` capabilities that cannot be translated to `termcap` format.
-c  Lists the capabilities that are common between the two entries. Capabilities that are not set are ignored. This flag can be used as a quick check to see if it is desirable to use the -u flag.

-d  Lists the capabilities that are different between terminals. You can use this flag to pinpoint the differences between similar terminal entries.

-I  (uppercase i) Uses the terminfo capability names when producing the source listing.

-1  (numeral) Prints the capabilities one to a line. by default, the fields are printed several to a line to a maximum width of 60 characters.

-L  Uses the long C variable name listed in /usr/include/term.h file to produce the source listing.

-n  Compares two entries and lists the capabilities that do not exist in either. If you do not specify a TermName parameter, the system uses the TERM environment variable for both TermName parameters. You can use this as a quick check to see if anything was left out of the description.

-r  Instructs the infocomp command to output terminfo capabilities that cannot be translated to termcap format. This flag is valid only with the -C flag.

-s  Sorts the output from the infocomp command within each capability type (Boolean, numeric, and string) and according to the argument below:

  d   Sort in the order specified in the terminfo database.
  i   Sort by terminfo name.
  l   Sort by the long C variable name.
  c   Sort by the termcap name.

If you do not specify an option with the -s flag, the command sorts each capability alphabetically by the terminfo name within each type. If you specify the -C or the -L flags with the -s flag, the capabilities are sorted by the termcap name or the long C variable name, respectively.

-u  Compares two or more terminal descriptions and produces new descriptions using the use attribute.

-v  Prints out tracing information on standard error.

-V  Prints out the version of the program in use on standard error and exits.

-w Width  Changes the output to the specified number of characters per line. The output includes as many fields as possible that can fit within the specified number of characters.

  Note: Fields are not truncated.

Examples
1. To list the common capabilities between the aixterm and lft terminals, enter:
   ```bash
   infocomp -c aixterm lft
   ```
2. To list all of the capabilities that are possible but do not currently exist for the current terminal, enter:
   ```bash
   infocomp -n
   ```
3. To produce a source listing for the lft terminal in terminfo format, enter:
   ```bash
   infocomp -l lft
   ```
4. To produce a source listing for the terminal description my_term that is located in /tmp using as much of the lft description as possible, enter:
   ```bash
   infocomp -A /tmp -u my_term lft
   ```

File

/usr/share/lib/terminfo  Contents the compiled terminal description database.
Related Information

The `tic` and `captoinfo` commands.

The `terminfo` file format.

infocenter Command

Purpose

Starts the IBM eServer pSeries Information Center in a browser window.

Syntax

`infocenter`

Description

The `infocenter` command starts the IBM eServer pSeries Information Center in a browser window. The Information Center is the central point from which AIX and IBM eServer pSeries documentation, AIX FAQs, the Message Center, and How-to documentation can be accessed. The Information Center can also be started by clicking the Information Center icon on the Help panel available from the CDE toolbar.

Note: The Information Center is available only by means of an HTML browser that has access to the Internet. The `infocenter` command (and the Information Center icon) activates the browser and directs the browser to the Web address of the Information Center.

The Web address of the Information Center is in the `/usr/lpp/bosinst/bos.vendor.profile` file. If the Information Center Web address is provided in this file, it will be in the following format:

```
export INFORMATION_CENTER_URL=information center url
```

If a line similar to this does not exist in the `/usr/lpp/bosinst/bos.vendor.profile` file, or if the file does not exist, the user will receive a message indicating that the Information Center cannot start because an Information Center URL was not found.

Files

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>/usr/sbin/infocenter</code></td>
<td>Starts the Information Center in a browser window.</td>
</tr>
<tr>
<td><code>/usr/lpp/bosinst/bos.vendor.profile</code></td>
<td>Contains vendor profile information including the Web address used by the <code>infocenter</code> command.</td>
</tr>
</tbody>
</table>

install Command

Purpose

Installs a command.

Syntax

```
```

Description

The `install` command installs a specified file in a specific place within a file system. It is most often used in makefiles. When replacing files, the `install` command copies (or moves) each file into the appropriate directory, thereby retaining the original owner and permissions based on the behavior of the `cp` and `mv` commands.
commands. An attempt is made to change the destination to owner bin and group bin. The -O Owner and -G Group flags can be used to specify a different owner or group. The install command writes a message telling you exactly which files it is replacing or creating and where they are going.

You must be a super-user if you want to specify the ownership of the installed file with the -O or -G flags.

If you do not specify the Directory parameter, the install command searches a set of default directories (/usr/bin, /etc, and /usr/lib, in that order) for a file with the same name as the File parameter. The first time it finds one, it overwrites it with File and issues a message indicating that it has done so. If a match is not found, the install command issues a message telling you there was no match and exits with no further action. If the File parameter does not exist in the current directory, the install command displays an error message and exits with a nonzero value.

If any directories are specified on the command line, the install command searches them before it searches the default directories.

Flags

-**-c DirectoryA** Installs a new command file in the DirectoryA variable only if that file does not already exist there. If it finds a copy of File there, it issues a message and exits without overwriting the file. This flag can be used alone or with the -s, -M, -O, -G, or -S flag.

-**-f DirectoryB** Forces installation of File in DirectoryB whether or not File already exists. If the file being installed does not already exist, the command sets the permission code and owner of the new file to 755 and bin, respectively. This flag can be used alone or with the -o, -s, -M, -O, -G, or -S flag.

-**-G Group** Specifies a different group for the destination file. The default group is bin.

-**-i** Ignores the default directory list and searches only those directories specified on the command line. This flag cannot be used with the -c, -f, or -m flags.

-**-m** Moves the File parameter to the directory instead of being copied. Cannot be used with the -c, -f, -i, or -n flag.

-**-M Mode** Specifies the mode of the destination file.

-**-n DirectoryC** Installs the File parameter in the DirectoryC variable if it is not in any of the searched directories, and sets the permissions and owner of the file to 755 and bin, respectively. This flag cannot be used with the -c, -f, or -m flag.

-**-o** Saves the old copy of the File parameter by copying it into a file called OLDFile in the same directory. This flag cannot be used with the -c flag.

-**-O Owner** Specifies a different owner of the destination file. The default owner is bin.

-**-s** Suppresses the display of all but error messages.

-**-S** Causes the binary to be stripped after installation.

Examples

1. To replace a command that already exists in one of the default directories, enter:

   ```
   install fixit
   ```

   This replaces the fixit file if it is found in the /usr/bin, /etc, or /usr/lib directory. Otherwise, the fixit file is not installed. For example, if /usr/bin/fixit exists, then this file is replaced by a copy of the file fixit in the current directory.

2. To replace a command that already exists in a specified or default directory and to preserve the old version, enter:

   ```
   install -o fixit /etc /usr/games
   ```

   This replaces the fixit file if it is found in the /etc or /usr/games directory or in one of the default directories. Otherwise the fixit file is not installed. If the file is replaced, the old version is preserved by renaming it OLDfixit in the directory in which it was found.
3. To replace a command that already exists in a specified directory, enter:

   `install -i fixit /home/jim/bin /home/joan/bin /usr/games`

   This replaces the `fixit` file if it is found in the `/home/jim/bin`, `/home/joan/bin`, or `/usr/games` directory. Otherwise, the file is not installed.

4. To replace a command found in a default directory or install it in a specified directory if it is not found, enter:

   `install -n /usr/bin fixit`

   This replaces the `fixit` file if it is found in one of the default directories. If the file is not found, it is installed as `/usr/bin/fixit`.

5. To install a new command, enter:

   `install -c /usr/bin fixit`

   This creates a new command by installing a copy of the `fixit` file as `/usr/bin/fixit`, but only if this file does not already exist.

6. To install a command in a specified directory whether or not it already exists, enter:

   `install -f /usr/bin -o -s fixit`

   This forces the `fixit` file to be installed as `/usr/bin/fixit` whether or not it already exists. The old version, if any, is preserved by moving it to `/usr/bin/OLDfixit` (a result of the `-o` flag). The messages that tell where the new command is installed are suppressed (a result of the `-s` flag).

**Compatibility**

For compatibility with Berkeley Software Distribution (BSD), two `install` commands exist. See the `installbsd` command.

**Files**

`/usr/bin/install` Contains the `install` command.

**Related Information**

The `chgrp`, `chmod`, `chown`, `cp`, `installbsd`, `make`, `mv`, `strip` command.

---

**install_all_updates Command**

**Purpose**

Updates installed software to the latest level on media and verifies the current recommended maintenance level.

**Syntax**

```
install_all_updates [-d Device] [p] [i] [c] [r] [-n] [s] [x] [-v] [-N] [-S] [-Y] [-V] [-D]
```

**Description**

`install_all_updates` examines currently installed software and attempts to update it to the latest level that is available on the media. `install_all_updates` will not install any filesets that are present on the media,
but not installed on the system, unless these filesets are installed as requisites of other selected filesets. For installp images, all installp requisites are enforced.

Notes:
1. Currently, install_all_updates processes installp images and rpm images. Because the rpm utility does not support automatic installation of requisites, some rpm software may not be installable with install_all_updates.
2. install_all_updates verifies the current recommended maintenance level by using the "oslevel" utility and checking with the latest recommended maintenance level known to this version of install_all_updates.
3. If install_all_updates locates an update to the install utilities (the bos.rte.install files), it first installs the update and then reinvores itself to process the remaining updates. The "-l" flag can be used to update the install utilities only, this is useful when attempting to view an accurate preview.
4. install_all_updates applies all installp updates unless the COMMIT flag (-c) is specified. For more information of APPLY vs. COMMIT please see the installp man page.
5. install_all_updates will by default instruct installp to automatically install requisites and to do any necessary filesystem expansions. The "-n" will override the install requisite default, and "-x" will override the filesystem expansion default.
6. The following flags apply to installp updates only: -c, -n, -x, -v, -S, and -V.

Flags

-c Instructs installp to commit all newly installed updates. Updates are applied by default (Please see the installp man page for more explanation on applying vs. committing updates).

-d Device Specifies where the installation media can be found. This can be a hardware device such as tape or cdrom or it can be a directory that contains installation images. When installation media is a tape device it should be specified as no-rewind-on-close and no-retension-on-open.

-D Turns on install_all_updates debug output. This flag is for debugging the install_all_updates utility and should not be used for normal operations.

-i Update install utilities only.

-n Instructs installp to not automatically install requisites. Automatic installation of requisites is the default behavior.

-N Skip updating install utilities first.

Note: This flag is not recommended unless you are debugging a related problem.

-p Performs a preview of an action by running all preinstallation checks for the specified action. No software changes are made.

-r Update rpm images (if possible). This flag is not set by default.

-s Skip recommended maintenance level verification. The verification is performed by default.

-S Instructs installp to suppress multi-volume processing of cdrom media.

-v Instructs installp to verify that all installed files in the fileset have the correct checksum value after the installation. This operation may require more time to complete the installation.

-V Instructs installp to run in verbose output mode.

-x Instructs installp to not automatically expand file systems. Automatic expansion of file systems is the default.

-Y Agrees to all software license agreements which are required for software installation.

Exit Status

0 All lppmgr related operations completed successfully.

>0 An error occurred.

Security

Only the root user can execute install_all_updates.
Examples
1. To install all installp updates on device /dev/cd0 and to verify the current recommended maintenance level, enter:
   ```bash
   install_all_updates -d /dev/cd0
   ```
2. To commit install all installp updates and install any installable rpm updates in directory /images, enter:
   ```bash
   install_all_updates -d /images -rc
   ```
3. To install the latest level of install utilities on device /dev/cd0 (bos.rte.install installp fileset), enter:
   ```bash
   install_all_updates -d /dev/cd0 -1
   ```

Files
/usr/sbin/install_all_updates Contains the install_all_updates command.

Related Information
The installp command, lslpp command, rpm command

install_assist Command

Purpose
Starts the Installation Assistant application.

Syntax
install_assist

Description
The install_assist command starts Installation Assistant, an application designed to simplify the customization of your system after a Base Operating System (BOS) installation. The Installation Assistant guides you through post-installation tasks and, in some cases, automatically installs software packages for you. The Installation Assistant has two interfaces, ASCII and graphical. The interface that displays is based on your terminal type (defined in the TERM environment variable).

If your terminal type is not set, the first menu displayed by the ASCII Installation Assistant requires you to enter your terminal type (tty). If you enter a terminal type that is not valid, this menu redisplays until a valid type is entered. If you enter a valid terminal type that does not match your terminal, the next screen displayed could be unreadable. In this case, press the break key sequence to return to the Set Terminal Type screen. For most terminal types, the break key sequence is Ctrl-C.

On a system with an ASCII interface, the newly installed BOS reboots and starts the Installation Assistant to guide you through completing configuration tasks. You must have root user authority to use the Installation Assistant. To access the Installation Assistant later, type install_assist on the command line. You can also access it from a graphics system through the SMIT smit assist fast path. If there are outstanding software license agreements that must be accepted before you can continue to use the machine, the Installation Assistant prompts you to view and accept these agreements.

On a system with a graphical interface, the newly installed BOS reboots and the Configuration Assistant starts to guide you through the configuration tasks. If there are outstanding software license agreements that must be accepted before you can continue to use the machine, the Configuration Assistant prompts you to view and accept these agreements. To access the Configuration Assistant later, type configassist on the command line.
Most Installation Assistant tasks create or add to the `smit.log` and `smit.script` files in your home directory. (These are the same files appended when you run a SMIT session.) The commands built and run by the Installation Assistant tasks are added to the end of the `smit.log` file along with the command output. The time, name of the task, and the command (flags and parameters included) are added to the end of the `smit.script` file in a format that can easily be used to create executable shell scripts.

Example
1. To start the Installation Assistant, type:
   ```bash
   install_assist
   ```
2. To access the Configuration Assistant, type:
   ```bash
   configassist
   ```
3. Access the Installation Assistant from a graphical interface, use the SMIT `smit assist` fast path.

Files
- `smit.log` specifies detailed information on your session, with time stamps.
- `smit.script` specifies the task commands run during your session, with time stamps.

Related Information
The `configassist` command.

**Configuring the Operating System** in *AIX 5L Version 5.2 Installation Guide and Reference*

**install_mh Command**

**Purpose**
Sets up mailbox directories.

**Syntax**
```bash
install_mh [-auto] [-help]
```

**Description**
The `install_mh` command sets up mailbox directories. The `install_mh` command is not started by the user. The `install_mh` command is called by other programs only.

The `install_mh` command starts automatically the first time you run any Message Handler (MH) command. The `install_mh` command prompts you for the name of your mail directory. If the directory does not exist, the `install_mh` command queries you if it should be created. Upon receiving a positive response, the `install_mh` command creates the `$HOME/.mh_profile` file and places the `Path: profile` entry in it. This entry identifies the location of your mailbox by specifying the directory path for your MH directory, `UserMHDirectory`.

**Flags**
- `-auto` Creates the standard MH path without prompting.
- `-help` Lists the command syntax, available switches (toggles), and version information.

**Note:** For MH, the name of this flag must be fully spelled out.
**Files**

$HOME/.mh_profile

Contains the MH user profile.

**Related Information**


---

**install_wizard Command**

**Purpose**

Invokes the Web-based System Manager Install Wizard or the SMIT install menu.

**Syntax**

```
install_wizard [ -d Media ]
```

**Description**

The `install_wizard` command invokes the Web-based System Manager Install Wizard or the SMIT install menu. This provides an easy path to the install interfaces. These interfaces show media content for `installp`, UDI, or ISJE products and launch the appropriate installer.

**Flags**

- `-d device or directory`

The device or directory containing the images to install.

**Example**

To invoke the Web-based System Manager Install Wizard, insert an install CD in cd1 and type:

```
install_wizard -d /dev/cd1
```

**Files**

```
/usr/sbin/install_wizard
```

**Related Information**

The `installp` command.

---

**installbsd Command**

**Purpose**

Installs a command (BSD version of the `install` command).

**Syntax**

```
/installbsd [ -c ] [ -g Group ] [ -m Mode ] [ -o Owner ] [ -s ] BinaryFileDestination
```

**Description**

The `installbsd` command installs the file specified by the `BinaryFile` parameter by moving it to a file or directory specified by the `Destination` parameter. Use of the `-c` flag copies the `BinaryFile` rather than moving it. If the specified `Destination` parameter is a directory, the `BinaryFile` is moved into the directory. If
the specified Destination parameter already exists as a file, the **installbsd** command removes that file before the BinaryFile is moved. The **installbsd** command does not move a file onto itself.

Installing the file **/dev/null** creates an empty file.

**Flags**

- **-c**
  Copies the file specified by the BinaryFile parameter to the file or directory specified by the Destination parameter.

- **-g** Group
  Specifies a group for the file specified by the Destination parameter. The default group is staff.

- **-m** Mode
  Specifies a mode for the file specified by the Destination parameter. The default mode is 755. The specified mode can be an octal number or a symbolic value.

- **-o** Owner
  Specifies the owner for the file specified by the Destination parameter. The default owner is the root user.

- **-s**
  Causes the file specified by the BinaryFile parameter to be stripped after installation.

**Examples**

To install a new command called **fixit**, enter:

```
installbsd -c mike fixit /usr/bin
```

This command sequence installs a new command by copying the program **fixit** to **/usr/bin/fixit**, with user **mike** as the owner.

**Files**

- **/usr/ucb/install**
  Hard-link to the **/usr/bin/installbsd** file.

- **/usr/bin/installbsd**
  Contains the **installbsd** command.

**Related Information**

The **chgrp** command, **chmod** command, **chown** command, **cp** command, **install** command, **mv** command, **strip** command.

---

**installp Command**

**Purpose**

Installs available software products in a compatible installation package.

**Syntax**

**To Install with Apply Only or with Apply and Commit**

```

**To Commit Applied Updates**

```

Alphabetical Listing of Commands 49
To Reject Applied Updates
installp [-U | [eLogFile | [vNumber | [b] | [g] | [p] | [v] | [x] | [O] | [r] | [s] | [u] ] ] [w] { FilesetName | [Level]... | [ListFile ]}

To Deinstall (Remove) Installed Software
installp [-u | [eLogFile | [vNumber | [d] | [g] | [p] | [v] | [x] | [O] | [r] | [s] | [u] ] ] [w] { FilesetName | [Level]... | [ListFile ]}

To Clean Up a Failed Installation:
installp [-C | [b] | [eLogFile ]]

To List All Installable Software on Media
installp [[-4 | [-L] | [eLogFile ] | [d] | [Device] | [-B] | [-I] | [-q] | [-E] | [-BlockSize] | [O] | [s] | [u] ] ]

To List All Customer-Reported Problems Fixed with Software or Display All Supplemental Information
installp [-A | [-I] | [eLogFile ] | [d] | [Device] | [-B] | [-I] | [-q] | [-E] | [-BlockSize] | [O] | [s] | [u] ] ] { FilesetName | [Level]... | [ListFile | all ]

To List Installed Updates That Are Applied But Not Committed
installp [-s | [eLogFile ] | [O] | [r] | [s] | [u] ] ] [w] { FilesetName | [Level]... | [ListFile | all ]

To List Platform Specific Installable Software on Media
installp [[-4 | [-L] | [-M | Platform ] | [eLogFile ] | [d] | [Device] | [-B] | [-I] | [-q] | [-E] | [-BlockSize] | [O] | [s] | [u] ] ]

Description

Note: The noclobber option of the Korn or C shell should be unset in the environment from which an installation is performed.

The installp command installs and updates software.

A fileset is the lowest installable base unit. For example, bos.net.tcp.client 4.1.0.0 is a fileset. A fileset update is an update with a different fix ID or maintenance level. For example, bos.net.tcp.client 4.1.0.2 and bos.net.tcp.client 4.1.1.0 are both fileset updates for bos.net.tcp.client 4.1.0.0.

When a base level (fileset) is installed on the system, it is automatically committed. You can remove a fileset regardless of the state (committed, broken, committed with applied updates, committed with committed updates, etc.).

When a fileset update is applied to the system, the update is installed. The current version of that software, at the time of the installation, is saved in a special save directory on the disk so that later you can return to that version if desired. After a new version of a software product has been applied to the system, that version becomes the currently active version of the software.

Updates that have been applied to the system can be either committed or rejected at a later time. The installp -s command can be used to get a list of applied updates that can be committed or rejected.

When updates are committed with the -c flag, the user is making a commitment to that version of the software product, and the saved files from all previous versions of the software product are removed from the system, thereby making it impossible to return to a previous version of the software product. Software can be committed at the time of installation by using the -ac flags. Note that committing already applied updates does not change the currently active version of a software product. It merely removes saved files for previous versions of the software product.
When a base level is removed with the -u flag, the files that are part of the software product and all its updates are removed from the system. Most cleanup of system configuration information pertaining to the product is also done, but this is dependent on the product and may not always be complete.

When a software product update is rejected with the -r flag, the active version of the software product is changed to the version immediately previous to the rejected update. Files saved for the rejected update and any updates that were applied after it are removed from the system.

A software product that is to be removed from the system can be in any state. Any product updates can be in either the applied or committed state, and they will also be removed.

If a previously interrupted installation leaves any software in a state of either applying or committing, it is necessary to perform cleanup with the -C flag before any further installations will be allowed. Although the `installp -C` command accepts software product names on the command line without returning an error, an attempt is always made to clean up all products when the -C flag is used. An attempt is made to clean up any incomplete installations by removing those parts that were previously completed. An attempt is also made to return to the previous version of the software product, if one exists, as the currently active version. If this cannot be done, the software product is marked as broken, and unpredictable results can occur if the user attempts to use it. Therefore, it is advisable for the user to reinstall any broken software products or updates.

The -t flag specifies an alternate location for a save directory that holds files being replaced by an update. This option is primarily useful in the following two circumstances.

- You have enough local disk space for saving replaced files but you do not want to permanently expand the root and /usr file systems.
  
  In this case, you can choose to create a separate file system for the alternate save directory. When you are satisfied with the updated system and have committed all applied updates, disk space can be retrieved by deleting the save file system.

- If you do not have enough local disk space for saving replaced files but you have access to ample disk space on a remote system, then you can specify a directory that is mounted from a remote file system.

If a remote file system is used, commit the updates as soon as possible. You may want to initiate the installation action as an apply and commit operation with the -ac flags. If you want to apply only to be capable of rejecting any unwanted updates, then test the newly installed updates as soon as possible and then commit or reject them.

Take into account the following considerations when using an alternate save directory:

- It is recommended that you use the same alternate save location on each invocation of the `installp` command.

- If an alternate save directory is used for an apply operation, make sure that the file system containing that directory remains mounted. It is highly recommended that any necessary mounts be done automatically on a reboot.

- If an alternate save directory is missing on a commit operation, the commit takes place, and a warning is given stating that the save directory could not be deleted. It is then your responsibility to delete the save directories that are no longer used in order to retrieve that disk space.

- If an alternate save directory is missing on reject, the reject operation cannot be done because the saved files are missing. An error is given, and the entire reject operation is cancelled. If the missing save directory is not caused by a temporary situation (for example, the inability to contact a remote directory on the network,) your only options are to commit the updates or leave them in an applied state permanently.

- When doing a system backup, you are responsible for backing up any alternate save directories that do not reside in the root volume group.
The installation process safeguards users with a remote save directory from the possibility of two different systems using the same remote directory. However, use directory pathnames that easily and uniquely identify each user’s system. For example, you might add the system's hostname somewhere in the pathname.

Do not create a mksysb backup of a system with a remote save directory and then try to restore the mksysb image onto a system other than the original. In this case, using a mksysb image to install several like systems causes multiple ownership of the same remote save directory.

The installp -A command can be used to obtain a list of the Authorized Program Analysis Report (APAR) numbers and summaries for all customer-reported problems that are fixed in the specified software package. The installp -i command can be used to display supplemental information contained in files that can be a part of the specified software package.

To list all the software products and updates on the specified installation media, use the installp -l command. The output of the installp command with the -l flag resembles the following:

```
# Fileset Name Level I/U Q Content
#================================================================
# X11.adt.include 4.1.0.0 I N usr
  # AIXwindows Application Development Toolkit Include
# X11.adt.lib    4.1.0.0 I N usr
  # AIXwindows Application Development Toolkit Libraries
# X11.adt.motif   4.1.0.0 I N usr
  # AIXwindows Application Development Toolkit Motif
# X11.adt.bitmaps 4.1.0.0 I N usr
  # AIXwindows Application Development Toolkit Bitmap
# X11.adt.ext     4.1.0.0 I N usr
  # AIXwindows Application Development Toolkit for X Ext
# X11.adt.imake   4.1.0.0 I N usr
  # AIXwindows Application Development Toolkit imake
# X11.apps.rte    4.1.0.0 I N usr
  # AIXwindows Runtime Configuration Applications
# X11.apps.msmi   4.1.0.0 I N usr
  # AIXwindows msmi Application
```

The fields have the following meanings:

- **Fileset Name**: Name of the fileset to be installed.
- **Level**: Level of the fileset to be installed.
I/U Indicates the type of package of which the fileset is a part. The fileset may belong to an installation package or to one of several types of update packages. The package types are as follows:

I Indicates an installation package.

S Indicates a single update.

SR Indicates a required update. Whenever the installp command encounters a required update, the update is automatically included in the input list.

SF Indicates a required update. Whenever the installp command encounters a required update, the update is automatically included in the input list. Reserved for updates to the installp fileset.

M Indicates a maintenance package. This is a packaging update that contains only a list of other updates to be applied. This package delivers no files.

ML Indicates an update package that identifies a new maintenance level for the product. This is a cumulative set of all updates since the previous product level.

Q Quiescent (quiet) column. A Y indicates that running processes can be affected by the installation of this fileset. Refer to the documentation supplied with the software product. An N indicates that running processes are not affected by the installation of this fileset. A B indicates bosboot and quiescent. A b indicates bosboot and not quiescent.

Content Content column:

`usr,root` /usr and root file systems (AIX 3.2 and later)

`usr` /usr file system only (AIX 3.2 and later)

`share` /usr/share file system only (AIX 3.2 and later)

Output from the `installp -s` command, which is used to get a list of applied software fileset updates and updates that are available to be either committed or rejected, resembles the following:

**Installp Status**

<table>
<thead>
<tr>
<th>Name</th>
<th>Part</th>
<th>Level</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>bos.net.tcp.client</td>
<td>USR</td>
<td>4.1.0.2</td>
<td>APPLIED</td>
</tr>
<tr>
<td>bos.net.tcp.client</td>
<td>ROOT</td>
<td>4.1.0.2</td>
<td>APPLIED</td>
</tr>
<tr>
<td>bos.rte.commands</td>
<td>USR</td>
<td>4.1.0.1</td>
<td>APPLIED</td>
</tr>
<tr>
<td>bos.rte.misc_cmds</td>
<td>USR</td>
<td>4.1.0.1</td>
<td>APPLIED</td>
</tr>
<tr>
<td>bos.rte.tty</td>
<td>USR</td>
<td>4.1.0.1</td>
<td>APPLIED</td>
</tr>
</tbody>
</table>

The fields have the following meanings:

Name: Name of the installed software product fileset.

Part: The part of the fileset where:

ROOT: root file system

SHARE: /usr/share file system

USR: /usr file system.

Level: The level of the installed software product option.

State: The state of the installed software product option.

The software products and updates to be installed can be identified in one of three ways:

- by the keyword **all**, which indicates that all software contained on the specified installation media is to be installed
- by a list of software product names (each of which can optionally be followed by a level) that indicates the software to be installed
• by the -f flag followed by a file name, where each line in the file is an entry containing a software product name, optionally followed by a level, or is a comment line that begins with a # and is ignored

**Note:** The `installp` program uses the `sysck` command to verify files after restoring them. The `sysck` command does not recognize the following special characters in file names: `-`, `,`, `;`, `\`, `"`, `$`, `&`, `( ), `l`, `{`, `} `, `<`, `>`, and `?`. If a file name contains one of these characters, installation fails.

The `FilesetName` parameter can be used to specify an entire software product or any separately installable filesets within the software package. For example, `bos.net` is the name of a software package, and the separately installable filesets within that software package are `bos.net.ncs.client`, `bos.net.nfs.client`, and `bos.net.tcp.client`. If the user specifies `bos.net` for the `FilesetName` parameter, then all of the separately installable filesets listed are installed. If the user specifies `bos.net.tcp.client` for the `FilesetName` parameter, then only that fileset is installed.

The `Level` parameter indicates the level of the software product or update that is to be installed. The `Level` parameter is of the form `vv.rr.mmmmm.mttt.fffpppppppp` where:

- `vv` is a numeric field of 1 to 2 digits that represents the version number.
- `rr` is a numeric field of 1 to 2 digits that represents the release number.
- `mmmm` is a numeric field of 1 to 4 digits that represents the modification level.
- `tttt` is a numeric field of 1 to 4 digits that represents the fix level.
- `pppppppp` is a character field of 1 to 9 characters that represents the fix ID.

If a user is installing an installation package from installation media that contains only installation packages it is not usually necessary to specify the level. More than one software product installation package with different levels does not often exist on the same installation medium, but when this does occur `installp` installs the specified software product at the latest software product level when `Level` is not specified with `FilesetName`. For installation media that contain either update packages only or contain both installation and update packages, all applicable update packages that are present on the installation media for the specified `FilesetName` are also installed when `Level` is not specified. For installation media that contain both installation and update packages the user can request the installation of only installation packages or only update packages by specifying the `-I` or `-B` flags, respectively. If the user wants to install only some of the updates on the installation medium for a specific software product both `FilesetName` and `Level` for each of the updates to be installed for that software product must be specified.

An example of what might be entered to install TCP/IP and one of its updates that are both contained in the `/usr/sys/inst.images` directory would be the following:

```
installp -a -d/usr/sys/inst.images bos.net.tcp.client 4.1.0.0
bos.net.tcp.client 4.1.0.2
```

**Note:** In the event that there are duplicate filesets at the same level, `installp` will use the first one that it finds in the install table of contents (.toc). This situation can occur when `bffccreate` is used to extract images from different media to the same installation directory. For this reason, make sure that update images are not extracted to the same directory as base level images for the same fileset at the same level.

A summary report is given at the end of the `installp` output that lists the status of each of the software products that were to be installed. An example summary report for the previous `installp` command follows:

```
Installp Summary
----------------
Name           Level   Part   Event   Result
bos.net.tcp.client  4.1.0.0  USR    APPLY  SUCCESS
bos.net.tcp.client  4.1.0.0  ROOT   APPLY  SUCCESS
bos.net.tcp.client  4.1.0.2  USR    APPLY  SUCCESS
```
Note: If a previously installed level of a fileset update is in the broken state, the -acgN flags must be used when that fileset update is installed again.

Summary Report Values
The summary report identifies the name of the product option and the part of the product. Other information given includes the requested action (event) and the result of that action.

Event Values
The Event column of the summary report identifies the action that has been requested of the installp command. The following values may be found in this column:

<table>
<thead>
<tr>
<th>Event</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>APPLY</td>
<td>An attempt was made to apply the specified fileset.</td>
</tr>
<tr>
<td>COMMIT</td>
<td>An attempt was made to commit the specified fileset update.</td>
</tr>
<tr>
<td>REJECT</td>
<td>An attempt was made to reject the specified fileset update.</td>
</tr>
<tr>
<td>CLEANUP</td>
<td>An attempt was made to perform cleanup for the specified fileset.</td>
</tr>
<tr>
<td>DEINSTALL</td>
<td>An attempt was made to remove the specified fileset.</td>
</tr>
</tbody>
</table>

Result Values
The Result column of the summary report gives the result of installp performing the requested action. It can have the following values:

<table>
<thead>
<tr>
<th>Result</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>SUCCESS</td>
<td>The specified action succeeded.</td>
</tr>
<tr>
<td>FAILED</td>
<td>The specified action failed.</td>
</tr>
<tr>
<td>CANCELLED</td>
<td>Although preinstallation checking passed for the specified option, it was necessary to cancel the specified action before it was begun. Interrupting the installation process with Ctrl-c can sometimes cause a canceled action, although, in general, a Ctrl-c interrupt causes unpredictable results.</td>
</tr>
</tbody>
</table>

Flags

- **-A**
  
  Displays the APAR number and summary of all customer-reported problems that are fixed in the specified software package. No installation is attempted.

- **-a**
  
  Applies one or more software products or updates. This is the default action. This flag can be used with the -c flag to apply and commit a software product update when installed.

- **-b**
  
  Prevents the system from performing a bosboot in the event that one is needed.

- **-B**
  
  Indicates that the requested action should be limited to software updates.

- **-C**
  
  Cleans up after an interrupted installation and attempts to remove all incomplete pieces of the previous installation. Cleanup should be performed whenever any software product or update is in a state of either applying or committing and can be run manually as needed. For backwards compatibility other flags and parameters can be accepted with installp -C, but are ignored because all necessary cleanup is attempted.

- **-c**
  
  Commits all specified updates that are currently applied but not committed. When an update is committed all other software products it is dependent on must also be committed (unless they are already in the committed state). The specified software product is dependent on any software product that is a prerequisite or corequisite of the specified product. The commit will fail and error messages will be given if any requisite software products are not in the committed state. The -g flag can be used to automatically commit requisite software product updates.
-D
Deletes the installation image file after the software product or update has been successfully installed. When the -g flag is specified, the installation image files for any products that are automatically included will also be deleted. This flag is valid only with the -a or -ac flags and is not valid with the -Or flag. This flag is also only valid when the device is a directory and an installation image file on the system where the installation is taking place.

-d Device
Specifies where the installation media can be found. This can be a hardware device such as tape or diskette, it can be a directory that contains installation images, or it can be the installation image file itself. When the installation media is a product tape or Corrective Service tape, specified the tape device as no-rewind-on-close and no-retension-on-open. Examples of this would be /dev/rmt0.1 for a high density tape, or /dev/rmt0.5 for a low density tape. Use the options specified by the tape supplier. The default device is /dev/r1d0.

-e LogFile
Enables event logging. The -e flag enables the user to append certain parts of the installp command output to the file specified by the LogFile variable. By default the output of the installp command goes to stdout and stderr, unless SMIT or VSM is used, in which case the output goes to the smit.log. The LogFile variable must specify an existing, writable file, and the file system in which the file resides must have enough space to store the log. The log file does not wrap.

Not all output is appended. Copyright information is still displayed to the user. Any error messages are displayed on the screen and sent to the file specified by the LogFile variable. A results summary of the installp command invocation is also displayed on the screen and sent to the LogFile. This flag is primarily used by NIM and BOS install to limit the output shown to the user, but keep useful information for later retrieval.

-E
Displays software license agreements. This flag is only valid with the -a or -l flags. If the -E flag is specified with the -a flag, a new section is emitted showing the pending license agreements associated with the selected filesets. If the -E flag is specified with the -l flag, output is emitted showing the license agreements associated with all filesets on the media.

-F
This option can be used to force the installation of a software product even if there exists a previously installed version of the software product that is the same as or newer than the version currently being installed. The -F flag is not valid with update packages or the -g flag. When you use the -F flag, the -l flag is implicit.

-f ListFile
Reads the names of the software products from ListFile. If ListFile is a - (dash), it reads the list of names from the standard input. Software fileset names, optionally followed by a level, should be one per line of text, and any text following the second set of white spaces or tabs on a line is ignored. Output from the installp -I command is suitable for input to this flag.

-g
When used to install or commit, this flag automatically installs or commits, respectively, any software products or updates that are requisites of the specified software product. When used to remove or reject software, this flag automatically removes or rejects dependents of the specified software. The -g flag is not valid when used with the -F flag.

Note: This flag also automatically pulls in a superseding update present on the media if the specified update is not present. This flag causes the newest update to be installed for a given fileset, when there are multiple superseding updates for the same fileset on the installation media.

-I (uppercase i)
Indicates that the requested action should be limited to base level filesets.

-i
Displays on standard output the lpp.instr, lpp.doc, lpp.REAMDE, and README files on the installation media for the software product, if they exist. This flag can take a significant amount of time for a large number of filesets.

-J
This flag is used when the installp command is executed from the System Management Interface Tool (SMIT) menus.

-l (lowercase L)
Lists all the software products and their separately installable options contained on the installation media to standard output. No installation occurs. The -l flag is not valid with the -Or flag.
-L Displays the contents of the media by looking at the table of contents (TOC) and displaying the information in colon-separated output. This flag is used by smit and vsm to list content of the media. The format provided:
  package:fileset:v.r.m.f:PTF:type:state:supersede:\
  sup_ptf:sup_state:latest_sup:quiesce:Descr:\
  netfs_vendor_id:netfs_prod_id:netfs_prod_ver

-M Platform Specifies the Platform value. Any of the following values may be used to list the installable software packages:
  R  Specifies POWER-based platform packages only.
  N  Specifies neutral packages, that is, packages that are not restricted to the POWER-based platform.
  A  Specifies all packages.

-N Overrides saving of existing files that are replaced when installing or updating. This flag is valid only with the -ac flags. If there is a failure in the system during the installation, there is no recovery of replaced files when this flag is used.

-O[r][s][u]] Installs the specified part of the software product. The r indicates the / (root) part to be installed, the s indicates the /usr/share part to be installed, and the u indicates the /usr part to be installed. The -O flag is not needed with standard systems because without this flag all parts are installed by default. This flag is needed for use with the installation of diskless or dataless workstations and is designed for use by the nim command. The -Or option is not valid with the -d or -l flags.

-p Performs a preview of an action by running all preinstallation checks for the specified action. This flag is only valid with apply, commit, reject, and remove (-a, -c, -r, and -u) flags.

-Q Suppresses errors and warnings concerning products failing to install due to instrequsites. This flag applies only to AIX 4.2 or later.

-q Specifies quiet mode, which suppresses the prompt for the device, except for media volume change.

-r Rejects all specified software updates that are currently applied but not committed. When a software update is rejected any other software product that is dependent on it (that is, those software products that have the specified software product as a requisite) must also be rejected. The -g flag can be used to reject automatically dependent software updates. The keyword all is not valid with the reject flag (-r). For backwards compatibility, the -R flag is also accepted as a reject flag. The -R cannot be used to remove base level filesets; use the -u flag.

-s Lists information about all software products and updates that have been applied but not committed. This list comprises the software that is available to be either committed or rejected.

-S Suppresses multiple volume processing when the installation device is a CD-ROM. Installation from a CD_ROM is always treated as a single volume, even if the CD-ROM contains information for a multiple volume CD set. This same suppression of multiple volume processing is performed if the INU_SINGLE_CD environment is set.

-t SaveDirectory Specifies an alternate save directory location for files being replaced by an update.

The -t flag is only valid with an apply or an apply/commit operation for updates. This flag is not valid with the -N flag.

The -t flag is useful when there is insufficient space in the default file systems (/ and /usr) or when it is undesirable to permanently expand these file systems. It may be desirable for the specified directory to be a remote file system. A remote file system must have ample space, because the installp command cannot expand remote file systems.

-u Removes the specified software product and any of its installed updates from the system. The product can be in either the committed or broken state. Any software products that are dependent on the specified product must also be explicitly included in the input list unless the -g flag is also specified. Removal of any bos.rte fileset is never permitted.
-v

Verifies that all installed files in the fileset have the correct checksum value after the installation. Installed files are always verified for correct file size after installation. Use this flag after network or remote device installations. If any errors are reported, it might be necessary to install the software product again. Post-installation requisite consistency checks are also started by this flag.

-V Number

Specifies the verbose option that provides four levels of detail for preinstallation output. The valid values for the Number parameter are 2, 3, or 4. The default level of verbosity, without the use of the -V flag, prints an alphabetically ordered list of FAILURES, WARNINGS, and SUCCESSES from preinstallation processing. Requisite failures are reported with emphasis on the real cause of the failure. Extraneous requisites for failed filesets are not displayed. The preinstallation output is modified by levels 2 through 4 as described below:

2  Prints alphabetically ordered list of FAILURES and WARNINGS. Requisite failures are displayed with additional information describing requisite relationships between selected filesets and the requisites causing them to fail. Failing requisites suppressed under Level 1 are displayed. Preinstallation SUCCESSES are displayed in the order in which they are processed.

3  Level 3 is the same as Level 2, with the exception that additional requisite information is displayed for SUCCESSES.

4  Level 4 is the same as Level 3 for SUCCESSES and WARNINGS. Requisite failures are displayed in a format depicting detailed requisite relationships.

Note: If verbosity level 2 or higher is used, the files that are restored on to the system is shown in the output. Because this will make installp's output much more verbose, make sure that your / (root) filesystem does not become full when the /smit.log becomes large (if using smit to run installp).

-w

Does not wildcard FilesetName. Use this flag from smit so it only installs the fileset chosen and will not install filesets that match. For example, if you choose foo.rte, foo.rte.bar is not automatically pulled in, as it would be by default, without the -w flag. This flag applies only to AIX 4.2 or later.

-X

Attempts to expand any file systems where there is insufficient space to do the installation. This option expands file systems based on current available space and size estimates that are provided by the software product package. Note that it is possible to exhaust available disk space during an installation even if the -X flag is specified, especially if other files are being created or expanded in the same file systems during an installation. Also note that any remote file systems cannot be expanded.

-Y

Agrees to required software license agreements for software to be installed. This flag is only valid with the -a flag.

-z BlockSize

Indicates in bytes the block size of the installation media. The default value of Size is 512.

FilesetName

This is the name of the software product to be installed and can specify either an entire software product or any separately installable filesets within the software product. This can be used to specify the name of a fileset or fileset update.

Level

This indicates the level of the software product or update that is to be installed and is of the form vv.rr.mmmm.fff. If a fileset update has an additional fix ID (also know as ptf id), that ID should also be specified in the Level as in vv.rr.mmmm.fff.pppppp.

Return Values

A zero (0) return value indicates that all attempted installations were successful, or that no processing was required for the requested action on the requested filesets (for example, if a requested fileset was already installed).

A nonzero return value indicates that some part of the installation was not successful.
A summary report is given at the end of the `installp` output that lists the status of each of the software products that were to be installed. For those software products that could not be installed or whose installation failed, the user can search for the cause in the more detailed information that is continually displayed from the `installp` command during the installation process.

**Security**

Privilege Control: Only the root user can run this command.

Auditing Events:

<table>
<thead>
<tr>
<th>Event</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>INSTALLP.Inst</td>
<td>Success or failure of the apply, commit, reject, and cleanup operations.</td>
</tr>
</tbody>
</table>

**Examples**

1. To list all software products and installable options contained on an installation cartridge tape, type:
   ```bash
   installp -L -d /dev/rmt0.1
   ```

2. To list all customer-reported problems fixed by all software products on an installation tape, type:
   ```bash
   installp -A -d /dev/rmt0.1 all
   ```

3. To install (automatically committed) all filesets within the `bos.net` software package (located in the `/usr/sys/inst.images` directory) and expand file systems if necessary, type:
   ```bash
   installp -aX -d /usr/sys/inst.images bos.net
   ```

4. To reinstall and commit the NFS software product option that is already installed on the system at the same level (from tape), type:
   ```bash
   installp -acF -d /dev/rmt0.1 bos.net.nfs.client 4.1.0.0
   ```

5. To install (apply only) certain updates that are contained on diskette for the TCP/IP software product, type:
   ```bash
   installp -a bos.net.tcp.client 4.1.0.2 bos.net.tcp.server 4.1.0.1
   ```

6. To remove a fileset named `bos.net.tcp.server`, type:
   ```bash
   installp -u bos.net.tcp.server
   ```

7. To specify an alternate storage directory on a remote file system for a BOSNET TCP/IP update with `-t/temp_space`, see the following example: the save directory becomes `/temp_space/My_Hostname/usr/lpp/bos.net/bos.net.nfs.client/4.1.1.0.save`.
   ```bash
   mount Server_Name:/Save_Area /temp_space
   ```
   ```bash
   installp -a -t /temp_space/My_Hostname \ bosnet.nfs.client 4.1.1.0
   ```

8. In order to capture a log file of all output from the `installp` command, the `script` command can be used as in the following example. Output is written to the `typescript` file in the current directory.
   ```bash
   script
   installp ...
   <Ctrl>d
   ```
   ```bash
   or
   installp ... 2>&1 | tee /tmp/inst.out
   ```
   In the second example, output is written to the screen and a copy is saved.

9. To preview (without performing) the installation of the Application Developer bundle of software using the `installp` command, type:
    ```bash
    installp -pacgXd /dev/rmt0.1 -f /usr/sys/inst.data/sys_bundles \ /App_Dev.bnd
    ```

10. To install TCP/IP and one of its updates that are both contained in the `/usr/sys/inst.images`, type:
A summary report is given at the end of the installp output that lists the status of each of the software products that were to be installed. An example summary report for the previous installp command follows:

```
Installp Summary
----------------
Name           Level Part Event Result
-----------------------------------------------------------------
bos.net.tcp.client 4.1.0.0 USR APPLY     SUCCESS
bos.net.tcp.client 4.1.0.0 ROOT APPLY    SUCCESS
bos.net.tcp.client 4.1.0.2 USR APPLY    SUCCESS
```

Note: This summary is also saved in /var/adm/sw/installp.summary until the next installp invocation. The header file inuerr.h in the /usr/include directory describes the fields making up the records in the installp.summary file.

11. To list software products (located in the /usr/sys/inst.images directory) that are installable on POWER-based machines, type:

```
installp -l -MR -d /usr/sys/inst.images
```

Files

- /dev/rfd0
- /dev/rmt
- /usr/sys/inst.images directory

Specifies the default restore device.

Specifies the raw streaming tape interface.

Contains files in backup format for use in installing or updating a complete set or subset of software products.

Related Information

The bffcreate command, inudocm command, inutoc command, lppchk command, lsip command, sysck command.

**installms Command**

**Purpose**

installms - Installs the Cluster Systems Management (CSM) management server software.

**Syntax**

```
installms [-h]
installms [-f] [-v | -V] {-x | -p pkg_path[pkg_path...] [Attr=value ...]}
```

**Description**

The installms command installs the CSM management server software. It automatically installs both CSM and its prerequisite software. Before running the installms command, a supported version of the operating system must be installed.

The command performs the following tasks:

- Creates the /csminstall directory structure that is required for CSM files.

  **Note:** If the /csminstall directory does not exist, it will be created by the installms command. However, it is recommended that you create the /csminstall directory in a separate file system before running the installms command. See the CSM for Linux: Planning and Installation Guide for more information on requirements for /csminstall.

- Optionally copies the required file sets to the correct /csminstall subdirectories.
• Copies the CSM scripts and files that are required to install CSM on the nodes of the cluster to the correct \texttt{/csminstall} subdirectories.
• Sets global cluster system attribute values.

If the \texttt{−p} flag is used, the \texttt{installms} command copies the required file sets from the specified location into the correct subdirectories of \texttt{/csminstall}. If the \texttt{−x} flag is used, the \texttt{installms} command assumes that the files have already been copied. When the \texttt{installms} command installs the management server software, it uses the file sets that have been copied to the \texttt{/csminstall} subdirectories.

\textbf{Flags}

\begin{description}
\item [\texttt{−c}] Forces the command to copy all required files from the \texttt{pkg\_path}. Normally, \texttt{installms} only copies files that are newer or the same as those in \texttt{/csminstall}, or files that do not exist in \texttt{/csminstall}. The \texttt{−c} flag forces \texttt{installms} to copy all files regardless of their version.
\item [\texttt{−f}] Forces installation regardless of the status of the existing installation.
\item [\texttt{−h}] Writes the command’s usage statement to standard output.
\item [\texttt{−p pkg\_path}] Specifies one or more colon-separated directories where installation packages are located. This flag cannot be used with \texttt{−x}.
\end{description}

When this option is specified, the command handles ISO CD images if they are found. The command will copy the files as if copying from a directory and mount one image at a time. The CD images do not have to be in order. The command also handles ISO images of CSM CDs, for example:

\begin{verbatim}
installms −p /tmp/image1:/tmp/image2:/tmp/image3 −n node1
\end{verbatim}

\begin{description}
\item [\texttt{−v | −V}] Writes the command’s verbose messages to standard output.
\item [\texttt{−x}] Specifies to not copy software packages and to use packages in the \texttt{/csminstall} subdirectory. This flag cannot be used with \texttt{−p}.
\end{description}

\textbf{Parameters}

\begin{description}
\item [\texttt{Attr=value} ...] Specifies one or more \texttt{Attr=value} pairs separated by spaces. \texttt{Attr=value} pairs must be specified last on the command line. Valid attributes are:
\item [\texttt{RemoteShell=full\_path\_name\_of\_shell}] The \texttt{RemoteShell} attribute value determines the default remote shell for the \texttt{dsh} command if the DSH\_REMOTE\_CMD variable is not set. It also determines which shell to set up on the nodes during installation. The default attribute value for the management server is \texttt{/usr/bin/rsh}.
\item [\texttt{SetupRemoteShell={yes | no}}] The \texttt{SetupRemoteShell} attribute value controls whether the remote shell is set up on the management server and the nodes. If it is set to \texttt{yes} or \texttt{1}, then the remote shell is set up. The remote shell is not set up if the attribute is set to \texttt{no} or \texttt{0}. The default value for this attribute is \texttt{1}.
\end{description}

\textbf{Security}

The command requires root access to the cluster management server.

\textbf{Exit Status}

\begin{description}
\item [0] The command has run successfully.
An error has occurred.

Environment Variables

**DSH_FANOUT**
Sets the maximum number of concurrent remote shell commands. This can also be set by the `−f` flag.

**DSH_LIST**
Specifies a file that contains definitions of the set of nodes that comprise the node list.

**DSH_LOG**
Enables logging when set to the absolute path and file name of the log file. This variable is overridden by the `--log` and `--noLogging` flags.

**DSH_PATH**
Sets the path that is used on the remote nodes. If DSH_PATH is not set, the default path for the remote shell is used. For example, `DSH_PATH=$PATH` sets the path on the remote node to the same path that is used on the source node.

**DSH_REMOTE_CMD**
Specifies the path of the remote shell executable to use instead of the default.

**DSH_REMOTE_OPTS**
Includes the options specified in the remote command when the command is forwarded to the remote nodes.

**DSH_REPORTSDIRECTORY**
Enables reporting and output file writing when set to the reports directory path. This variable is overridden by the `--reports` and the `--noReporting` flags.

**DSH_TIMEOUT**
Specifies the time, in seconds, to wait for response from the remote shell. This can also be set with the `−t` flag.

**WCOLL**
Equivalent to **DSH_LIST**.

Standard Error

When the **dshbak** filter is used and standard error messages are generated, all error messages on standard error appear before all standard output messages. This is true with and without the `−c` flag.

Examples

1. To install CSM management server software using CSM file sets that were previously copied to the `/tmp/csm` directory, enter:
   ```
   installms -p /tmp/csm
   ```

2. To install CSM and the packages contained in the current directory on the management server using the force option, enter:
   ```
   installms -f -p .
   ```

3. To install CSM on the management server using the AIX CSM version 1.3.0 packages previously copied to the `/csminstall/AIX/csm/1.3.0/packages` directory, enter:
   ```
   installms -x
   ```

Location

`/opt/csm/bin/installms`

62 Commands Reference, Volume 3
Related Information
- The csmconfig, definename, uninstallms, and updatenode commands.

instfix Command

Purpose
Installs filesets associated with keywords or fixes.

Syntax
instfix [ -T | -M Platform ] [ -S String ] [ -S ] [ -k Keyword | -f File ] [ -p ] [ -d Device ] [ -a ] [ -c ] [ -f ] [ -q ] [ -t ] 

Description
The instfix command allows you to install a fix or set of fixes without knowing any information other than the Authorized Program Analysis Report (APAR) number or other unique keywords that identify the fix.

Any fix can have a single fileset or multiple filesets that comprise that fix. Fix information is organized in the Table of Contents (TOC) on the installation media. After a fix is installed, fix information is kept on the system in a fix database.

The instfix command can also be used to determine if a fix is installed on your system.

Note: Return codes for the instfix command are documented in the /usr/include/inuerr.h file, which is shipped with the bos.adt.include fileset. There is also a general failure code of 1 and a single reference to EACCES (13) from /usr/include/errno.h.

Flags
- -a
  Displays the symptom text associated with a fix. Can be combined with the -i, -k, or -f flag.
- -c
  Displays colon-separated output for use with -f flag. Output includes keyword name, fileset name, required level, installed level, status, and abstract. Status values are:
  - Down level
  = Correct level
  + Superseded
  ! Not installed
- -d Device
  Specifies the input device. Required for all flags except -i and -a.
- -F
  Returns failure unless all filesets associated with the fix are installed.
- -f File
  Specifies the input file containing keywords or fixes. Use - (dash) for standard input. The -T flag produces a suitable input file format for -f.
- -i
  Displays whether fixes or keywords are installed. Use this flag with the either the -k or the -f flag. Installation is not attempted when the -i flag is used. If you do not specify the -k or the -f flag, all known fixes are displayed.
- -k Keyword
  Specifies an APAR number or keyword to be installed. Multiple keywords can be entered. A list of keywords entered with the -k flag must be contained in quotation marks and separated with spaces.
- -M Platform
  Specifies that any of the Platform values may be used to list the fixes for that particular platform.
  R Specifies POWER-based platform fixes only.
  N Specifies neutral fixes, that is, fixes that are not restricted to the POWER-based platform.
  A Specifies all fixes.
-p Displays filesets associated with keywords. This flag is used with either the -k or the -f flag. Installation is not attempted when the -p flag is used.

-q Specifies quiet mode. Use this flag with the -i flag. If you use the -c flag, no heading is displayed, otherwise there is no output.

-s String Searches for and displays fixes on media containing a specified string.

-S Suppresses multiple volume processing when the installation device is a CD-ROM. Installation from a CD_ROM is always treated as a single volume, even if the CD-ROM contains information for a multiple volume CD set. This same suppression of multiple volume processing is performed if the INU_SINGLE_CD environment is set.

-T Displays the entire list of fixes present on the media.

-tType Used with the -i flag to limit searches to a given type. Valid types are:

  f     fix

  p     preventive maintenance

-v Used with the -i flag to specify verbose mode. Displays information about each fileset associated with a fix or keyword.

Security
Privilege Control: You must be the root user to install using the instfix command, but any user can run the instfix command to query the fix database.

Examples
1. To install all filesets associated with fix IX38794 from the tape mounted on /dev/rmt0.1, type:

   instfix -k IX38794 -d /dev/rmt0.1

2. To install all fixes on the media in the tape drive, type:

   instfix -T -d /dev/rmt0.1 | instfix -d /dev/rmt0.1 -f-

   The first part of this command lists the fixes on the media, and the second part of this command uses the list as input.

3. To list all keyword entries on the tape containing the string SCSI, type:

   instfix -s SCSI -d /dev/rmt0.1

4. To inform the user on whether fixes IX38794 and IX48523 are installed, type:

   instfix -i -k "IX38794 IX48523"

5. To create a list of filesets associated with fix IX12345 for bffs in the /bffs directory, type:

   instfix -p -k IX12345 -d /bffs | installp -acgX -f -d /bffs

   This sequence passes the list of fixes to the installp command to be applied and committed. The installp command extends filesystems as needed with the flags shown. This example shows that you can select other installp flags. The instfix command calls installp if the -p flag is not used.

6. To list all of the fixes that are not restricted to the POWER-based platform, type:

   instfix -T -MN -d /dev/cd0

Files
/usr/sbin/instfix Contains the instfix command.
/usr/lib/objrepos/fix Specifies the path to the Object Data Manager database.
Related Information

The `installp` command.

inucp Command

Purpose
Performs simple copy operations for the `installp` command. This command is used by the `installp` command and the install scripts.

Syntax

```
inucp -s StartDirectory [-e FinalDirectory ] ListFile ProductName
```

Description
The `inucp` command copies the files in a file tree with its root at `StartDirectory` to the appropriate place on the `FinalDirectory` root.

Before replacing files that may already exist in the `FinalDirectory` file tree, the `inusave` command should be called to save the files until needed by the `inurecv` command.

The `ListFile` parameter specifies a list, one per line, of all the files for `ProductName`. `ListFile` is the full path name of the file that contains the relative path names of files that the product needs to have copied.

The `ProductName` parameter specifies the name of the software product to be copied.

Flags

- `-e FinalDirectory` Indicates the root of the file tree that the files are to be copied to. The `FinalDirectory` should be the base of the file tree. The default directory is the `/` (root) directory when this flag is not specified.

- `-s StartDirectory` Indicates the root of the file tree that the files are to be copied from.

Environment Variables

**INUEXPAND** This flag is set to 1 by the `installp` command if file systems are to be expanded if necessary to do the copy (that is, the `-X` flag was passed). It is set to 0 if file systems are not to be expanded. If this environment variable is not set, the default is not to expand file systems.

**INUTEMPDIR** This flag is set by the `installp` command to the path of the current temporary directory. If this flag is not set the default is `/tmp`.

Error Codes
The `inucp` command returns the following error codes, which are defined in `inuerr.h`.

- **INUACCS** One or both of `StartDirectory` and `FinalDirectory` are not directories.
- **INUBADAR** Could not archive files in `lpp.acf` file.
- **INUBADC1** The copy operation failed.
- **INUBADMIN** Unrecognizable flag specified.
- **INUGOOD** No error conditions occurred.
- **INUNOAP2** Could not access the `ListFile`.
- **INUNODIR** No write access to `FinalDirectory`.
- **INUNOLPP** One or both of `StartDirectory` and `FinalDirectory` do not have the necessary permissions.
- **INUNOMK** Could not create a needed directory.
- **INUNOSPC** Insufficient space for the copy and `INUEXPAND` was not set.
Security
Privilege Control: You must be the root user to run this command.

Examples
To copy all the files listed in the /usr/lpp/X11/inst_root/al list from the /usr/lpp/X11/inst_root file tree to the root directory, enter:

```
inucp -s /usr/lpp/X11/inst_root /usr/lpp/X11/inst_root/al X11
```

Related Information
The `installp` command, `inurecv` command, `inurest` command, `inusave` command.

------

**inudocm Command**

**Purpose**
Displays contents of files containing supplemental information.

**Syntax**
```
inudocm [ -d Device ] [ -q ] { ProductName ... | all }
```

**Description**

*Note:* This command is used by the `installp` command and is not recommended as a way to get README information.(See `installp -l`.)

The `inudocm` command is used to display supplemental information. The files from the media that are displayed, if they exist, are the `lpp.doc` file, the `lpp.instr` file, the `lpp.README` file and the `README` file.

The `ProductName` parameter specifies the name of the software product being checked. Specify `all` to display information about all software products that are known to the system.

**Flags**

- `-d Device` Specifies where the installation media can be found. The `Device` parameter can specify a hardware device, such as a tape or diskette drive, a directory that contains installation images, or an installation image file. The default device is `/dev/rfd0`.
- `-q` Specifies quiet mode, which suppresses prompts.

**Security**
Privilege Control: Only a root user can run this command.

**Examples**
To display the update instructions for the `snaserv` software product on `/dev/rfd0`, enter:
```
inudocm snaserv
```
Files

/usr/sbin/inudocm
Contains the inudocm command.
/usr/lpp/ProductName/lpp.instr
Specifies the update instructions for the software product.
/usr/lpp/ProductName/lpp.README
Specifies special instructions for the software product.
/usr/lpp/README
Specifies special instructions for the software product.
/usr/lpp/ProductName/lpp.doc
Specifies the updates to the documentation for the software product.

Related Information
The installp command, restore command.

inulag Command

Purpose
Acts as the front end to the subroutines to manage license agreements.

Syntax

\texttt{inulag -r [\textbf{-n} FilesetName | \textbf{-s} FileName | \textbf{-p} Product] [\textbf{-d} Description [\textbf{-m} MessageSpecification] \textbf{-f} File}}

\texttt{inulag -l [\textbf{-c} | \textbf{-v}] [\textbf{-n} FilesetName | \textbf{-s} FileName | \textbf{-p} Product | \textbf{-a}]}

\texttt{inulag -u [\textbf{-n} FilesetName | \textbf{-s} FileName | \textbf{-p} Product]}

\texttt{inulag -A}

\texttt{inulag -D}

Description
The inulag command manages software license agreements. The basic forms are license agreement registration, license agreement listing, license agreement deactivation, license agreement validation, and license agreement revalidation.

The -r flag manages software license agreement registration of a fileset installed with installp or an independently-installed product installed through another installer. The path to a file that is always installed with an independently-installed product must be specified with the -s flag when the license agreement is registered.

The -l flag lists software license agreement registrations. If the -c flag is specified, the path to the software license agreement file is displayed rather than the contents of the file.

The -q flag queries for existence of software license agreements. A return code of 0 is returned if a license agreement exists. If the -a flag is also specified, then a return code of 0 is returned if there is a pending license agreement.

The -u flag removes the listing of software license agreements for a fileset or independently-installed product.

The -D flag forces revalidation of software license agreements upon the next system reboot.
Flags

-a
Used with the -I flag to show products that have a pending license agreement.

-A
Registers agreements for all pending license agreements.

-c
Used with the -I flag for colon-separated listing. Cannot be used with the -v flag.

-d Description
Specifies the default description for the fileset or product to which license applies.

-D
Forces the revalidation all license agreements on the next reboot.

-f File
Specifies the pathname specification for the license agreement. A "%L" in the specification is a substitution pattern for the current locale. en_US is the default locale. A "%!" in the specification matches the first two characters of the locale unless the current locale is zh_CN, in which all five characters of the locale designation are used.

-l
Lists software license agreements.

-m MessageSpecification
Specifies the message catalog for a translated description of the form "catalog,set number,message number".

-n FilesetName
Specifies the name of a fileset registered in the software vital product database governed by the license agreement.

-p Product
Specifies the product id, a nontranslatable alphanumeric string that uniquely identifies a product.

-q
Queries for license agreements. Does not show output. The value of 0 is returned if a license agreement exists. The -q flag can be used with other flags to query for particular license agreements or pending license agreements.

-r
Registers a software license agreement. Requires the -f flag for the path to the agreement file and either the -n flag or the -s flag to indicate the fileset name or signature file containing software subject to the agreement. The -r flag cannot be used with the -l, -q, or the -u flag. License agreements are registered as pending (status="P") during system installation, and NIM SPOT installation unless the environment variable ACCEPT_LICENSES is set to yes.

-s FileName
Specifies a signature file unique to installed software that identifies software not registered in the software vital product database that is governed by the license agreement. This is for use by software products not registering into the software vital product database. This form exists for the purpose of identifying software installed but not registered in the software vital product database. The FileName includes the full path to the file.

-u
Removes a license agreement. This does not actually remove the license agreement file, rather it changes the status of a license agreement associated with a fileset to inactive. Inactive license agreements do not need to be reagreed to, but they do not show up when listing installed software licenses.

-v
Used with the -I flag for verbose listing. Cannot be used with the -c flag.

Security
The agreement database is writable only by root. As a result, all flags other than the -I flag can only be used by a user operating with root user authority.

Related Information
The installp command, lspp command, nim command.

Optional Software Products and Service Updates in AIX 5L Version 5.2 Installation Guide and Reference

inurecv Command

Purpose
Recovers files saved by the nusave command.
Syntax

```
inurecv [ProductName] [OptionList]
```

Description

The `inurecv` command recovers files and archive constituent files saved from a previous `inusave` command. It uses the `update.list` and `archive.list` files from the directory specified by the `INUSAVEDIR` environment variable. The `inurecv` command recovers files saved by program-provided installation or update procedures.

The `inurecv` command is primarily called by the `install` -r command and the `installp` -C command to recover the files for a rejected program or a program that needs to be cleaned up.

The `inurecv` command is used to recover all the files for an installable program by separate calls to `inurecv` for the root, `/usr`, and `/usr/share` file trees. The save directories for the root, `/usr`, and `/usr/share` parts of an installation are:

- `/lpp/PackageName/FilessetName/V.R.M.F.save`
- `/usr/lpp/PackageName/FilessetName/V.R.M.F.save`
- `/usr/share/lpp/PackageName/FilessetName/V.R.M.F.save`

respectively, when set up by the `install` command. `Level` refers to the level of the software product and has the format of `vv.rr.mmmm.ffff.ppppppppp`, where `vv` = version, `rr` = release, `mmmm` = modification, `ffff` = fix, and `pppppppppp` = fix ID (only for Version 3.2 images).

Parameters

- **OptionList** Specifies the full path name of a stanza file that contains the names of the separately installable options, such as `bosnet.tcp.obj`, that are to be recovered for the `ProductName` software product. The option names in the `OptionList` file must be specified one per line.
- **ProductName** Specifies the installable software product, such as `bosnet`, whose files are to be recovered.

Environment Variables

- **INUEXPAND** This flag is set to 1 by the `install` command if file systems are to be expanded if necessary to do the recover (that is, the `--X` flag was passed to `install`). It is set to 0 if file systems are not to be expanded. If this environment variable is not set, the default is not to expand file systems.
- **INUSAVE** This flag is set to 1 by the `install` command if files are to be saved (that is, the `--N` flag was not passed), and otherwise set to 0. The `inurecv` command attempts to recover files if `INUSAVE` is set to 1. If `INUSAVE` is set to 0, `inurecv` performs no recovery and exits with a return code of `INUGOOD`. If this environment variable is not set, the default is to attempt to recover files.
- **INUSAVEDIR** The full path name to the directory where files are saved. If this environment variable is not set, then the directory used is `/usr/lpp/ProductName/inst_updt.save`.
- **OMDIR** The Object Data Manager object repository where the software vital product data is saved. If this environment variable is not set, the default directory used is `/etc/objrepos`.

Error Codes

- **INUADC1** A copy of a file from one directory to another was unsuccessful.
- **INUGOOD** No error conditions occurred.
- **INUNORP1** Unsuccessful replacement of a file in an archive file during program recovery.
- **INUNOSAV** The save directory does not exist.
- **INUNOSVF** A file that was saved in the save directory was not found.
Security
Privilege Control: Only the root user can run this command.

Examples
To recover all files previously saved for the `snaserv` program, enter:
inurecv snaserv

Files

/lpp/PackageName/FilessetName/V.R.M.F.save
Files saved for the root file tree.

/usr/lpp/PackageName/FilessetName/V.R.M.F.save
Files saved for the /usr file tree.

/usr/share/lpp/PackageName/FilessetName/V.R.M.F.save
Files saved for the /usr/share file tree.

Related Information
The `installp` command, `inusave` command.

inurest Command

Purpose
Performs simple archive and restore operations for the `installp` command and shell scripts. This command is used by the `installp` command and the install scripts.

Syntax

inurest [-d Device] [-q] ListFile ProductName

Description
The `inurest` command restores or archives all files listed in the file specified by the `ListFile` parameter.

If files are to be archived, there must be an archive control file, `/usr/lpp/ProductName/lpp.acf`, which contains entries in the following form:

ComponentFile LibraryFile.a

If the archive control file exists, the `inurest` command compares each of the file names in the `ListFile` file to the component files listed in `/usr/lpp/ProductName/lpp.acf`. Whenever the `inurest` command finds a match, the file name is added to a list of files that are archived. This list is then used to archive the restored files into a copy of the corresponding archive. When the archive is finished, the copy replaces the original file.

The `ListFile` parameter specifies the full path name of a file containing the relative path names, one per line, of files that a product needs to have restored.

The `ProductName` parameter specifies the software product to be restored.

Flags

-d Device Specifies the input device. The default device is the `/dev/rfd0` device.
-q Specifies quiet mode. Suppresses the prompt from `restore`. 
Environment Variables

INUEXPAND This flag is set to 1 by the installp command if file systems are to be expanded if necessary to do the restore (that is, the -X flag was passed). It is set to 0 if file systems are not to be expanded. If this environment variable is not set, the default is not to expand file systems.

INULIBDIR This is the directory where files that are specific to software product installation reside. If INULIBDIR is not set the /usr/lpp/ProductName directory is used.

INUTEMPDIR The directory to use for temporary space that is needed during the execution of this command. If this environment variable is not set, then the directory used is /tmp.

Error Codes

INUBADRC Restoration of an updated version of files was unsuccessful.
INUBADMIN Unusable flag was specified.
INUCHDIR Cannot change directory.
INUGOOD No error conditions occurred.
INUNOAP2 Unable to access the apply list.
INUNORP2 Failed replacing a constituent file in the archive file.
INUTOOFW One or more parameters are missing.
INUTOOMN Too many parameters are specified.

Security

Privilege Control: Only the root user can run this command.

Examples

To restore all the files listed in the ac file for the snaserv program, enter:

inurest /usr/lpp/snaserv/ac snaserv

Files

$INULIBDIR/lpp.acf Archive control file.

Related Information

The installp command, inucp command, inurecv command, inusave command.

inurid Command

Purpose

Removes information used for installation of diskless/dataless clients from the inst_root directories of installed software.

Syntax

inurid [-q | -r]

Description

The inurid command is used to remove files stored in the inst_root directories of installed software.

The names of these directories are of the forms: /usr/lpp/ProductName/inst_root for software products and /usr/lpp/PackageName/OptionName/v.r.m.f/inst_root for AIX Version 4 updates.
When this command is called, the **inst_root** directories are removed for all products and updates in the committed state. Also, an indicator is stored in the Software Vital Product Data indicating that the proper **inst_root** directory information is to be removed after the completion of each future installation action, for example, actions performed by the **installp** command.

**Attention:** One reason a user may want to remove **inst_root** directories is to save disk space. The implication of removing these directories is that the system cannot be used as a Shared Product Object Tree (SPOT) server of diskless/dataless clients. Also, once **inst_root** directories are removed from a system, there is no way to retrieve the directories. Therefore, the system cannot later be converted to a SPOT server without reinstalling the entire operating system.

**Flags**

- **-q** Queries whether **inst_root** directories have been removed from the system. A return value of 0 indicates that **inst_root** directories have not been removed and a return value of 1 indicates that the **inst_root** directories have been removed.
- **-r** Requests **inst_root** directories be removed from the system.

**Security**

Privilege Control: You must be the root user to run this command.

**Files**

```bash
/usr/lib/instl/inurid
```

Contains the **inurid** command.

**Related Information**

The **installp** command.

### inusave Command

**Purpose**

Saves files that are installed or updated during an installation procedure. This command is used by the **installp** command and the install scripts.

**Syntax**

```bash
inusave [ListFile] [ProductName]
```

**Description**

The **inusave** command saves the files and archived files that are listed in the file specified by the **ListFile** parameter for the **ProductName** software product. The **inusave** command is designed for use with the **installp** command.

The **inusave** command creates the `/usr/lpp/PackageNametFilesetName/V.R.M.F.save` directory if it does not already exist, where **Level** has the form `vv.rr.mmmm.ffff` and `vv` = the version, `rr` = the release, `mmmm` = the modification, and `ffff` = fix. This is the directory in which the installation procedures store saved files. The save directory is defined by the **INUSAVEDIR** environment variable.

The save directories for the `/ (root)`, `/usr`, and `/usr/share` parts of an installation are:

- `/lpp/PackageNametFilesetName/V.R.M.F.save`
- `/usr/lpp/PackageNametFilesetName/V.R.M.F.save`
• /usr/share/lpp/PackageName/FilessetName/V.R.M.F.save

respectively, when set up by the installp command. The installp command calls inusave for each of these three directories. The ListFile parameter is the full path name of the file that lists the files that are to be saved if a current copy exists.

If a file named in the ListFile file already exists, the inusave command copies file to the $INUSAVEDIR/update.n file, where n is an integer assigned by the inusave command. If the file does not exist, the inusave command assumes that this entry in the ListFile parameter represents either a new file or a file to be archived or processed by the archive procedure described later in this section.

The inusave command maintains a list of saved files in the $INUSAVEDIR/update.list file. This file is a stanza file with an entry for each saved file. Entries in the update.list file resemble the following:

/usr/bin/chkey:
update.n = update.1
option = bosnet.nfs.obj
_id = 209
_reserved = 0
_scratch = 0
lpp_id = 72
private = 0
file_type = 0
format = 1
loc0 = /usr/bin/chkey
size = 7800
checksum = 44561

/usr/bin/domainname:
update.n = update.2
option = bosnet.nfs.obj
_id = 210
_reserved = 0
_scratch = 0
lpp_id = 72
private = 0
file_type = 0
format = 1
loc0 = /usr/bin/domainname
size = 2526
checksum = 12439

In the previous example /usr/bin/chkey (the name of the stanza) is the name of an original file that was saved and update.1 is the name of the file in the $INUSAVEDIR directory to which it was copied. The file /usr/bin/chkey belongs to the bosnet.nfs.obj installable option of the software product bosnet. The stanza name and the first two items in the stanza (update.n and option) exist for each stanza in the update.list file. The remaining items in the stanza, which may vary, are information from the Software Vital Product Data (SWVPD) database.

An archived constituent file is saved if there is a valid archive control file, lpp.acf, in the current directory. If the lpp.acf file exists, the inusave command compares each of the file names in ListFile to the constituent file names in lpp.acf. When it finds a match, the inusave command uses the ar command to extract the constituent file from its associated archive file. It then moves the file to the $INUSAVEDIR/archive.n file, where n is an integer selected by the inusave command.

The inusave command maintains a list of the extracted files that have been saved in the $INUSAVEDIR/archive.list file. This file is a stanza file with an entry for each saved constituent file. Entries in the archive.list file resemble the following:
/prodx.filea:
archive.n = archive.1
arc_name = /usr/lib/productx/libprodx.a
option = productx.option1.obj
_id = 833
_reserved = 0
_scratch = 0
_lpp_id = 7
_private = 0
_file_type = 0
_format = 1
_loc0 = /prodx.filea
_loc1 = "h11,h12"
_loc2 = "/usr/lpp/productx.filea/s11,/usr/lpp/productx.filea/s12"
_size = 1611
_checksum = 62793

In the previous example /prodx.filea (the name of the stanza) is the name of the original constituent file that was saved and archive.1 is the name of the file in the SINUSAVEDIR directory to which it was copied. The /usr/lib/productx/libprodx.a is the full path name of the archive file defined in the lpp.acf archive control file. The constituent file /prodx.filea belongs to the productx.option1.obj installable option of the software product productx. The stanza name and the first three items in the stanza (archive.n, arc_name, and option) will exist for each stanza in the archive.list file. The remaining items in the stanza, which may vary, are information from the SWVPD database.

Parameters

ListFile Specifies the full path name of the file containing a list of relative path names, one per line, of files that are to be saved.

ProductName Specifies the installable software product whose files are to be saved.

Environment Variables

INUEXPAND This flag is set to 1 by the installp command if file systems are to be expanded if necessary to do the save (that is, the -X flag was passed to installp). It is set to 0 if file systems are not to be expanded. If this environment variable is not set, the default is not to expand file systems.

INUSAVE This flag is set to 1 by the installp command if files are to be saved (that is, the -N flag was not passed to installp). It is set to 0 if files are not to be saved. If this environment variable is not set, the default is to save files.

INUSAVEDIR The full path name to the directory where files are to be saved. If this environment variable is not set, then the directory to be used is /usr/lpp/ProductName/inst_updt.save.

INUTEMPDIR The directory to use for temporary space that is needed during the execution of this command. If this environment variable is not set, then the directory used is /tmp.

Error Codes

The following error codes are defined in /usr/include/inuerr.h:

INUBADSC A save directory could not be created.
INUBADC2 A file could not be copied from one directory to another.
INUGOOD No error conditions occurred.
INUNOAP1 Could not access ListFile.
INUTOOFW One or more parameters were missing.
INUTOOMN Too many parameters were specified.

Security

Privilege Control: Only the root user can run this command.
Examples
To save all the files listed in the snaserv.al file of the snaserv program, enter:

`inusave /usr/lpp/snaserv/snaserv.al snaserv`

Files

```
/usr/lpp/PackageName/lpp.acf
    Specifies the archive control file.
/lpp/PackageName/FilesetName/V.R.M.F.save
    Specifies the save directory for the root.
/usr/lpp/PackageName/FilesetName/V.R.M.F.save
    Specifies the save directory for the /usr files.
/usr/share/lpp/PackageName/FilesetName/V.R.M.F.save
    Specifies the save directory for the /usr/share files.
```

Related Information
The `installp` command, `inurecv` command.

inutoc Command

Purpose

Creates a .toc file for directories that have backup format file install images. This command is used by the `installp` command and the install scripts.

Syntax

```
inutoc [ Directory ]
```

Description

The `inutoc` command creates the .toc file in Directory. If a .toc file already exists, it is recreated with new information. The default installation image Directory is `/usr/sys/inst.images`. The `inutoc` command adds table of contents entries in the .toc file for every installation image in Directory.

The `installp` command and the `bffcreate` command call this command automatically upon the creation or use of an installation image in a directory without a .toc file.

Error Codes

```
INUBADIR    Usage error or Directory did not specify a directory.
INUCHDIR    Unable to change directories to Directory.
INUCRTOC    Could not create the .toc file.
INUGOOD     No errors occurred.
INUSYSFL    A system call failed.
```

Security

Privilege Control: Only the root user can run this command.

Examples

1. To create the .toc file for the `/usr/sys/inst.images` directory, enter:

```
inutoc
```
2. To create a .toc file for the /tmp/images directory, enter:
   
inutoc /tmp/images

Files

/usr/sys/inst.images     The default directory to create a .toc file.
.toc                     The file created by this command in the specified directory.

Related Information

The bffcreate command, installp command.

inuumsg Command

Purpose
Displays specific error or diagnostic messages provided by a software product’s installation procedures. This command is used by the installp command and the install scripts.

Syntax

inuumsg Number [ Argument1 ] [ , Argument2 ] [ , Argument3 ] [ , Argument4 ]

Description
The inuumsg command displays error or diagnostic messages for a software product's installation procedures. Rather than each procedure having its own text, messages are maintained in a central message catalog, /usr/lpp/msg/$LANG/inuumsg.cat. When you run the inuumsg command and specify the message Number, the error message is displayed. Up to four string arguments, Argument1 to Argument4, can be substituted into the message in the appropriate location.

Return Values

0 Indicates the message was found and displayed.
1 Indicates the message was not found and not displayed.

Security
Privilege Control: Only the root user can run this command.

Examples

To see error message number 3, enter:
inuumsg 3

Files

/usr/lpp/msg/$LANG/inuumsg.cat     The message catalog.

Related Information

The installp command.
invscout Command

Purpose
Surveys the host system for currently installed microcode or Vital Product Data (VPD).

Syntax

```
invscout [ -v ] [ -c ] [ -r ] [ -m machine_type_and_model ] [ -s serial_number ] [ -call
microcode_catalog_path ] [ -g ] [ -q ] [ -h ]
```

Description
The `invscout` command executes one instance of the stand-alone version of the Inventory Scout process. The `invscout` command starts the server daemon side of a client-server version.

The Inventory Scout process supports two survey types:
- Microcode Survey
- Vital Product Data (VPD) Survey (-v)

Microcode Survey

A Microcode Survey gathers data from the host system on currently installed microcode for `invscout`-supported systems, devices and adapters. It compares the gathered microcode levels to the latest levels available, and stores the data in a [Microcode Survey Upload File] that can be displayed locally via the webSM GUI, or uploaded to a Web server via the Internet.

A Microcode Survey also produces a [Microcode Survey Formatted Text Report File]. This file can be printed or displayed on a monitor and contains a subset of the information recorded in the upload file. This subset includes information about the `invscout` execution itself and the levels of currently installed microcode. The `-r` flag causes this report also to be sent to the screen from where the command was invoked.

All of the previous reports can contain information on the following:
- system microcode
- service microcode
- device and adapter microcode

VPD Survey (-v)

A VPD Survey stores the system VPD in a [VPD Survey Upload File] that can be uploaded to a Web server via the Internet. Once on a Web server, a CGI forwards the file to a repository and produces a Web page indicating the status of the operation.

No formatted text report is available for VPD Surveys.

Survey Results Concatenation (-c)

This option concatenates two or more [Microcode Survey Upload Files] into a single [Microcode Survey Concatenated Upload File] or two or more VPD Survey Upload Files into a single [VPD Survey Concatenated Upload File]. A Concatenated Upload File can be uploaded to a Web server using the Internet and processed by the server CGI to give the same results as would have been obtained by uploading and processing all the component files individually. The input files can be any valid upload files but, typically, this operation is done to simplify the task of uploading the results from several host systems.
The version of the command executing the concatenation and the versions of the commands that produced the files to be concatenated must all be the same.

Microcode Survey Upload Files cannot be concatenated with VPD Survey Upload Files.

Versions 2.1.0.0 and subsequent versions of this command do not require concatenation of Microcode Survey Upload Files, because the files are processed locally.

To concatenate a set of existing Microcode Survey upload files, do the following:
1. Copy the files into the Microcode Survey Concatenation Input Directory
2. Execute: `invscout -c`
3. Find the output Microcode Survey Concatenated Upload File in the same directory as the upload file for a Microcode Survey.

To concatenate a set of existing VPD Survey upload files, do the following:
1. Copy the files into the VPD Survey Concatenation Input Directory
2. Execute: `invscout -v -c`
3. Find the output VPD Survey Concatenated Upload File in the same directory as the upload file for a VPD Survey.

Flags

- **v**
  Sets the survey or concatenation type to VPD (the default is Microcode).
- **c**
  Concatenates existing survey upload files (the default is to perform a new survey).
- **r**
  For a Microcode Survey, sends a copy of the formatted text report file to the screen from which the command was invoked. This flag is ignored if either the -v or the -c flag is used.
- **m** `machine_type_and_model`
  For a VPD survey, allows input of the host platform machine type and model for hosts that use/require this information.
- **s** `serial_number`
  For a VPD survey, allows input of the host serial number for hosts that use/require this information.
- **catl** `microcode_catalog_path`
  Overrides the default location of the microcode catalog path.
- **g**
  Displays the versions of this command and of the logic database currently in use.
- **q**
  Suppresses most run-time messages.
- **h**
  Generates a help (usage) statement. If this flag is used, all other flags are ignored.

Exit Status

This command returns the following exit values:

0 Indicates successful completion.
Non-zero Indicates an error occurred.

If an error occurs, the command writes an error log.

Security

This command is owned by root, and is installed with the setuid bit ON so that any user can run it.
Examples

1. To run one Microcode Survey and send the results to a formatted text report file and an upload file, type:
   `invscout`

2. To run one VPD Survey and send the results to an upload file, type:
   `invscout -v`

3. To concatenate previously produced Microcode Survey upload files into a single upload file, type:
   `invscout -c`

   **Note:** Only applicable to Versions of this command prior to 2.1.0.0.

4. To concatenate previously produced VPD Survey upload files into a single upload file, type:
   `invscout -v -c`

Files

- `/usr/sbin/invscout` Contains the `invscout` command.
- `/var/adm/invscout/host.mup` Microcode Survey Upload File. The `host` variable is the host name of the system represented in the file.
- `/var/adm/invscout/host.vup` VPD Survey Upload File. The `host` variable is the host name of the system represented in the file.
- `/var/adm/invscout/invs.vpd.con.inp` VPD Survey Concatenation Input Directory
- `/var/adm/invscout/invs.con.mup` Microcode Survey Concatenated Upload File.
- `/var/adm/invscout/invs.con.vup` VPD Survey Concatenated Upload File.
- `/var/adm/invscout/invs.err` Error log written if the command encounters an error.
- `/var/adm/invscout/invscout.log` Log file.
- `/var/adm/invscout/tmp` Holds `invscout` temporary files. All files in this directory are deleted at the start of every execution of this command.

Related Information

The `invscout` command.

invscoutd Command

Purpose

Launches a permanent Inventory Scout server daemon.

Syntax

```
invscout [\[-o\] [\[-p\] Portno ] [\[-b\] Bufsize ] [\[-t\] Timeout ] [\[-v\] Verblev ]
```

Description

The `invscoutd` command implements a permanent Inventory Scout server daemon on one machine in a user's local network. The usual client is a Java applet running in the user's Web browser, which was downloaded from a central Inventory Scout CGI application.
Daemon initialization involves reading command line options and several local Inventory Scout companion files. When in operation, each client-server transaction involves reading from a well-known socket for a text string and returning a text report over the same socket.

The daemon maintains a record of its actions in a log file. Depending on the specified verbosity level, the log lines may contain startup and shutdown banners, traces of each call, detailed internal program traces, and error statements. Depending on the specified verbosity level, startup banners may also be written to stderr.

Protocols
Client connections to the daemon's socket use the Internet TCP/IP protocol. In a transaction, the invoking client applet sends an action request, as a URL-encoded text string, to the server daemon. The request is by any ASCII control character (x00 to x1F), which triggers the processing of the request.

Some requests require the client to pass additional data. In these cases, the additional data immediately follow the termination byte for a length specified in the action request.

With one exception (ACTION=PING), the server daemon always returns a pseudo MIME format text report written back over the same socket connection. The pseudo MIME format is used even for error results. The daemon terminates the returned text and the transaction itself by closing the socket, resulting in an end-of-file (EOF) indication to the invoking client. The client should close the socket at its end of the connection as soon as the EOF is received.

URL-encoded message
The action request string is a standard URL-encoded string. For example:
"ACTION=actionword&NAME1=value1&NAME2&NAME3=word%xx+word+word\0"

Supported Field Names and Values

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning/Use</th>
<th>Supported Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACTION</td>
<td>See the action request table that follows.</td>
<td>The left-hand column of the action request table constitutes a list of supported Values.</td>
</tr>
<tr>
<td>MRDM</td>
<td>Allows the client to provide a (cleartext) password for any ACTION that uses/requires this information. The value is case sensitive.</td>
<td>Any ASCII string (case sensitive).</td>
</tr>
<tr>
<td>DATALEN</td>
<td>This name must be present if additional binary data immediately follow an ACTION string termination byte, and must be absent if no additional data follow the termination byte. The integer value provided specifies the number of additional data bytes. If the client attempts to write more data than this, if the action does not accept the DATALEN parameter and discards any additional data, or if the action processor detects an early error, the daemon may prematurely close the client-to-server socket pipe. A transaction with n greater than a specific maximum value will immediately return an error code (see the -d command line option).</td>
<td>Any integer up to the value implied by the presence or absence of the -d command line option</td>
</tr>
<tr>
<td>CLIENT</td>
<td>Allows the client to identify itself for any ACTION that uses/requires this information.</td>
<td>The HSC value instructs Inventory Scout to allow certain actions that are only allowed when under the control of an HMC Inventory Scout master.</td>
</tr>
</tbody>
</table>
Supported Field Names and Values

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning/Use</th>
<th>Supported Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODEL</td>
<td>Allows the client to inform the server of the server's model number for VPD surveys that use/require this information.</td>
<td>Any ASCII string of up to 25 characters (restrictions apply with some machines)</td>
</tr>
<tr>
<td>SERIAL</td>
<td>Allows the client to inform the server of the server's serial number for VPD surveys that use/require this information.</td>
<td>Any ASCII string of up to 25 characters (restrictions apply with some machines)</td>
</tr>
</tbody>
</table>

Notes:
1. Field names and their values are separated by equal signs (=).
2. Name=Value pairs are separated by an & character.
3. The Name field is always case insensitive.
4. The Value field is case insensitive, unless documented otherwise.
5. The ACTION=keyword pair must always be present.
6. A string between ampersands without an equal sign is parsed as a Name with an Empty value.
7. Spaces can be represented by + (plus signs).
8. Binary characters may be coded as the escape sequence of a percent sign followed by exactly two hexadecimal chars (%xx). This escape sequence must also be used to code URL metacharacters like the &, = (equal sign), and + (plus sign) within a Value.
9. The control character termination byte must always be sent by the client.

Action Requests

<table>
<thead>
<tr>
<th>Action</th>
<th>MRDM</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PING</td>
<td>not required</td>
<td>The daemon immediately closes the socket, causing an immediate EOF in the client. This is the only action that does not return a result code or text of any kind. Example:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;action=ping\0&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&lt;EOF&gt;</td>
</tr>
<tr>
<td>ECHO</td>
<td>not required</td>
<td>The daemon returns a text report consisting of the original unparsed request string followed by a linefeed. A password (MRDM) is not required but will be echoed if provided, along with everything else. Additional data (DATALEN) are not required but will be echoed if present, as is, after the request string. For the ECHO request, DATALEN will be silently truncated to a maximum of 2000. Example:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;action=ECHO&amp;MRDM=xyz&amp;datalen=5\0abcde&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;RESULT=0\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;action=ECHO&amp;MRDM=xyz&amp;datalen=5\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;abcde&quot;&lt;EOF&gt;</td>
</tr>
</tbody>
</table>
### Action Requests

<table>
<thead>
<tr>
<th>Action</th>
<th>MRDM</th>
<th>Description</th>
</tr>
</thead>
</table>
| URLDECODE    | not required | The daemon returns a text report of the request string after parsing, and an exact copy of any subsequent data. A password (MRDM) is not required but will be parsed and returned if provided. Additional data (DATALEN) are not required but will be parsed and returned if provided; however, any actual additional data beyond the request string will be discarded. Each numbered line of the report exhibits one parsed Name=Value pair from the original string. Example:  
  
  "action=UrlDecode&subaction=xyz\0"
  
  "RESULT=0\n"
  
  "\n"
  
  " 0: ACTION UrlDecode\n"
  
  " 1: SUBACTION xyz\n"
  
  <EOF>                                                                                                                                                                                                                                           |
| TESTPWD      | required   | The daemon returns RESULT=0 if the MRDM password is valid. Otherwise it returns RESULT=2. Additional data (DATALEN) are not accepted and will be discarded if present. Example:  
  
  "ACTION=TESTPWD&MRDM=thepassword\0"
  
  "RESULT=0\n"
  
  "\n"
  
  <EOF>                                                                                                                                                                                                                                           |
| VERSIONS     | not required | The daemon reports the current version numbers of the Inventory Scout itself. Additional data (DATALEN) are not accepted and will be discarded if present. Example:  
  
  "ACTION=VERSIONS\0"
  
  "RESULT=0\n"
  
  "\n"
  
  "1.2.3.4\n"
  
  "5.6.7.8\n"
  
  <EOF>                                                                                                                                                                                                                                           |
| CATALOG      | required   | The daemon updates the scout's microcode catalog file with the file data passed. Both a password and the data length parameter must be included in the request string. The daemon does not necessarily have to execute as root for this action but it must have file write permissions to /var/adm/invscout/microcode/catalog.mic. Example:  
  
  "ACTION=CATALOG&MRDM=xyz&DATALEN=17042\0"
  
  "...17042 bytes of ascii data..."
  
  "RESULT=0\n"
  
  "\n"
  
  <EOF>                                                                                                                                                                                                                                           |
| MCODES       | required   | The daemon executes the Microcode Survey Option. Additional data (DATALEN) are not accepted and will be discarded if present. Example:  
  
  "ACTION=MCODES&MRDM=xyz\0"
  
  "RESULT=0\n"
  
  "\n"
  
  "Report Line 1\n"
  
  "Report Line 2\n"
  
  :                                                                                                                                                                                                                                                  |
  
  :                                                                                                                                                                                                                                                  |
  
  "Report Line N\n"
  
  <EOF>                                                                                                                                                                                                                                           |
Action Requests

<table>
<thead>
<tr>
<th>Action</th>
<th>MRDM</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>VPDS</td>
<td>required</td>
<td>The daemon executes the VPD Survey Option. Additional data (DATALEN) are not accepted and will be discarded if present. Example: &quot;ACTION=VPDS&amp;MRDM=xyz\0&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;RESULT=0\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;Report Line 1\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;Report Line 2\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&quot;Report Line N\n&quot;</td>
</tr>
<tr>
<td></td>
<td></td>
<td>&lt;EOF&gt;</td>
</tr>
</tbody>
</table>

Results

The daemon returns a text result in a pseudo MIME format. It returns a header consisting of one or more Name=Value pairs, each on a line by itself. The first Name=Value pair always is the result code in the form RESULT=number. The result code always is returned for every action, except the PING action.

Internal scout result codes applicable only to the Java applet client are not documented in the following information.

An optional free-form text report may follow the header lines depending on the result code. If there is a free-form text report, the header is first terminated by an empty line, such as two adjacent linefeeds.

In any event, the result report is terminated by an EOF indicator after reading the last of the report text from the socket. The EOF also signifies the end of the transaction itself.

Result Codes

<table>
<thead>
<tr>
<th>Result=</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Complete success.</td>
</tr>
<tr>
<td>1</td>
<td>Daemon aborted due to memory allocation error. This can happen in either the parent server daemon or one of the service children.</td>
</tr>
<tr>
<td>2</td>
<td>Service child daemon aborted because the required password (MRDM=password) was missing or not valid.</td>
</tr>
<tr>
<td>3</td>
<td>Service child daemon aborted because the action name-value pair (ACTION=keyword) was missing or not valid.</td>
</tr>
<tr>
<td>4</td>
<td>Service child daemon aborted because it was unable to reset its user ID to invscout.</td>
</tr>
<tr>
<td>21</td>
<td>Service child daemon aborted due to overflow of socket input buffer. The text report part of the result is a native language error message. Client must reduce the length of the request string, or kill and restart the daemon with an increased buffer size.</td>
</tr>
<tr>
<td>22</td>
<td>Service child daemon aborted due to socket read error. The text report part of the result is a native language error message including the system's I/O errno string. A logfile entry will also contain the system's errno string.</td>
</tr>
<tr>
<td>23</td>
<td>Service child daemon aborted due to socket read timeout. The text report part of the result is a native language error message. Client must send a control character termination byte after the end of the request string, and must always send as many data bytes as specified in the DATALEN parameter. The timeout period may be changed with the -t command line argument.</td>
</tr>
<tr>
<td>24</td>
<td>Service child daemon aborted due to premature EOF while reading request string. The text report part of the result is a native language error message. Client must send a termination byte after the end of the request string before closing the socket connection.</td>
</tr>
</tbody>
</table>
### Flags

Specify any arguments, beginning with a hyphen (-). Space is not allowed between a flag and its value.

- `-o`  Overwrites an existing logfile. If the `-o` flag is not specified, new logfile lines are appended to any existing logfile.

- `-p Portno`  Changes this server’s port number from the default 808 to `Port`.

- `-b Bufsize`  Inventory Scout commands are specified as URL-encoded strings read from a TCP/IP socket into a 1024 byte fixed length buffer. The `-b` flag can change the buffer size to `Bufsize` bytes if future protocol changes require a larger read buffer.

- `-t Timeout`  The client applet writes a control character termination byte at the end of the URL-encoded request string to indicate the end of the request. If the `invscoutd` daemon does not receive the termination byte within a timeout period, it aborts the transaction and closes the socket. Similarly the client must send all bytes of the additional data specified in the `DATALEN` parameter with sufficient speed to prevent timeout between read blocks. The `-t` option changes the default timeout period from 30 seconds to `Timeout` seconds.

- `-v Verblev`  The amount of detail written to the logfile and stderr depends on the verbosity level of the daemon. Each level incorporates the messages in the lower levels; increasing the verbosity level increases the number and types of messages that are written. The verbosity level is an integer ranging from 0 to 25. The `-v` flag changes the verbosity level from the default 18 to `Verblev`.

### Verbosity Levels

<table>
<thead>
<tr>
<th>Level</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>All error and status messages disabled.</td>
</tr>
<tr>
<td>5</td>
<td>Only fatal error messages are written. Fatal errors result in the death of the server. Usually, similar messages are written to both the <code>Logfile</code> and stderr.</td>
</tr>
<tr>
<td>10</td>
<td>All error messages are written. These include nonfatal errors such as protocol errors, as well as fatal errors. Nonfatal error messages are usually written only to the <code>Logfile</code>.</td>
</tr>
<tr>
<td>15</td>
<td>This level includes startup and shutdown banner messages. Simple banner messages are usually written to both the <code>Logfile</code> and stderr.</td>
</tr>
<tr>
<td>18</td>
<td>This level includes call trace status messages. Every client call results in a single trace message. This is the default level for the <code>invscoutd</code> daemon. Trace messages are written only to the <code>Logfile</code>.</td>
</tr>
<tr>
<td>20</td>
<td>This level includes program trace messages. Program traces are fairly detailed program execution status messages typically used for debugging purposes. This level is not suitable for usual production execution because over time, it floods the <code>Logfile</code> with large amounts of text. Trace messages are written only to the <code>Logfile</code>.</td>
</tr>
</tbody>
</table>
Verbosity Levels

<table>
<thead>
<tr>
<th>Level</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>This is the maximum level and includes extensive program debug messages. This level is not suitable for usual production execution. Trace messages are written only to the Logfile.</td>
</tr>
</tbody>
</table>

Exit Status

This command returns the following exit values:

- 0: Indicates successful initialization
- Non-zero: Indicates unsuccessful initialization

Security

The daemon must execute as effective user ID 0 (root). It is owned by root, and is installed with the "setuid" bit ON so that any user can launch it. At certain execution points, however, service children of the daemon reset their user ID to the authentication user ID invscout. The daemon will not execute unless the user invscout has been created on the host system.

By default, an accompanying cleartext password is required from the client for most operations. If the client's password does not match the system password for the authentication user ID invscout, the action exits with a return code. The authentication user ID cannot be changed.

Files

- /usr/sbin/invscoutd: Contains the invscoutd command
- /etc/security/password: Host system password file
- /var/adm/invscout/invscout.log: Log file

Related Information

The `invscout` command.

ioo Command

Purpose

Manages Input/Output tunable parameters.

Syntax

- `ioo [-p | -r] { o Tunable [ =NewValue ] }`
- `ioo [-p | -r] { d Tunable }
- `ioo [-p | -r] -D`
- `ioo [-p | -r] -a`
- `ioo -?`
- `ioo -h [ Tunable ]`
ioo [-L] Tunable

ioo [-x] Tunable

Note: Multiple -o, -d, -x and -L flags are allowed.

Description

Note: The ioo command can only be executed by root.

The ioo command configures Input/Output tuning parameters. This command sets or displays current or next boot values for all Input/Output tuning parameters. This command can also make permanent changes or defer changes until the next reboot. Whether the command sets or displays a parameter is determined by the accompanying flag. The -o flag performs both actions. It can either display the value of a parameter or set a new value for a parameter.

If a process appears to be reading sequentially from a file, the values specified by the minpgahead parameter determine the number of pages to be read ahead when the condition is first detected. The value specified by the maxpgahead parameter sets the maximum number of pages that are read ahead, regardless of the number of preceding sequential reads.

The operating system allows tuning of the number of file system bufstructs (numfsbuf) and the amount of data processed by the write-behind algorithm (numclust).

Attention: Misuse of the ioo command can cause performance degradation or operating-system failure. Before experimenting with ioo, you should be thoroughly familiar with Performance Overview of the Virtual Memory Manager (VMM).

Flags

-h [Tunable] Displays help about the Tunable parameter if one is specified. Otherwise, displays the ioo command usage statement.

-a Displays current, reboot (when used in conjunction with -r) or permanent (when used in conjunction with -p) value for all tunable parameters, one per line in pairs tunable = value. For the permanent option, a value is only displayed for a parameter if its reboot and current values are equal. Otherwise NONE displays as the value.

-d Tunable Resets Tunable to its default value. If a Tunable needs to be changed (that is it is currently not set to its default value) and is of type Bosboot or Reboot, or if it is of type Incremental and has been changed from its default value, and -r is not used in combination, it is not changed but a warning displays.

-D Resets all tunables to their default value. If tunables needing to be changed are of type Bosboot or Reboot, or are of type Incremental and have been changed from their default value, and -r is not used in combination, they are not changed but a warning displays.

-o Tunable [=NewValue] Displays the value or sets Tunable to NewValue. If a Tunable needs to be changed (the specified value is different than current value), and is of type Bosboot or Reboot, or if it is of type Incremental and its current value is bigger than the specified value, and -r is not used in combination, it is not changed but a warning displays.

When -r is used in combination without a NewValue, the nextboot value for tunable displays. When -p is used in combination without a NewValue, a value displays only if the current and next boot values for he Tunable are the same. Otherwise NONE displays as the value.
-p Specifies that the changes apply to both the current and reboot values when used in combination with the -o, -d or -D flags. Turns on the updating of the /etc/tunables/nextboot file in addition to the updating of the current value. These combinations cannot be used on Reboot and Bosboot type parameters, their current value cannot be changed.

When used with -a or -o without specifying a new value, the values display only if the current and next boot values for a parameter are the same. Otherwise, NONE displays as the value.

-r Makes changes apply to reboot values when used in combination with the -o, -d or -D flags. That is, turns on the updating of the /etc/tunables/nextboot file. If any parameter of type Bosboot is changed, the user is prompted to run bosboot.

When used with -a or -o without specifying a new value, next boot values for tunables display instead of current values.

-L [Tunable] Lists the characteristics of one or all tunables, one per line, using the following format:

<table>
<thead>
<tr>
<th>NAME</th>
<th>DEPENDENCIES</th>
<th>CUR</th>
<th>DEF</th>
<th>BOOT</th>
<th>MIN</th>
<th>MAX</th>
<th>UNIT</th>
<th>TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>minpgahead</td>
<td></td>
<td>2</td>
<td>2</td>
<td>0</td>
<td>4K</td>
<td>4K</td>
<td>pages</td>
<td>D</td>
</tr>
<tr>
<td>maxpgahead</td>
<td></td>
<td>8</td>
<td>8</td>
<td>0</td>
<td>4K</td>
<td>4K</td>
<td>pages</td>
<td>D</td>
</tr>
<tr>
<td>pd_npages</td>
<td></td>
<td>64K</td>
<td>64K</td>
<td>64K</td>
<td>1</td>
<td>512K</td>
<td>4KB</td>
<td>pages</td>
</tr>
<tr>
<td>maxrandert</td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>512K</td>
<td>4KB</td>
<td>pages</td>
<td>D</td>
</tr>
<tr>
<td>numfsbufs</td>
<td></td>
<td>186</td>
<td>186</td>
<td>186</td>
<td>16</td>
<td>16</td>
<td>16</td>
<td>16</td>
</tr>
</tbody>
</table>

where:
- CUR = current value
- DEF = default value
- BOOT = reboot value
- MIN = minimal value
- MAX = maximum value
- UNIT = tunable unit of measure
- TYPE = parameter type
  D (for Dynamic), S (for Static), R (for Reboot),
  M (for Mount), I (for Incremental) and C (for Connect)
- DEPENDENCIES = list of dependent tunable parameters, one per line

-x [Tunable] Lists characteristics of one or all tunables, one per line, using the following (spreadsheet) format:

| tunable, current, default, reboot, min, max, unit, type, {dtunable} |

where:
- current = current value
- default = default value
- reboot = reboot value
- min = minimal value
- max = maximum value
- unit = tunable unit of measure
- type = parameter type
  D (for Dynamic), S (for Static), R (for Reboot),
  B (for Bosboot), M (for Mount), I (for Incremental)
  and C (for Connect)
- dtunable = space separated list of dependent tunable parameters

-? Displays the iio command usage statement.

Any change (with -o, -d or -D) to a parameter of type Mount will result in a message being displayed to warn the user that the change is only effective for future mountings.

Any change (with -o, -d or -D flags) to a parameter of type Connect will result in inetd being restarted, and a message being displayed to warn the user that the change is only effective for future socket connections.
Any attempt to change (with -o, -d or -D) a parameter of type **Bosboot** or **Reboot** without -r, will result in an error message.

Any attempt to change (with -o, -d or -D but without -r) the current value of a parameter of type **Incremental** with a new value smaller than the current value, will result in an error message.

**Tunable Parameters Type**

All the tunable parameters manipulated by the tuning commands (**no**, **nfso**, **vmo**, **ioo**, and **schedo**) have been classified into these categories:

- **Dynamic** If the parameter can be changed at any time
- **Static** If the parameter can never be changed
- **Reboot** If the parameter can only be changed during reboot
- **Bosboot** If the parameter can only be changed by running bosboot and rebooting the machine
- **Mount** If changes to the parameter are only effective for future file systems or directory mounts
- **Incremental** If the parameter can only be incremented, except at boot time
- **Connect** If changes to the parameter are only effective for future socket connections

For parameters of type **Bosboot**, whenever a change is performed, the tuning commands automatically prompt the user to ask if they want to execute the **bosboot** command. For parameters of type **Connect**, the tuning commands automatically restart the **inetd** daemon.

Note that the current set of parameters managed by the **ioo** command only includes Static, Dynamic, Mount and Incremental types.

**Compatibility Mode**

When running in pre 5.2 compatibility mode (controlled by the **pre520tune** attribute of **sys0**, see [Tuning Enhancements for AIX 5.2](#) in the AIX 5L Version 5.2 Performance Management Guide), reboot values for parameters, except those of type **Bosboot**, are not really meaningful because in this mode they are not applied at boot time.

In pre 5.2 compatibility mode, setting reboot values to tuning parameters continues to be achieved by imbedding calls to tuning commands in scripts called during the boot sequence. Parameters of type **Reboot** can therefore be set without the -r flag, so that existing scripts continue to work.

This mode is automatically turned ON when a machine is MIGRATED to AIX 5L Version 5.2. For complete installations, it is turned OFF and the reboot values for parameters are set by applying the content of the `/etc/tunables/nextboot` file during the reboot sequence. Only in that mode are the -r and -p flags fully functional. See [Kernel Tuning](#) in AIX 5L Version 5.2 Performance Tools Guide and Reference for more information.
Tunable Parameters

**hd_pbuf_cnt**

**Purpose:**
Specifies the total number of pbufs that the LVM uses.

**Values:**
- Default: For the 32-bit kernel, the default is \((4 + \text{number of physical disks with an open logical volume}) \times 64\). For the 64-bit kernel, the default is \((4 + \text{number of physical disks with an open logical volume}) \times 128\).
- Range:
- Type: Incremental

**Diagnosis:**
Value of `hd_pendqblked` is non-zero, which indicates that LVM had to wait for pbufs. To see value of `hd_pendqblked`, use the `vmstat -v` command.

**Tuning:**
Useful to increase if there are a substantial number of simultaneous I/Os and value of `hd_pendqblked` is non-zero.

**Refer To:**
[File System Buffer Tuning](#)

**lvm_bufcnt**

**Purpose:**
Specifies the number of LVM buffers for raw physical I/Os.

**Values:**
- Default: 9
  - Range: 1 to 64
  - Type: Dynamic

**Diagnosis:**
Applications performing large writes to striped raw logical volumes are not obtaining the desired throughput rate.

**Tuning:**
LVM splits large raw I/Os into multiple buffers of 128 K a piece. Default value of 9 means that about 1 MB I/Os can be processed without waiting for more buffers. If a system is configured to have striped raw logical volumes and is doing writes greater than 1.125 MB, increasing this value may help the throughput of the application. If performing larger than 1 MB raw I/Os, it might be useful to increase this value.

**Refer To:**
[File System Buffer Tuning](#)
maxpgahead

**Purpose:**
Specifies the maximum number of pages to be read ahead when processing a sequentially accessed file.

**Values:**
- Default: 8 (the default should be a power of two and should be greater than or equal to minpgahead)
- Range: 0 to 4096
- Type: Dynamic

**Diagnosis:**
Observe the elapsed execution time of critical sequential-I/O-dependent applications with the time command.

**Tuning:**
Because of limitations in the kernel, do not exceed 512 as the maximum value used. The difference between minfree and maxfree should always be equal to or greater than maxpgahead. If execution time decreases with higher maxpgahead, observe other applications to ensure that their performance has not deteriorated.

**Refer To:**
Sequential Page Read-Ahead

maxrandwrt

**Purpose:**
Specifies a threshold (in 4 KB pages) for random writes to accumulate in RAM before subsequent pages are flushed to disk by the write-behind algorithm. The random write-behind threshold is on a per-file basis.

**Values:**
- Default: 0
- Range: 0 to largest_file_size_in_pages
- Type: Dynamic

**Diagnosis:**
`vmstat -n` shows page out and I/O wait peaks on regular intervals (usually when the sync daemon is writing pages to disk).

**Tuning:**
Useful to set this value to 1 or higher if too much I/O occurs when `syncd` runs. Default is to have random writes stay in RAM until a sync operation. Setting maxrandwrt ensures these writes get flushed to disk before the sync operation has to occur. However, this could degrade performance because the file is then being flushed each time. Tune this option to favor interactive response time over throughput. After the threshold is reached, all subsequent pages are then immediately flushed to disk. The pages up to the threshold value stay in RAM until a sync operation. A value of 0 disables random write-behind.

**Refer To:**
Sequential and Random Write Performance
**minpgahead**

**Purpose:**
Specifies the number of pages with which sequential read-ahead starts.

**Values:**
- Default: 2
- Range: 0 to 4096 (should be a power of two)
- Type: Dynamic

**Diagnosis:**
Observe the elapsed execution time of critical sequential-I/O-dependent applications with time command.

**Tuning:**
Useful to increase if there are lots of large sequential accesses. Observe other applications to ensure that their performance has not deteriorated. Value of 0 may be useful if I/O pattern is purely random.

**Refer To:**
Sequential Page Read-Ahead

**numclust**

**Purpose:**
Specifies the number of 16 k clusters processed by the sequential write-behind algorithm of the VMM.

**Values:**
- Default: 1
- Range: 0 to any positive integer
- Type: Dynamic

**Diagnosis:**
N/A

**Tuning:**
Useful to increase if there is a need to keep more pages in RAM before scheduling them for I/O when the I/O pattern is sequential. May be appropriate to increase if striped logical volumes or disk arrays are being used.

**Refer To:**
Sequential and Random Write Performance

**numfsbufs**

**Purpose:**
Specifies the number of file system bufstructs.

**Values:**
- Default: 186 (value is dependent on the size of the bufstruct)
- Range:
- Type: Mount

**Diagnosis:**
N/A

**Tuning:**
If the VMM must wait for a free bufstruct, it puts the process on the VMM wait list before the start I/O is issued and will wake it up once a bufstruct has become available. May be appropriate to increase if striped logical volumes or disk arrays are being used.

**Refer To:**
File System Buffer Tuning
pd_npages

Purpose:
Specifies the number of pages that should be deleted in one chunk from RAM when a file is deleted.

Values:
Default: 65536
Range: 1 to largest filesize_in_pages
Type: Dynamic

Diagnosis:
Real-time applications that experience sluggish response time while files are being deleted.

Tuning:
Tuning this option is only useful for real-time applications. If real-time response is critical, adjusting this option may improve response time by spreading the removal of file pages from RAM more evenly over a workload.

Refer To:
File System Buffer Tuning

sync_release_ilock

Purpose:
If set, will cause a sync() to flush all I/O to a file without holding the i-node lock, and then use the i-node lock to do the commit.

Values:
Default: 0 (off)
Range 0 or 1
Type: Dynamic

Diagnosis:
I/O to a file is blocked when the syncd daemon is running.

Tuning:
Default value of 0 means that the i-node lock is held while all dirty pages of a file are flushed.

Refer To:
File Synchronization Performance Tuning
j2_inodeCacheSize

Purpose:
Controls the amount of memory JFS2 will use for the inode cache.

Values:
Default:
- 400 (AIX 5.2F)
- 100 (AIX 5.3)
Range: 1 to 1000 (enforced by /proc/sys. Controlled by /proc/sys/fs/jfs2/inode_cache_size.
Type: Dynamic

Diagnosis:
N/A

Tuning:
This tunable does not explicitly indicate the amount that will be used, but is instead a scaling factor. It is used in combination with the size of the main memory to determine the maximum memory usage for the inode cache. The valid values for this tunable are between 1 and 1000, inclusive. This value represents a maximum size. The system may not reach the maximum size. If the tunable is lowered, a best effort will be made to lower the size. It may not be possible to lower the size immediately, so shortly after tuning the size of the cache may be higher than the maximum. It is not recommended to set the values above 400, but the interface is provided in case it helps certain workloads. Values above 400 may exhaust the kernel heap. Similarly, low values (values below 100) may be too few depending on the workload and demands on the system. This may result in errors such as "File table full" being returned to the application. Also, on the 32-bit kernel, the ideal maximum may never be reached due to a restricted kernel heap. If this value is changed, the value for metadata_cache_size may need to be reconsidered if tuning for a specific workload. The inode cache controls the inode data stored in memory, so if the workload uses a large number of files, increasing the maximum size of the inode cache may help. If the workload uses few files, but the files are large, increasing the maximum size of the metadata cache may help; use the metadata_cache_size tunable for that.
j2_metadataCacheSize

**Purpose:**
Controls the amount of memory JFS2 will use for the metadata cache.

**Values:**
- Default: 8
- Range: 1 to 1000 (enforced by `/proc/sys`. Controlled by `/proc/sys/fs/jfs2/inode_cache_size`.
- Type: Dynamic

**Diagnosis:**
N/A

**Tuning:**
This tunable does not explicitly indicate the amount that will be used, but is instead a scaling factor; it is used in combination with the size of the main memory to determine the maximum memory usage for the inode cache. The valid values for this tunable are between 1 and 1000, inclusive. This value represents a maximum size. The system may not reach the maximum size. If the tunable is lowered, a best effort will be made to lower the size. It may not be possible to lower the size immediately, so shortly after tuning the size of the cache may be higher than the maximum. It is not recommended to set the values above 400, but the interface is provided in case it helps certain workloads. Values above 400 may exhaust the kernel heap. Similarly, low values (values below 100) may be too few depending on the workload and demands on the system. This may result in extremely slow access times. Also, on the 32-bit kernel, the ideal maximum may never be reached due to a restricted kernel heap. If this value is changed, the value for `inode_cache_size` may need to be reconsidered if tuning for a specific workload. The inode cache controls the inode data stored in memory, so if the workload uses a large number of files, increasing the maximum size of the inode cache may help; use the `inode_cache_size` tunable for that. If the workload uses few files, but the files are large, increasing the maximum size of the metadata cache may help.

j2_maxPageReadAhead

**Purpose:**
Specifies the maximum number of pages to be read ahead when processing a sequentially accessed file on Enhanced JFS.

**Values:**
- Default: 8
- Range: 
- Type: Dynamic

**Diagnosis:**
N/A

**Tuning:**
The difference between minfree and maxfree should always be equal to or greater than `j2_maxPageReadAhead`. If run time decreases with higher a `j2_maxPageReadAhead` value, observe other applications to ensure that their performance has not deteriorated.
**j2_maxRandomWrite**

**Purpose:**
Specifies a threshold for random writes to accumulate in RAM before subsequent pages are flushed to disk by the Enhanced JFS's write-behind algorithm. The random write-behind threshold is on a per-file basis.

**Values:**
- Default: 0
- Range:
- Type: Dynamic

**Diagnosis:**
N/A

**Tuning:**
Useful if too many pages are flushed out by `syncd`.

**j2_minPageReadAhead**

**Purpose:**
Specifies the minimum number of pages to be read ahead when processing a sequentially accessed file on Enhanced JFS.

**Values:**
- Default: 2
- Range:
- Type: Dynamic

**Diagnosis:**
N/A

**Tuning:**
Useful to increase if there are lots of large sequential accesses. Observe other applications to ensure that their performance has not deteriorated. Value of 0 may be useful if I/O pattern is purely random.

**j2_nBufferPerPagerDevice**

**Purpose:**
Specifies the number of file system bufstructs for Enhanced JFS.

**Values:**
- Default: 512
- Range:
- Type: Mount

**Diagnosis:**
Using `vmstat -v`, the value of xpagerbufwaitcnt increases fast.

**Tuning:**
If the kernel must wait for a free bufstruct, it puts the process on a wait list before the start I/O is issued and will wake it up once a bufstruct has become available. May be appropriate to increase if striped logical volumes or disk arrays are being used.
**j2_nPagesPerWriteBehindCluster**

**Purpose:** Specifies the number of pages per cluster processed by Enhanced JFS's write behind algorithm.

**Values:**
- Default: 32
- Range: 
- Type: Dynamic

**Diagnosis:** N/A

**Tuning:** Useful to increase if there is a need to keep more pages in RAM before scheduling them for I/O when the I/O pattern is sequential. May be appropriate to increase if striped logical volumes or disk arrays are being used.

**j2_nRandomCluster**

**Purpose:** Specifies the distance apart (in clusters) that writes have to exceed in order for them to be considered as random by the Enhanced JFS's random write behind algorithm.

**Values:**
- Default: 0
- Range: 
- Type: Dynamic

**Diagnosis:** N/A

**Tuning:** Useful to increase if there is a need to keep more pages in RAM before scheduling them for I/O when the I/O pattern is random and random write behind is enabled (j2_maxRandomWrite).

**jfs_clread_enabled**

**Purpose:** This tunable controls whether JFS uses clustered reads on all files.

**Values:**
- Default: 0
- Range: 0 - 1
- Type: Dynamic

**Diagnosis:** N/A

**Tuning:** In general, this option is not needed, but it may benefit certain workloads that have relatively random read access patterns.
jfs_use_read_lock

Purpose:
Controls whether JFS uses a shared lock when reading from a file. If this option is turned off, two processes cannot disrupt each other's read.

Values:
Default: 1
Range: 0 - 1
Type: Dynamic

Diagnosis:
N/A

Tuning:
Certain workloads may benefit from this.

Examples
1. To list the current and reboot value, range, unit, type and dependencies of all tunables parameters managed by the ioo command, type:
   
   ioo -L

2. To turn sync_release_ilock on, type:
   
   ioo -o sync_release_ilock=1

3. To display help on j2_nPagesPerWriteBehindCluster, type:
   
   ioo -h j2_nPagesPerWriteBehindCluster

4. To set maxrandwrt to 4 after the next reboot, type:
   
   ioo -r -o maxrandwrt=4

5. To permanently reset all ioo tunable parameters to default, type:
   
   ioo -p -D

6. To list the reboot value of all ioo parameters, type:
   
   ioo -a

7. To list (spreadsheet format) the current and reboot value, range, unit, type and dependencies of all tunables parameters managed by the ioo command, type:
   
   ioo -x

Related Information
The niso, no, schedo, tuncheck, tunchange, tundefault, tunrestore, tunsave, and vmo command.


Tuning Enhancements for AIX 5.2 in AIX 5L Version 5.2 Performance Management Guide.

iostat Command

Purpose
Reports Central Processing Unit (CPU) statistics and input/output statistics for the entire system, adapters, tty devices, disks and CD-ROMs.

Syntax
iostat [-S] [-a] [-d] [-t] [-T] [-m] [ PhysicalVolume ... ] [ Interval [ Count ] ]
Description

The **iostat** command is used for monitoring system input/output device loading by observing the time the physical disks are active in relation to their average transfer rates. The **iostat** command generates reports that can be used to change system configuration to better balance the input/output load between physical disks and adapters.

The first report generated by the **iostat** command provides statistics concerning the time since the system was booted. Each subsequent report covers the time since the previous report. All statistics are reported each time the **iostat** command is run. The report consists of a tty and CPU header row followed by a row of tty and CPU statistics. On multiprocessor systems, CPU statistics are calculated system-wide as averages among all processors.

If the **-s** flag is specified, a system-header row is displayed followed by a line of statistics for the entire system. The hostname of the system is printed in the system-header row.

If the **-a** flag is specified, an adapter-header row is displayed followed by a line of statistics for the adapter. This will be followed by a disk-header row and the statistics of all the disks/CD-ROMs connected to the adapter. Such reports are generated for all the disk adapters connected to the system.

A disks header row is displayed followed by a line of statistics for each disk that is configured. If the **PhysicalVolume** parameter is specified, only those names specified are displayed.

If the **PhysicalVolume** parameter is specified, one or more alphabetic or alphanumeric physical volumes can be specified. If the **PhysicalVolume** parameter is specified, the tty and CPU reports are displayed and the disk report contains statistics for the specified drives. If a specified logical drive name is not found, the report lists the specified name and displays the message **Drive Not Found**. If no Logical Drive Names are specified, the report contains statistics for all configured disks and CD-ROMs. If no drives are configured on the system, no disk report is generated. The first character in the **PhysicalVolume** parameter cannot be numeric.

The **Interval** parameter specifies the amount of time in seconds between each report. The first report contains statistics for the time since system startup (boot). Each subsequent report contains statistics collected during the interval since the previous report. The **Count** parameter can be specified in conjunction with the **Interval** parameter. If the **Count** parameter is specified, the value of count determines the number of reports generated at **Interval** seconds apart. If the **Interval** parameter is specified without the **Count** parameter, the **iostat** command generates reports continuously.

The **iostat** command is useful in determining whether a physical volume is becoming a performance bottleneck and if there is potential to improve the situation. The % utilization field for the physical volumes indicates how evenly the file activity is spread across the drives. A high % utilization on a physical volume is a good indication that there may be contention for this resource. Since the CPU utilization statistics are also available with the **iostat** report, the percentage of time the CPU is in I/O wait can be determined at the same time. Consider distributing data across drives if the I/O wait time is significant and the disk utilization is not evenly distributed across volumes.

**Note:** Some system resource is consumed in maintaining disk I/O history for the **iostat** command. Use the **sysconfig** subroutine, or the System Management Interface Tool (SMIT) to stop history accounting.

Reports

The **iostat** command generates four types of reports, the tty and CPU Utilization report, the Disk Utilization report, the System throughput report and the Adapter throughput report.

**tty and CPU Utilization Report:** The first report generated by the **iostat** command is the tty and CPU Utilization Report. For multiprocessor systems, the CPU values are global averages among all processors. Also, the I/O wait state is defined system-wide and not per processor. The report has the following format:
This information is updated at regular intervals by the kernel (typically sixty times per second). The tty report provides a collective account of characters per second received from all terminals on the system as well as the collective count of characters output per second to all terminals on the system.

**Methods Used to Compute CPU Disk I/O Wait Time:** Operating system version 4.3.3 and later contain enhancements to the method used to compute the percentage of CPU time spent waiting on disk I/O (wio time). The method used in AIX 4.3.2 and earlier versions of the operating system can, under certain circumstances, give an inflated view of wio time on SMPs. The wio time is reported by the commands `sar (%wio)`, `vmstat (wa)` and `iostat (% iowait)`.

The method used in AIX 4.3.2 and earlier versions is as follows: At each clock interrupt on each processor (100 times a second per processor), a determination is made as to which of the four categories (usr/sys/wio/idle) to place the last 10 ms of time. If the CPU was busy in usr mode at the time of the clock interrupt, then usr gets the clock tick added into its category. If the CPU was busy in kernel mode at the time of the clock interrupt, then the sys category gets the tick. If the CPU was not busy, a check is made to see if any I/O to disk is in progress. If any disk I/O is in progress, the wio category is incremented. If no disk I/O is in progress and the CPU is not busy, the idle category gets the tick. The inflated view of wio time results from all idle CPUs being categorized as wio regardless of the number of threads waiting on I/O. For example, systems with just one thread doing I/O could report over 90 percent wio time regardless of the number of CPUs it has.

The method used in AIX 4.3.3 and later is as follows: The change in operating system version 4.3.3 is to only mark an idle CPU as wio if an outstanding I/O was started on that CPU. This method can report much lower wio times when just a few threads are doing I/O and the system is otherwise idle. For example, a system with four CPUs and one thread doing I/O will report a maximum of 25 percent wio time. A system with 12 CPUs and one thread doing I/O will report a maximum of 8 percent wio time. NFS client reads/writes go through the VMM, and the time that biods spend in the VMM waiting for an I/O to complete is now reported as I/O wait time.

**Disk Utilization Report:** The second report generated by the `iostat` command is the Disk Utilization Report. The disk report provides statistics on a per physical disk basis. The report has a format similar to the following:

- `% tm_act` Indicates the percentage of time the physical disk was active (bandwidth utilization for the drive).
- `Kbps` Indicates the amount of data transferred (read or written) to the drive in KB per second.
- `tps` Indicates the number of transfers per second that were issued to the physical disk. A transfer is an I/O request to the physical disk. Multiple logical requests can be combined into a single I/O request to the disk. A transfer is of indeterminate size.
- `Kb_read` The total number of KB read.
- `Kb_wrtn` The total number of KB written.

Statistics for CD-ROM devices are also reported.

For large system configurations where a large number of disks are configured, the system can be configured to avoid collecting physical disk input/output statistics when the `iostat` command is not
executing. If the system is configured in the above manner, the first Disk report displays the message Disk History Since Boot Not Available instead of the disk statistics. Subsequent interval reports generated by the iostat command contain disk statistics collected during the report interval. Any tty and CPU statistics after boot are unaffected. If a system management command is used to re-enable disk statistics keeping, the first iostat command report displays activity from the interval starting at the point that disk input/output statistics were enabled.

System Throughput Report: This report is generated if the -s flag is specified. This report provides statistics for the entire system. This report has the following format:

<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kbps</td>
<td>Indicates the amount of data transferred (read or written) in the entire system in KB per second.</td>
</tr>
<tr>
<td>tps</td>
<td>Indicates the number of transfers per second issued to the entire system.</td>
</tr>
<tr>
<td>Kb_read</td>
<td>The total number of KB read from the entire system.</td>
</tr>
<tr>
<td>Kb_wrtt</td>
<td>The total number of KB written to the entire system.</td>
</tr>
</tbody>
</table>

Adapter Throughput Report: This report is generated if the -a flag is specified. This report provides statistics on an adapter-by-adapter basis. This report has the following format:

<table>
<thead>
<tr>
<th>Metric</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kbps</td>
<td>Indicates the amount of data transferred (read or written) in the adapter in KB per second.</td>
</tr>
<tr>
<td>tps</td>
<td>Indicates the number of transfers per second issued to the adapter.</td>
</tr>
<tr>
<td>Kb_read</td>
<td>The total number of KB read from the adapter.</td>
</tr>
<tr>
<td>Kb_wrtt</td>
<td>The total number of KB written to the adapter.</td>
</tr>
</tbody>
</table>

Disk Input/Output History: To improve performance on, the collection of disk input/output statistics has been disabled. To enable the collection of this data, type:

```bash
chdev -l sys0 -a iostat=true
```

To display the current settings, type:

```bash
lsattr -E -l sys0 -a iostat
```

If the collection of disk input/output history is disabled, the first disk report of iostat output displays the message Disk History Since Boot Not Available instead of disk statistics. As before, subsequent interval reports generated by the iostat command contain disk statistics collected during the report interval.

Flags

- **-a** Displays the adapter throughput report.
- **-d** The -d flag is exclusive of the -t flag and displays only the disk utilization report.
- **-s** Displays the system throughput report.
- **-m** The -m flag will print the path statistics for the following:
  - Paths to MPIO (Multi-Path I/O) enabled devices.
  - Paths in the ESS machines.

The throughput is per device. The throughput for all the paths to that device follow the throughput of that device.

For ESS machines, the vpaths will be treated as disks and hdisks will be treated as Paths. Internally the vpaths are actually disks and hdisks are the paths to them. For MPIO enabled devices, the path name will be represented as Path0, Path1, Path2 and so on. The numbers 0, 1, 2, and so on are the path IDs provided by the lspath command. Since paths to a device can be attached to any adapter, the adapter report will report the path statistics under each adapter. The disk name will be a prefix to all the paths. For all MPIO enabled devices, the adapter report will print the path names as hdisk10_Path0, hdisk0_Path1, and so on. For all ESS Machines, the adapter report will print the path names as vpath0_hdisk3, vpath10_hdisk25, and so on.

- **-t** The -t flag is exclusive of the -d flag and displays only the tty and cpu usage reports.
-T  Prints the time-stamp next to each line of output of iostat. The time-stamp displays in the HH:MM:SS format.

Notes:
- The -s and -a flags can both be specified to display both the system and adapter throughput reports.
- If the -a flag is specified with the -t flag, the tty and CPU report is displayed, followed by the adapter throughput report. Disk Utilization reports of the disks connected to the adapters, will not be displayed after the Adapter throughput report.
- If the -a flag is specified with the -d flag, tty and CPU report will not be displayed. If Physical Volume parameter is specified, the Disk Utilization Report of the specified Physical volume will be printed under the corresponding adapter to which it belongs.

Examples
1. To display a single history since boot report for all tty, CPU, and Disks, type:
   ```
   iostat
   ```
2. To display a continuous disk report at two second intervals for the disk with the logical name disk1, type:
   ```
   iostat -d disk1 2
   ```
3. To display six reports at two second intervals for the disk with the logical name disk1, type:
   ```
   iostat disk1 2 6
   ```
4. To display six reports at two second intervals for all disks, type:
   ```
   iostat -d 2 6
   ```
5. To display six reports at two second intervals for three disks named disk1, disk2, disk3, type:
   ```
   iostat disk1 disk2 disk3 2 6
   ```
6. To print the System throughput report, type:
   ```
   iostat -s
   ```
7. To print the Adapter throughput report, type:
   ```
   iostat -a
   ```
8. To print the System and Adapter throughput reports, with only the tty and CPU report (no disk reports), type:
   ```
   iostat -sat
   ```
9. To print the System and Adapter throughput reports with the Disk Utilization reports of hdisk0 and hdisk7, enter
   ```
   iostat -sad hdisk0 hdisk7
   ```
10. To display time stamp next to each line of output of iostat, type:
    ```
    iostat -T
    ```

File
/usr/bin/iostat
Contains the iostat command.

Related Information
The `vmstat` command.

The `/dev/kmem` special file.

The `knlist` subroutine, `sysconfig` subroutine.
The Input and Output Handling Programmer's Overview in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs describes the files, commands, and subroutines used for low-level, stream, terminal, and asynchronous I/O interfaces.

**ipcrm Command**

**Purpose**
Removes message queue, semaphore set, or shared memory identifiers.

**Syntax**
```
ipcrm [-m SharedMemoryID | -M SharedMemoryKey | -q MessageID | -Q MessageKey | -s SemaphoreID | -S SemaphoreKey]
```

**Description**
The `ipcrm` command removes one or more message queues, semaphore sets, or shared memory identifiers.

**Flags**
- `-m SharedMemory ID` Removes the shared memory identifier `SharedMemoryID`. The shared memory segment and data structure associated with `SharedMemoryID` are also removed after the last detach operation.
- `-M SharedMemoryKey` Removes the shared memory identifier, created with the key `SharedMemoryKey`. The shared memory segment and data structure associated with it are also removed after the last detach operation.
- `-q MessageID` Removes the message queue identifier `MessageID` and the message queue and data structure associated with it.
- `-Q MessageKey` Removes the message queue identifier, created with the key `MessageKey`, and the message queue and data structure associated with it.
- `-s SemaphoreID` Removes the semaphore identifier `SemaphoreID` and the set of semaphores and data structure associated with it.
- `-S SemaphoreKey` Removes the semaphore identifier, created with the key `SemaphoreKey`, and the set of semaphores and data structure associated with it.

The `msgctl`, `shmctl`, and `semctl` subroutines provide details of the remove operations. The identifiers and keys can be found by using the `ipcs` command.

**Examples**
To remove the shared memory segment associated with `SharedMemoryID` 18602, enter:
```
ipcrm -m 18602
```

**Related Information**
The `ipcs` command.

The `msgget` subroutine, `semctl` subroutine, `semget` subroutine, `shmctl` subroutine, `shmget` subroutine.

**ipcs Command**

**Purpose**
Reports interprocess communication facility status.

**Syntax**
```
```

**Description**
The `ipcs` command writes to the standard output information about active interprocess communication facilities. If you do not specify any flags, the `ipcs` command writes information in a short form about currently active message queues, shared memory segments, semaphores, remote queues, and local queue headers.

The column headings and the meaning of the columns in an `ipcs` command listing follow. The letters in parentheses indicate the flags that cause the corresponding heading to appear. The designator `all` means the heading is always displayed. These flags only determine what information is provided for each facility. They do not determine which facilities are listed.

- **T (all)** the type of facility. There are three facility types:
  - q message queue
  - m shared memory segment
  - s semaphore

- **ID (all)** the identifier for the facility entry.

- **KEY (all)** the key used as a parameter to the `msgget` subroutine, the `semget` subroutine, or the `shmget` subroutine to make the facility entry.

**Note:** The key of a shared memory segment is changed to `IPC_PRIVATE` when the segment is removed until all processes attached to the segment detach from it.
MODE

(all) the facility access modes and flags. The mode consists of 11 characters that are interpreted as follows:

The first two characters can be the following:

R If a process is waiting on a msgrecv system call.
S If a process is waiting on a msgsnd system call.
D If the associated shared memory segment has been removed. It disappears when the last process attached to the segment detaches it.
C If the associated shared memory segment is to be cleared when the first attach is run.
- If the corresponding special flag is not set.

The next nine characters are interpreted as three sets of 3 bits each. The first set refers to the owner's permissions; the next to permissions of others in the user group of the facility entry; and the last to all others. Within each set, the first character indicates permission to read, the second character indicates permission to write or alter the facility entry, and the last character is currently unused.

The permissions are indicated as follows:

r If read permission is granted.
w If write permission is granted.
a If alter permission is granted.
- If the indicated permission is not granted.

OWNER
(all) The login name of the owner of the facility entry.
GROUP
(all) The name of the group that owns the facility entry.
CREATOR
(a,c) The login name of the creator of the facility entry.
CGROUP
(a,c) The group name of the creator of the facility entry.

Note: For the OWNER, GROUP, CREATOR, and CGROUP, the user and group IDs display instead of the login names.

CBYTES
(a,o) The number of bytes in messages currently outstanding on the associated message queue.
QNUM
(a,o) The number of messages currently outstanding on the associated message queue.
QBYTES
(a,b) The maximum number of bytes allowed in messages outstanding on the associated message queue.
LSPID
(a,p) The ID of the last process that sent a message to the associated queue. If the last message sent was from a process in a node other than the node that holds the queue, LSPID is the PID of the kernel process that actually placed the message on the queue, not the PID of the sending process.
LRPID
(a,p) The ID of the last process that received a message from the associated queue. If the last message received was from a process in a node other than the node that holds the queue, LRPID is the PID of the kernel process that actually received the message on the queue, not the PID of the receiving process.
STIME
(a,t) The time when the last message was sent to the associated queue. For remote queues, this is the server time. No attempt is made to compensate for time-zone differences between the local clock and the server clock.
RTIME
(a,t) The time when the last message was received from the associated queue. For remote queues, this is the server time. No attempt is made to compensate for any time-zone differences between the local clock and the server clock.
CTIME
(a,t) The time when the associated entry was created or changed. For remote queues, this is the server time. No attempt is made to compensate for any time-zone differences between the local clock and the server clock.
NATTCH
(a,o) The number of processes attached to the associated shared memory segment.
SEGSZ
(a,b) The size of the associated shared memory segment.
CPI
(a,p) The process ID of the creator of the shared memory entry.
Flags

- a  Uses the -b, -c, -o, -p and -t flags.
- b  Writes the maximum number of bytes in messages on queue for message queues, the size of segments for shared memory, and the number of semaphores in each semaphores set.
- c  Writes the login name and group name of the user that made the facility.
- CCoreFile  Uses the file specified by the CoreFile parameter in place of the /dev/mem file. The CoreFile parameter is a memory image file produced by the Ctrl-(left)Alt-Pad1 key sequence.
- l  When used with the -S flag, this flag writes the list of SIDs unwrapped.
- m  Writes information about active shared memory segments.
- nKernel  Uses the specified Kernel (the /usr/lib/boot/unix file is the default).
- o  Writes the following usage information:
  - Number of messages on queue
  - Total number of bytes in messages in queue for message queues
  - Number of processes attached to shared memory segments
- p  Writes process number information:
  - Process number of the last process to receive a message on message queues
  - Process number of last process to send a message on message queues
  - Process number of the creating process
  - Process number of last process to attach or detach on shared memory segments
- P  Writes the list of SIDs (segment IDs) associated with the shared memory ID, along with the number of bytes pinned to that segment and an indication of whether the segment is large-page enabled or not. If the segment is large-page enabled, a 'Y' is displayed, otherwise a 'N' is displayed.
- q  Writes information about active message queues.
- s  Writes information about active semaphore set.
- S  Writes the list of SID attached to shared memory id.
- t  Writes time information:
  - Time of the last control operation that changed the access permissions for all facilities
  - Time of the last msgsnd and msgrcv on message queues
  - Time of the last shmat and shmdt on shared memory
  - Time of the last semop on semaphore sets
- T  Writes the output of the -t flag with the date.

Notes:
1. If the user specifies either the -C or -N flag, the real and effective UID/GID is set to the real UID/GID of the user invoking ipcs.
2. Values can change while ipcs is running; the information it gives is guaranteed to be accurate only when it was retrieved.
Example

Example output from entering `ipcs` without flags:

IPC status from /dev/mem as of Mon Aug 14 15:03:46 1989

<table>
<thead>
<tr>
<th>T</th>
<th>ID</th>
<th>KEY</th>
<th>MODE</th>
<th>OWNER</th>
<th>GROUP</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>q 0</td>
<td>0x00010381</td>
<td>-Rwr-rw-</td>
<td>root</td>
<td>system</td>
</tr>
<tr>
<td>0</td>
<td>q 65537</td>
<td>0x00010307</td>
<td>-Rrw-rw-</td>
<td>root</td>
<td>system</td>
</tr>
<tr>
<td>0</td>
<td>q 65538</td>
<td>0x00010311</td>
<td>-Rrw-rw-</td>
<td>root</td>
<td>system</td>
</tr>
<tr>
<td>0</td>
<td>q 65539</td>
<td>0x0001032f</td>
<td>-Rrw-rw-</td>
<td>root</td>
<td>system</td>
</tr>
<tr>
<td>0</td>
<td>q 65540</td>
<td>0x0001031b</td>
<td>-Rrw-rw-</td>
<td>root</td>
<td>system</td>
</tr>
<tr>
<td>0</td>
<td>q 65541</td>
<td>0x00010339</td>
<td>--rw-rw-</td>
<td>root</td>
<td>system</td>
</tr>
<tr>
<td>0</td>
<td>q 6</td>
<td>0x0002fe03</td>
<td>-Rrw-rw-</td>
<td>root</td>
<td>system</td>
</tr>
</tbody>
</table>

Message Queues:

- `q` 0 0x00010381 -Rrw-rw- root system
- `q` 65537 0x00010307 -Rrw-rw- root system
- `q` 65538 0x00010311 -Rrw-rw- root system
- `q` 65539 0x0001032f -Rrw-rw- root system
- `q` 65540 0x0001031b -Rrw-rw- root system
- `q` 65541 0x00010339 --rw-rw- root system
- `q` 6 0x0002fe03 -Rrw-rw- root system

Shared Memory:

- `m` 65537 0x00000000 DCrw----- root system
- `m` 720898 0x00010300 DCrw-rw-rw root system
- `m` 65539 0x00000000 DCrw----- root system

Semaphores:

- `s` 131072 0x4d02086a --ra--- root system
- `s` 65537 0x00000000 --ra----- root system
- `s` 131072 0x000133d0 --ra----- 7003 30720

Files

- `/usr/lib/boot/unix` Specifies the system kernel image.
- `/dev/mem` Specifies memory.
- `/etc/passwd` Specifies user names.
- `/etc/group` Specifies group names.
- `/usr/include/sys/ipc.h` Contains the header file.

Related Information

The `ipcrm` command.

The `svmon` command.

The `msgrcv` subroutine, `msgsnd` subroutine, `semop` subroutine, `shmat` subroutine, `shmdt` subroutine.


ipfilter Command

Purpose

Extracts different operation headers from an ipreport output file and displays them in a table. Some customized nfs information regarding requests and replies is also provided.

Syntax

```
ipfilter [-f [untxca]] [-s [untxca]] [-n [-d milliseconds]] ipreport_output_file
```

Description

The `ipfilter` command extracts specific information from an ipreport output file and displays it to a table. The operation headers currently recognized are: udp, nfs, tcp, ipx, icmp, atm. The `ipfilter` command has three different types of reports:

- A single file (`ipfilter.all`) that displays a list of all the selected operations. The table displays packet number, Time, Source and Destination, Length, Sequence #, Ack #, Source Port, Destination Port, Network Interface, and Operation Type.
**Individual files for each selected header** (ipfilter.udp, ipfilter.nfs, ipfilter.tcp, ipfilter.ipx, ipfilter.icmp, ipfilter.atm). The information is the same as ipfilter.all.

- A file **nfs.rpt** that reports on nfs requests and replies. The table contains: Transaction ID #, Type of Request, Status of Request, Call Packet Number, Time of Call, Size of Call, Reply Packet Number, Time of Reply, Size of Reply, and Elapsed millisecond between call and reply.

**Flags**

- **u n t x c a** Specifies operation headers (udp, nfs, tcp, ipx, and icmp and atm respectively).
- **-d milliseconds** Only Call/Reply pairs whose elapsed time is greater than milliseconds are to be shown.
- **-f [ u n t x c a ]** Selected operations are to be shown in ipfilter.all.
- **-n** Generates an nfs.rpt.
- **-s [ u n t x c ]** Separate files are to be produced for each of the selected operations.

**Related Information**
The iptrace daemon, ipreport command.

**ipreport Command**

**Purpose**
Generates a packet trace report from the specified packet trace file.

**Syntax**
/usr/sbin/ipreport [-e] [-r] [-n] [-s] LogFile

**Description**
The /usr/sbin/ipreport command generates a trace report from the specified trace file created by the iptrace command. The LogFile parameter specifies the name of the file containing the results of the Internet Protocol trace. This file is created by the iptrace command.

**Flags**

- **-e** Generates the trace report in EBCDIC format. The default format is ASCII.
- **-r** Decodes remote procedure call (RPC) packets.
- **-n** Includes a packet number to facilitate easy comparison of different output formats.
- **-s** Prepends the protocol specification to every line in a packet.

**Related Information**
The iptrace command, trpt command.

**ipsec_convert Command**

**Purpose**
Converts IP Security tunnel export files to a format that can be imported by the IBM Secure Network Gateway.

**Syntax**
ipsec_convert SNG22 | FW31 [-f export_directory]
Description
IP Security allows the importing of IBM Secure Network Gateway 2.2 and IBM Firewall 3.1 tunnels using the `imptun` command. However, these firewall products do not allow the reverse capability. The `ipsec_convert` command allows for this capability by translating exported IP Security tunnels to IBM Firewall tunnels. The translated files will be placed in the current directory.

Flags

SNG22 | FW31 Specifies whether the format of the resulting files will be in the format of IBM Secure Network Gateway 2.2 or IBM Firewall 3.1 format.
-f Specifies the directory where the exported IPSec files are located.

Related Information
The `imptun` command.

ipsecstat Command

Purpose

Syntax
`ipsecstat [-c] [-d] [-A] [-E]`

Description
The `ipsecstat` command, used without flags, displays the status of the IP Security devices, the crypto algorithms installed for IP Security, and the statistics of IP Security packets.

The command can be used with flags to only list the status of IP Security devices, to only list the installed algorithms, or to reset statistic counters (to zero).

Flags

-c Resets statistics counters (after displaying current value). The -c flag cannot be used with any other flags.
-d Lists only the status of the IP Security devices. The -d flag cannot be used with any other flags.
-A Lists only the installed authentication algorithms. The -A flag cannot be used with any other flags.
-E Lists only the installed encryption algorithms. The -E flag cannot be used with any other flags.

ipsectrcbuf Command

Purpose
Lists the contents of tracing buffers in the IP Security subsystem.

Syntax
`ipsectrcbuf [-I {0|1|2}]`
Description
The IP Security subsystem maintains a memory resident trace buffer to help debug if there is a problem. The content of the buffer, a fixed number of the most recent trace messages, will be in a system dump or can be listed by running this command with no arguments.

Flags
-\( d \)  Sets the IP Security trace level. By default, of the nine IP Security trace hooks, only IPSEC_ERROR trace messages are put into the buffer. To enable or disable the other trace hooks, use the \( d \) flag with one of the following values:

\[
\begin{align*}
0 & \quad \text{Only IPSEC_ERROR trace messages are written to the buffer. This is the default.} \\
1 & \quad \text{IPSEC_FILTER, IPSEC_CAPSUL, IPSEC_CRYPTO, IPSEC_TUNNEL, as well as IPSEC_ERROR trace messages are written to the buffer.}\ \\
2 & \quad \text{All IP Security trace messages are put into the buffer (that includes IPSEC_FILTER_INFO, IPSEC_CAPSUL_INFO, IPSEC_CRYPTO_INFO, and IPSEC_TUNNEL_INFO as well as those in level 1).}
\end{align*}
\]

iptrace Daemon

Purpose
Provides interface-level packet tracing for Internet protocols.

Syntax

Description
The /usr/sbin/iptrace daemon records Internet packets received from configured interfaces. Command flags provide a filter so that the daemon traces only packets meeting specific criteria. Packets are traced only between the local host on which the iptrace daemon is invoked and the remote host.

If the iptrace process was started from a command line without the System Resource Controller (SRC), it must be stopped with the kill \( -15 \) command. The kernel extension loaded by the iptrace daemon remains active in memory if iptrace is stopped any other way.

The LogFile parameter specifies the name of a file to which the results of the iptrace command are sent. To format this file, run the ipreport command. The ipreport command may display the message TRACING DROPPED xxxx PACKETS. This count of dropped packets indicates only the number of packets that the iptrace command was unable to grab because of a large packet, the size of which exceeded the socket-receive buffer size. This message does NOT mean that the packets are being dropped by the system.

Note: The file specified by the LogFile parameter should not reside on an NFS-mounted file system. Specifying an output file on an NFS-mounted file system can cause the iptrace daemon to hang. You may be unable to kill the iptrace daemon if it hangs, requiring that you restart the system.

Flags
-\( a \)  Suppresses ARP packets.
-\( b \)  Changes the \( d \) or \( s \) flags to bidirectional mode.
-d Host  Records packets headed for the destination host specified by the Host variable. The Host variable can be a host name or an Internet address in dotted-decimal format.

If used with the -b flag, the -d flag records packets both going to and coming from the host specified by the Host variable.

-e  Enables promiscuous mode on network adapters that support this function.

-i Interface  Records packets received on the interface specified by the Interface variable.

-L Log_size  This option causes iptrace to log data in such that the LogFile is copied to LogFile.old at the start and also every time it becomes approximately Log_size bytes long.

-P Protocol_list  Records packets that use the protocol specified by the Protocol_list variable which is a comma separated list of protocols. The Protocols can be a decimal number or name from the /etc/protocols file.

-p Port_list  Records packets that use the port number specified by the Port_list variable which is a comma separated list of ports. The Port_list variable can be a decimal number or name from the /etc/services file.

-s Host  Records packets coming from the source host specified by the Host variable. The Host variable can be a host name or an Internet address in dotted-decimal format.

If used with the -b flag, the -s flag records packets both going to and coming from the host specified by the Host variable.

-u  Unloads the kernel extension that was loaded by the iptrace daemon at startup.

Examples

1. To start the iptrace daemon with the System Resource Controller (SRC), enter:
   
   startsrc -s iptrace -a "/tmp/nettrace"

   To stop the iptrace daemon with SRC enter the following:
   
   stopsrc -s iptrace

2. To record packets coming in and going out to any host on every interface, enter the command in the following format:
   
   iptrace /tmp/nettrace

   The recorded packets are received on and sent from the local host. All packet flow between the local host and all other hosts on any interface is recorded. The trace information is placed into the /tmp/nettrace file.

3. To record packets received on an interface from a specific remote host, enter the command in the following format:
   
   iptrace -i en0 -p telnet -s airmail /tmp/telnet.trace

   The packets to be recorded are received on the en0 interface, from remote host airmail, over the telnet port. The trace information is placed into the /tmp/telnet.trace file.

4. To record packets coming in and going out from a specific remote host, enter the command in the following format:
   
   iptrace -i en0 -s airmail -b /tmp/telnet.trace

   The packets to be recorded are received on the en0 interface, from remote host airmail. The trace information is placed into the /tmp/telnet.trace file.

Related Information

The [preport](#) command, the [tcpdump](#) command.

The [etc/protocols](#) file format, [etc/services](#) file format.
isC2host Command

Purpose
Determine the C2 status of a system.

Syntax
isC2host [-i | -s]

Description
The isC2host command returns the configuration status of the host machine. If the host has been configured to operate in C2 mode, the command exits with a zero (true) code. If the host has not been configured to operate in C2 mode, the command exits with a non-zero (false) code.

This command may be used in shell scripts where the security status of the host must be known.

The -i option is used to determine the installation status of the system. The C2 status of the system is determined by examining the ODM database, and the exit status indicates whether or not the system was installed in C2 mode.

The -s option is used to initialize AIX in C2 mode and may only be issued by the root user. The C2 status of the system is determined by examining the ODM database. On a system that has not been installed with C2, as indicated by the ODM, this option performs no operation.

Flags
- Determine the C2 installation status of the system.
- Set the C2 status of the system from the ODM.

Subcommands

Exit Status

0  When used with no options, the system has been initialized to operate in C2 mode. When used with the -s flag, the system was successfully initialized according to the C2 mode setting defined in the ODM database. When used with the -i flag, the system was installed with C2 enabled.

1  When used with no options, the system has not been initialized to operate in C2 mode. When used with the -s flag, the system could not be initialized to operate in the security mode that was defined in the ODM. When used with the -i flag, the system was installed with C2 enabled but is not currently operating in C2 mode.

2  When used with the -s option, the isC2host command was executed by a non-root user. When used with the -i option, the system was not installed with C2 enabled.

3  The isC2host command was executed with an invalid command line option.

Files
/usr/sbin/isC2host  Contains the isC2host command.
Related Information
The `ch2admin` command, `ls2admin` command, `mk2admin` command, `rm2admin` command.

The `sysconfig()` subroutine.

The Object Data Manager subsystem.

**isCChost Command**

**Purpose**
Determine the Common Criteria enabled status of a system.

**Syntax**
```
isCChost [-i] [-s]
```

**Description**
The `isCChost` command returns the configuration status of the host machine. If the host has been configured to operate in Common Criteria enabled mode, the command exits with a zero (true) code. If the host has not been configured to operate in Common Criteria enabled mode, the command exits with a non-zero (false) code.

This command may be used in shell scripts where the security status of the host must be known.

The `-i` option is used to determine the installation status of the system. The Common Criteria enabled status of the system is determined by examining the ODM database, and the exit status indicates whether or not the system was installed in Common Criteria enabled mode.

The `-s` option is used to initialize AIX in Common Criteria enabled mode and may only be issued by the root user. The Common Criteria enabled status of the system is determined by examining the ODM database. On a system that has not been installed with Common Criteria enabled, as indicated by the ODM, this option performs no operation.

**Flags**
- `-i` Determine the Common Criteria enabled installation status of the system.
- `-s` Set the Common Criteria enabled status of the system from the ODM.

**Subcommands**

**Exit Status**

0 When used with no options, the system has been initialized to operate in Common Criteria enabled mode. When used with the `-s` flag, the system was successfully initialized according to the Common Criteria enabled mode setting defined in the ODM database. When used with the `-i` flag, the system was installed with Common Criteria enabled enabled.

1 When used with no options, the system has not been initialized to operate in Common Criteria enabled mode. When used with the `-s` flag, the system could not be initialized to operate in the security mode that was defined in the ODM. When used with the `-i` flag, the system was installed with Common Criteria enabled but is not currently operating in Common Criteria enabled mode.

2 When used with the `-s` option, the `isCChost` command was executed by a non-root user. When used with the `-i` option, the system was not installed with Common Criteria enabled.
The `isCChost` command was executed with an invalid command line option.

**Files**

`/usr/sbin/isCChost` Contains the `isCChost` command.

**Related Information**

The `chCCadmin` command, `lsCCadmin` command, `mkCCadmin` command, `rmCCadmin` command.

The `sysconfig()` subroutine.

The `Object Data Manager` subsystem.

---

**istant Command**

**Purpose**

Examines i-nodes.

**Syntax**

`istant (FileName | i-nodeNumber Device)`

**Description**

The `istant` command displays the i-node information for a particular file. You can specify the file either by providing a file or directory name with the `FileName` parameter or by providing an i-node number with the `i-nodeNumber` parameter and a device name with the `Device` parameter. You can specify the `Device` parameter as either a device name or as a mounted file system name.

If you specify the `FileName` parameter, the `istant` command writes the following information about the file:

- Device where the file resides
- i-node number of the file, on that device
- File type, such as normal, directory, and block device
- File access permissions
- Name and identification number of the owner and group

**Note:** The owner and group names for remote files are taken from the local `/etc/passwd` file.

- Number of links to the file
- If the i-node is for a normal file, length of the file
- If the i-node is for a device, major and minor device designations
- Date of the last i-node update
- Date of the last file modification
- Date of the last reference to the file

If you specify the `i-nodeNumber` and `Device` parameters, the `istant` command also displays, in hexadecimal values, the block numbers recorded in the i-node.

**Note:** The `Device` parameter cannot refer to a remote device.
Examples

1. To display the information in the i-node corresponding to the /usr/bin/ksh file, enter:
   istat /usr/bin/ksh

   This command displays the i-node information for the /usr/bin/ksh file. The information looks similar to the following:
   Inode 10360 on device 10/6 File
   Protection: r-xr-xr-x
   Owner: 2(bin) Group: 2(bin)
   Link count: 2 Length 372298 bytes
   Last updated: Wed May 13 14:08:13 1992
   Last modified: Wed May 13 13:57:00 1992

2. To display i-node information by specifying a file i-node number, enter:
   istat 10360 /dev/hd2

   This command displays the information contained in the i-node identified by the number 10360 on the /dev/hd2 device. In addition to the information shown in Example 1, this displays:
   Block pointers (hexadecimal):
   2a9a 2a9b 2a9c 2a9d 2a9e 2a9f 2aa0 2aa1

   These numbers are addresses of the disk blocks that make up the /usr/bin/ksh file.

Files

/usr/bin/istat Contains the istat command.

Related Information

The fsdb command.

The filesystems file, fs/sys.h file.


Files Overview in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices provides information on working with files.

Directory Overview in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices provides an introduction on i-nodes and how they are used by the file system.

jobs Command

Purpose
Displays status of jobs in the current session.

Syntax
jobs [l|n|p] [JobID ...]
Description

The **jobs** command displays the status of jobs started in the current shell environment. If no specific job is specified with the **JobID** parameter, status information for all active jobs is displayed. If a job termination is reported, the shell removes that job’s process ID from the list of those known by the current shell environment.

The **/usr/bin/jobs** command does not work when operating in its own command execution environment, because that environment does not have applicable jobs to manipulate. For this reason, the **jobs** command is implemented as a Korn shell or POSIX shell regular built-in command.

If the **-p** flag is specified, output consists of one line for each process ID. If no flags are specified, standard output is a series of lines with the following fields:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>job-number</strong></td>
<td>Indicates the process group number to use with the <strong>wait</strong>, <strong>fg</strong>, <strong>bg</strong>, and <strong>kill</strong> commands. When used with these commands, prefix the job number with a % (percent sign).</td>
</tr>
<tr>
<td><strong>current</strong></td>
<td>A + (plus sign) identifies the job that will be used as a default for the <strong>fg</strong> or <strong>bg</strong> commands. This job ID can also be specified using the %+ (percent sign, plus) or %% (double percent sign). A - (minus sign) identifies the job that becomes the default if the current default job exits. This job ID can also be specified using %- (percent sign, minus). For other jobs, the <strong>current</strong> field is a space character. Only one job can be identified with a +, and only one job can be identified with a -. If there is a single suspended job, that will be the current job. If there are at least two suspended jobs, then the previous job is also suspended.</td>
</tr>
<tr>
<td><strong>state</strong></td>
<td>Displays one of the following values (in the POSIX locale):</td>
</tr>
<tr>
<td>Running</td>
<td>Indicates that the job has not been suspended by a signal and has not exited.</td>
</tr>
<tr>
<td>Done</td>
<td>Indicates that the job completed and returned exit status 0.</td>
</tr>
<tr>
<td>Done (<strong>code</strong>)</td>
<td>Indicates that the job completed normally and that it exited with the specified non-zero exit status code. This code is expressed as a decimal number.</td>
</tr>
<tr>
<td>Stopped</td>
<td>Indicates that the job was suspended.</td>
</tr>
<tr>
<td>Stopped (<strong>SIGTSTP</strong>)</td>
<td>Indicates that the <strong>SIGTSTP</strong> signal suspended the job.</td>
</tr>
<tr>
<td>Stopped (<strong>SIGSTOP</strong>)</td>
<td>Indicates that the <strong>SIGSTOP</strong> signal suspended the job.</td>
</tr>
<tr>
<td>Stopped (<strong>SIGTTIN</strong>)</td>
<td>Indicates that the <strong>SIGTTIN</strong> signal suspended the job.</td>
</tr>
<tr>
<td>Stopped (<strong>SIGTTOU</strong>)</td>
<td>Indicates that the <strong>SIGTTOU</strong> signal suspended the job.</td>
</tr>
<tr>
<td><strong>command</strong></td>
<td>The associated command that was given to the shell.</td>
</tr>
</tbody>
</table>

If the **-I** flag is specified, a field containing the process group ID is inserted before the **state** field. Also, more processes in a process group may be output on separate lines, using only the **job-number** and **command** fields.

Flags

- **-I** (lowercase L) Provides more information about each job listed. This information includes the job number, current job, process group ID, state, and the command that initiated the job.
-n Displays only jobs that have stopped or exited since last notified.
-p Displays the process IDs for the process group leaders for the selected jobs.

By default the jobs command displays the status of all stopped jobs, all running background jobs, and all jobs whose status has changed but not been reported by the shell.

**Exit Status**
The following exit values are returned:

<table>
<thead>
<tr>
<th>Exit Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Successful completion.</td>
</tr>
<tr>
<td>&gt;0</td>
<td>An error occurred.</td>
</tr>
</tbody>
</table>

**Examples**
1. To display the status of jobs in the current environment, enter:

   jobs -l

   The screen displays a report similar to the following output:

   +[4] 139 Running CC – C foo c&
   -[3] 465 Stopped mail morris
   [2] 687 Done(1) foo.bar&

2. To display the process ID for the job whose name begins with "m," enter:

   job -p %m

   Using the jobs reported in Example 1, the screen displays the following process ID:

   465

**Files**

- `/usr/bin/ksh` Contains the Korn shell `jobs` built-in command.
- `/usr/bin/jobs` Contains the `jobs` command.

**Related Information**
The `bg` command, `csh` command, `fg` command, `kill` command, `ksh` command, `wait` command.

---

**join Command**

**Purpose**
Joins the data fields of two files.

**Syntax**

```
join [ -a FileNumber | -v FileNumber ] [ -e String ] [ -o List ] [ -t Character ] [ -1 Field ] [ -2 Field ]
```

**Description**
The `join` command reads the files specified by the `File1` and `File2` parameters, joins lines in the files according to the flags, and writes the results to standard output. The `File1` and `File2` parameters must be text files. Both `File1` and `File2` must be sorted in the collating sequence of sort `-b` on the field that they are being joined by before invoking the `join` command.
One line appears in the output for each identical join field appearing in both files. The join field is the field in the input files examined by the `join` command to determine what will be included in the output. The output line consists of the join field, the rest of the line from the file specified by the `File1` parameter, and the rest of the line from the file specified by the `File2` parameter. Specify standard input in place of either the `File1` or `File2` parameter by substituting a `-` (dash) as the file name. Both input files cannot be specified with a `-` (dash).

Fields are usually separated by a space, a tab character, or a new-line character. In this case, the `join` command treats consecutive separators as one and discards leading separators.

**Flags**

- `-1 Field` Joins the two files using the field specified by the `Field` variable in the `File1` input file. The value of the `Field` variable must be a positive decimal integer.

- `-2 Field` Joins the two files using the field specified by the `Field` variable in the `File2` input file. The value of the `Field` variable must be a positive decimal integer.

- `-a FileNumber` Produces an output line for each line in the file specified by the `FileNumber` variable whose join fields do not match any line in the other input file. The output lines are produced in addition to the default output. The value of the `FileNumber` variable must be either 1 or 2, corresponding to the files specified by the `File1` and `File2` parameters, respectively. If this flag is specified with the `-v` flag, this flag is ignored.

- `-e String` Replaces empty output fields with the string specified by the `String` variable.

- `-o List` Constructs an output line to comprise the fields specified in the `List` variable. One of the following forms applies to the `List` variable:

  \[FileNumber.Field\]

  Where `FileNumber` is a file number and `Field` is a decimal-integer field number. Separate multiple fields with a `,` (comma) or space characters with quotation marks around the multiple fields.

  \[0 (zero)\]

  Represents the join field. The `-o 0` flag essentially selects the union of the join fields.

- `-t Character` Uses the character specified by the `Character` parameter as the field separator character in the input and the output. Every appearance of the character in a line is significant. The default separator is a space. With default field separation, the collating sequence is that of the `sort` command. If you specify `-t`, the sequence is that of a plain sort. To specify a tab character, enclose it in single quotation marks.

- `-v FileNumber` Produces an output line for each line in the file specified by the `FileNumber` variable whose join fields do not match any line in the other input file. Default output is not produced. The value of the `FileNumber` variable must be either 1 or 2, corresponding to the files specified by `File1` and `File2` parameters, respectively. If this flag is specified with the `-a` flag, the `-a` flag is ignored.

**Exit Status**

This command returns the following exit values:

- `0` Successful completion.
- `>0` An error occurred.

**Examples**

**Note:** The vertical alignment shown in the following examples might not be consistent with your output.

1. To perform a simple join operation on two files where the first fields are the same, type:

   ```
   join phonedir names
   ```
If the phonedir file contains the following names:

Adams A.  555-6235
Dickerson B.  555-1842
Erwin G.  555-1234
Jackson J.  555-0256
Lewis B.  555-3237
Norwood M.  555-5341
Smartt D.  555-1540
Wright M.  555-1234
Xandy G.  555-5015

and the names file contains these names and department numbers:

Erwin Dept. 389
Frost Dept. 217
Nicholson Dept. 311
Norwood Dept. 454
Wright Dept. 520
Xandy Dept. 999

the join command displays:

Erwin G.  555-1234  Dept. 389
Norwood M.  555-5341  Dept. 454
Wright M.  555-1234  Dept. 520
Xandy G.  555-5015  Dept. 999

Each line consists of the join field (the last name), followed by the rest of the line found in the phonedir file and the rest of the line in the names file.

2. To display unmatched lines with the join command, type:

```
join -a phonedir names
```

If the phonedir and names files are the same as in Example 1, the join command displays:

Erwin G.  555-1234  Dept. 389
Frost Dept. 217
Nicholson Dept. 311
Norwood Dept. 454
Wright Dept. 520
Xandy Dept. 999

This command performs the same join operation as in Example 1, and also lists the lines of names that have no match in the phonedir file. The names Frost and Nicholson are included in the listing, even though they do not have entries in the phonedir file.

3. To display selected fields with the join command, type:

```
join -o 2.3,2.1,1.2,1.3 phonedir names
```

This displays the following fields in the order given:

<table>
<thead>
<tr>
<th>Field 3 of names</th>
<th>Department number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Field 1 of names</td>
<td>Last name</td>
</tr>
<tr>
<td>Field 2 of phonedir</td>
<td>First initial</td>
</tr>
<tr>
<td>Field 3 of phonedir</td>
<td>Telephone number</td>
</tr>
</tbody>
</table>

If the phonedir file and names files are the same as in Example 1, the join command displays:

<table>
<thead>
<tr>
<th>389</th>
<th>Erwin G. 555-1234</th>
</tr>
</thead>
<tbody>
<tr>
<td>454</td>
<td>Norwood M. 555-5341</td>
</tr>
<tr>
<td>520</td>
<td>Wright M. 555-1234</td>
</tr>
<tr>
<td>999</td>
<td>Xandy G. 555-5015</td>
</tr>
</tbody>
</table>

4. To perform the join operation on a field other than the first, type:
sort +2 -3 phonedir | join -1 3 - numbers

This command combines the lines in the phonedir and numbers files, comparing the third field of the phonedir file to the first field of the numbers file.

First, this command sorts the phonedir file by the third field, because both files must be sorted by their join fields. The output of the sort command is then piped to the join command. The - (dash) by itself causes the join command to use this output as its first file. The -1 3 flag defines the third field of the sorted phonedir file as the join field. This is compared to the first field of numbers because its join field is not specified with a -2 flag.

If the numbers file contains:

- 555-0256
- 555-1234
- 555-5555
- 555-7358

then this command displays the names listed in the phonedir file or each telephone number:

- 555-0256 Jackson J.
- 555-1234 Erwin G.
- 555-1234 Wright M.

Note that the join command lists all the matches for a given field. In this case, the join command lists both Erwin G. and Wright M. as having the telephone number 555-1234. The number 555-5555 is not listed because it does not appear in the phonedir file.

**Files**

- `/usr/bin/join` Contains the join command.
- `/usr/lib/nls/loc/*.src` Contains collation information.

**Related Information**

The `awk` command, `comm` command, `cut` command, `paste` command, `sort` command.


**joinvg Command**

**Purpose**

Joins a snapshot volume group back into its original volume group.

**Syntax**

```
joinvg [-f] VGroupName
```

**Description**

Joins a snapshot volume group that was created with the `splitvg` command back into its original volume group. The snapshot volume group is deleted and the disks reactivated in the original volume group. Any stale partitions will be resynchronized by a background process.
Flags

-flags

Forces the join when disks in the snapshot volume group are not active. The mirror copy on the inactive disks will be removed from the original volume group.

VGname

The original volume group name with the splitvg command.

Security

Access Control: You must have root authority to run this command.

Examples

To join the original volume group, testvg, with the snapshot volume group snapvg, type:

joinvg testvg

Files

/usr/sbin

Directory where the joinvg command resides.

Related Information

The splitvg and recreatevg commands.

kdb Command

Purpose

Allows examining of a system dump or a running kernel.

Syntax

kdb [flags] [SystemImageFile [KernelFile] [KernelModule] ... ]]

Description

The kdb command is an interactive utility for examining an operating system image or the running kernel. The kdb command interprets and formats control structures in the system and provides miscellaneous functions for examining a dump.

The SystemImageFile parameter specifies the file that contains the system image. The value can indicate a system dump, the name of a dump device, or the /dev/pmem special file. The default SystemImageFile is /dev/pmem.

The KernelFile parameter specifies the AIX kernel that kdb will use to resolve kernel symbol definitions. A kernel file must be available. When examining a system dump it is imperative that the kernel file be the same as the kernel that was used to take the system dump. The default for the KernelFile is /unix.

The KernelModule parameters specify the file names of any additional kernel modules which kdb is to use for resolving symbol definitions not found in the kernel file itself.

Root permissions are required for execution of the kdb command on the active system. This is required because the special file /dev/pmem is used. To run the kdb command on the active system, type:

kdb

Note: Stack tracing of the current process on a running system does not work.
To invoke the **kdb** command on a system image file, type:

```
kdb SystemImageFile
```

When kdb starts, it looks for a `.kdbinit` file in the user’s home directory and in the current working directory. If a `.kdbinit` file exists in either of these locations, kdb will execute all the commands inside the file as if they were entered at the interactive kdb prompt. If a `.kdbinit` file exists in both of these locations, the file in the home directory will be processed first followed by the file in the current working directory (unless the current directory is the home directory, in which case the file is processed only once).

**Flags**

- **-c CommandFile** Specifies a different name for the startup script file. If this option is used, then kdb will search for the `CommandFile` parameter in the home and current directories, instead of the `.kdbinit` file.

- **-cp** Causes kdb to print out each command in the startup script files as that command is executed. This may be used to aid in the debugging of `.kdbinit` files (or any other file specified with the `-c` flag). Each command will be printed with a + (plus) sign in front of it.

- **-h** Displays a short help message in regard to command line usage and a brief listing of the available command line options.

- **-i HeaderFile** Makes all of the C structures defined in the `HeaderFile` parameter available for use with the kdb `print` subcommand. This option requires a C compiler to be installed on the system. If the `HeaderFile` variable needs additional `.h` files to compile, these may have to be specified with separate `-i` options as well.

- **-k Module** Instructs kdb to use the `Module` parameter as an additional kernel module for resolving symbol definitions not found in the kernel itself. Using this option is equivalent to specifying the kernel module with the `KernelModule` parameter.

- **-l** Disables the inline pager (that is, the `more` (`^C to quit`) ? prompt) in kdb. In this case the `set scroll` subcommand in kdb has no effect, and the inline pager is always disabled regardless of the scroll setting.

- **-m Image** Instructs kdb to use the `Image` parameter as the system image file. Using this option is equivalent to specifying the system image file with the `SystemImageFile` parameter.

- **-script** Disables the inline pager (that is, the `more` (`^C to quit`) ? prompt) and disables printing of most status information when kdb starts. This option facilitates parsing of the output from the `kdb` command by scripts and other programs that act as a front end for kdb.

- **-u Kernel** Instructs kdb to use the `Kernel` as the kernel file for resolving symbol definitions. Using this option is equivalent to specifying the kernel with the `KernelFile` parameter.

- **-v** Displays a list of all Component Dump Tables (CDTs) in the system dump file when kdb starts. CDTs list which memory regions are actually included in the system dump. If kdb is used on a live system, this option is ignored.

- **-w** Examines a kernel file directly instead of a system image. All kdb subcommands which normally display memory locations from the system image file will instead read data directly from `KernelFile`. Subcommands which write memory are not available.

**Examples**

The following examples demonstrate invocation options for the **kdb** command

1. To invoke the **kdb** command with the default system image and kernel image files, type:

   ```
kdb
   ```

   The **kdb** program returns a (0)> prompt and waits for entry of a subcommand.

2. To invoke the **kdb** command using a dump file named `/var/adm/ras/vmcore.0` and the UNIX kernel file named `/unix`, type:

   ```
kdb /var/adm/ras/vmcore.0 /unix
   ```

   The **kdb** program returns a (0)> prompt and waits for entry of a subcommand.
Files
/usr/sbin/kdb Contains the kdb command.
/dev/pmem Default system image file
/unix Default kernel file

Related Information
AIX 5L Version 5.2 KDB Kernel Debugger and kdb command

kdestroy Command

Purpose
Destroys a Kerberos credentials cache.

Syntax
kdestroy [ [-q] [-c cache_name | -e expired_time] ]

Description
The kdestroy command deletes a Kerberos credentials cache file.
If you specify the -e flag, the command checks all of the credentials cache files in the default cache directory (/var/krb5/security/creds) and deletes any file which contains only expired tickets, provided the tickets have been expired for the specified expired_time.

Flags
- -c cache_name Specifies the name of the credentials cache you want to destroy. The default credentials cache is destroyed if you do not specify a command flag.

If the KRB5CCNAME environment variable is set, its value is used to name the default credentials (ticket) cache.

This flag is mutually exclusive with the -e flag.
- -e expired_time Specifies that all credentials cache files containing expired tickets be deleted if the tickets have been expired at least as long as the expired_time value.

The expired_time is expressed as nwnndnmns, where:

n represents a number
w represents weeks
d represents days
h represents hours
m represents minutes
s represents seconds

You must specify the expired_time components in this order but you can omit any component. For example, 4h5m represents four hours and 5 minutes and 1w2h represents 1 week and 2 hours. If you only specify a number, the default is hours.

-q Suppress the beep when kdestroy fails to destroy the ticket.
Security
To delete a credentials cache, the user must be the owner of the file or must be a root (uid 0) user.

Examples
1. To delete the default credentials cache for the user, type:
   kdestroy
2. To delete all credentials cache with expired tickets older than one day, type:
   kdestroy -e 1d

Files
/usr/krb5/bin/kdestroy
/var/krb5/security/creds/krb5cc_[uid] default credentials cache ([uid] is the UID of the user.)

Related Information
The kinit command, klist command, and env command.

keyadd Command

Purpose
keyadd retrieves objects from the source keystore and adds them to the destination keystore.

Syntax
keyadd [-S servicename] -l label -s source_keystore [-d destination_keystore] [username]

Description
The keyadd command retrieves the objects named by label from the source keystore and adds them to the destination keystore. In a keystore, a user may have the private key, public key and the certificate stored using the same label. All objects matching a label are copied regardless of the object type. If an object with the same label already exists in the destination keystore, the command returns an error. This forces the user to explicitly remove an existing object instead of blindly destroying it.

Attention: Generally, there is no way to recover a destroyed object.

The -S option specifies which end-entity services and libraries to use while adding the objects from the keystore. Available services are defined in /usr/lib/security/pki/ca.cfg. When invoked without -S, keydelete will use the default service, which is local. It is an error to specify a servicename which does not have an entry in the /usr/lib/security/pki/ca.cfg file.

The -l option must be specified. This label uniquely identifies an object in the keystore to be copied. The -s option must also be specified.

If the -d option is not given, the username’s default keystore file will be used as the destination keystore. The user’s default keystore location is /var/pki/security/keys/<username>.

If no username is given, the current user’s username will be used. The user will be prompted for the password of the destination keystore and the source keystore. If the destination keystore does not exist, one will be created and the user will be asked to enter the destination keystore password again for confirmation.
Flags

- S servicename Specifies which service module to use.
- l label Specifies the label associated with the key to be added.
- s source_keystore Specifies the location of the source keystore.
- d destination_keystore Specifies the location of the destination keystore.

Exit Status

0 The command completed successfully.
>0 An error occurred.

Security

This is a setuid command. In order to list the contents of a keystore the user must know the password of the private keystore.

Root and invokers belonging to group security are allowed to list anybody’s keystore. However, they can only successfully complete this operation if they know the password to the keystore. A non-privileged user is only allowed to list the keystore that he owns.

Audit

This command records the following event information:

KEY_Add <username>

Examples

To copy a keystore object labeled as label from /var/pki/security/keys/src.keystore to /var/pki/security/keys/dst.keystore, enter:

$ keyadd -s /var/pki/security/keys/src.keystore -d /var/pki/security/keys/dst.keystore -l label pkitest

Files

/usr/lib/security/pki/policy.cfg

/usr/lib/security/pki/ca.cfg

Related Information

The certadd, certcreate, certdelete, certget, certlink, certlist, certrevoke, certverify, keydelete, keylist, keypasswd and mksecpki commands.

keycomp Command

Purpose

Compiles a keyboard mapping file into an input method keymap file.

Syntax

keycomp <Infile> Outfile
Description
The keycomp command reads a textual description of the keyboard from standard input and produces a binary file that maps the keys to standard output. The binary file is used by the Input Method to translate key strokes into character strings.

You can bind characters and strings to keys on a keyboard with specified combinations of modifier keys called keyboard states, or you can specify particular key and state combinations as unbound (return nothing). All input keys are represented by keysyms, which stand for the key symbols that are usually used in the AIXwindows environment to represent keyboard input.

Any combination of modifier keys is possible when you press a key on the keyboard, but usually the keys are mapped into a smaller set of states. This state mapping can be specified.

Keycomp Source File
The input file used by the keycomp command consists of one or more lines. The items on the line are separated by a space. Each line begins with a keysym or a hexadecimal value for a keysym. The hexadecimal value represents keyboard input in the AIXwindows environment. Items following the keysym represent the binding for a particular combination of the Ctrl, Alt, Shift, Lock, and Alt Graphic keys.

An item can be one of the following:
• Character surrounded by single quotation marks
• String surrounded by double quotation marks
• Keysym allowing mapping to other keysyms
• U indicating that the entry is unbound

Hexadecimal ( \xXX), octal ( \oOOO), and decimal ( \dDDD) notations of a byte can be contained in character and string items.

Keyboard States
Modifier keys (Shift, Lock, Ctrl, Alt, and Alt Graphics keys) change the state of the keyboard. They are used to select one item from a line corresponding to the input keysym. A value that is a combination of bits, each bit corresponding to a modifier key, indicates the state of a keyboard. The modifier keys increase in significance in the following order: Shift, Lock, Ctrl, Alt, and Alt Graphic modifier keys.

The bit combination or state value of a keyboard is mapped to one item of a line. The mapping is defined by the line beginning with the ^M control, which can contain only numbers. The first number after the ^M control is the item number. The numbers that follow the first number represent keyboard states, and they are all mapped to the item. See "Examples."

Flags
<InFile Specifies a source file to be compiled by the keycomp command.
>OutFile Specifies the name of the keymap file to be created.

Examples
1. The following is an example of a line for XK_a keysym input:

   XK_a\'a'  XK_A  XK_A  XK_a \'x01'  U "hello"
   A , (comma) can, but need not, follow each item. Regardless of whether a comma follows an item, a space or tab must separate the items.
Blank lines and lines beginning with the # character, except control statements, are ignored. All text
between the # and the following line is ignored unless the # is part of a string enclosed in single or
double quotation marks. Therefore, you can place comments at the end of a line that contains only a
single item.

2. The following line shows that the keyboard states Ctrl, Ctrl+Shift, and Ctrl+Shift+Lock are all mapped
to the third item:

```
%M 3 4 5 7
```

Files

- `/usr/include/x11/keysymdef.h`: Contains standard keysym definitions.
- `/usr/include/x11/aix_keysym.h`: Contains unique keysym definitions.
- `/usr/bin/keycomp`: Contains the `keycomp` command.
- `/usr/lib/nls/loc/*.imkeymap.src`: Contains imkeymap source information.
- `/usr/lib/nls/loc/*.imkeymap`: Maps a keysym/modifier to a string.

Related Information

The `IMInitializeKeymap` subroutine.


The `National Language Support Overview` in *AIX 5L Version 5.2 National Language Support Guide and
Reference*.

---

**keydelete Command**

**Purpose**

Deletes an object (key, certificate, etc) identified by the label from a keystore. If the label is ALL, all objects
are deleted.

**Syntax**

```
keydelete [ -S ServiceName ] -l Label [ -p PrivateKeystore ] [ UserName ]
```

**Description**

The `keydelete` command deletes an object (key, certificate, etc) identified by the `Label`. If the `Label` is ALL, all objects are deleted. The `-S` flag specifies which end-entity services and libraries to use while deleting the objects from the keystore. Available services are defined in `/usr/lib/security/pki/ca.cfg`. When invoked without `-S`, `keydelete` uses the default service, which is `local`. An error is returned if a `ServiceName` is specified which does not have an entry in the `/usr/lib/security/pki/ca.cfg` file.

The `-l` flag must be specified. The `Label` is a variable length text string that is used to map a key in the
keystore to the certificate which contains the matching public key. If the `Label` is ALL, all the objects in the
keystore are deleted.

If the `-p` flag is not given, the username’s default keystore file is used. The user’s default keystore location is `/var/pki/security/keys/<UserName>`.

If no `UserName` is given, the current user’s user name is used. The user is prompted for the password of
the keystore.
Flags

-S ServiceName  Specifies which service module to use.
-I Label        Specifies the label associated with the key to be added.
-p PrivateKeystore  Specifies the location of the source destination keystore.

Arguments

username - Specifies the user whose key is going to be deleted.

Security

This is a privileged (set-UID root) command.

In order to list the contents of a keystore, the user must know the password of the private keystore.

root and invokers belonging to group security are allowed to list anybody’s keystore. However, they can only successfully complete this operation if they know the password to the keystore. A non-privileged user is only allowed to list the keystore that he owns.

Audit

This command records the following event information:

KEY_Delete <UserName>

Examples

1. To delete a keystore object with a label signcert from the invoker’s default keystore, type:
   keydelete -l signcert

2. To delete all the objects from the invoker’s default keystore, type:
   keydelete -l ALL

3. To delete a keystore object with a label signcert from the keystore /home/bob/bob.keystore, type:
   keydelete -p /home/bob/bob.keystore -l signcert

Files

/usr/lib/security/pki/ca.cfg

Related Information

The keyadd, keylist and keypasswd commands.

keyenvoy Command

Purpose

Acts as an intermediary between user processes and the keyserv daemon.

Syntax

/usr/sbin/keyenvoy

Description

The keyenvoy command acts as an intermediary by some Remote Procedure Call (RPC) programs between their user processes and the keyserv daemon. An intermediary is necessary because the keyserv daemon talks only to root processes. This program cannot be run interactively.
Files

/usr/sbin/keyenvoy  Contains the keyenvoy command.

Related Information

The keyserv daemon.


keylist Command

Purpose

keylist lists the keystore labels in a private keystore.

Syntax

keylist [-S servicename] [-v | -c] [-p privatekeystore] [username]

Description

The keylist command lists the keystore labels in a private keystore. The -S option specifies which end-entity services and libraries to use while listing the labels in the keystore. Available services are defined in /usr/lib/security/pki/ca.cfg. When invoked without -S, keylist will use the default service, which is local. It is an error to specify a servicename which does not have an entry in the /usr/lib/security/pki/ca.cfg file. The user optionally may provide the location of the private keystore. If not given, the default location will be used. If the -c option is given, the type of the keystore object corresponding to the label will be specified by one letter symbol. The following are the symbols denoting the keystore object types:

P = Public Key
p = Private Key
T = Trusted Key
S = Secret Key
C = Certificate
t = Trusted Certificate
U = Useful Certificate

If the -v option is used, the type of the object for a label will be given in non-abbreviated version (for example, Public Key, Secret Key).

If required, the user will be prompted for the password of the underlying service keystore.
Flags

-\texttt{S servicename} Specifies which service module to use.
-\texttt{p privatekeystore} Specifies the location of the keystore.
-\texttt{v} Specifies that the output is in verbose mode.
-\texttt{c} Specifies a concise output.

Arguments

\textit{username} Specifies the AIX user whose key labels is going to be queried.

Exit Status

0 Successful completion.
>0 An error occurred.

Security

This is a privileged (set-UID root) command.

In order to list the contents of a keystore the user must know the password of the private keystore.

Root and invokers belonging to group security are allowed to list anybody’s keystore. However, they can only successfully complete this operation if they have the knowledge of the password to the keystore.

A non-privileged user is only allowed to list the keystore that he owns.

Audit

This command records the following event information:

\texttt{KEY\_List <username>}

Examples

1. To list the labels in keystore \texttt{/var/security/pki/keys/bob}, enter:
   \begin{verbatim}
   $ keylist -c -p /var/pki/security/keys/bob bob
   PpC label1
   PpC label2
   \end{verbatim}

2. To list labels/objects in verbose mode, enter:
   \begin{verbatim}
   $ keylist -v -p /var/pki/security/keys/bob bob
   \end{verbatim}

Files

\texttt{/usr/lib/security/pki/policy.cfg}

\texttt{/usr/lib/security/pki/ca.cfg}

Related Information

The \texttt{certadd} \texttt{certcreate} \texttt{certdelete} \texttt{certget} \texttt{certlink} \texttt{certlist} \texttt{certrevoke} \texttt{certverify} \texttt{keyadd} \texttt{keydelete} \texttt{keypasswd} and \texttt{mksecpki} commands.
keylogin Command

Purpose
Decrypts and stores the user’s secret key.

Syntax
/usr/bin/keylogin

Description
The keylogin command prompts users for their passwords. Then, the keylogin program decrypts the user’s secret key, which is stored in the /etc/publickey file. The decrypted key is then stored by the local keyserv daemon to be used by any secure Remote Procedure Call (RPC) service, such as the Network File System (NFS).

The decrypted key given to the local keyserv daemon may eventually reach a time out and become invalid for that particular login session. The user can use the keylogin command again to refresh the key held by the keyserv daemon.

Files
/etc/publickey Contains public or secret keys for NIS maps.

Related Information
The chkey command, newkey command.

The keyserv daemon.


List of NIS Commands


keylogout Command

Purpose
Deletes stored secret key.

Syntax
keylogout [-f]

Description
The keylogout command deletes the key stored by the key server process keyserv. Further access to the key is revoked; however, current session keys may remain valid until they expire or are refreshed.
Deleting the keys stored by `keyserv` will cause any background jobs or scheduled jobs that need secure RPC services to fail. Since only one copy of the key is kept on a machine, do not place a call to this command in your `logout` file since it will affect other sessions on the same machine.

**Flags**

- `-f` Forces `keylogout` to delete the secret key for the superuser. By default, `keylogout` by the superuser is disallowed because it would break all RPC services, such as NFS, that are started by the superuser.

**Related Information**

The `at` command, `chkey` command, `login` command, `keylogin` command, `newkey` command. The `keyserv` daemon.

**keypasswd Command**

**Purpose**

`keypasswd` manages the passwords which are used to access a user's private keystore.

**Syntax**

`keypasswd [-S servicename] [-p privatekeystore | -k username]`

**Description**

The `keypasswd` command allows a user to change the password of a private keystore. The user will be asked to enter the old and new password of the keystore. The `-S` option specifies which end-entity services and libraries to use while changing the password. Available services are defined in the `/usr/lib/security/pki/ca.cfg` file. When invoked without `-S`, `keypasswd` will use the `local` service. You will get an error if you specify a servicename which does not have an entry in the `/usr/lib/security/pki/ca.cfg` file. The `-p` option specifies the private keystore for which the password is going to be changed. The `-k` option specifies the user's default private keystore. You will get an error if you specify both the `-k` and `-p` options.

**Flags**

- `-S servicename` Specifies which service module to use.
- `-p privatekeystore` Specifies the private keystore whose password is going to be changed.
- `-k` Specifies that the keystore to be used is that of username.

**Security**

This is a privileged (set-UID root) command.

To change the privileged password of a keystore one must know the password of the keystore.

Root and invokers belonging to group security are allowed to change the password of any keystore as long as they know the password of the keystore. A non-privileged user is allowed to change only the keystore file that they own.
Audit
This command records the following event information:

KEY_Password <username>

Examples
1. To change the password of the default private keystore that is owned by Bob, enter:
   
   $ keypasswd
   
   where the invoker is Bob.
2. To change the password of any other private keystore, enter:
   
   $ keypasswd -p bob.keystore

Files
/usr/lib/security/ca.cfg
/usr/lib/security/policy.cfg

Related Information
The certadd, certcreate, certdelete, certget, certlink, certlist, certrevoke, certverify, keyadd, keydelete, keylist, and mksecpki commands.

ke yserv Daemon

Purpose
Stores public and private keys.

Syntax
/usr/sbin/keyserv [ -n ]

Description
The keyserv daemon stores the private encryption keys of each user logged into the system. When a user types in a password during a keylogin, the secret key is decrypted. The decrypted key is then stored by the keyserv daemon. These decrypted keys enable the user to access secure network services such as secure Network File System (NFS).

When the keyserv daemon starts, it reads the key for the root directory from the /etc/rootkey file. This daemon keeps the secure network services operating normally. For instance, after a power failure, when the system restarts itself, it gets the key for the root directory from the /etc/rootkey file.

Flags

- n Prevents the keyserv daemon from reading the key for the root directory from the /etc/rootkey file. Instead, the keyserv daemon prompts the user for the password to decrypt the root directory's key stored in the network information service map and then stores the decrypted key in the /etc/rootkey file for future use. This option is useful if the /etc/rootkey file ever goes out of date or is corrupted.

Examples
1. To start the keyserv daemon enabling the system to get the key for the root directory from the /etc/rootkey file, enter:
2. A System Resource Controller (SRC) command can also enable the system to get the key for the root directory from the /etc/.rootkey file as follows:

   startsrc -s keyserv

   This command sequence starts a script that contains the keyserv daemon.

3. To prevent the keyserv daemon from reading the key for the root directory from the /etc/.rootkey file, enter:

   chssys -s keyserv -a '-n'

   This command passes the -n argument to the keyserv daemon if SRC is used to start the daemon.

Files

/etc/.rootkey Stores the encrypted key for the root directory.

Related Information

The chssys command, keyenvoy command, startsrc command.


NIS Reference


kill Command

Purpose

Sends a signal to running processes.

Syntax

To Send Signal to Processes

   kill [ -s { SignalName | SignalNumber } ] ProcessID ...

   kill [ - SignalName | - SignalNumber ] ProcessID ...

To List Signal Names

   kill [ - [ ExitStatus ] ]

Description

The kill command sends a signal (by default, the SIGTERM signal) to a running process. This default action normally stops processes. If you want to stop a process, specify the process ID (PID) in the
ProcessID variable. The shell reports the PID of each process that is running in the background (unless you start more than one process in a pipeline, in which case the shell reports the number of the last process). You can also use the `ps` command to find the process ID number of commands.

A root user can stop any process with the `kill` command. If you are not a root user, you must have initiated the process you want to stop.

SignalName is recognized in a case-independent fashion, without the SIG prefix.

If the specified SignalNumber is 0, the `kill` command checks the validity of the specified PID.

Flags

-`s{SignalName | SignalNumber}` Specifies the signal as a signal number or a signal name, such as `SIGKILL` -9 or `SIGTERM` -15.
-`-SignalName` Specifies a signal name, such as `SIGHUP`. This is obsolete syntax.
-`-SignalNumber` Specifies a signal number. This is obsolete syntax.

Note: To specify the negative PID with the default signal in this syntax, you must specify `-` as a signal. Otherwise the first operand is interpreted as a `SignalNumber`.

ProcessID Specifies a decimal integer representing a process or process group to be signaled. If PID is a positive value, the `kill` command sends the process whose process ID is equal to the PID. If the PID value is 0, the `kill` command sends the signal to all processes having a process group ID equal to the process group ID of the sender. The signal is not sent to processes with a PID of 0 or 1. If the PID is -1, the `kill` command sends the signal to all processes owned by the effective user of the sender. The signal is not sent to processes with a PID of 0 or 1. If it is a negative number but not -1, the `kill` command sends the signal to all processes that have a process group ID equal to the absolute value of the PID.

-`-l` Lists all signal names supported by the implementation
-`-lExitStatus` Lists signal names stripped of the common SIG prefix. If `ExitStatus` is an decimal integer value, the signal name corresponding to that signal is displayed. If `ExitStatus` is a value of the exit status corresponding to a process that was terminated by a signal, the signal name corresponding to the signal that terminated the process is displayed.

Exit Status

This command returns the following exit values:

0 At least one matching process was found for each `ProcessID` operand, and the specified signal was successfully processed for at least one matching process.

>0 An error occurred.

Examples

1. To stop a given process, enter:
   ```
   kill 1095
   ```
   This stops process 1095 by sending it the default `SIGTERM` signal. Note that process 1095 might not actually stop if it has made special arrangements to ignore or override the `SIGTERM` signal.

2. To stop several processes that ignore the default signal, enter:
   ```
   kill -kill 2098 1569
   ```
This sends signal 9, the **SIGKILL** signal, to processes 2098 and 1569. The **SIGKILL** signal is a special signal that normally cannot be ignored or overridden.

3. To stop all of your processes and log yourself off, enter:

    ```bash
    kill -kill 0
    ```

   This sends signal 9, the **SIGKILL** signal, to all processes having a process group ID equal to the senders process group ID. Because the shell cannot ignore the **SIGKILL** signal, this also stops the login shell and logs you off.

4. To stop all processes that you own, enter:

    ```bash
    kill -9 -1
    ```

   This sends signal 9, the **SIGKILL** signal, to all processes owned by the effective user, even those started at other work stations and that belong to other process groups. If a listing that you requested is being printed, it is also stopped.

5. To send a different signal code to a process, enter:

    ```bash
    kill -USR1 1103
    ```

   The name of the **kill** command is misleading because many signals, including **SIGUSR1**, do not stop processes. The action taken on **SIGUSR1** is defined by the particular application you are running.

   **Note:** To send signal 15, the **SIGTERM** signal with this form of the **kill** command, you must explicitly specify -15 or **SIGTERM**.

**Files**

```
/usr/include/sys/signal.h
```

  Specifies signal names.

**Related Information**

The **csh** command, **ksh** command, **ps** command, **sh** command.

The **kill** subroutine, **sigaction** subroutine.

---

**killall Command**

**Purpose**

Cancels all processes except the calling process.

**Syntax**

```
killall [ ] [ Signal]
```

**Description**

The **killall** command cancels all processes that you started, except those producing the **killall** process. This command provides a convenient means of canceling all processes created by the shell that you control. When started by a root user, the **killall** command cancels all cancellable processes except those processes that started it. If several Signals are specified, only the last one is effective.

If no signal is specified, the **killall** command sends a **SIGKILL** signal.
Flags
- Sends a **SIGTERM** signal initially and then sends a **SIGKILL** signal to all processes that survive for 30 seconds after receipt of the signal first sent. This gives processes that catch the **SIGTERM** signal an opportunity to clean up. If both - and -**Signal** are set, the **killall** command sends the specified signal initially and then sends a **SIGKILL** signal to all processes that survive for 30 seconds after receipt of the signal first sent.
-**Signal** Sends the specified **Signal** number or **SignalName**.

Examples
1. To stop all background processes that have started, enter:
   ```
   killall
   ```
   This sends all background processes the **kill** signal 9 (also called the **SIGKILL** signal).
2. To stop all background processes, giving them a chance to clean up, enter:
   ```
   killall -
   ```
   This sends signal 15, the **SIGTERM** signal; waits 30 seconds, and then sends signal 9, the **SIGKILL** signal.
3. To send a specific signal to the background processes, enter:
   ```
   killall -2
   ```
   This sends signal 2, the **SIGINT** signal, to the background processes.

Related Information
The **kill** command.
The **signal** subroutine.

**kinit** Command

Purpose
Obtains or renews the Kerberos ticket-granting ticket.

Syntax
```bash
kinit [\[-l lifetime\]] [\[-r renewable_life\]] [\[rf\]] [\[-p\]] [\[-A\]] [\[-S start_time\]] [\[-S target_service\]] [\[-k\]] [\[-t keytab_file\]] [\[-R\]] [\[-V\]] [\[-c cachename\]] [\[-l principal\]]
```

Description
The **kinit** command obtains or renews a Kerberos ticket-granting ticket. The Key Distribution Center (KDC) options specified by the [kdcdefault] and [realms] in the Kerberos configuration file (**kdc.conf**) are used if you do not specify a ticket flag on the command line.

If you are not renewing an existing ticket, the command reinitializes the credentials cache and will contain the new ticket-granting ticket received from the KDC. If you do not specify the **Principal** name on the command line and you do specify the -**s** flag, the **Principal** name is obtained from the credentials cache.
The new credentials cache becomes the default cache unless you specify the cache name using the -**c** flag.

The ticket **Time** value for the -**l**, -**r** and -**s** flags is expressed as **ndhnmns** where:

- **n** represents a number
files

\texttt{kinit} -v
\texttt{kinit} -t
\texttt{kinit} -s
\texttt{kinit} -R
\texttt{kinit} -r
\texttt{kinit} -k
\texttt{kinit} -l
\texttt{kinit} -k
\texttt{kinit} -c
\texttt{kinit} -f
\texttt{kinit} -A
\texttt{kinit} -p
\texttt{kinit} -l
\texttt{kinit} -k
\texttt{kinit} -f
\texttt{kinit} -c
\texttt{kinit} -A

You must specify the components in this order but you can omit any component, for example \texttt{4h5m} represents four hours and 5 minutes and \texttt{1d2s} represents 1 day and 2 seconds.

**Flags**

- \texttt{-A} Specifies that the ticket contain a list of client addresses. The ticket will contain the local host address list if this option is not specified. When an initial ticket contains an address list, it can be used only from one of the addresses in the the address list.

- \texttt{-c cachename} Specifies the name of the credentials cache to use. The default credentials cache is used if this flag is not specified. If the KRB5CCNAME environment variable is set, its value is used to name the default ticket cache. Any existing contents of the cache i are destroyed by \texttt{kinit}.

- \texttt{-f} Specifies that the ticket is to be forwardable. To forward the ticket, this flag must be specified.

- \texttt{-k} Specifies to obtain the key for the ticket principal from a key table. If you do not specify this flag, you are prompted to enter the password for the ticket principal.

- \texttt{-l lifetime} Specifies the ticket end time interval. The ticket cannot be used after the interval expires unless the ticket is renewed. The interval default time is 10 hours.

- \texttt{-p} Specifies that the ticket is to be proxiable. To make the ticket proxiable, this flag must be specified.

- \texttt{principal} Specifies the ticket principal. The principal is obtained from the credentials cache if the principal is not specified on the command line.

- \texttt{-r renewable_life} Specifies the renew time interval for a renewable ticket. The ticket cannot be renewed after the interval expires. The renew time must be greater than the end time. If this flag is not specified, the ticket is not renewable, although you can still generate a renewable ticket if the requested ticket lifetime exceeds the maximum ticket lifetime.

- \texttt{-R} Specifies to renew an existing ticket. No other flags may be specified when renewing an existing ticket.

- \texttt{-s start_time} Specifies a request for a postdated ticket, valid starting at \texttt{start_time}.

- \texttt{-S target_service} Specifies an alternate service name to use when getting initial tickets.

- \texttt{-t keytab_file} Specifies the key table name. The default key table is used if this flag is not specified and the \texttt{-k} flag is specified. The \texttt{-t} flag implies the \texttt{-k} flag.

- \texttt{-v} Specifies that the ticket granting ticket in the cache be passed to the kdc for validation. If the ticket is within its requested time range, the cache is replaced with the validated ticket.

**Examples**

1. To obtain a ticket-granting ticket with a lifetime of 10 hours, which is renewable for five days, type:
   \texttt{kinit -l 10h -r 5d my_principal}

2. To renew an existing ticket, type:
   \texttt{kinit -R}

**Files**

- \texttt{/usr/krb5/bin/kinit}
- \texttt{/var/krb5/security/creds/krb5cc_[uid]}
- \texttt{/etc/krb5/krb5.keytab}
- \texttt{/var/krb5/krb5kdc/kdc.conf}
  
  default credentials cache ([uid] is the UID of the user.)
  default location for the local host’s keytab file.
  Kerberos KDC configuration file.
Related Information
The `klist` command, `kdestroy` command, and `env` command.

klist Command

Purpose
Displays the contents of a Kerberos credentials cache or key table.

Syntax
```
```

Description
The `klist` command displays the contents of a Kerberos credentials cache or key table.

Flags
- `-a` Displays all tickets in the credentials cache, including expired tickets. Expired tickets are not listed if this flag is not specified. This flag is valid only when listing a credentials cache.
- `-c` Lists the tickets in a credentials cache. This is the default if neither the `-c` nor the `-k` flag is specified. This flag is mutually exclusive with the `-k` flag.
- `-e` Displays the encryption type for the session key and the ticket.
- `-f` Displays the ticket flags using the following abbreviations:
  - F Forwardable ticket
  - f Forwarded ticket
  - P Proxiable ticket
  - p Proxy ticket
  - D Postdateable ticket
  - d Postdated ticket
  - R Renewable ticket
  - I Initial ticket
  - i Invalid ticket
  - H Hardware preauthentication used
  - A Preauthentication used
  - O Server can be a delegate
- `name` Specifies the name of the credentials cache or key table. The default credentials cache or key table is used if you do not specify a filename.

  If you do not specify a name indicating a cache name or keytab name, `klist` displays the credentials in the default credentials cache or keytab file as appropriate. If the KRB5CCNAME environment variable is set, its value is used to name the default credentials (ticket) cache.

- `-k` Lists the entries in a key table. This flag is mutually exclusive with the `-c` flag.
- `-K` Displays the encryption key value for each key table entry. This flag is valid only when listing a key table.
- `-n` Displays the numerical internet address instead of the host name. The default without the `-n` is host name. This command is used in conjunction with the `-a` flag.
- `-s` Suppresses command output but sets the exit status to 0 if a valid ticket-granting ticket is found in the credentials cache. This flag is valid only when listing a credentials cache.
- `-t` Displays timestamps for key table entries. This flag is valid only when listing a key table.
Examples
1. To list all of the entries in the default credentials cache, type:
   klist
2. To list all of the entries in the `/etc/krb5/my_keytab` key table with timestamps, type:
   klist -t -k etc/krb5/my_keytab

Files
/usr/krb5/bin/klist
/var/krb5/security/creds/krb5cc_[uid]
/etc/krb5/krb5.keytab
default credentials cache ([uid] is the UID of the user.)
default location for the local host's keytab file.

Related Information
The `kinit` command, `kdestroy` command, and `env` command.

**kmodctrl Command**

**Purpose**
Loads or unloads the kernel extension `/usr/lib/drivers/kmobip6`.

**Syntax**
```
 kmodctrl [-k kextname] [-lq] [-u]
```

**Description**
The kernel extension `/usr/lib/drivers/kmobip6` contains support for the Mobile IPv6 functionality. This kernel extension must be loaded in order to configure the system as a mobile IPv6 home agent or correspondent node. Normally this command will be run automatically by the `/etc/rc.mobip6` script if mobile IPv6 has been enabled using system management.

**Flags**
- `-k` Specifies an alternate path for the mobility kernel extension.
- `-l` Loads the mobility kernel extension.
- `-q` Checks whether the kernel extension is loaded.
- `-u` Unloads the mobility kernel extension.

**Exit Status**
- `0` The command completed successfully.
- `>0` An error occurred.

**Security**
You must be the root user or a member of the system group to execute this command.

**Examples**
1. The following example loads the kmobip6 kernel extension:
   ```
   kmodctrl -l
   ```
2. The following example unloads the kmobip6 kernel extension. This will disable all mobile IPv6 functionality on the system:
   kmodctrl -u
3. The following example queries whether the kmobip6 kernel extension is loaded:
   kmodctrl -q

Related Information
The mobip6ctrl command, mobip6reqd daemon, ndpd-router command, rc.mobip6 command.
The Mobile IPv6 section in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

kpasswd Command

Purpose
Changes the password for a Kerberos principal.

Syntax
kpasswd [Principal]

Description
The kpasswd command changes the password for a specified Kerberos principal. It prompts for the current principals password, which is used to obtain a changepw ticket from the KDC for the user's Kerberos realm. If kpasswd successfully obtains the changepw ticket, the user is prompted twice for the new password and the password is changed.

If the principal is governed by a policy that specifies for example length and/or number of character classes required in the new password, the new password must conform to the policy.

You may not change the password for a ticket-granting service principal (krbtgt/domain) using the kpasswd command.

Parameters
Principal Specifies the principal for which password you want to change. If you do not specify the principal on the command line, the principal is obtained from the default credentials cache.

Security
When requesting a password change, you must supply both the current password and the new password.

Files
/usr/krb5/bin/kpasswd
/var/krb5/security/creds/krb5cc_[uid] default credentials cache ([uid] is the UID of the user.)

krlogind Daemon

Purpose
Provides the server function for the rlogin command.
Syntax
/usr/sbin/krlogind [-n] [-s]

Note: The krlogind daemon is normally started by the inetd daemon. It can also be controlled from the command line, using SRC commands.

Description
The /usr/sbin/krlogind daemon is the server for the rlogin remote login command. The server provides a remote login facility.

Changes to the krlogind daemon can be made by using Web-based System Manager, the System Management Interface Tool (SMIT) or System Resource Controller (SRC), by editing the /etc/inetd.conf or /etc/services file. Entering krlogind at the command line is not recommended. The krlogind daemon is started by default when it is uncommented in the /etc/inetd.conf file.

The inetd daemon get its information from the /etc/inetd.conf file and the /etc/services file.

After changing the /etc/inetd.conf or /etc/services file, run the refresh -s inetd or kill -1 InetdPID command to inform the inetd daemon of the changes to its configuration file.

Service Request Protocol
When the krlogind daemon receives a service request, the daemon initiates the following protocol:

1. The krlogind daemon checks the source port number for the request. If the port number is not in the range 512 through 1023, the krlogind daemon terminates the connection.

2. The krlogind daemon uses the source address of the initial connection request to determine the name of the client host. If the name cannot be determined, the krlogind daemon uses the dotted-decimal representation of the client host address.

3. The krshd daemon attempts to validate the user using the following steps:
   • makes sure that Kerberos 5 is a valid authentication method if the incoming ticket is a Kerberos 5 ticket. If the incoming ticket is a Kerberos 4 ticket, the connection fails. Kerberos 4 is not supported for rlogin.
   • calls vvalid_user with the local account name as well as the DCE principal.

Error Messages
The following error messages are associated with the krlogind daemon:

Try again: A fork command made by the server has failed.
/usr/bin/shell: No shell. The shell specified for the shell variable cannot be started. The shell variable may also be a program.

Flags
-n Disables transport-level keep-alive messages. The messages are enabled by default.
-s Turns on socket level debugging.

Manipulating the krshd Daemon
The krshd daemon is a subserver of the inetd daemon, which is a subsystem of the System Resource Controller (SRC). The krshd daemon is a member of the tcpip SRC subsystem group. Using the chaunthent command will comment/uncomment the kshell line in the /etc/inetd.conf file and restart the
inetd daemon depending on whether Kerberos 5 or Kerberos 4 is configured/unconfigured. This daemon should be manipulated using the chauthent/lsauthent commands. Direct modification of the inetd.conf file's kshell entry is not recommended.

Related Information
The rlogin command.

The inetd daemon, rshd daemon, syslogd daemon.

The pty special file.

The kvalid_user subroutine.

The /etc/inetd.conf file format.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

Network Overview in AIX 5L Version 5.2 System Management Guide: Communications and Networks.


Krshd Daemon

Purpose
Provides the server function for remote command execution.

Syntax
/usr/sbin/krshd

Note: The rshd daemon is normally started by the inetd daemon. It can also be controlled from the command line, using SRC commands.

Description
The /usr/sbin/krshd daemon is the server for the rcp and rsh commands using Kerberos authentication. The krshd daemon provides remote execution of shell commands. These commands are based on requests from privileged sockets on trusted hosts. The shell commands must have user authentication. The krshd daemon listens at the kshell socket defined in the /etc/services file.

Changes to the krshd daemon can be made using the System Management Interface Tool (SMIT) or System Resource Controller (SRC), by editing the /etc/inetd.conf or /etc/services file. Entering krshd at the command line is not recommended. The krshd daemon is started by default when it is uncommented in the /etc/inetd.conf file.

The inetd daemon gets its information from the /etc/inetd.conf file and the /etc/services file.

After changing the /etc/inetd.conf or /etc/services file, run the refresh -s inetd or kill 1 InetdPID command to inform the inetd daemon of the changes to its configuration file.

Service Request Protocol
When the krshd daemon receives a service request, it initiates the following protocol:

1. The krshd daemon checks the source port number for the request. If the port number is not in the range 0 through 1023, the krshd daemon terminates the connection.
2. The **krshd** daemon reads characters from the socket up to a null byte. The string read is interpreted as an ASCII number (base 10). If this number is nonzero, the **krshd** daemon interprets it as the port number of a secondary stream to be used as standard error. A second connection is created to the specified port on the client host. The source port on the local host is also in the range 0 through 1023.

3. The **krshd** daemon uses the source address of the initial connection request to determine the name of the client host. If the name cannot be determined, the **krshd** daemon uses the dotted decimal representation of the client host's address.

4. The **krshd** daemon retrieves the following information from the initial socket:
   - A Kerberos service ticket.
   - A null-terminated string of at most 16 bytes interpreted as the user name of the user on the client host.
   - Another null-terminated string interpreted as a command line to be passed to a shell on the local server host.
   - A null-terminated string of at most 16 bytes interpreted as the user name to be used on the local server host.
   - If the service ticket was a Kerberos 5 ticket, the daemon will expect either a Kerberos 5 TGT or a null string.

5. The **krshd** daemon attempts to validate the user using the following steps:
   - makes sure that Kerberos 5 is a valid authentication method if the incoming ticket is a Kerberos 5 ticket. Likewise, if the incoming ticket is a Kerberos 4 ticket, the Kerberos 4 authentication method must be configured.
   - calls **kvalid_user** with the local account name as well as the DCE Principal.

6. Once **krshd** validates the user, the **krshd** daemon returns a null byte on the initial connection. If the connection is a Kerberos 5 ticket and the TGT is sent, the command line passes to the **k5dcelogin** command, (which upgrades it to full DCE credentials). If the TGT is not sent or if the connection is a Kerberos 4 ticket, the command line passes to the user's local login shell. The shell then inherits the network connections established by the **krshd** daemon.

   The **krshd** daemon is controlled by using the System Management Interface Tool (SMIT) or by changing the **/etc/inetd.conf** file. Entering **krshd** at the command line is not recommended.

**Manipulating the krshd Daemon**

The **krshd** daemon is a subserver of the **inetd** daemon, which is a subsystem of the System Resource Controller (SRC). The **krshd** daemon is a member of the tcip SRC subsystem group. Using the **chauthent** command will comment/uncomment the kshell line in the **/etc/inetd.conf** file and restart the **inetd** daemon depending on whether Kerberos 5 or Kerberos 4 is configured/unconfigured. This daemon should be manipulated using the **chauthent/isauthent** commands. Direct modification of the **inetd.conf** file's kshell entry is not recommended.

**Related Information**

The **rsh** command.

The **inetd** daemon.

The **kvalid_user** function.

The **/etc/hosts.equiv** file format, **/etc/inetd.conf** file format, and **/etc/services** file format.

Network Overview in **AIX 5L Version 5.2 System Management Guide: Communications and Networks**.

Secure Rcmds in **AIX 5L Version 5.2 System User's Guide: Communications and Networks**.
ksh Command

Purpose
Invokes the Korn shell.

Syntax
ksh [ + | - ] ( e h k m n t u v x ) [ -c Option ... ] [ -s | -t ] File [ Parameter ]

Note: Preceding a flag with + (plus) rather than - (minus) turns off the flag.

Description
The ksh command invokes the Korn shell, which is an interactive command interpreter and a command programming language. The shell carries out commands either interactively from a terminal keyboard or from a file.

The Korn shell is backwardly compatible with the Bourne shell (invoked with the bash command) and contains most of the Bourne shell features as well as several of the best features of the C shell.

For more information about the Korn shell, refer to Enhanced Korn Shell in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices.

Note: The ksh wait built in behaves in a manner similar to the parent wait() API.

An enhanced version of the korn shell, called ksh93, is also available. The enhanced Korn shell has additional features that are not available in the default Korn shell. For information regarding these additional features, refer to Enhanced Korn Shell in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices.

Flags
-a
Exports automatically all subsequent parameters that are defined.
-c String
Causes the Korn shell to read commands from the String variable. This flag cannot be used with the -s flag or with the File [ Parameter ] parameter.
-e
Executes the ERR trap, if set, and exits if a command has a nonzero exit status. This mode is disabled while reading profiles.
-f
Disables file name substitution.
-h
Designates each command as a tracked alias when first encountered.
-i
Indicates that the shell is interactive. An interactive shell is also indicated if shell input and output are attached to a terminal (as determined by the ioctl subroutine). In this case, the TERM environment variable is ignored (so that the kill 0 command does not kill an interactive shell) and the INTR signal is caught and ignored (so that a wait state can be interrupted). In all cases, the QUIT signal is ignored by the shell.
-k
Places all parameter assignment arguments in the environment for a command, not just those arguments that precede the command name.
-m
Runs background jobs in a separate process and prints a line upon completion. The exit status of background jobs is reported in a completion message. On systems with job control, this flag is turned on automatically for interactive shells.
-n
Reads commands and checks them for syntax errors, but does not execute them. This flag is ignored for interactive shells.
-o Option

Prints the current option settings and an error message if you do not specify an argument. You can use this flag to enable any of the following options:

**allexport**
Same as the -a flag.

**errexit**
Same as the -e flag.

**bgnice**
Runs all background jobs at a lower priority. This is the default mode.

**emacs**
Enter an emacs-style inline editor for command entry.

**gmacs**
Enter a gmacs-style inline editor for command entry.

**ignoreeof**
Does not exit the shell when it encounters an end-of-file character. You must use the **exit** command, or override the flag and exit the shell by pressing the Ctrl-D key sequence more than 11 times.

**keyword**
Same as the -k flag.

**markdirs**
Appends a / (slash) to all directory names that are a result of filename substitution.

**monitor**
Same as the -m flag.

**noclobber**
Prevents redirection from truncating existing files. When you specify this option, use the redirection symbol > (right caret, pipe symbol) to truncate a file.

**noexec**
Same as the -n flag.

**noglob**
Same as the -f flag.

**nolog**
Prevents function definitions from being saved in the history file.

**nounset**
Same as the -u flag.

**privileged**
Same as the -p flag.

**verbose**
Same as the -v flag.

**trackall**
Same as the -h flag.

**vi**
Enter the insert mode of a vi-style inline editor for command entry. Entering escape character 033 puts the editor into the move mode. A return sends the line.

**viraw**
Processes each character as it is typed in vi mode.

**xtrace**
Same as the -x flag.

You can set more than one option on a single **ksh** command line.

- **r**
Runs a restricted shell. With a restricted shell you cannot:
  - Change the current working directory.
  - Set the value of the **SHELL**, **ENV**, or **PATH** variable.
  - Specify the pathname of a command that contains a / (slash).
  - Redirect output of a command with > (right caret), > (right caret, pipe symbol), < (left caret, right caret), or >> (two right carets).

- **s**
Causes the **ksh** command to read commands from the standard input. Shell output, except for the output of the special commands, is written to file descriptor 2. This parameter cannot be used with the -c flag or with the **File[Parameter]** parameter.
Exits after reading and executing one command.
-t

Treats unset parameters as errors when substituting.
-u

Prints shell input lines as they are read.
-v

Prints executed commands and their arguments.
-x

Files

/usr/bin/ksh Contains the path name to the Korn shell.
/tmp/sh* Contains temporary files that are created when a shell is opened.

Related Information

The env command.
The profile file format.


kvno Command

Purpose
Displays the current key version number for a principal.

Syntax

kvno [-e etype] service 1 service2...

Description
The kvno command displays the current key version number for a principal (service 1 service2...). The security policy must allow a service ticket to be obtained for the principal. The current network identity is used when requesting the service ticket.

Flags

-e etype Specifies which encryption type to get the current key version.

service 1 service2... Specifies the principal for which you want to display the current key version number.

Security
The security policy must allow a service ticket to be obtained for the principal.

Files

/usr/krb5/bin/kvno

Related Information

The klist command.
last Command

Purpose
Displays information about previous logins.

Syntax
last [-f FileName] [-T Time] [-n Number | -Number] [ Name ... ] [ Terminal ... ]

Description
The last command displays, in reverse chronological order, all previous logins and logoffs still recorded in the /var/adm/wtmp file. The /var/adm/wtmp file collects login and logout records as these events occur and holds them until the records are processed by the acctcon1 and acctcon2 commands as part of the daily reporting procedures. When the time daemon, timed, changes the system time, it logs entries in wtmp under the pseudo-user "date". An entry starting with "date |" is logged before the change, and one starting with "date {" is logged after the change. This allows for accurate accounting of logins that span a time change.

The list can be restricted to:
- The number of lines specified either with the -Number parameter or with the -n flag.
- Logins or logoffs by the users specified by the Name parameter.
- Logins or logoffs from the terminals specified by the Terminal parameter.
- A terminal can be named fully or abbreviated as a tty. For example, you can specify either the tty0 terminal or the 0 terminal.

Note: If you specify both a Name and Terminal parameter, the last command displays all logins and logoffs meeting either criterion.

For each process, the last command displays the:
- Time the session began
- Duration
- Terminal (tty) used

If applicable, the following information is included:
- Terminations due to rebooting
- Sessions that are still continuing

If the last command is interrupted, it indicates how far the search has progressed in the /var/adm/wtmp file. If interrupted with a quit signal, the command indicates how far the search has progressed and then continues the search. The quit signal can be any one of the following:

#define SIGQUIT 3 /**< (*) quit, generated from terminal special char */
#define SIGKILL 9 /**< kill (cannot be caught or ignored) */
#define SIGTERM 15 /**< software termination signal */

The kill command sends the default SIGTERM signal when it is invoked without any option. If you want to send the SIGQUIT signal, enter the following:

kill -3 (Process ID)

See the kill command for more information.
Flags

- f FileName  Specifies an alternate file from which to read logins and logoffs.
- n            Specifies the number of lines to be displayed on the list.
- T Time       Displays users logged in at the given Time value. The Time variable is specified in the decimal form [[CC][YY]MMDDhhmm][.SS] where:

  CC  Specifies the first two digits of the year.
  YY  Specifies the last two digits of the year.
  MM  Specifies the month of the year (01 to 12).
  DD  Specifies the day of the month (01 to 31).
  hh  Specifies the hour of the day (00 to 23).
  mm  Specifies the minute of the hour (00 to 59).
  SS  Specifies the second of the minute (00 to 59).

Examples

1. To display all the recorded logins and logoffs by user root or from the console terminal, type:
   last root console
2. To display the time between reboots of the system, type:
   last reboot
   The reboot pseudo-user logs in when the system starts again.
3. To display all the users still logged in at 10.30 am on April 15th, enter:
   last -t 04151030
4. To display 10 lines in the list, type:
   last -n 10

Files

/usr/bin/last           Contains the last command.
/var/adm/wtmp          Contains connect-time accounting data, including login, logoff, and shutdown records.

Related Information

The acctcon1, accton2 command, lastlogin command.

For more information about the Accounting System, the preparation of daily and monthly reports, and the accounting files, see the Accounting Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.

Setting Up an Accounting System in AIX 5L Version 5.2 System Management Guide: Operating System and Devices describes the steps you must take to establish an accounting system.

lastcomm Command

Purpose

Displays information about the last commands executed.
Syntax

`lastcomm [ Command ] [ Name ] [ Terminal ]`

Description

The `lastcomm` command displays information, in reverse chronological order, about all previously executed commands that are still recorded in the summary files in the `/var/adm/pacct` directory. You need to run the `/usr/sbin/acct/startup` command before you can execute the `lastcomm` command.

The list the `lastcomm` command displays can be restricted to:

- Commands specified by the `Command` parameter.
- Commands executed by the user specified by the `Name` parameter.
- Commands from the terminal specified by the `Terminal` parameter.

A terminal can be named fully or abbreviated as a tty. For example, you can specify either the tty0 terminal or the 0 terminal.

For each process, the following information is displayed:

- The name of the user who ran the process.
- Any flags the accounting facilities collected when the command executed. The following are valid flags:
  - `S`: The root user executed the command.
  - `F`: The command ran after a fork, but without a following subroutine.
  - `C`: The command ran in PDP-11 compatibility mode.
  - `D`: The command terminated with the generation of a core file.
  - `X`: The command was terminated with a signal.
- The name of the command under which the process was called.
- The seconds of CPU time used by the process.
- The time the process was started.

Examples

1. To display information about all previously executed commands recorded in the `/var/adm/pacct` file, enter:
   ```
   lastcomm
   ```

2. To display information about commands named `a.out` executed by the root user on the ttyd0 terminal, enter:
   ```
   lastcomm a.out root ttyd0
   ```

Files

- `/usr/bin/lastcomm`: Contains the `lastcomm` command.
- `/var/adm/pacct`: The directory that contains the current accounting summary files.

Related Information

The `acctcms` command.

For more information about the Accounting System, the preparation of daily and monthly reports, and the accounting files, see the Accounting Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.

Setting Up an Accounting System in AIX 5L Version 5.2 System Management Guide: Operating System and Devices describes the steps you must take to establish an accounting system.
lastlogin Command

Purpose
Reports the last login date for each user on the system.

Syntax
/usr/sbin/acct/lastlogin

Description
The lastlogin command updates the /var/adm/acct/sum/loginlog file to show the last date each user logged in. Normally, the runacct command, running under the cron daemon, calls this command and adds the information to the daily report. However, the lastlogin command can also be entered by a user who is a member of the ADM group.

Note: You should not share accounting files among nodes in a distributed environment. Each node should have its own copy of the various accounting files.

Security
Access Control: This command should grant execute (x) access only to members of the ADM group.

Files
/usr/sbin/acct The path to the accounting commands.
/var/adm/wtmp The login and logout history file.
/var/adm/acct/sum Cumulative directory for daily accounting records.

Related Information
The runacct command.
The cron daemon.

For more information about the Accounting System, the preparation of daily and monthly reports, and the accounting files, see the Accounting Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.

Setting Up an Accounting System in AIX 5L Version 5.2 System Management Guide: Operating System and Devices explains the steps you must take to establish an accounting system.

lb_admin Command

Purpose
Administers the registration of NCS-based servers in location broker databases.

Syntax
lb_admin [ -nq ] [ -version ]

Description
The lb_admin tool administers the registrations of NCS-based servers in global location broker (GLB) or local location broker (LLB) databases. A server registers universal unique identifiers (UUIDs) specifying an
object, a type, and an interface, along with a socket address specifying its location. A client can locate servers by issuing lookup requests to GLBs and LLBs. The **lb_admin** tool can be used to look up information, add new entries, and delete existing entries in a specified database.

The **lb_admin** tool is useful for inspecting the contents of location broker databases and for correcting database errors. For example, if a server terminates abnormally without unregistering itself, use **lb_admin** to manually remove its entry from the GLB database.

When accepting input or displaying output, **lb_admin** uses either character strings or descriptive textual names to identify objects, types, and interfaces. A character string directly represents the data in a UUID in the format

```
xxxxxxxxxxxx.xx.xx.xx.xx.xx.xx.xx
```

where each x is a hexadecimal digit. Descriptive textual names are associated with UUIDs in the **uuidname.txt** file.

The **lb_admin** command examines or modifies only one database at a time. This is referred to as the **current database**. The **use_broker** command selects the type of location broker database, GLB or LLB. The **set_broker** command selects the host whose GLB or LLB database is to be accessed. If one replica of a replicated GLB database is modified, the modifications are propagated to the other replicas of that database.

**Flags**

- `-nq`  
  Do not query for verification of wildcard expansions in unregister operations.

- `-version`  
  Display the version of NCS that this **lb_admin** belongs to, but do not start the tool.

**Subcommands**

In the **lookup**, **register**, and **unregister** commands, the **object**, **type**, and **interface** arguments can be either character strings representing UUIDs or textual names corresponding to UUIDs, as described earlier.

- `a[dd]`  
  Synonym for **register**.
c[lean] Finds and deletes obsolete entries in the current database. When issuing this command, lb_admin attempts to contact each server registered in the database. If the server responds, the entry for its registration is left intact in the database. If the server does not respond, lb_admin tries to look up its registration in the LLB database at the host where the server is located, tells the result of this lookup, and asks if the entry is to be deleted. If a server responds, but its UUIDs do not match the entry in the database, lb_admin tells this result and asks if the entry is to be deleted.

There are two situations in which it is likely that a database entry should be deleted:

- The server does not respond. lb_admin succeeds in contacting the LLB at the host where the server is located, but the server is not registered with that LLB. The server is probably no longer running.
- Server responds, but its UUIDs do not match the entry in the database. The server that responded is not the one that registered the entry.

Entries that meet either of these conditions are probably safe to delete.

In other situations, it is best not to delete the entry unless it can be verified directly that the server is not running (for example, by listing the processes running on its host).

When lb_admin asks to delete an entry, there are four ways to respond. A y[es] response deletes the entry. A n[o] response leaves the entry intact in the database. After a yes or a no, lb_admin proceeds to check the next entry in the current database. A g[o] response invokes automatic deletion, in which all eligible entries are deleted and all ineligible entries are left intact, without the user being queried, until all entries have been checked. A q[uit] response terminates the clean operation.

d[edit] Synonym for unregister.

h[elp] [Command] or ? [Command] Displays a description of the specified Command or, if none is specified, list all of the lb_admin commands.

l[ookup] Object Type Interface Looks up and displays all entries with matching Object, Type, and Interface fields in the current database. An asterisk can be used as a wildcard for any of the arguments. If all the arguments are wildcards, lookup displays the entire database.

q[uit] Exits the lb_admin session.

r[egister] Object Type Interface Location Annotation [Flag] Adds the specified entry to the current database. Use an asterisk to represent the nil UUID in the Object, Type, and Interface fields.

The location is a string in the format Family:Host[Port], where Family is an address family. Host is a host name, and Port is a port number. Possible values for Family include ip. A leading # can be used to indicate that a host name is in the standard numeric form. For example, ip:vienna[1756] and ip:#192.5.5.5[1791] are acceptable location specifiers.

The Annotation is a string of up to 64 characters annotating the entry. Use double quotation marks to delimit a string that contains a space or contains no characters. To embed a double quotation mark in the string, precede it with a backslash.

The Flag is either local (the default) or global, indicating whether the entry should be marked for local registration only or for registration in both the LLB and GLB databases. The Flag is a field that is stored with the entry but does not affect where the entry is registered. The set_broker and use_broker commands select the particular LLB or GLB database for registration.
Sets the host for the current LLB or GLB. If specifying global as the BrokerSwitch, set_broker sets the current GLB; otherwise, it sets the current LLB. The host is a string in the format Family:Host, where Family is an address family and Host is a host name. Possible values for Family include ip. A leading # can be used to indicate that a host name is in the standard numeric form. For example, ip:prague and ip:#192.5.5.5 are acceptable host specifiers.

Issue use_broker, not this command, to determine if subsequent operations will access the LLB or the GLB.

Sets the timeout period used by lb admin for all of its operations. With an argument of short or long, set_timeout sets the timeout accordingly. With no argument, it displays the current timeout value.

The location is a string in the format Family:Host[Port], where Family is an address family, Host is a host name, and Port is a port number. Possible values for Family include ip. A leading # can be used to indicate that a host name is in the standard numeric form. For example, ip:vienna[1756] and ip:#192.5.5.5[1791] are acceptable location specifiers.

An asterisk can be used as a wildcard in the Object, Type, And Interface fields to match any value for the field. Unless queries have been suppressed by invoking lb admin with the -no option, unregister allows deletion of each matching entry. A y(es) response deletes the entry. A n(o) response leaves the entry in the database. A g[o] response deletes all remaining database entries that match, without querying. A q[uit] response terminates the unregister operation, without deleting any additional entries.

Selects the type of database that subsequent operations will access, GLB or LLB. The BrokerSwitch is either global or local. If a BrokerSwitch is not supplied, use_broker determines if the current database is global or local.

Use set_broker to select the host whose GLB or LLB is to be accessed.

Related Information

The drm_admin (NCS) command

The llbd (NCS) daemon, llbd (NCS) daemon, lrglbd (NCS) daemon.

lb_find Command

Purpose

Gets a list of global location broker (GLB) server daemons and their attributes.

Syntax

lb_find [-q] [-v] [-dl]

Description

The lb_find command sends out inquiries to the NCS location broker daemons and gathers the responses. The results are analyzed to determine whether the global location broker is replicatable, and which cell each daemon serves. After ten seconds, the results are summarized, showing the GLB broker type, the server host's network address, a cell name of either default or alternate_N, and the cell's UUID.
Flags

-q  Queries for a GLB server, using the standard RPC mechanism. At most, one GLB server is printed, and only servers in the current machine’s cell are searched. The program exits with a status of 0 if a GLB server is found; otherwise the status is nonzero.

-v  Prints out the NCS version string.

-dl  Turns on RPC debugging while searching for GLB servers.

Examples

A network contains one glbd in each of two NCS cells and one nrglbd in a third cell.

/etc/ncs/lb_find

sent to broadcast address 192.92.110.255

waiting for replies

received response from glb daemon at ip:stimpy(192.92.110.43) port 1072.

received response from glb daemon at ip:oscar(192.92.110.16) port 1168.

received response from glb daemon at ip:vmess(192.92.110.21) port 1114.

......

replicatable     ip:stimpy     default 333b91c50000.0d.0
                 0.00.87.84.00.00.00

replicatable     ip:oscar     alternate_1 54bdad9a4000.0d.0
                 0.01.83.0f.00.00.00

non_replicatable ip:vmess     alternate_2 5c0e4acb8fa7.02.c
                 0.5c.6e.15.00.00.00

Related Information

The lb_admin command.

The glbd (NCS) daemon, llbd (NCS) daemon, nrglbd (NCS) daemon.

lbxproxy Command

Purpose

Low BandWidth X proxy.

Syntax

lbxproxy [ -display Display ] [ -help ] [ -display Display ] [ -motion Number ] [ -terminate | -reset ] [ -reconnect | -d ] [ -nolbx ] [ -nocomp ] [ -nodelta ] [ -notags ] [ -negfx ] [ -noimage ] [ -nosquish ] [ -nointernsc ] [ -naomfiles ] [ -atomsfiles ] [ -file File ] [ -nowinattr ] [ -nograbcmap ] [ -norgbfile ] [ -rgbfile Path ] [ -tagcachesize ] [ -zlevel Level ] [ -compstats ] [ -nozeropad ] [ -cheaterrors ] [ -cheatevents ]

Description

The lbxproxy command accepts client connections, multiplexes them over a single connection to the X server, and performs various optimizations on the X protocol to make it faster over low bandwidth and/or high latency connections. Applications that would like to take advantage of the Low Bandwidth extension to
X (LBX) must make their connections to an lbxproxy. These applications need to know nothing about LBX, they simply connect to the lbxproxy as if were a regular server.

For authentication/authorization, lbxproxy passes the credentials presented by the client along to the server. Since X clients connect to lbxproxy, it is important that the user’s .Xauthority file contain entries with valid keys associated with the network ID of the proxy. lbxproxy does not get involved with how these entries are added to the .Xauthority file. The user is responsible for setting it up.

The lbxproxy program has various flags, all of which are optional.

If :<Display> is specified, the proxy uses the Display port when listening for connections. The display port is an offset from port 6000, identical to the way in which regular X display connections are specified. If no port is specified on the command line, lbxproxy defaults to port 63. If the port that the proxy tries to listen on is in use, the proxy exits with an error message.

At startup, lbxproxy pre-interns a configurable list of atoms. This allows lbxproxy to intern a group of atoms in a single round trip and immediately store the results in its cache. While running, lbxproxy uses heuristics to decide when to delay sending window property data to the server. The heuristics depend on the size of the data, the name of the property, and whether a window manager is running through the same lbxproxy. Atom control is specified in the AtomControl file, set up during installation of lbxproxy, with command line overrides.

The file is a simple text file. There are three forms of lines: comments, length control, and name control. Lines starting with a ! (exclamation point) are treated as comments. A line of the form z length specifies the minimum length in bytes before property data is delayed. A line of the form options atomname controls the given atom, where options is any combination of the following characters: i means the atom should be pre-interned; and w means data for properties with this name should be delayed only if a window manager is also running through the same lbxproxy.

**Flags**

- **-atomsfile File**  
  Overrides the default AtomControl file.

- **-cheaterrors**  
  Allows cheating on X protocol for the sake of improved performance. The X protocol guarantees that any replies, events or errors generated by a previous request are sent before those of a later request. This puts substantial restrictions on when lbxproxy can short circuit a request. The -cheaterrors flag allows lbxproxy to violate X protocol rules with respect to errors. Use at your own risk.

- **-cheatevents**  
  The -cheatevents flag allows lbxproxy to violate X protocol rules with respect to events as well as errors. Use at your own risk.

- **-compstats**  
  Reports stream compression statistics every time the proxy resets or receives a SIGHUP signal.

- **-display Display**  
  Specifies the address of the X server supporting the LBX extension. If this flag is not specified, the display is obtained by the DISPLAY environment variable.

- **-help**  
  Prints a brief help message about the command line flags.

- **-i**  
  Causes all remaining arguments to be ignored.

- **-motion Number**  
  Specifies the maximum Number of events that can be in flight. A limited number of pointer motion events are allowed to be in flight between the server and the proxy at any given time. The default is 8.

- **-noatomsfile**  
  Disables reading of the AtomControl file.

- **-nocomp**  
  Disables stream compression.

- **-nodelta**  
  Disables delta request substitutions.

- **-nogfx**  
  Disables reencoding of graphics requests (not including image related requests).

- **-nograbmap**  
  Disables colormap grabbing.

- **-noimage**  
  Disables image compression.

- **-nointernsc**  
  Disables short circuiting of InternAtom requests.

- **-nolbx**  
  Disables all LBX optimizations.
-norgfile
Disables color name to RGB resolution in proxy.

-nosquish
Disables squishing of X events.

-notags
Disables usage of tags.

-nowinattr
Disables GetWindowAttributes/GetGeometry grouping into one round trip.

-nozeropad
Indicates to not zero out unused pad bytes in X requests, replies, and events.

-reconnect
Causes lbxproxy to reset (see -reset) and attempt to reconnect to the server when its connection to the server is broken. The default behavior of lbxproxy is to exit.

-rrgbfile Path
Specifies an alternate RGB database Path for color name to RGB resolution.

-tagsizesize
Sets the size of the proxy’s tag cache (in bytes).

-[terminate|reset]
The default behavior of lbxproxy is to continue running as usual when it’s last client exits.
The -terminate option will cause lbxproxy to exit when the last client exits. The -reset option will cause lbxproxy to reset itself when the last client exits. Resetting causes lbxproxy to clean up it’s state and reconnect to the server.

-zlevel Level
Set the Zlib compression level (used for stream compression). The default is 9.
1 = worst compression, fastest.
9 = best compression, slowest.

---

**Id Command**

**Purpose**
Links object files.

**Syntax**

```
Id [ [D]Number ] [ [e]Label ] [ [G] ] [ [H]Number ] [ [K] ] [ [m] ] [ [M] ] [ [o]Name ] [ [r] ] [ [s] ] [ [S]Number ] [ [T]Number ] [ [u]Name ] ... [ [v] ] [ [z] ] [ [Z]String ] ... [ [b]Option ] ... [ [L]Directory ] ... [ [FileID ... ] ] [ [Name ... ] ] [ InputFile ... ]
```

**Description**
The **Id** command, also called the linkage editor or binder, combines object files, archives, and import files into one output object file, resolving external references. It produces an executable object file that can be run. In addition, if you specify the **Id** command without the -s flag, you can use the output file as an **InputFile** parameter in another call to the **Id** command. By default, the **Id** command creates and places its output in the **a.out** file.

The **Id** command can relink a program without requiring that you list all input object files again. For example, if one object file from a large program has changed, you can relink the program by listing the new object file and the old program on the command line, along with any shared libraries required by the program. See **Examples** on page 173.

The **Id** command links input files in the order you specify on the command line. If you specify a file more than once, only the first occurrence of the file is processed. You must specify at least one input file, either with the -bl (uppercase letter I), -bimport, -bkeepfile, -f, or -l (lowercase letter L) flag or as an **InputFile** parameter. (The -bl, -bimport, or -bkeepfile flag is the -b flag used with the l, import, or keepfile option.)

Use the cc command to link files when you are producing programs that run under the operating system. Because the cc command calls the **Id** command with common options and necessary support libraries, you do not need to specify them on the command line. (This information is read from the /etc/xlC.cfg or /etc/vac.cfb configuration file.)

**Linking Mode**
The **Id** command can link 32-bit objects and programs as well as 64-bit objects and programs, but 32-bit and 64-bit objects may not be linked together. To specify the mode for linking, you can use the **OBJECT_MODE** environment variable or the -b32 or -b64 options.
Archive Files
Archive files are composite objects, which usually contain import files and object files, including shared objects. If an archive file contains another archive file or a member whose type is not recognized, the id command issues a warning and ignores the unrecognized member. If an object file contained in an archive file has the F_LOADONLY bit set in the XCOFF header, the id command ignores the member. This bit is usually used to designate old versions of shared objects that remain in the archive file to allow existing applications to load and run. New applications link with the new version of the shared object, that is, another member of the archive.

Shared Objects
A shared object, usually created by another call to the id command, is an object file with the F_SHROBJ bit set in the XCOFF header. A shared object defines external symbols that are resolved at run time. If you specify the -bnsos or -bnoautoimp option, the id command processes a shared object as an ordinary object file, and if the file is stripped, the link fails.

Ordinarily, a shared object used as input is only listed in the loader section of the output file if a symbol in the shared object is actually referenced. When the run-time linker is used, however, you might want shared objects to be listed even if there are no symbols referenced. When the -brelt option is used, all shared objects listed on the command-line that are not archive members are listed in the output file. The system loader loads all such shared objects when the program runs, and the symbols exported by these shared objects may be used by the run-time linker. Shared objects that are archive members are not loaded automatically unless automatic loading is enabled by an import file in the archive. To enable automatic loading, see "Import and Export File Format (-bl: and -be: Flags)" on page 169.

Import and Export Files
Import files are ASCII files that identify the external symbols to resolve at run time. An import file identifies the shared object defining the imported symbols. The system loader finds and resolves those symbols at run time. If the first line of an import file begins with #! (#, exclamation point), you can specify the file on the command line as an ordinary InputFile. Otherwise, you must use the -bl or -bimport option to specify the import file.

Export files are ASCII files that identify external symbols that are made available for another executable object file to import. The file format of an export file is the same as the file format of an import file.

Libraries
Libraries are files whose names end in .a, or possibly .so. To designate a library, you can specify an absolute or relative path name or use the -l (lowercase letter L) flag in the form -lName. The last form designates a libName.a file, or in dynamic mode, a libName.so file, to be searched for in several directories. These search directories include any directories specified by -L flags and the standard library directories /usr/lib and /lib.

Note: If you specify a shared object, or an archive file containing a shared object, with an absolute or relative path name, instead of with the -lName flag, the path name is included in the import file ID string in the loader section of the output file. You can override this behavior with the -bnoipath option.

Processing
The id command processes all input files in the same manner, whether they are archives or not. It includes the symbol tables of all objects, discarding only symbol definitions that duplicate existing symbols. Unlike some other versions of the id command, you do not need to order archive files so references precede definitions. Furthermore, you do not need to list an archive file more than once on the command line.

The order of the id command flags does not affect how they are processed, except for the flags used with input object files, libraries, and import files. These flags are: -L, -f, -I (lowercase letter L), -bkeepfile, and -bl (uppercase letter i). The flags are processed in the following order:
1. The -L flag adds a directory to the list of search directories to locate libraries specified by the -l (lowercase letter L) flag. The directories are searched in the order specified. All -L flags are processed before any -I flags are processed.

2. The ld command processes the InputFile parameters, the files specified by the -f flag and libraries specified by the -l (lowercase letter L) flag in the order specified.

3. The ld command processes import files specified by the -bl (uppercase letter i) flag in the order specified after processing all other object files and libraries. You can specify an import file as an input file without the -bl flag if it is necessary to process the file before processing some object files. In this case, the first line of the import file must begin with the #! (#, exclamation point) symbols, and the import file is processed with other input files as described in step 2.

4. The -bkeepfile option names an input file on which the ld command does not perform garbage collection. If the specified input file is also specified as an InputFile parameter or listed in a file specified by the -f flag, the -bkeepfile option does not affect the order in which the file is processed. Otherwise, the file is processed in order along with other input files, as described in step 2.

An output file produced by the ld command has execute permission set, unless you specify the -r flag or -bnox option or errors were reported while linking. An existing output file is not overwritten if any severe errors occurred, or if the output file was specified as an input file and any errors occurred.

Symbols
The ld command uses the following predefined symbols to provide special address locations and can be declared in C syntax as extern char name[ ]. The symbol names are:

- _text
- _etext
- _data
- _edata
- _end or end

Specifies the first location of the program.
Specifies the first location after the program.
Specifies the first location of the data.
Specifies the first location after the initialized data.
Specifies the first location after all data.

The only way to use these symbols is to take their addresses. If an input file redefines any of these symbols, there may be unpredictable results. An additional predefined symbol, _ptrgl, is used by compilers to implement calls using function pointers.

Garbage Collection
By default, the ld command performs garbage collection, deleting control sections (CSECTs) that are not referenced when generating the output file.

A CSECT is an indivisible unit of coding or data. A CSECT references another CSECT if it contains a relocation entry (RLD) referring to a symbol contained in the other CSECT. A referenced CSECT causes all CSECTs it references to be referenced as well. In addition, a CSECT is referenced if it contains exported symbols, symbols specified with the -u flag, or the symbol designated as the entry point with the -e flag.

If a symbol is not referenced but is needed in the output file, you can export the symbol, specify the symbol with the -u flag, or suppress garbage collection. To suppress garbage collection, use the -r flag or -bnogc option. To suppress garbage collection for individual object files, use the -bkeepfile option or the -bgcbypass option. Even when garbage collection is suppressed, unreferenced internal symbols are deleted.

Ignored and Unsupported Flags
For compatibility with other versions of the ld command, some flags are recognized but ignored. These flags produce a message stating that the flag and its operand were ignored. An ignored flag does not cause the ld command to stop without further processing. The following flags are ignored:
Flags that the `ld` command does not support result in an error message. After all unsupported flags are diagnosed, the `ld` command stops without further processing.

**Flags**

The `ld` command conforms to the XPG Utility Syntax Guidelines, except that the argument – only applies to the next operand, not to the remaining operands on the command line. For example, in the command line:

```
ld -- -s -v
```

The `-s` is treated as a filename and the `-v` is treated as a flag. To have `-v` treated as a filename, specify:

```
ld -- -s -- -v
```

**Note:** Enter a flag with an operand with or without a space between the flag and the operand. You can specify numeric values in decimal, octal (with a leading 0), or hexadecimal (with a leading 0x or 0X) format. If you specify conflicting flags on the command line, the `ld` command accepts the latest flag and ignores earlier ones.

- `-bOption` Sets special processing options. This flag can be repeated. For more information on these options, see "Options (-bOptions)" on page 161.
- `-DNumber` Sets the starting address for the initialized data (the data section) of the output file to `Number`. If the specified number is -1, the data section starts immediately after the text section. By default, the data section begins at location 0. **Note:** The system loader relocates the data section at run time, so the specified number only affects addresses listed in address maps or printed by utilities such as the `dump` or `nm` command.
- `-eLabel` Sets the entry point of the executable output file to `Label`. The default entry point is `__start` (double underscore `start`).
- `-fFileid` Specifies a file containing a list of input files to process. FileId must contain a list of input file names. Each line in FileId is treated as if it were listed separately on the `ld` command line. Lines in the file can contain shell pattern characters * (asterisk), [ (left bracket), ] (right bracket), and ? (question mark), which are expanded using the `glob` subroutine and can designate multiple object files.
- `-G` Produces a shared object enabled for use with the run-time linker. The `-G` flag is equivalent to specifying the `erok`, `rtl`, `nortlib`, `nosymbolic`, `nautoexec`, and `M:SRE options with the `-b` flag. Subsequent options can override these options.
- `-HNumber` Aligns the text, data, and loader sections of the output file so that each section begins on a file offset that is a multiple of `Number`. If the specified number is 1, no alignment occurs. If the specified number is 0, the loader section is aligned on a word boundary, and the text and data sections are aligned on a boundary so as to satisfy the alignment of all CSECTs in the sections. The default value is 0.

If the specified `Number` causes any CSECTS to be unaligned within the output file, the `ld` command issues a warning and the output executable file may not load or run.

- `-K` Aligns the header, text, data, and loader sections of the output file so that each section begins on a page boundary. This flag is equivalent to specifying `-HNumber`, where `Number` is the page size of the machine on which `ld` is running.
-IName
In dynamic mode, processes the libName.so or libName.a file. In all cases, directories specified by the -L flag or in the standard library directories (/usr/lib and /lib) are searched to find the file. In dynamic mode, the first directory containing either libName.so or libName.a satisfies the search. If both files are found in the same directory, libName.a is used. To preference to libName.so, you must specify the rtl option as well. You can repeat this flag. For more information about dynamic mode, see "Run-time Linking" on page 169.

Note: The first definition of a symbol is kept, even if no reference to the symbol has been seen when the archive is read. In other versions of the ld command, a symbol defined in an archive is ignored if no reference to the symbol has been seen when the archive is read.

-LDirectory
Adds Directory to the list of search directories used for finding libraries designated by the -l (lowercase letter L) flag. The list of directories, including the standard library directories, is also recorded in the output object file loader section for use by the system loader unless you use the -blibpath or -bnolibpath option. You can repeat this flag.

-m or -M
Lists to standard output the names of all files and archive members processed to create the output file. Shared objects and import files are not listed.

-oName
Names the output file Name. By default, the name of the output file is a.out.

-r
Produces a nonexecutable output file to use as an input file in another ld command call. This file may also contain unresolved symbols. The -r flag is equivalent to specifying the erok, noglink, nox, and nogc options with the -b flag. (Subsequent options can override these options.)

-s
Strips the symbol table, line number information, and relocation information when creating the output file. Stripping saves space but impairs the usefulness of the debuggers. You can also strip an existing executable by using the strip command.

Note: Non-shared objects cannot be linked if they are stripped. A shared object can be stripped, but a shared object cannot be used when linking statically.

-SNumber
Sets the maximum size (in bytes) allowed for the user stack when the output executable program is run. This value is saved in the auxiliary header and used by the system loader to set the [soft ulimit]. The default value is 0.

For more information on large user stacks and 32-bit programs, see "Large Program Support Overview" in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

-TNumber
Sets the starting address of the text section of the output file to Number. The default value is 0.

Note: The system loader relocates the text section at run time, so the specified number affects only addresses listed in address maps or printed by utilities such as the nm or the dump command.

-uName
Prevents garbage collection of the external symbol Name. If the specified symbol does not exist, a warning is reported. You can repeat this flag.

-v
Writes additional information about binder command execution to the loadmap file.

-z
Functions the same as the -K flag.

-ZString
Prefixes the names of the standard library directories with String when searching for libraries specified by the -l (lowercase letter L) flag. For example, with the -Ztest and -lxyz flags, the ld command looks for the /test/usr/lib/libxyz.a and /test/lib/libxyz.a files. When the -ZString flag is used, the standard library directories are not searched. This flag has no effect on the library path information saved in the loader section of the output file. This flag is useful when developing a new version of a library. You can repeat this flag.

The Binder
The ld command verifies the command-line arguments and calls the binder (by default the /usr/ccs/bin/bind file), passing a generated list of binder subcommands. The binder program actually links the files. Although the binder is usually called by the ld command, you can start the binder directly. In this case, the binder reads commands from standard input.

Two options affect the calling of the binder. The binder option specifies which binder to call, and the nobind option prevents the ld command from calling a binder. Other binder options affect the binder subcommands that are generated.
If the `ld` command does not detect any errors in the options or command-line arguments, it calls the binder. The binder is called with a command line of the form:

```
bind [quiet_opt] [loadmap_opt]
```

The default value for `quiet_opt` is `quiet` and the default value for the `loadmap_opt` is the null string, so the default command line is:

```
/usr/ccs/bin/bind quiet
```

**Options (-bOptions)**

The following values are possible for the `Options` variable of the `-b` flag. You can list more than one option after the `-b` flag, separating them with a single blank.

**Notes:**

1. In the following list of binder options, two option names separated by the word or are synonymous.
2. The `FileID` indicates a path name. You can use either a relative or a full path name.
3. For a non-repeatable option that is followed by an argument, you can negate the option using a null argument. That is, specify only the option and the colon.
4. If you specify conflicting options, the last one takes precedence.

- **32** Specifies 32-bit linking mode. In this mode, all input object files must be XCOFF32 files, or an error is reported. XCOFF64 archive members are ignored. For import or export files specifying the mode of certain symbols, 64-bit symbols are ignored. If both `-b32` and `-b64` options are specified, the last specified option is used. If neither option is specified, the mode is determined from the value of environment variable `OBJECT_MODE`.

- **64** Specifies 64-bit linking mode. In this mode, all input object files must be XCOFF64 files, or an error will be reported. XCOFF32 archive members are ignored. For import or export files specifying the mode of certain symbols, 32-bit symbols are ignored. If both `-b32` and `-b64` options are specified, the last specified option is used. If neither option is specified, the mode is determined from the value of environment variable `OBJECT_MODE`.

- **asis** Processes all external symbols in mixed case. This is the default. To process all external symbols in uppercase, see the `caps` option that follows.

- **autoexp** Automatically exports some symbols from the output module without having to list them in an export file. (This option does not export all symbols from the output module. Use the `-bexpall` option to export all symbols.) This is the default. Use this option when linking a main program. The linker assumes that you are linking a main program when you do not specify a module type (with the `M` or `modtype` option) beginning with S and you do not use the `noentry` option.

When you use the `autoexp` option, if any shared object listed on the command-line imports a symbol from the special file . (dot), and the module being linked contains a local definition of the symbol, the symbol is exported automatically.

Other symbols are also exported automatically when you link with the `rtl` option. If a symbol defined in the module being linked has one or more additional definitions exported from a shared object listed on the command-line, and if any of the definitions is a BSS symbol, the symbol is exported automatically. If the definition in the module being linked is a BSS symbol, the symbol is exported with the `nosymbolic` attribute. Otherwise, the symbol is exported with the `symbolic` attribute. If the symbol is listed in an export file with another export attribute, the explicit attribute is used.

If the `autoexp` option would automatically export a symbol, but the symbol is listed in an export file with the `list` attribute, the symbol is not exported.

- **autoimp or so** Imports symbols from any shared objects specified as input files. The shared objects are referenced but not included as part of the output object file. This is the default.

- **autoload:** Automatically load archive member when the `-brtl` option is used.

Alphabetical Listing of Commands  161
bigtoc

Generates extra code if the size of the table of contents (TOC) is greater than 64KB. Extra code is needed for every reference to a TOC symbol that cannot be addressed with a 16-bit offset. Because a program containing generated code may have poor performance, reduce the number of TOC entries needed by the program before using this option. The default is the nobigtoc option.

bindcmds:FileID

Writes a copy of the binder commands generated by the Id command to FileID. You can redirect the resultant file as standard input to the binder program when the binder program is called as a standalone program. By default, no file is produced.

binder:FileID

Uses FileID as the binder called by the Id command. The default binder is the /usr/ccs/bin/bind file.

bindopts:FileID

Writes a copy of the binder program arguments to FileID. You can use the resultant file to start the binder program as a standalone program. By default, no file is produced.

C:FileID or calls:FileID

Writes an address map of the output object file to FileID. Symbols are sorted by section and then by address. For each symbol listed in the map, references from the symbol to other symbols are listed. By default, no file is produced. To learn more about the calls option, see “Address Maps” on page 172.

caps

Processes all external symbols in uppercase. The default is the asis option.

comprld or crld

Combines multiple relocation entries (RLDs) at the same address into a single RLD when possible. This is the default.

cror15

Uses the cror 15,15,15 (0x4def7b82) instruction as the special no-op instruction following a call instruction. The default value is ori 0, 0, 0 (0x60000000). See the nop option.

Use this option when linking object files on the current level of the system that you intend to relink on AIX 3.1.

cror31

Uses the cror 31,31,31 (0xfffff82) instruction as the special no-op instruction following a call instruction. The default value is ori 0, 0, 0 (0x60000000). See the nop option.

Use this option when linking object files on the current level of the system that you intend to relink on AIX 3.2.

D: Number [/dsa] or maxdata:Number[/dsa]

Sets the maximum size (in bytes) allowed for the user data area (or user heap) when the executable program is run. This value is saved in the auxiliary header and used by the system loader to set the soft data ulimit. The default value is 0. When this option is used, the specified number of bytes are reserved for the user data area. The program may not explicitly map objects, using shmat or mmap functions, to virtual addresses that are reserved for the user data area.

For 32-bit programs, the maximum value allowed by the system is 0x80000000 for programs running under Large Program Support and 0x00000000 for programs running under Very Large Program Support. See “Large Program Support Overview” in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs. When a non-zero value is specified, the user data area begins in segment 3, and the program uses as many segments as necessary to satisfy the maxdata value specified.

For 64-bit programs the maxdata option provides a guaranteed maximum size for the programs data heap. Any value can be specified but the data area cannot extend past 0x0FFFFFFFFFFFFFF regardless of the maxdata value specified.

dbg:Option or debugopt:Option

Sets a special debugging or control option. By default, no debug option is set.

delcsect

Deletes all symbols in a CSECT if any symbol in the CSECT was defined by a previously read object file. This option prevents more than one instance of the same function from existing in the same program. For example, if a.o defines function a() and b.o defines functions a() and b(), linking a.o and b.o with the -delcsect option deletes symbols a() and b() from b.o. Thus, two instances of a() do not exist. The default is the nodelcsect option.
**dynamic or shared**

Cause the linker to process subsequent shared objects in dynamic mode. This is the default. In dynamic mode, shared objects are not statically included in the output file. Instead, the shared objects are listed in the loader section of the output file. When you specify the **rtl** option and dynamic mode is in effect, files ending in .so as well as .a satisfy searches for libraries specified with the -l (lowercase L) flag. When both are in effect, preference is given to .so instead of .a when present in same directory. Otherwise, if only dynamic is set and not **rtl**, preference is given to .a instead of .so.

**E:** *FileID or export:* *FileID*

Exports the external symbols listed in the file *FileID*. Exported symbols are listed in the loader section of the output file. There is no default export file.

**ernotok or f**

Reports an error if there are any unresolved external references. This is the default.

**erok**

Produces the output object file without errors even if there are unresolved external references. The default is the **ernotok** option.

**errmsg**

Writes error messages to standard error if the error level of the message is greater than or equal to the value of the **halt** option and the **quiet** option is used or standard output is redirected. This is the default.

**ex1:** *FileID, ex2:* *FileID, ex3:* *FileID, ex4:* *FileID, and ex5:* *FileID**

Provide user exits in the typical binder subcommand sequence. Each file specified by *FileID* must contain a list of binder subcommands, which will be run as follows:

**ex1:** *FileID*

Before reading any *InputFiles*

**ex2:** *FileID*

Immediately before symbol resolution

**ex3:** *FileID*

Immediately after symbol resolution

**ex4:** *FileID*

Immediately before writing the output file

**ex5:** *FileID*

Immediately after writing the output file

**expall**

Exports all global symbols, except imported symbols, unreferenced symbols defined in archive members, and symbols beginning with an underscore (_). You can export additional symbols by listing them in an export file. This option does not affect symbols exported by the **autoexp** option.

When you use this option, you might be able to avoid using an export file. On the other hand, using an export file provides explicit control over which symbols are exported, and allows you to use other global symbols within your shared object without worrying about conflicting with names exported from other shared objects. The default is **noexpall**.

**export:** *FileID**

Functions the same as the **E:** *FileID* option.

**f**

Functions the same as the **ernotok** option.

**gc**

Performs garbage collection. Use the **nogc**, **gc bypass**, or **keepfile** option to prevent garbage collection for some or all object files. This is the default.

**gcbypass:** *Number**

Specifies the number of files to bypass when garbage collecting if the **gc** option is specified. This option is ignored if the **nogc** option is used. If *Number* is 0, this option is equivalent to the **gc** option and garbage collection is performed for all files. The default value is 0.

**glink:** *FileID**

Uses the global linkage prototype code specified by *FileID*. Global-linkage interface code is generated for each imported or undefined function. In 32-bit mode, the default is the `/usr/lib/glink.o` file. In 64-bit mode, the default is the `/usr/lib/glink64.o` file.
**h: Number or halt: Number**

Specifies the maximum error level for binder command processing to continue. The default value is 4. If any binder subcommand has a return value greater than Number, no additional binder subcommands are processed. If the halt level value is 8 or greater, the output file may not be executable if it is produced at all. Return values are:

- 0: No error
- 4: Warning
- 8: Error
- 12: Severe error
- 16: Internal program error

**l: FileID or import: FileID**

(Uppercase i) Imports the symbols listed in FileID. There is no default import file.

**initfini: [Initial] [:Termination] [:Priority]**

Specifies a module initialization and termination function for a module, where Initial is an initialization routine, Termination is a termination routine, and Priority is a signed integer, with values from -2,147,483,648 to 2,147,483,647. You must specify at least one of Initial and Termination, and if you omit both Termination and Priority, you must omit the colon after Initial as well. If you do not specify Priority, 0 is the default. This option can be repeated.

This option sorts routines by priority, starting with the routine with the smallest (most negative) priority. It invokes initialization routines in order, and termination routines in reverse order.

This option invokes routines with the same priority in an unspecified order, but if multiple initfini options specify the same priority and both an initialization and termination routine, it preserves the relative order of the routines. For example, if you specify the options initfini:i1:f1 and initfini:i2:f2, then function i1 and i2 are invoked in an unspecified order, but if i1 is invoked before i2 when the module is loaded, f2 will be invoked before f1 when the module is unloaded.

**Note:** IBM will only use priorities in the following inclusive ranges:

-2,147,483,640 to -2,147,000,000
-1,999,999,999 to -1,000,000,000
-99,999,999 to -50,000,000
0 to 50,000,000
99,999,999 to 99,999,999
1,000,000,000 to 1,999,999,999
2,147,000,000 to 2,147,483,640

**ipath**

For shared objects listed on the command-line, rather than specified with the -l flag, use the path component when listing the shared object in the loader section of the output file. This is the default.

**keepfile: FileID**

Prevents garbage collection of FileID. By default, the binder deletes unreferenced CSECTS in all files. You can repeat this option.
lazy Enables lazy loading of a module’s dependent modules. This option adds a `-lrl` option following other flags and options. If the `-lrl` option is specified, the `-blazy` option is ignored and lazy loading is not enabled.

When a module is linked, a list of its dependent modules is saved in the module’s loader section. The system loader automatically loads the dependent modules after the module is loaded. When lazy loading is enabled, loading is deferred for some dependents until a function is called in the module for the first time.

A module is lazy loaded when all references to the module are function calls. If variables in the module are referenced, the module is loaded in the typical way.

**Note:** Be careful while comparing function pointers if you are using lazy loading. Usually a function has a unique address to compare two function pointers to determine whether they refer to the same function. When using lazy loading to link a module, the address of a function in a lazy loaded module is not the same address computed by other modules. Programs that depend upon the comparison of function pointers should not use lazy loading.

For more information about lazy loading, refer to “Shared Libraries and Lazy Loading” in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

I: FileID or loadmap:FileID (Lowercase L) Writes each binder subcommand and its results to FileID. By default, no file is produced. Uses Path as the library path when writing the loader section of the output file. Path is neither checked for validity nor used when searching for libraries specified by the -l flag. Path overrides any library paths generated when the -l flag is used.

If you do not specify any -l flags, or if you specify the nolibpath option, the default library path information is written in the loader section of the output file. The default library path information is the value of the `LIBPATH` environment variable if it is defined, and `/usr/lib:/lib`, otherwise.

loadmap:FileID Functions the same as the I:FileID option.

m: UR Sets the SGETUREGS flag for the linker. When the SGETUREGS flag is set, the contents of the registers are stored in a buffer. This option is used by coredump system call.

M: ModuleType or modtype: ModuleType Sets the two-character module-type field and the shared object flag in the object file. The module type is not checked by the binder, but it should be set to one of the following values:

1L Single use. Module requires a private copy of the data section for each load.

RE Reusable. Module requires a private copy of the data area for each process dependent on the module.

RO Read only. Module is read-only, and can be used by several processes at one time.

If an S prefix is used on any of the preceding options, the shared flag in the object file is set. The system loader attempts to share a single instance of the data section of an RO module. Otherwise, the module type is ignored by the system loader. The default value is 1L.

map:FileID or R: FileID Writes an address map of the output object file to FileID. Symbols are sorted by section and then by address. By default, no file is produced. To learn more about the map option, see "Address Maps" on page 172.

maxdata: Number [dsa] Functions the same as the D: Number [dsa] option.

maxstack: Number or S: Number Functions the same as the -S flag.

modtype: ModuleType Functions the same as the M: ModuleType option.

nl or noloadmap Does not write the binder subcommands and their results to a load map file. This is the default.

noautoexp Prevents automatic exportation of any symbols. The default is the autoexp option.
noautoimp or nso
Links any unstripped, shared objects as ordinary object files. When you use this option, the loader section of shared objects is not used. The default is the autoimp or so option.

Note: By using either of these flags, you statically link a shared object file into an application. Any application that is statically linked is not binary portable from any fix or release level to any other fix or release level.

nobigtoc
Generates a severe error message if the size of the TOC is greater than 64KB. If an output file is produced, it will not execute correctly. This is the default.

nobind
Omits calling the binder. Instead, the id command writes the generated list of binder subcommands to standard output. By default, the id command calls the binder.

nocomprld or nocrl
Does not combine multiple relocation entries (RLDs) at the same address into a single RLD. The default is the comprld or crld option.

nodelcsect
Allows all symbols in the CSECT to be considered during symbol resolution, even if some symbol in the CSECT is defined in a previously read object file. For more information, see the delcsect option. The nodelcsect option is the default.

noexpall
Does not export symbols unless you list them in an export file or you export them with the autexp option. This is the default.

nodeentry
Indicates that the output file has no entry point. To retain any needed symbols, specify them with the -u flag or with an export file. You can also use the -r flag or the nogc or gcbt options to keep all external symbols in some or all object files. If neither the noentry nor the nox option is used and the entry point is not found, a warning is issued.

noerrmsg
Does not write error messages to standard error. Use this option if you specify the noquiet option and you pipe standard output to a command such as tee or pg.

nogc
Prevents garbage collection. CSECTs in all object files that contain global symbols are kept, whether they are referenced or not. The default is the gc option.

noglink
Prevents the id command from inserting global linkage code. By default, the binder inserts the global linkage code.

noipath
For shared objects listed on the command-line, rather than specified with the -I flag, use a null path component when listing the shared object in the loader section of the output file. A null path component is always used for shared objects specified with the -I flag. This option does not affect the specification of a path component by using a line beginning with #! in an import file. The default is the ipath option.

nolibpath
Overrides any previous library path generated by the -L flag or specified by the libpath option. Instead, the default library path information is written in the loader section of the output file. The default library path information is the value of the LIBPATH environment variable if it is defined, and /usr/lib:/lib otherwise.

noloadmap
Functions the same as the nl option.

nom
Does not list the object files used to create the output file. This option overrides the -m flag. This is the default.

noobjreorder
Does not use the depth-first CSECT reordering logic. The CSECTs in the output file are arranged in the same order that the object files and library files were specified on the command line, except as follows:

- CSECTs are placed in their correct text, data, or BSS section of the object file, based on the storage-mapping class field of each CSECT.
- All CSECTs with a storage-mapping class of XMC_TC (TOC address constant) or XMC_TD (TOC variable) are grouped together.

If both the noobjreorder and noorder options are specified, the noorder option takes precedence. The default is the reorder option.
nop: Nop

Specifies the no-op instruction used after branches to local routines. Nop can be one of the special values cror15, cror31, ori, or an eight-digit hexadecimal number. The ori instruction is the default. Specifying the -bnop:cror15 option is equivalent to specifying the -bcror15 option; specifying the -bnop:cror31 option is equivalent to specifying the -bcror31 option. If you specify one of the special nop options, all previous nop options are overridden.

If Nop is an eight-digit hexadecimal number, it specifies an arbitrary machine instruction. This machine instruction overrides any previously specified special value for Nop instruction. When you use this form, you can repeat this option.

The last machine instruction specified is the instruction generated by the binder after intramodule branches. Other specified machine instructions are recognized as no-op instructions, but are converted to the preferred no-op instruction.

noquiet

Wipes each binder subcommand and its results to standard output. The default is the quiet option.

noreorder

Does not reorder CSECTs, except to combine all XMC_TD (TOC address constant) and XMC_TD (TOC variable) CSECTs and place them in the data section, and combine all BSS symbols and place them in the bss section. All other CSECTs are placed in the text section, so text and data are mixed in the output file. When the noreorder option is used, the text section of the output file may no longer be position-independent and the system loader will not load a module if the text section is not position-independent. Therefore, avoid using this option for programs and kernel extensions. If both noobjreorder and noreorder options are specified, the noreorder option takes precedence. The default is the reorder option.

nortllib

Disables run-time linking for the output file. This option implies the nortlib and nosymbolic- options. Furthermore, additional actions described under the rtl option are not taken. This is the default.

nortl

Disables run-time linking for the output file. This option implies the nortlib and nosymbolic- options. Furthermore, additional actions described under the rtl option are not taken. This is the default.

nostrip

Does not generate a stripped output file. Thus, the symbol table and relocation information is written in the output file. This option overrides the -s flag. This is the default.

nosymbolic

Assigns the nosymbolic attribute to most symbols exported without an explicit attribute. For more information, see "Attributes of Exported Symbols" on page 171. The default is the nosymbolic- option.

nosymbolic-

Assigns the nosymbolic- attribute to most symbols exported without an explicit attribute. For more information, see "Attributes of Exported Symbols." This is the default.

notextro or nro

Does not check to ensure that there are no load time relocation entries for the text section of the output object file. This is the default.

notypchk

Does not check function-parameter types between external functional calls. The default is the typchk option.

nov

Does not write additional information to the load map file. This option is the default and overrides the -v flag.

nox

Does not make the output file executable. Neither the auxiliary header nor the loader section is written. Flags and options that specify values written in the auxiliary header or loader section have no effect when this option is used. The default is the x option.

nro

Functions the same as the notextro option.

nso

Functions the same as the noautoimp option.

pD: Origin

Specifies Origin as the address of the first byte of the file page containing the beginning of the data section. For example, if the data section begins at offset 0x22A0 in the object file, and p0:0x20000000 is specified, the first byte of the data section is assigned address 0x20000000. This assumes a page size of 4096 (0x1000) bytes.

pT: Origin

Specifies Origin as the address of the first byte of the file page containing the beginning of the text section. For example, if the text section begins at offset 0x264 in the object file, and pT:0x10000000 is specified, the first byte of the text section is assigned address 0x10000264.

quiet

Does not write binder subcommands and their results to standard output. This is the default.
Functions the same as the map:map option.

Reorders CSECTs as part of the save command processing. The reorder process arranges CSECTs of the same storage-mapping class by proximity of reference. This is the default.

Renames the external symbol Symbol to NewName. In effect, it is as if all definitions and references to Symbol in all object files were renamed to NewName before the files were processed. By default, symbols are not renamed.

Ensures that there are no load time relocation entries for the text section of the resultant object file. The default is the ro option.

Enables run-time linking for the output file. This option implies the rtl and symmetric options.

When dynamic mode is in effect (see the dynamic and static options), the rtl option allows input files specified with the -l flag to end in .so as well as in .a.

All input files that are shared objects are listed as dependents of your program in the output files loader section. The shared objects are listed in the same order as they were specified on the command line.

A shared object contained in an archive is only listed if the archive specifies automatic loading for the shared object member. You specify automatic loading for an archive member foo.o by creating a file with the following lines:

```bash
# autoload
#! (foo.o)
```

and adding the file as a member to the archive. You may also specify automatic loading for an archive member foo.o by using the -bautoload option:

-bautoload:<archive_name>(foo.o)

You may specify additional archive members with additional -bautoloads.

Includes a reference to the run-time linker. The run-time linker is defined in librta.a, and an implicit -rtl flag is added automatically to the command line. This option (implied by the rtl option) must be used when linking a main program or no run-time linking will occur.

Shared objects do not have to be linked with this option. The default is the nortllib option.

Functions the same as the -S flag.

Writes an address map of the object file to FileID. Symbols are listed alphabetically. For each symbol listed in the map, references from the symbol to the other symbols are listed. By default, no file is produced. To learn more about the scalls option, see "Address Maps" on page 172.

Functions the same as the dynamic option.

Writes an address map of the object file to FileID. Symbols are listed alphabetically. By default, no file is produced. To learn more about the smap option, see "Address Maps" that follows.

Functions the same as the autoimp option.
stabcmpct: Level

Specifies the level of compaction for stabstrings in the debug section. Stabstrings are strings that are longer than eight characters. Each substring in the symbol table has its own offset in the debug section. The following values are valid for Level:

0  Does not compact. Separate copies of duplicate stabstrings are written to the debug section.

1  Deletes duplicates. Each stabstring is written once to the .debug section. Duplicate stabstrings in the symbol table specifies the same offset into the debug section.

2  Renumbers the stabstrings and deletes most duplicates. (In some instances, multiple stabstrings can exist. They describe the same type but use different type numbers.) The scope of a type number is the entire output file, rather than a single input file as indicated by a C_FILE symbol table entry.

If the binder does not recognize a stabstring, it returns an error message and the resulting executable file does not have valid stabstrings. The rest of the file is unaffected by the error.

static

Causes the linker to process subsequent shared objects in static mode. In static mode, shared objects are statically linked in the output file. Furthermore, files ending in .so are not found when searching for libraries specified with the -I flag.

sxref: FileID

Writes an address map of the object file to FileID. Symbols are listed alphabetically. For each symbol listed in the map, references to the symbol from other symbols are listed. By default, no file is produced. To learn more about the sxref option, see “Address Maps” that follows.

symbolic

Assigns the symbolic attribute to most symbols exported without an explicit attribute. For more information, see “Attributes of Exported Symbols” that follows. The default is the nosymbolic- option.

textro

Same as the ro option.

typchk

Performs function-parameter type checking between external functional calls.

Parameter-type checking information can be included in object files by compilers and assemblers. This is the default.

For more information on type checking, see the “XCOFF (a.out) File Format” in AIX 5L Version 5.2 Files Reference.

x

Makes the output file executable, if no errors exist. This is the default option.

X or xref: FileID

Writes an address map of the object file to FileID. Symbols are sorted by section and then by address. For each symbol listed in the map, references to the symbol from other symbols are listed. By default, no file is produced. To learn more about the xref option, see “Address Maps” that follows.

Run-time Linking

By default, references to symbols in shared objects are bound at link time. That is, the output module associates an imported symbol with a definition in a specific shared object. At load time, the definition in the specified shared object is used even if other shared objects export the same symbol.

You can cause your program to use the run-time linker, allowing some symbols to be rebound at load time. To create a program that uses the run-time linker, link the program with the -brtl option. The way that shared modules are linked affects the rebinding of symbols.

You can build shared objects enabled for run-time linking by using the -G flag. You can fully enable run-time linking for existing shared objects by relinking them with the rtl_enable command, as long as they have not been stripped.

Import and Export File Format (-bl: and -bE: Flags)

Each line within an import or export file must list a single symbol, followed by optional keywords or an optional address. Keywords are svc, svc32, svc3264, svc64, syscall, syscall32, syscall3264, syscall64, symbolic, nosymbolic, nosymbolic-, list, cm, bss, and weak. Only the keyword weak may be used in conjunction with another keyword.
In an import file, you can specify an address after the symbol to map data CSECTs to a shared memory segment and eliminate the need to use the assembler. You can also use one of the keywords cm, bss, or weak to specify the storage class of an imported symbol. When the autoexp option is used, the storage class of an imported symbol affects which symbols are automatically exported. If any other keyword is specified in an import file, the keyword is ignored.

In an export file, you can use the svc or syscall keyword after a name to indicate that it is a system call. This is needed when linking kernel extensions. You can use the symbolic, nosymbolic, or nosymbolic-keyword to associate an attribute with an exported symbol. For more information, see "Attributes of Exported Symbols" on page 171. You can use the list keyword to cause a symbol to be listed in the loader section of the output file, although it will not be marked as an exported symbol. This can be used for applications that want to process some symbols at run time. Listed symbols are not processed by the system loader or the run-time linker. A symbol address and the keywords cm and bss are ignored in an export file. The weak keyword can be used to specify weak symbol binding, and may be used with another attribute.

The id command treats import and export files according to the following guidelines:

- A blank line is ignored.
- A line beginning with an * (asterisk) is a comment and is ignored.
- A line beginning with a # (#, blank space) provides operands to the setopt binder subcommand (-bdg:Option). For example, a line containing # verbose causes the binder to list each symbol as it is read from the file. These option settings are active only while processing the file. The #32, #64, #no32, and #no64 options can be used to specify whether the listed symbols should be used for 32-bit links, 64-bit links, or both.

32-bit and 64-bit Import File Options

32 This option is used in an import or export file to specify that subsequent symbols should be processed when linking in 32-bit mode, but ignored when linking in 64-bit mode. If no 32 or 64 option is specified, all symbols are processed in both 32- and 64-bit modes.

64 This option is used in an import or export file to specify that subsequent symbols should be processed when linking in 64-bit mode, but ignored when linking in 32-bit mode. If no 32 or 64 option is specified, all symbols are processed in both 32- and 64-bit modes.

no32 or no64 Override a previous 32 or 64. Subsequent symbols are processed in both 32- and 64-bit modes.

- When processing an import file, a line beginning with a #! (#, exclamation point) provides the shared library name to be associated with subsequent import symbols. The line can occur more than once and applies to subsequent symbols until the next line beginning with #! is read. This file name information is placed in the loader section of the XCOFF object file. It is used by the system loader to locate the appropriate object file at execution time. If the import file name is path/file (member), the file name placed in the loader section is determined based on the import file name and the contents of the #! line of the import file, as follows:

#! (Nothing after the #!) Use null path, null file, and null number. This is treated as a deferred import by the system loader.

#! () Use path, ifile, and nmember. This line can be used if the import file is specified as an InputFile parameter on the command line. The file must begin with #! in this case. This line can also be used to restore the default name if it was changed by another #! line.

#! path/file (member) Use the specified path, file, and member.
#! path/file Use the specified path and file, and a null member.
#! file Use a null path, the specified file, and a null member. At run time, a list of directories is searched to find the shared object.

#! (member) Use path, ifile, and the specified member. At run time, a list of directories is searched to find the shared object.

#! file (member) Use a null path and specified file and member. At run time, a list of directories is searched to find the shared object.
#! .

(A single dot) This name refers to the main executable. Use this file name when you are creating a shared object that imports symbols from multiple main programs with different names. The main program must export symbols imported by other modules, or loading will fail. This import file name can be used with or without the run-time linker.

#! ..

(Two dots) Use this name to list symbols that will be resolved by the run-time linker. Use this file name to create shared objects that will be used by programs making use of the run-time linker. If you use a module that imports symbols from .. in a program that was not linked with the rtlib option, symbols will be unresolved, and references to such symbols will result in undefined behavior.

To automatically load archive members when the -brtl option is used, you can create an import file as follows. If shr.so is a shared object in an archive, create an import file:

```
# autoload
#! (shr.so)
```

You can list additional member names on additional lines, if appropriate. You do not need to list symbol names in the import file because the symbols imported from shr.so will be read from shr.so itself.

For more information on creating a shared library, see “How to Create a Shared Library” in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs. For more information on loading and binding, see the load subroutine in AIX 5L Version 5.2 Technical Reference: Base Operating System and Extensions Volume 1.

Attributes of Exported Symbols

When you use run-time linking, a reference to a symbol in the same module can only be rebound if the symbol is exported with the proper attribute. References to symbols with the symbolic attribute cannot be rebound. References to symbols with the nosymbolic attribute can be rebound. References to symbols with the nosymbolic- attribute can be rebound if the symbols are variables. For function symbols, calls using a function pointer can be rebound, while direct function calls cannot be rebound. The nosymbolic- attribute is the default and is provided for compatibility with previous versions of the operating system, but its use is not recommended.

If you are not using the run-time linker, avoid using the nosymbolic attribute because intra-module function calls will be made indirectly through a function descriptor using global-linkage code. Otherwise, the attribute of exported symbols has no effect for modules used with programs that do not use the run-time linker.

You can specify an explicit export attribute for symbols listed in an export file. Most symbols without an explicit attribute are exported with the default export attribute, as specified with the symbolic, nosymbolic, or nosymbolic- options.

The weak export attribute will mark the associated symbol's mapping type with L_WEAK in the loader section.

Imported symbols may only have the weak export attribute. If a symbol is imported from another module, all references to the symbol can be rebound. However, if a symbol is imported at a fixed address, all references are bound to this fixed address and cannot be rebound by the run-time linker. The system loader must resolve deferred imports. The run-time linker never resolves or rebinds references to deferred imports.

For exports of non-imported symbols, the following rules are used.

- If a symbol has the list attribute, it is listed in the loader section symbol table, but the L_EXPORT flag is not set in the symbol table entry. The run-time linker ignores such symbols.
- If a symbol was exported with an explicit attribute, the explicit attribute is used.
- If the symbol is a BSS symbol, it is exported with the nosymbolic attribute.
• Otherwise, the symbol is exported with the global attribute, as specified by the symbolic, nosymbolic, or nosymbolic- option. The default global attribute is nosymbolic-.

Address Maps
The ld command generates address maps, listing the layout of symbols in the output object file. If you use the map (or R) option, unresolved symbols and imported symbols are listed first, followed by the symbols in each section in address order. If you use the calls (or C) option, each symbol that is listed is followed by a list of references from that symbol to other symbols. If you use the xref (or X) option, each symbol that is listed is followed by a list of references to that symbol from other symbols. If you use the smap, scalls, or sxref option, the address map contains the same information as listed by the map, calls, or xref option, respectively, but symbols are listed in alphabetical order.

Internal symbols, with a storage class C_HIDEXT, are printed with the characters < and > (angle brackets) surrounding the symbol name. Names of external symbols, with a storage class C_EXT, are printed without the angle brackets, and those with a storage class of C_WEAKEXT, are printed with the characters { and } surrounding the symbol name.

Information listed about each symbol includes:
• An indication of whether the symbol is imported, exported, or the entry point. An * (asterisk) is used to mark the entry point, I is used to mark imported symbols, and E is used to mark exported symbols.
• Its address (except for imported symbols)
• Length and alignment (for CSECTs and BSS symbols)
• Storage-mapping class
• Symbol type
• Symbol number (used to differentiate between symbols of the same name)
• Symbol name
• Input file information

Storage-mapping classes and symbol types are defined in the /usr/include/syms.h file. In the address maps, only the last two characters are shown, except that storage-mapping class XMC_TC0 is shown as T0.

The input file information depends on the type of input file. For object files, source files names obtained from C_FILE symbols table entries are listed. If the object is from an archive file, the object file name is listed in the following format:
ArchiveFileName[ObjectName]

A shared object name is listed between { } (braces). If a shared object is defined by an import file, the name of the import file is listed before the shared object name.

Import symbols have a symbol type of ER, but they have associated file input information. Undefined symbols are also listed with a symbol type of ER, but all other columns, except the symbol number, are left blank.

The -T and -D flags (or pT or PD options) affect the addresses printed in these address maps. For machine-level debugging, it is helpful to choose address so that symbols are listed with the same addresses that they have at run time. For a 32-bit program that does not use privately loaded shared objects, you can choose the proper addresses by specifying the -bpT:0x10000000 and -bpD:0x20000000 options. These options are defined by default in the /etc/xlC.cfg or /etc/vac.cfg file.
Environment Variables

The following environment variables affect the execution of the *ld* command:

**LIBPATH**

If **LIBPATH** is defined, its value is used as the default library path information. Otherwise, the default library path information is `/usr/lib:/lib`. If no `-L` flags are specified and no `-bibpath` option is specified, the default library path information is written in the loader section of the output file. Regardless of any options specified, **LIBPATH** is not used when searching for libraries that are specified from the command line.

**TMPDIR**

If the output file already exists or it is on a remote file system, the *ld* command generates a temporary output file. The temporary output file is created in the directory specified by **TMPDIR**. If **TMPDIR** is not defined, the temporary output file is created in the `/tmp` directory if the output file is remote, or in the same directory as the existing output file.

**OBJECT_MODE**

If neither the `-b32` nor `-b64` option is used, the **OBJECT_MODE** environment variable is examined to determine the linking mode. If the value of **OBJECT_MODE** is 32 or 64, 32-bit or 64-bit mode is used, respectively. If the value is 32_64 or any other value, the linker prints an error message and exits with a non-zero return code. Otherwise, 32-bit mode is used.

Examples

1. To link several object files and produce an *a.out* file to run under the operating system, type:

   ```
   ld /usr/lib/crt0.o pgm.o subs1.o subs2.o -lc
   ```

   The `-lc` (lowercase letter L) links the *libc.a* library. A simpler way to accomplish this is to use the *cc* command (the compiler) to link the files as follows:

   ```
   cc pgm.o subs1.o subs2.o
   ```

2. To specify the name of the output file, type:

   ```
   cc -o pgm pgm.o subs1.o subs2.o
   ```

   This creates the output in the file *pgm*.

3. To relink *pgm* if only the object file *subs1.o* has changed, type:

   ```
   cc -o pgm subs1.o pgm
   ```

   The CSECTs that originally came from object files *pgm.o* and *subs2.o* are read from the file *pgm*. This technique can speed the linking process if a program consists of many input files, but only a few files change at a time.

4. To link with library subroutines, type:

   ```
   cc pgm.o subs1.o subs2.o mylib.a -ltools
   ```

   This links the object modules *pgm.o*, *subs1.o*, and *subs2.o*, the subroutines from the *mylib.a* archive, and the subroutine from the library specified by `-l` (lowercase letter L) flag. (This means the `/usr/lib/libtools.a` file).

5. To generate a shared object, type:

   ```
   ld -o shrsb.o subs1.o subs2.o -bE:shrsb.exp -bM:SRE -lc
   ```

   This links the object files *subs1.o*, *subs2.o*, and the subroutines from the library *libc.a* specified by `-lc` flag. It exports the symbols specified in the file *shrsb.exp* and stores the linked shared object in file *shrsb.o*. The `-bM:SRE` sets the shared object flag in the linked object file.

6. To link with the shared object *shrsb.o* generated previously, type:

   ```
   cc -o pgm pgm.o shrsb.o -L .'
   ```

   This links the object file *pgm.o* with the exported symbols of *shrsb.o*. The linked output is stored in the object file *pgm*. The `-L `.' adds the current directory to the library search path that the system loader uses to locate the *shrsb.o* shared object. At run time, this program is loaded only if it is run
from a directory containing an instance of the shrsub.o file or if the shrsub.o file is found in the /usr/lib
standard library directory. To allow the program to be run from anywhere, use the option -L `pwd`.
The list of directories searched by the system loader can be seen using the dump command.

7. To link a program using the libc.a library as a non-shared library, type:

```
cc -o pgm pgm.o -bnso -bI:/lib/syscalls.exp
```

This links pgm.o with the necessary support libraries and names the output file pgm. For the cc
command, the libc.a library is a necessary support library and is usually link-edited to the user’s
program as a shared library. In this example, the -bnso option directs the ld command to link with the
libc.a library as a non-shared library, and the -bI:/lib/syscalls.exp directs the ld command to import
the system call functions that are actually contained in the kernel or /usr/lib/boot/unix file. Whenever
linking with the -bnso option, any symbols that were both imported and exported (that is, passed
through) in a shared object must be explicitly imported, as is done by the -bI:/lib/syscalls.exp
option in this example.

Note: Any time that /usr/lib/libc.a is linked non-shared, the flag -bI:/lib/syscalls.exp must be used.
The application can also have to be linked again whenever an updated release of the operating
system is installed. Any application that is statically linked is not binary portable from any fix or
release level to any other fix or release level.

Files

<table>
<thead>
<tr>
<th>Path</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/bin/ld</td>
<td>Contains the ld command.</td>
</tr>
<tr>
<td>/usr/lib/lib*.a</td>
<td>Specifies libraries used for linking programs.</td>
</tr>
<tr>
<td>a.out</td>
<td>Specifies the default output file name.</td>
</tr>
</tbody>
</table>

Related Information

The `ar` command, `as` command, `nm` command, `dump` command, `rtl_enable` command, `strip` command.

The a.out file format.

The `glob` subroutine, `load` subroutine, `loadbind` subroutine, `loadquery` subroutine, `unload` subroutine.

How to Create a Shared Library in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Large Program Support Overview in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Shared Libraries and Lazy Loading in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Shared Library Overview in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

XCOFF (a.out) File Format in AIX 5L Version 5.2 Files Reference.

Idd Command

Purpose

Lists dynamic dependencies.
Syntax

```
ldd FileName
```

Description

The `ldd` command lists the path names of all dependencies. The command will report dependencies on only valid XCOFF files.

Parameters

```
FileName
```

Specifies the file whose dependencies will be listed.

Exit Status

```
0       The command completed successfully.
>0      An error occurred.
```

Examples

1. To display dependencies on `/usr/bin/dbx`, enter:

```
ldd /usr/bin/dbx
```

The output looks like the following:

```
/usr/bin/dbx needs:
    /usr/lib/libc.a(shr.o)
    /usr/lib/libdbx.a(shr.o)
    /unix
    /usr/lib/libcrypt.a(shr.o)
    /usr/lib/libpthdebug.a(shr.o)
```

Files

```
/usr/bin/ldd
```

Contains the `ldd` command.

Related Information

The `dump` command.

---

Idedit Command

Purpose

Modifies an XCOFF executable file header.

Syntax

```
Idedit [-b Option [-V]] File
```

Description

You can use the `Idedit` command to modify various fields in an XCOFF header or the auxiliary header of an executable file. The `Idedit` command makes it possible to mark or unmark an application as a 'large page data' program. The `Idedit` command also makes it possible to add or modify the values of MAXDATA and MAXSTACK without relinking.
The format of the -b flag is similar to the format used by the link editor, the ld command. The -b flag can be used multiple times on the command line.

If no flags are specified, the ldedit command displays a usage message using the standard error output.

**Flags**

-b Option  
Modifies an executable as specified by Option. The possible values for Option are:

- lndata  
Marks a file as a 'large page data' executable.

- nolndata  
Unmarks a file as a 'large page data' executable.

- maxdata:value  
Sets the MAXDATA value. value is an octal number when it starts with 0, a hexadecimal number when it starts with 0x, and a decimal number in all other cases.

- maxdata:valueDSA  
Sets the MAXDATA value and the DSA bit. value is an octal number when it starts with 0, a hexadecimal number when it starts with 0x, and a decimal number in all other cases.

- maxstack:value  
Sets the MAXSTACK value. value is an octal number when it starts with 0, a hexadecimal number when it starts with 0x, and a decimal number in all other cases.

-V  
Prints the version of the ldedit command on the standard error output.

**Related Information**

The ld command, dump command.

---

**learn Command**

**Purpose**

Provides computer-aided instruction for using files, editors, macros, and other features.

**Syntax**

```bash
learn[-Directory] [Subject LessonNumber]
```

**Description**

The learn command provides computer-aided instruction for using files, editors, macros, and other features. The first time you invoke the command, the system provides introductory information about the learn command. Otherwise, the learn command begins at the point where you left the last learn command session.

You can bypass the default action of the learn command by specifying the Subject parameter. The learn command starts with the first lesson of the subject you specify. You can specify any of the following subjects:

- Files
- Editors
- More files
- Macros
- EQN (the enquiry character)
- C (the language)
Note: You can only run the EQN lesson on a hardcopy terminal that is capable of 1/2 line motion. The /usr/share/lib/learn/eqn/init file contains a detailed list of the supported terminals.

When you enter the learn command, the system searches the /usr/share/lib/learn directory for the appropriate lesson file. Use the -Directory flag to identify a different search directory.

Subcommands
- The bye subcommand terminates a learn command session.
- The where subcommand tells you of your progress; the where m subcommand provides more detail.
- The again subcommand re-displays the text of the lesson.
- The again LessonNumber subcommand lets you review the lesson.
- The hint subcommand prints the last part of the lesson script used to evaluate a response; the hint m subcommand prints the entire lesson script.

Parameters
- -Directory Allows you to specify a different search directory. By default, the system searches for lesson files in the /usr/share/lib/learn directory.
- LessonNumber Identifies the number of the lesson.
- Subject Specifies the subject you want instruction on.

Examples
To take the online lesson about files, enter:
learn files

The system starts the learn program and displays instructions for how to use the program.

Files
- /usr/share/lib/learn Contains the file tree for all dependent directories and files.
- /tmp/pl* Contains the practice directories.
- $HOME/.learnrc Contains the startup information.

Related Information
The ex command.


leave Command

Purpose
Reminds you when you have to leave.

Syntax
leave [ [+][-][hmn] ]
Description

The leave command waits until the specified time and then reminds you that you have to leave. You are reminded at 5 minutes and at 1 minute before the actual time, again at that time, and at every minute thereafter. When you log off, the leave command exits just before it would have displayed the next message.

If you do not specify a time, the leave command prompts with when do you have to leave? A reply of newline causes the leave command to exit; otherwise, the reply is assumed to be a time. This form is suitable for inclusion in a .login or .profile file.

The leave command ignores interrupt, quit, and terminate operations. To clear the leave command, you should either log off or use the kill-9 command and provide the process ID.

Flags

+ Specify to set the alarm to go off in the indicated number of hours and minutes from the current time.

hhmm Specifies a time of day in hours and minutes (based on a 12- or 24-hour clock) or, if preceded by the +, a set number of hours and minutes from the current time for the alarm to go off. All times are converted to a 12-hour clock and assumed to relate to the next 12 hours.

Examples

To remind yourself to leave at 3:45, enter:

`leave 345`

Related Information

The calendar command.

lecstat Command

Purpose

Displays operational information about an Asynchronous Transfer Mode network protocol (ATM) Local Area Network (LAN) Emulation Client.

Syntax

`lecstat [-a -c -q -r -s -t -v] Device_Name`

Description

This command displays ATM LAN Emulation Client (LEC) operational information gathered by a specified LEC device. If a LEC device name is not entered, statistics for all available LEC’s appear. Select a flag to narrow down your search results. You can display specific categories of information such as Configuration, LE_ARP Cache Entries, Virtual Connections, and Statistics, or you can choose to display all of the information categories.

You can also toggle debug tracing on or off and reset statistics counters.

Parameters

`Device_Name` The name of the LE Client device, for example, ent1.
Flags

-\texttt{a} Requests all the LE Client information. This flag does not reset statistics counters or toggle trace.
-\texttt{c} Requests the configuration.
-\texttt{q} Requests the LE_ARP cache.
-\texttt{r} Resets the statistics counters after reading.
-\texttt{s} Requests the statistics counters.
-\texttt{t} Toggles full debug trace on or off.
-\texttt{v} Requests the list of virtual connections.

The following information appears for all valid calls and contains the following fields:

**Device Type**
Displays a description of the LAN Emulation Client (example: Ethernet or Token Ring)

**LAN MAC Address**
Displays the LAN Emulation Client’s 6-byte Ethernet or Token Ring MAC address.

**ATM Address**
Displays the LAN Emulation Client’s 20-byte Asynchronous Transfer Mode (ATM) address.

**Elapsed Time**
Displays the real time period which has elapsed since statistics were last reset.

**Driver Flags**
The current LAN Emulation Client (LEC) device driver NDD status flags. Example status flags:

- Broadcast: Allowing broadcast packets.
- Dead: Requires re-open.
- Debug: Internal debug tracing enabled.
- Limbo: Attempting ELAN recovery.
- Running: Fully operational on the ELAN.
- Up: Device has been opened.

**Configuration Information**
Selected with the -\texttt{a} or -\texttt{c} flags. Displays the network administrator’s pre-configured attributes, as well as the current ELAN configuration values as defined by the LANE Servers.

**Lane LE_ARP Table Entries**
Selected with the -\texttt{a} or -\texttt{q} flags. Displays the current LE Client ARP cache. Included are the type of entry, it’s state, the remote LAN MAC address or route descriptor, the remote ATM address and some descriptive values.

**Example Types**

- BUS-PP: Broadcast and Unknown Server (point-to-point).
- BUS-MP: Broadcast and Unknown Server (multi-point).
- Data: Data (point-to-point).
- LES-PP: LE Server (point-to-point).
- LES-MP: LE Server (multi-point).

**Example States**

- Arping: Attempting to locate remote client/server via LE_ARP.
- Connected: Fully connected to the remote client/server.
- Flushing: Flushing the data path to the client/server.
- Known: Remote address is known but no connection yet.
- Unknown: Remote address in unknown and not able to LE_ARP yet.

**Lane Servers and Statistics**
Selected with the -\texttt{a} or -\texttt{s} flags. Displays the current Transmit, Receive, and General statistics for this LE Client, as well as the ATM addresses of the current and available LANE Servers.

**Lane connections**
Selected with the -\texttt{a} or -\texttt{v} flags. Displays the current list of virtual connections in use by this LE
Client. Included are virtual path and channel values, remote ATM address, and some descriptive values such as whether this connection was started by the remote, whether it is a duplicate connection, or whether the remote station is proxied by another LE Client.

**Exit Status**

If you specify an invalid *Device_Name*, this command produces error messages stating that it could not connect to the device. Examples of an invalid device error message might be:

LECSTAT: No LANE device configured.
LECSTAT: Device is not a LANE device.
LECSTAT: Device is not available.

**Related Information**

The `atmstat` command, `entstat` command, `mpcstat` command, and `tokstat` command.

---

**lex Command**

**Purpose**

Generates a C or C++ language program that matches patterns for simple lexical analysis of an input stream.

**Syntax**

```
lex [-C] [-t] [-v] [-n] [File ..]
```

**Description**

The `lex` command reads *File* or standard input, generates a C language program, and writes it to a file named `lex.yy.c`. This file, `lex.yy.c`, is a compilable C language program. A C++ compiler also can compile the output of the `lex` command. The `-C` flag renames the output file to `lex.yy.C` for the C++ compiler.

The C++ program generated by the `lex` command can use either STDIO or IOSTREAMS. If the `cpp` define `_CPP_IOSTREAMS` is true during a C++ compilation, the program uses IOSTREAMS for all I/O. Otherwise, STDIO is used.

The `lex` command uses rules and actions contained in *File* to generate a program, `lex.yy.c`, which can be compiled with the `cc` command. The compiled `lex.yy.c` can then receive input, break the input into the logical pieces defined by the rules in *File*, and run program fragments contained in the actions in *File*.

The generated program is a C language function called `yylex`. The `lex` command stores the `yylex` function in a file named `lex.yy.c`. You can use the `yylex` function alone to recognize simple one-word input, or you can use it with other C language programs to perform more difficult input analysis functions. For example, you can use the `lex` command to generate a program that simplifies an input stream before sending it to a parser program generated by the `yacc` command.

The `yylex` function analyzes the input stream using a program structure called a finite state machine. This structure allows the program to exist in only one state (or condition) at a time. There is a finite number of states allowed. The rules in *File* determine how the program moves from one state to another.

If you do not specify a *File*, the `lex` command reads standard input. It treats multiple files as a single file.

**Note:** Since the `lex` command uses fixed names for intermediate and output files, you can have only one program generated by `lex` in a given directory.
lex Specification File

The input file can contain three sections: definitions, rules, and user subroutines. Each section must be separated from the others by a line containing only the delimiter, `%%` (double percent signs). The format is:

```
definitions
%%
rules
%%
user subroutines
```

The purpose and format of each are described in the following sections.

Definitions

If you want to use variables in your rules, you must define them in this section. The variables make up the left column, and their definitions make up the right column. For example, if you want to define D as a numerical digit, you would write the following:

```
D [0-9]
```

You can use a defined variable in the rules section by enclosing the variable name in `{}` (braces), for example:

```
{D}
```

Lines in the definitions section beginning with a blank or enclosed in `%%` delimiter lines are copied to the `lex.yy.c` file. You can use this construct to declare C language variables to be used in the `lex` actions or to include header files, for example:

```
#include <math.h>
int count;
```

Such lines can also appear at the beginning of the rules section, immediately after the first `%%` delimiter, but they should not be used anywhere else in the rules section. If the line is in the definitions section of `File`, the `lex` command copies it to the external declarations section of the `lex.yy.c` file. If the line appears in the rules section, before the first rule, the `lex` command copies it to the local declaration section of the `yylex` subroutine in `lex.yy.c`. Such lines should not occur after the first rule.

The type of the `lex` external, `yytext`, can be set to either a null-terminated character array (default) or a pointer to a null-terminated character string by specifying one of the following in the definitions section:

```
%array (default)
%pointer
```

In the definitions section, you can set table sizes for the resulting finite state machine. The default sizes are large enough for small programs. You may want to set larger sizes for more complex programs.

```
%an Number of transitions is n (default 5000)
%en Number of parse tree nodes is n (default 2000)
%hn Number of multibyte character output slots (default is 0)
%kn Number of packed character classes (default 1000)
%mhn Number of multibyte “character class” character output slots (default is 0)
%nn Number of states is n (default 2500)
%on Number of output slots (default 5000, minimum 257)
%pn Number of positions is n (default 5000)
%vp Percentage of slots vacant in the hash tables controlled by %h and %m (default 20, range 0 <= P < 100)
%zn Number of multibyte character class output slots (default 0)
```
If multibyte characters appear in extended regular expression strings, you may need to reset the output array size with the %o argument (possibly to array sizes in the range 10,000 to 20,000). This reset reflects the much larger number of characters relative to the number of single-byte characters.

If multibyte characters appear in extended regular expressions, you must set the multibyte hash table sizes with the %h and %m arguments to sizes greater than the total number of multibyte characters contained in the lex file.

If no multibyte characters appear in extended regular expressions but you want ’.’ to match multibyte characters, you must set %z greater than zero. Similarly, for inverse character classes (for example, [^abc]) to match multibyte characters, you must set both %h and %m greater than zero.

When using multibyte characters, the lex.yy.c file must be compiled with the -qmbcs compiler option.

**Rules**

Once you have defined your terms, you can write the rules section. It contains strings and expressions to be matched by the yylex subroutine, and C commands to execute when a match is made. This section is required, and it must be preceded by the delimiter %% (double percent signs), whether or not you have a definitions section. The lex command does not recognize your rules without this delimiter.

In this section, the left column contains the pattern in the form of an extended regular expression, which will be recognized in an input file to the yylex subroutine. The right column contains the C program fragment executed when that pattern is recognized, called an action.

When the lexical analyzer finds a match for the extended regular expression, the lexical analyzer executes the action associated with that extended regular expression.

Patterns can include extended characters. If multibyte locales are installed on your system, patterns can also include multibyte characters that are part of the installed code set.

The columns are separated by a tab or blanks. For example, if you want to search files for the keyword KEY, you can write the following:

```c
(KEY) printf ("found KEY");
```

If you include this rule in File, the yylex lexical analyzer matches the pattern KEY and runs the printf subroutine.

Each pattern can have a corresponding action, that is, a C command to execute when the pattern is matched. Each statement must end with a ; (semicolon). If you use more than one statement in an action, you must enclose all of them in { } (braces). A second delimiter, %%, must follow the rules section if you have a user subroutine section. Without a specified action for a pattern match, the lexical analyzer copies the input pattern to the output without changing it.

When the yylex lexical analyzer matches a string in the input stream, it copies the matched string to an external character array (or a pointer to a character string), yytext, before it executes any commands in the rules section. Similarly, the external int, yyleng, is set to the length of the matched string in bytes (therefore, multibyte characters will have a size greater than 1).

For information on how to form extended regular expressions, see [Extended Regular Expressions in the lex Command] in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

**User Subroutines**
The lex library defines the following subroutines as macros that you can use in the rules section of the lex specification file:
The \texttt{winput, wunput,} and \texttt{woutput} macros are defined to use the \texttt{yyinput, yywunput,} and \texttt{yywoutput} subroutines coded in the \texttt{lex.yy.c} file. For compatibility, these \texttt{yy} subroutines subsequently use the \texttt{input, unput,} and \texttt{output} subroutines to read, replace, and write the necessary number of bytes in a complete multibyte character.

You can override these macros by writing your own code for these routines in the user subroutines section. But if you write your own, you must undefine these macros in the definition section as follows:

\begin{verbatim}
%{
  #undef input
  #undef unput
  #undef output
  #undef winput
  #undef wunput
  #undef woutput
  #undef yysetlocale
%
\end{verbatim}

There is no \texttt{main} subroutine in \texttt{lex.yy.c}, because the \texttt{lex} library contains the \texttt{main} subroutine that calls the \texttt{yylex} lexical analyzer, as well as the \texttt{yywrap} subroutine called by \texttt{yylex( )} at the end of \texttt{File}. Therefore, if you do not include \texttt{main( ), yywrap( ),} or both in the user subroutines section, when you compile \texttt{lex.yy.c,} you must enter \texttt{cclex.yy.c-ll,} where \texttt{ll} calls the \texttt{lex} library.

External names generated by the \texttt{lex} command all begin with the preface \texttt{yy}, as in \texttt{yyin, yyout, yylex,} and \texttt{yytext}.

**Finite State Machine**

The default skeleton for the finite state machine is defined in \texttt{/usr/ccs/lib/lex/ncform}. The user can use a personally configured finite state machine by setting an environment variable \texttt{LEXER=PATH}. The \texttt{PATH} variable designates the user-defined finite state machine path and file name. The \texttt{lex} command checks the environment for this variable and, if it is set, uses the supplied path.

**Putting Blanks in an Expression**

Normally, blanks or tabs end a rule and therefore, the expression that defines a rule. However, you can enclose the blanks or tab characters in " " (quotation marks) to include them in the expression. Use quotes around all blanks in expressions that are not already within sets of [ ] (brackets).

**Other Special Characters**

The \texttt{lex} program recognizes many of the normal C language special characters. These character sequences are:

<table>
<thead>
<tr>
<th>Sequence</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>\l a</td>
<td>Alert</td>
</tr>
<tr>
<td>\l b</td>
<td>Backspace</td>
</tr>
<tr>
<td>\l f</td>
<td>Form Feed</td>
</tr>
<tr>
<td>\l n</td>
<td>Newline (Do not use the actual new-line character in an expression.)</td>
</tr>
<tr>
<td>\l r</td>
<td>Return</td>
</tr>
<tr>
<td>\l t</td>
<td>Tab</td>
</tr>
<tr>
<td>\l v</td>
<td>Vertical Tab</td>
</tr>
</tbody>
</table>
When both expression. The expression occurs.

Sequence Meaning
\ Backslash
\digits The character with encoding represented by the one-, two-, or three-digit octal integer specified by digits.
\xdigits The character with encoding represented by the sequence of hexadecimal characters specified by digits.
\c Where c is none of the characters listed above, represents the character c unchanged.

Note: Do not use \0 or \x0 in lex rules.

When using these special characters in an expression, you do not need to enclose them in quotes. Every character, except these special characters and the operator symbols described in [Extended Regular Expressions in the lex Command] in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs, is always a text character.

Matching Rules
When more than one expression can match the current input, the lex command chooses the longest match first. When several rules match the same number of characters, the lex command chooses the rule that occurs first. For example, if the rules

```
integer    keyword action...;
[a-z]+     identifier action...;
```

are given in that order, and integers is the input word, lex matches the input as an identifier, because [a-z]+ matches eight characters while integer matches only seven. However, if the input is integer, both rules match seven characters. lex selects the keyword rule because it occurs first. A shorter input, such as int, does not match the expression integer, and so lex selects the identifier rule.

Matching a String Using Wildcard Characters
Because lex chooses the longest match first, do not use rules containing expressions like .*. For example:

```
'.*'    
```

might seem like a good way to recognize a string in single quotes. However, the lexical analyzer reads far ahead, looking for a distant single quote to complete the long match. If a lexical analyzer with such a rule gets the following input:

```
'first' quoted string here, 'second' here
```

it matches:

```
'first' quoted string here, 'second'
```

To find the smaller strings, first and second, use the following rule:

```
'[^'\n]*'
```

This rule stops after 'first'.

Errors of this type are not far reaching, because the . (period) operator does not match a new-line character. Therefore, expressions like .* (period asterisk) stop on the current line. Do not try to defeat this with expressions like [\.*\n]. The lexical analyzer tries to read the entire input file and an internal buffer overflow occurs.

Finding Strings within Strings
The lex program partitions the input stream and does not search for all possible matches of each expression. Each character is accounted for once and only once. For example, to count occurrences of both she and he in an input text, try the following rules:
where the last two rules ignore everything besides he and she. However, because she includes he, lex does not recognize the instances of he that are included in she.

To override this choice, use the action REJECT. This directive tells lex to go to the next rule. lex then adjusts the position of the input pointer to where it was before the first rule was executed and executes the second choice rule. For example, to count the included instances of he, use the following rules:

```
she      {s++;REJECT;}
he       {h++;REJECT;}
```

After counting the occurrences of she, lex rejects the input stream and then counts the occurrences of he. Because in this case she includes he but not vice versa, you can omit the REJECT action on he. In other cases, it may be difficult to determine which input characters are in both classes.

In general, REJECT is useful whenever the purpose of lex is not to partition the input stream but to detect all examples of some items in the input, and the instances of these items may overlap or include each other.

**Flags**

- `-C` Produces the lex.yy.C file instead of lex.yy.c for use with a C++ compiler. To get the I/O Stream Library, use the macro, _CPP_ISTREAMS, as well.
- `-n` Suppresses the statistics summary. When you set your own table sizes for the finite state machine, the lex command automatically produces this summary if you do not select this flag.
- `-t` Writes lex.yy.c to standard output instead of to a file.
- `-v` Provides a one-line summary of the generated finite-state-machine statistics.

**Exit Status**

This command returns the following exit values:

0 Successful completion.

>0 An error occurred.

**Examples**

1. To draw lex instructions from the file lexcommands and place the output in lex.yy.c, use the following command:

```
lex lexcommands
```

2. To create a lex program that converts uppercase to lowercase, removes blanks at the end of a line, and replaces multiple blanks by single blanks, including the following in a lex command file:

```
[A-Z]  putchar(yytext[0]+ 'a'-'A');
[ ]+$  putchar(' ');
```

**Files**

- `/usr/ccs/lib/libl.a` Contains the run-time library.
- `/usr/ccs/lib/lex/nform` Defines a finite state machine.
Related Information

The `yacc` command.

Creating an Input Language with the lex and yacc Commands in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Using the lex Program with the yacc Program in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Example Program for the lex and yacc Programs in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.


libxrx Command

Purpose
RX Netscape Navigator Plug-in.

Description
The RX Plug-in may be used with Netscape Navigator (3.0 or later) to interpret documents in the RX MIME type format and start remote applications.

The RX Plug-in reads an RX document, from which it gets the list of services the application wants to use. Based on this information, the RX Plug-in sets the various requested services, including creating authorization keys if your X server supports the SECURITY extension. It then passes the relevant data, such as the X display name, to the application through an HTTP GET request of the associated CGI script. The Web server then executes the CGI script to start the application. The client runs on the web server host connected to your X server. In addition when the RX document is used within the EMBED tag (a Netscape extension to HTML), the RX Plug-in uses the XC-APPGROUP extension, if it is supported by your X server, to cause the remote application to be embedded within the browser page from which it was launched.

Installation
To install the RX Plug-in so that Netscape Navigator can use it, find the file named `libxrx.so` in `/usr/lib/X11` and copy it to either the system Netscape plugin directory or `$HOME/netscape/plugins`. Do not install it as a symbolic link, this would confuse Netscape.

If you have configured Netscape Navigator to use the RX helper program (`xrx`), you must reconfigure it. Generally you simply need to remove or comment out the line you may have previously added in your mailcap file to use the RX helper program. Otherwise the plug-in will not be enabled. (The usual comment character for mailcap is `#`.) If you are already running Netscape Navigator, you need to exit and restart it after copying the plug-in library so the new plug-in will be found. Once this is done you can check that Navigator has successfully loaded the plug-in by checking the About Plug-ins page from the Help menu. This should show something like:

```
RX Plug-in

File name: /usr/local/lib/netscape/plugins/libxrx.so.6.3
X Remote Activation Plug-in
Mime Type Description Suffixes Enabled
application/x-rx X Remote Activation xrx Yes
```
Once correctly configured, Netscape Navigator will activate the RX Plug-in whenever you retrieve any document of the MIME type application/x-rx.

**Resources**

The RX Plug-in looks for resources associated with the widget **netscape.Navigator** (class **Netscape.TopLevelShell**) and understands the following resource names and classes:

**xrxHasFirewallProxy (class XrxHasFirewallProxy)**

Specifies whether an X server firewall proxy is running and should be used. Default is False.

**xrxInternalWebServers (class XrxInternalWebServers)**

The web servers for which the X server firewall proxy should not be used (only relevant when **xrxHasFirewallProxy** is “True”). Its value is a comma separated list of mask/value pairs to be used to filter internal web servers, based on their address. The mask part specifies which segments of the address are to be considered and the value part specifies what the result should match. For instance the following list:

255.255.255.0/198.112.45.0,
255.255.255.0/198.112.46.0

matches the address sets: 198.112.45.* and 198.112.46.*. More precisely, the test is (address and mask) == value.

**xrxFastWebServers (class XrxFastWebServers)**

The web servers for which LBX should not be used. The resource value is a list of address mask/value pairs, as previously described.

**xrxTrustedWebServers (class XrxTrustedWebServers)**

The web servers from which remote applications should be run as trusted clients. The default is to run remote applications as untrusted clients. The resource value is a list of address mask/value pairs, as previously described.

**Environment**

If the RX document requests X-UI-LBX service and the default X server does not advertise the LBX extension, the RX Plug-in will look for the environment variable **XREALDISPLAY** to get a second address for your X server and look for the LBX extension there. When running your browser through lbxproxy you will need to set **XREALDISPLAY** to the actual address of your server if you wish remote applications to be able to use LBX across the Internet.

If the RX document requests XPRINT service, the RX Plug-in will look for the variables **XPRINTER**, **PDPRINTER**, **LPDEST**, **PRINTER**, and **XPERVERLIST** to get the printer name and X Print server address to use. Note that although this set of variables allows to specify more than one server and printer, only the first pair will be used.

Finally, if you are using a firewall proxy, RX Plug-in will look for **PROXY_MANAGER** to get the address of your proxy manager (see **proxymngr**). When not specified it will use :6500 as the default.

**Related Information**

The **xrx** command, **lbxproxy** command, and the **proxymngr** command.

**line Command**

**Purpose**

Reads one line from the standard input.
Syntax
line

Description
The line command copies one line from standard input and writes it to standard output. It returns an exit value of 1 on an end-of-file and always writes at least a new-line character. Use this command within a shell command file to read from the workstation.

Exit Status
This command returns the following exit values:

0     Successful completion.
>0    End-of-file occurred on input.

Examples
To read a line from the keyboard and append it to a file, create a script file as follows:

```
echo 'Enter comments for the log:'
echo ': \c'
line >> log
```

This shell procedure displays the message:

Enter comments for the log:

and then reads a line of text from the workstation keyboard and adds it to the end of the log. The echo ': \c' command displays a colon prompt. See the echo command for information about the \c escape sequence.

Related Information
The [echo command](#), [ksh command](#), [sh command](#).

The [read subroutine](#).

The [Input and Output Handling Programmer's Overview](#) in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs describes the files, commands, and subroutines used for low-level, stream, terminal, and asynchronous I/O interfaces.

### link Command

**Purpose**
Perform a link subroutine.

**Syntax**
```
link File1 File2
```

**Description**
The link command performs the link subroutine on a specified file. The link command does not issue error messages when the associated subroutine is unsuccessful; you must check the exit value to determine if the command completed normally. It returns a value of 0 if it succeeds, a value of 1 if too few or too many parameters are specified, and a value of 2 if its system call is unsuccessful.
Attention: The link command allows a user with root user authority to deal with unusual problems, such as moving an entire directory to a different part of the directory tree. It also permits you to create directories that cannot be reached or escaped from. Be careful to preserve the directory structure by observing the following rules:

- Be certain every directory has a . (dot) link to itself.
- Be certain every directory has a .. (dot dot) link to its parent directory.
- Be certain every directory has no more than one link to itself or its parent directory.
- Be certain every directory is accessible from the root of its file system.

Note: If the . (dot) entry has been destroyed and the fsck command is unable to repair it (a rare occurrence), you can use the link command to restore the . (dot) entry of the damaged directory. Use the link Dir Dir/. command where the Dir parameter is the name of the damaged directory. However, use this only as a last resort when the directory is destroyed and the fsck command is unable to fix it.

Although the linked files and directories can be removed by the unlink command, it’s safer to use the rm or rmdir command.

Examples
To create an additional link for an existing file1, enter:
link file1 file2

Files
/usr/sbin/link Contains the link command.

Related Information
The fsck command, unlink command, ln command.

The link subroutine, unlink subroutine.


The Files Overview in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices provides information on working with files.


Lint Command

Purpose
Checks C and C++ language programs for potential problems.

Syntax
lint [-a | -b | -c | -C | -h | -Key | -ln | -oLibrary | -qDBCS | -p | -t | -u | -v | -w Class
| Class ... | ] [-x | -MA | -Nd | -N | Number | -N | Number | -N | Number | -N | Number | -Directory | ]
-DName [=Definition ] | [-UName ] File ...
Description

The lint command checks C and C++ language source code for coding and syntax errors and for inefficient or non-portable code. You can use this program to:

- Identify source code and library incompatibility.
- Enforce type-checking rules more strictly than does the compiler.
- Identify potential problems with variables.
- Identify potential problems with functions.
- Identify problems with flow control.
- Identify legal constructions that may produce errors or be inefficient.
- Identify unused variable and function declarations.
- Identify possibly non-portable code.

Note: Checking of C++ language files by the lint command requires the presence of the C Set++ Compiler package.

The inter-file usage of functions is checked to find functions that return values in some instances and not in others, functions called with varying numbers or types of arguments, and functions whose values are not used or whose values are used but not returned.

The lint command interprets file name extensions as follows:

- File names ending in .c are C language source files.
- File names ending in .C are C++ language source files.
- File names ending in .ln are non-ASCII files that the lint command produces when either the -c or the -o flag is used.

The lint command warns you about files with other suffixes and ignores them.

The lint command takes all the .c, .C, and .ln files and the libraries specified by -l flags and processes them in the order that they appear on the command line. By default, it adds the standard llib-ic.ln lint library to the end of the list of files. However, when you select the -p flag, the lint command uses the llib-port.ln portable library. By default, the second pass of lint checks this list of files for mutual compatibility; however, if you specify the -c flag, the .ln and llib-Lx.ln files are ignored.

The -c and -o flags allow for incremental use of the lint command on a set of C and C++ language source files. Generally, use the lint command once for each source file with the -c flag. Each of these runs produces a .ln file that corresponds to the .c file and writes all messages concerning that source file. After you have run all source files separately through the lint command, run it once more, without the -c flag, listing all the .ln files with the needed -i flags. This writes all inter-file inconsistencies. This procedure works well with the make command, allowing it to run the lint command on only those source files modified since the last time that set of source files was checked.

The lint and LINT preprocessor symbols are defined to allow certain questionable code to be altered or removed for the lint command. Therefore, the lint and LINT symbols should be thought of as a reserved word for all code that is planned to be checked by lint.

The following comments in a C and C++ language source program change the way the lint command operates when checking the source program:

/*NOTREACHED*/
Suppresses comments about unreachable code.
The **lint** command warning messages give file name and line number. As each file goes through the first pass, warnings for each file and each line number are reported.

If you have not specified the -c flag, the **lint** command collects information gathered from all input files and checks it for consistency. At this point, if it is not clear whether a message stems from a given source file or from one of its included files, the **lint** command displays the source file name followed by a question mark.

ANSI programs that include many standard header files may wish to set the -wD flag to reduce the quantity of warnings about prototypes not used, and the -n flag to disable checking against the ANSI standard library. For non-ANSI programs, it is advisable to specify the -wk flag to reduce the amount of warnings concerning the absence of function prototypes.

### Flags

-a

Suppresses messages about assignments of long values to variables that are not long.

-b

Suppresses messages about unreachable break statements.

-c

Causes the **lint** command to produce an .In file for every .c file on the command line. These .In files are the product of the first pass of the **lint** command only and are not checked for inter-function compatibility.

-C

Specifies to use the C++ libraries (in the /usr/lpp/xlC/lib directory).

-h

Does not try to detect bugs, improper style, or reduce waste.

-IKey

Includes the additional lib-lKey.In lint library. You can include a lint version of the lib-lm.In math library by specifying -lm on the command line or lib-Ldos.In library by specifying the -ldos flag on the command line. Use this flag to include local lint libraries when checking files that are part of a project having a large number of files. This flag does not prevent the **lint** command from using the lib-lc.In library. The lint library must be in the /usr/ccs/lib directory.

-n

Suppresses the check for compatibility with either the standard or the portable lint libraries. This applies for both the ANSI and extended mode libraries.
The lint command is used to detect possible errors in C source code. The command is run as follows:

```
```

- **-o Library**: Causes the lint command to create the `llib-ILibrary.ln` lint library. The `-c` flag nullifies any use of the `-o` flag. The lint library produced is the input that is given to the second pass of the lint command. The `-o` flag simply causes this file to be saved in the named lint library. To produce a `llib-ILibrary.ln` without extraneous messages, use the `-x` flag. The `-v` flag is useful if the source files for the lint library are just external interfaces (for example, the way the `llib-ic` file is written). These flag settings are also available through the use of lint command comment lines.

- **-p**: Checks for portability to other C language dialects.
- **-t**: Checks for problematic assignments when porting from 32 to 64 bit. Only the following cases are checked:
  - all shift / mask operations are flagged because some operations that work well in 32-bit may cause problems in 64-bit.
  - warnings are given for the following type of assignments.
    ```
    int = long
    int = ptr
    ```

- **-u**: Suppresses messages about functions and external variables that are either used and not defined or defined and not used. Use this flag to run the lint command on a subset of files of a larger program.

- **-v**: Suppresses messages about function parameters that are not used.

- **-w Class [Class...]**: Controls the reporting of warning classes. All warning classes are active by default, but can be individually deactivated by including the appropriate option as part of the `Class` argument. The individual options are listed as:
  - a: Non-ANSI features.
  - c: Comparisons with unsigned values.
  - d: Declaration consistency.
  - h: Heuristic complaints.
  - k: Use for K+R type source code.
  - l: Assignment of long values to variables that are not long.
  - n: Null-effect code.
  - o: Unknown order of evaluation.
  - p: Various portability concerns.
  - r: Return statement consistency.
  - s: Storage capacity checks.
  - u: Proper capacity checks.
  - A: Deactivate all warnings.
  - C: Constants occurring in conditionals.
  - D: External declarations are never used.
  - O: Obsolescent features.
  - P: Function prototype presence.
  - R: Detection of unreachable code.

- **-x**: Suppresses messages about variables that have external declarations but are never used.

- **-MA**: Enforces the ANSI C language standard rules. The default mode is equal to the extended C mode. The ANSI mode prepends the standard ANSI library function prototypes in place of the default extended mode C lint library. The ANSI mode enforces a stricter inter-file object reference and provides definition linkage checks.

- **-Nd Number**: Changes the dimension table size to `Number`. The default value of `Number` is 2000.

- **-NI Number**: Changes the number of type nodes to `Number`. The default value of `Number` is 8000.

- **-Nn Number**: Increases the size of the symbol table to `Number`. The default value of `Number` is 1500.
-NtNumber Changes the number of tree nodes to Number. The default value of Number is 1000.

In addition, the lint command recognizes the following flags of the cpp command (macro preprocessor):

-Directory Adds the Directory to the list of directories in which the lint command searches for the #include files.

-DName[=Definition] Defines the Name, as if by the #define file. The default of the Definition is the value of 1.

-qDBCS Sets multibyte mode specified by the current locale.

-UName Removes any initial definition of the Name, where the Name is a reserved symbol that is predefined by the particular preprocessor.

Exit Status
This command returns the following exit values:

0 Successful completion.
>0 An error occurred.

Examples
1. To check a C program for errors, enter:
   lint command.c

2. To suppress some of the messages, enter:
   lint -v -x program.c

   This checks program.c, but does not display error messages about unused function parameters (-v) or unused externals (-x).

3. To check the program against an additional lint library, enter:
   lint -lsubs program.c

   This checks program.c against both the /usr/ccts/lib/llib-lc.ln standard lint library and
   /usr/lib/llib-lsubs.ln lint library.

4. To check against the portable library and an additional library, enter:
   lint -lsubs -p program.c

   This checks program.c against both the /usr/ccts/lib/llib-port.ln portable lint library and
   /usr/lib/llib-lsubs.ln lint library.

5. To check against a nonstandard library only, enter:
   lint -lsubs -n program.c

   This checks program.c against only /usr/lib/llib-lsubs.ln.

Files
/usr/lib/lint[12] Programs
/usr/ccts/lib/llib-lansi Declarations for standard ANSI functions (source)
/usr/ccts/lib/llib-lansi.ln Declarations for standard ANSI functions (binary format)
/usr/ccts/lib/llib-lc Declarations for standard functions (source)
/usr/ccs/lib/llib-lc.ln Declarations for standard functions (binary format)
/usr/ccts/lib/llib-lcrses Declarations for curses functions (source)
/usr/ccs/lib/llib-lcrses.ln Declarations for curses functions (binary format)
/usr/ccts/lib/llib-lm Declarations for standard math functions (source)
/usr/ccs/lib/llib-lm.ln Declarations for standard math functions (binary format)
Declarations for portable functions (source)
Declarations for portable functions (binary format)
Directory containing C++ libraries
Temporary files

**Related Information**
The `cpp` command, `make` command.

---

### listdgrp Command

**Purpose**
Displays devices of a device class.

**Syntax**
```bash
listdgrp DeviceClass
```

**Description**
Lists information about devices where the `DeviceClass` parameter refers to a object class of Customized Devices in the Device Configuration database.

**Parameters**

- **DeviceClass**
  Specifies the device class whose members will be displayed.

**Exit Status**

- **0** The command completed successfully.
- **>0** An error occurred.

**Examples**

1. To list the devices in the `adapter` class, enter:
   ```bash
   listdgrp adapter
   ```
   The output looks similar to the following:
   ```
   a0
   sal
   siokma0
   fda0
   scsi0
   scsi1
   b10
   sioka0
   siota0
   ```

**Files**

- `/usr/bin/listdgrp`
  Contains the System V `listdgrp` command.

**Related Information**
The `chdev` command, `lsattr` command, `lsdev` command, `mkdev` command, `rmdev` command.
listvgbackup Command

Purpose
Lists or restores the contents of a volume group backup on a specified media.

Syntax
listvgbackup [\[-b\] blocks] [\[-f\] device] [\[-a\] | \[-c\] | \[-d\] | \[-r\] | \[-S\] | \[-d\] path] [\[-B\] | \[-D\] | \[-L\] | \[-V\] \[-s\] [file_list]]

Description
The listvgbackup command lists the contents of a volume group backup from tape, file, CD-ROM, or other source and can be used to restore files from a valid backup source. The listvgbackup command also works for multi-volume backups such as multiple CDs, DVDs, or tapes.

The listvgbackup \[-r\] and restorevgfiles commands perform identical operations and should be considered interchangeable.

Flags

- \-a
  Verifies the physical block size of the tape backup, as specified by the \-b block flag. You may need to alter the block size if necessary to read the backup. The \-a flag is valid only when a tape backup is used.

- \-b blocks
  Specifies the number of 512-byte blocks to read in a single input operation, as defined by the blocks parameter. If the blocks parameter is not specified, the number of blocks read will default to 100.

- \-B
  Prints the volume group backup log to stdout.

  This flag will display the past 256 backups (roughly). The log is in alog format and is kept in /var/adm/ras/vgbackuplog. Each line of the log is a semicolon-separated list of the file or device name, the command used to make backup, date, shrink size, full size of the backup, and recommended maintenance level (if any).

  Note: The shrink size is the size of the data on all filesystems. The full size is total size of each filesystem (unused + data).

- \-c
  Produces colon-separated output. This flag only works with the \-l and \-L flags.

- \-d path
  Specifies the directory path to which the files will be restored, as defined by the path parameter. If the \-d parameter is not used, the current working directory is used. This can be a problem if the current working directory is root. We recommend writing to a temporary folder instead of to root.

- \-D
  Produces debug output.

- \-I
  Displays useful information about a volume group backup.

  This flag requires the \-f device flag. This flag causes listvgbackup to display information such as volume group, date and time backup was made, uname output from backed up system, oslevel, recommended maintenance level, backup size in megabytes, and backup shrink size in megabytes. The shrink size is the size of the data on all filesystems. The full size is the total size of each filesystem (unused + data). The \-I flag also displays the logical volume and filesystem information of the backed up volume group, equivalent to running “lsvg \-l vgname”.

- \-L
  Displays lpp filesystem information about a mksysb backup only.

  This flag requires the \-f device flag and displays the equivalent information to that produced by invoking “lslpp \-I” on the running backed up system. This flag does not produce output about any volume group backup other than that produced by mksysb.

- \-f device
  Specifies the type of device containing the backup (file, tape, CD-ROM, or other source) as defined by the device parameter. When \-f is not specified, device will default to /dev/rmt0.
Specifies to restore the backup files, as defined by the file-list parameter. If the file-list parameter is not specified, then all files in the backup will be restored. If the -r flag is not used, then executing the listvgbackup command only lists the files in the specified backup.

-s
Specifies that the backup source is a user volume group and not rootvg.
-V
Verifies a tape backup.

This flag requires the -f device flag and works for tape devices only. The -V flag causes listvgbackup to verify the readability of the header of each file on the volume group backup and print any errors that occur to stderr.

Parameters

file_list
Identifies the list of files to be restored. This parameter is used only when the -r flag is specified. The full path of the files relative to the current directory should be specified in the space-separated list. All files in the specified directory will be restored unless otherwise directed. If you are restoring all files in a directory, we recommend writing to a temporary folder instead of to root.

Examples

1. To list the contents of the system backup located on the default device /dev/rmt0, enter:
   listvgbackup
2. To list the contents of the system backup located on device /dev/cd1, enter:
   listvgbackup -f /dev/cd1
3. To list the contents of the system backup located on device /dev/cd1, which is a user volume group that is not rootvg, enter:
   listvgbackup -f /dev/cd1 -s
4. To restore /etc/filesystems from the system backup located on device /dev/cd1, enter:
   listvgbackup -f /dev/cd1 -r /etc/filesystems
5. To restore all files in the /myfs/test directory of the non-rootvg backup, which is located on device /dev/cd1, and write the restored files to /data/myfiles, enter:
   listvgbackup -f /dev/cd1 -r -s -d /data/myfiles /myfs/test
6. To display colon separated lpp information about a mksysb backup tape located on /dev/rmt0, enter the following:
   lsmksysb -Lc -f /dev/rmt0
7. To display the volume group backup log to stdout, enter:
   lssavevg -B
8. To list volume group and general backup data about a backup located at /tmp/mybackup, enter:
   listvgbackup -l -f /tmp/mybackup
9. To verify the readability of each header on a volume group backup tape in /dev/rmt0, enter:
   lsmksysb -V -f /dev/rmt0

Files

/usr/bin/listvgbackup

Contains the listvgbackup command

Related Information

The restorevgfiles command.
**listX11input Command**

**Purpose**
Lists X11 input extension records entered into the Object Data Manager (ODM) database.

**Syntax**
listX11input

**Description**
The listX11input command lists all X11 input extension records entered in the ODM database.

**Error Codes**
*ODM could not open class*

The ODM database is not stored in the /usr/lib/objrepos directory.

**Related Information**
The addX11input command, deleteX11input command.

---

**llbd Daemon**

**Purpose**
Manages the information in the local location broker database.

**Syntax**
llbd [-family FamilyName] [-version]

**Description**
The llbd daemon is part of the Network Computing System (NCS). It manages the local location broker (LLB) database, which stores information about NCS-based server programs running on the local host.

A host must run the llbd daemon to support the location broker forwarding function or to allow remote access (for example, by the lb_admin tool) to the LLB database. In general, any host that runs an NCS-based server program should run an llbd daemon, and llbd should be running before any such servers are started. Additionally, any network or internet supporting NCS activity should have at least one host running a global location broker daemon (glbd).

The llbd daemon is started in one of two ways:

- Through the System Resource Controller (the recommended method), by entering on the command line:
  
  startsrc -s llbd

- By a person with root user authority entering on the command line:
  
  /etc/ncs/llbd &

TCP/IP must be configured and running on your system before you start the llbd daemon. (You should start the llbd daemon before starting the glbd or nrglbd daemon.)
Flags

-listen FamilyList
Restricts the address families on which an LLB listens. Use it only if you are creating a special configuration where access to an LLB is restricted to a subset of hosts in the network or internet.

The FamilyList is a list of the address families on which the LLB will listen. Names in this list are separated by spaces. Possible family names include ip.

If llbd is started without the -listen option, the LLB will listen on all address families that are supported both by NCS and by the local host.

-version
Displays the version of NCS that this llbd belongs to, but does not start the daemon.

Files

/etc/rc.ncs Contains commands to start the NCS daemons.

Related Information
The lb_admin command, startsrc command.
The glbd (NCS) daemon, nrglbd (NCS) daemon.

In Command

Purpose
Links files.

Syntax

To Link a File to a File
In [-f | -n] [-s] SourceFile [ TargetFile ]

To Link a File or Files to a Directory
In [-f | -n] [-s] SourceFile ... TargetDirectory

Description
The In command links the file designated in the SourceFile parameter to the file designated by the TargetFile parameter or to the same file name in another directory specified by the TargetDirectory parameter. By default, the In command creates hard links. To use the In command to create symbolic links, designate the -s flag.

A symbolic link is an indirect pointer to a file; its directory entry contains the name of the file to which it is linked. Symbolic links may span file systems and may refer to directories.

If you are linking a file to a new name, you can list only one file. If you are linking to a directory, you can list more than one file.

The TargetFile parameter is optional. If you do not designate a target file, the In command creates a new file in your current directory. The new file inherits the name of the file designated in the SourceFile parameter. See example 5.

Notes:
1. You cannot link files across file systems without using the -s flag.
2. If TargetDirectory is already a symbolic link to a directory, then the ln command treats the existing target as a file. This means that a command such as ln -fs somepath/lname symdir will not follow the existing symbolic link of symdir, instead it will create a new symbolic link from somepath/lname to symdir.

**Flags**

- **-f** Causes the ln command to replace any destination paths that already exist. If a destination path already exists and the -f flag is not specified, the ln command writes a diagnostic message to standard error without creating a new link and continues to link the remaining SourceFiles.
- **-n** Specifies that if the link is an existing file, do not overwrite the contents of the file. The -f flag overrides this flag. This is the default behaviour.
- **-s** Causes the ln command to create symbolic links. A symbolic link contains the name of the file to which it is linked. The referenced file is used when an open operation is performed on the link. A stat call on a symbolic link returns the linked-to file; an istat call must be done to obtain information about the link. The readlink call may be used to read the contents of a symbolic link. Symbolic links can span file systems and refer to directories.

**Note:** Absolute path names must be used when specifying the SourceFile parameter for the -s flag. If the absolute path name is not designated, unexpected results may occur when the SourceFile and the TargetFile parameters are located in different directories. The source file does not need to exist before creating the symbolic link.

**Exit Status**

This command returns the following exit values:

- **0** All specified files were successfully linked.
- **>0** An error occurred.

**Examples**

1. To create another link (alias) to a file, enter:

   ```bash
   ln -f chap1 intro
   ```

   This links chap1 to the new name, intro. If intro does not already exist, the file name is created. If intro does exist, the file is replaced by a link to chap1. Then both the chap1 and intro file names will refer to the same file. Any changes made to one also appear in the other. If one file name is deleted with the rm command, the file is not completely deleted since it remains under the other name.

2. To link a file to the same name in another directory, enter:

   ```bash
   ln index manual
   ```

   This links index to the new name, manual/index.

   **Note:** intro in example 1 is the name of a file; manual in example 2 is a directory that already exists.

3. To link several files to names in another directory, enter:

   ```bash
   ln chap2 jim/chap3 /home/manual
   ```

   This links chap2 to the new name /home/manual/chap2 and jim/chap3 to /home/manual/chap3.

4. To use the ln command with pattern-matching characters, enter:

   ```bash
   ln manual/*
   ```
This links all files in the manual directory into the current directory, . (dot), giving them the same names they have in the manual directory.

**Note:** You must type a space between the asterisk and the period.

5. To create a symbolic link, enter:

```
ln -s /tmp/toc toc
```

This creates the symbolic link, toc, in the current directory. The toc file points to the /tmp/toc file. If the /tmp/toc file exists, the `cat` toc command lists its contents.

To achieve identical results without designating the `TargetFile` parameter, enter:

```
ln -s /tmp/toc
```

## Files

`/usr/bin/ln` Contains the `ln` command.

## Related Information

The `cp` command, `mv` command, `rm` command.

The `link` subroutine, `readlink` subroutine, `stat` subroutine, `symlink` subroutine.


[Linking for Programmers](#) in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs discusses links from a programming viewpoint.

### locale Command

#### Purpose

Writes information to standard output about either the current locale or all public locales.

#### Syntax

```
locale [-O 64] [-a | -m] [-c | -k] Name ...
```

#### Description

The `locale` command writes information to standard output about either the current locale or all public locales. A public locale is a locale available to any application.

To write the name and value of each current locale category, do not specify any flags or variables. To write the names of all available public locales, specify the `-a` flag. To write a list of the names of all available character-mapping (charmap) files, specify the `-m` flag. These charmap filenames are suitable values for the `-f` flag specified with the `localedef` command.

To write information about specified locale categories and keywords in the current locale, specify the `Name` parameter. The `Name` parameter can be one of the following:
• A locale category, such as **LC_TYPE** or **LC_MESSAGES**
• A keyword, such as **yesexpr** or **decimal_point**
• The **charmmap** reserved word to determine the current character mapping

You can specify more than one **Name** parameter with the **locale** command.

If you specify the **locale** command with a locale category name and no flags, the **locale** command writes the values of all keywords in the locale category specified by the **Name** parameter. If you specify the **locale** command with a locale keyword and no flags, the **locale** command writes the value of the keyword specified by the **Name** parameter.

If the **Name** parameter is a locale category name or keyword, the -c and -k flags can determine the information displayed by the **locale** command.

**Flags**

- **-a**  
  Writes the names of all available public locales.
- **-c**  
  Writes the names of selected locale categories. If the **Name** parameter is a keyword, the **locale** command writes the name of the locale category that contains the specified keyword, and the value of the specified keyword. If the **Name** parameter is a locale category, the **locale** command writes the name of the specified locale category and the values of all keywords in the specified locale category.
- **-k**  
  Writes the names and values of selected keywords. If the **Name** parameter is a keyword, the **locale** command writes the name and value of the specified keyword. If the **Name** parameter is a locale category, the **locale** command writes the names and values of all keywords in the specified locale category.
- **-m**  
  Writes the names of all available character-mapping (charmap) files.
- **-ck**  
  Writes the name of the locale category, followed by the names and values of selected keywords. If the **Name** parameter is a keyword, the **locale** command writes the name of the locale category that contains the specified keyword, and the name and value of the specified keyword. If the **Name** parameter is a locale category, the **locale** command writes the name of the specified locale category and the names and values of all keywords in the specified locale category.
- **-O 64**  
  Displays locale information as seen by a 64 bit executable. This should be identical to information as seen by a 32 bit executable.

**Exit Status**

This command returns the following exit values:

- **0**  
  All the requested information was found and output successfully.
- **>0**  
  An error occurred.

**Examples**

1. To retrieve the names and values of all the current locale environment variables, enter:

   ```
   locale
   ```

   If **locale_x** and **locale_y** are valid locales on the system, as determined with **locale -a**, and if the locale environment variables are set as follows:

   ```
   LANG=locale_x
   LC_COLLATE=locale_y
   ```

   The **locale** command produces the following output:

   ```
   LANG=locale_x
   LC_TYPE=locale_x
   LC_COLLATE=locale_x
   LC_TIME=locale_x
   ```
When setting the locale variables, some values imply values for other locale variables. For example, if the `LC_ALL` locale variable is set to the `En_US` locale, all locale environment variables are set to the `En_US` locale. In addition, implicit values are enclosed in double quotes (" ). Explicitly set values are not enclosed in double quotes (" ). See "Understanding Locale Environment Variables" in AIX 5L Version 5.2 National Language Support Guide and Reference for more information.

2. To determine the current character mapping, enter:
   ```
   locale charmap
   ```
   If the `LC_ALL` locale variable is set to the C locale, the `locale` command produces the following output:
   ```
   ISO8859-1
   ```

3. To retrieve the value of the decimal_point delimiter for the current locale, enter:
   ```
   locale -ck decimal_point
   ```
   If the `LC_ALL` locale variable is set to the C locale, the `locale` command produces the following output:
   ```
   LC_NUMERIC
   decimal_point="."
   ```

Related Information
The `localedef` command.
Character Set Description (charmap) Source File Format and Locale Definition Source File Format in AIX 5L Version 5.2 Files Reference.

For specific information about the locale definition source file format, categories, and their locale variables, see the `LC_COLLATE` category, `LC_CTYPE` category, `LC_MESSAGES` category, `LC_MONETARY` category, `LC_NUMERIC` category, and `LC_TIME` category in AIX 5L Version 5.2 Files Reference.

localedef Command

Purpose
Converts locale and character set description (charmap) source files to produce a locale database.

Syntax
```
localedef [ -c ] [ -f Charmap ] [ -i SourceFile ] [ -L LinkOptions ] [ -m MethodFile ] LocaleName
```

Description
The `localedef` command converts source files that contain definitions of locale-dependent information (such as collation, date and time formats, and character properties) into a locale object file used at run-time. The locale object file created by the `localedef` command is then used by commands and subroutines that set the locale with the `setlocale` subroutine.
The -i SourceFile flag and variable specify the file that contains the source category definitions. If the -i flag is not specified, the file is read from standard input.

The -f CharMap flag and variable specify a file that maps character symbols to actual character encodings. Using the -f flag allows one locale source definition to be applicable to more than one code set. If the -f flag is not specified, the default value for the CharMap variable is ISO8859-1.

The LocaleName parameter specifies the locale name for the locale database generated by the localedef command from the specified source files. The LocaleName parameter can be either an absolute path name for the file location or a relative path name.

If a locale category source definition contains a copy statement and the statement names an existing locale installed in the system, the localedef command proceeds as though the source definition contained the valid category definition for the named locale.

**Notes:**
1. The localedef command uses the C compiler to generate the locale database. Therefore, to use this command you must have the C compiler installed.
2. When replacing systemwide databases, it is advisable to do a soft reboot to ensure that the new locale is used throughout the system.

If an error is detected, no permanent output is created.

If warnings occur, permanent output is created when the -c flag is specified. The following conditions cause warning messages to be issued:

- A symbolic name not found in the file pointed to by the CharMap variable is used for the descriptions of the LC_TYPE or LC_COLLATE categories. This is an error condition for other categories.
- The number of operands to the order_start keyword exceeds the COLL_WEIGHTS_MAX limit.
- Optional keywords not supported by the implementation are present in the source file.

**Flags**

- `-c` Forces the creation of locale tables even if warning messages have been issued.
- `-f CharMap` Specifies the name of a file containing a mapping of character symbols and collating element symbols to actual character encodings. A locale is associated with one and only one code set. If this flag is not specified, the ISO 8859-1 code set is assumed.
  
  **Note:** The use of certain system-provided CharMap files is fully supported. However, while correctly defined user-provided CharMap files may work properly, the result of such use is not guaranteed.
- `-i SourceFile` Specifies the path name of a file containing the locale category source definitions. If this flag is not present, source definitions are read from standard input.
- `-L LinkOptions` Passes the specified link options to the command used to build the locale.
- `-m MethodFile` Specifies the name of a method file that describes the methods to override when constructing a locale. The method file specifies user-supplied subroutines that override existing definitions, as well as a path name for the library containing the specified subroutines. The localedef command reads the method file and uses entry points when constructing the locale objects. The code set methods specified are also used in parsing the file pointed to by the CharMap variable.
  
  **Note:** To create a 64-bit locale, the method file must specify the path of the library as a single archive that has two shared objects, one 32-bit and the other 64-bit, that contain the specified subroutines. Specifying separate paths to the 32-bit and 64-bit shared objects causes the localedef command to fail due to incompatible XCOFF format.
- `LocaleName` Specifies the name of the locale to be created. This is the name that can subsequently be used to access this locale information.
Exit Status
The `localedef` command returns the following exit values:

0  No errors occurred and the locales were successfully created.
1  Warnings occurred and the locales were successfully created.
2  The locale specification exceeded limits or the code set or sets used were not supported by the implementation, and no locale was created.
3  The capability to create new locales is not supported.
>3  Warnings or errors occurred and no locales were created.

Examples
1. To create a locale called Austin from standard input and disregard warnings, enter:
   ```
   localedef -c Austin
   ```
2. To create a locale called Austin with Austin.src as source input, enter:
   ```
   localedef -i Austin.src Austin
   ```

Related Information
The `id` command, `locale` command.
The `setlocale` subroutine.


For specific information about the locale categories and their keywords, see the `LC_COLLATE` category, `LC_CTYPE` category, `LC_MESSAGES` category, `LC_MONETARY` category, `LC_NUMERIC` category, and `LC_TIME` category for the locale definition source file format in AIX 5L Version 5.2 Files Reference.


lock Command

Purpose
Reserves a terminal.

Syntax
```
lock [\-Timeout]
```

Description
The `lock` command requests a password from the user, reads it, and requests the password a second time to verify it. In the interim, the command locks the terminal and does not relinquish it until the password is received the second time or one of the following occurs:

- The timeout interval is exceeded.
- The command is killed by a user with appropriate permission.

The timeout default value is 15 minutes, but this can be changed with the \-Timeout flag.
Flags

- **Timeout**  Indicates the timeout interval in minutes, as specified by the `Timeout` parameter. The default value is 15 minutes.

Examples

1. To reserve a terminal under password control, enter:

   `lock`

   You are prompted for the password twice so the system can verify it. If the password is not repeated within 15 minutes, the command times out.

2. To reserve a terminal under password control, with a timeout interval of 10 minutes, enter:

   `lock -10`

Files

`/usr/bin/lock`  Contains the `lock` command.

Related Information

The `passwd` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the [AIX 5L Version 5.2 Security Guide](#).

lockd Daemon

**Purpose**
Processes lock requests.

**Syntax**

```
/usr/sbin/rpc.lockd [ -a NumberOfThreads ] [ -t TimeOut ] [ -g GracePeriod ]
```

**Description**

The `lockd` daemon processes lock requests that are either sent locally by the kernel or remotely by another lock daemon. The `lockd` daemon forwards lock requests for remote data to the server site lock daemon through the RPC package. The `lockd` daemon then asks the `statd` (status monitor) daemon for monitor service. The reply to the lock request is not sent to the kernel until both the `statd` daemon and the server site `lockd` daemon reply. The `statd` daemon should always be started before the `lockd` daemon.

If either the status monitor or the server site lock daemon is unavailable, the reply to a lock request for remote data is delayed until all daemons become available.

When a server recovers, it waits for a grace period for all client site `lockd` daemons to submit reclaim requests. The client site `lockd` daemons, on the other hand, are notified of the server recovery by the `statd` daemon. These daemons promptly resubmit previously granted lock requests. If a `lockd` daemon fails to secure a previously granted lock at the server site, the `lockd` daemon sends a SIGLOST signal to the process.

The `lockd` daemon is started and stopped with the following System Resource Controller (SRC) commands:
To modify the arguments passed to the lockd daemon when it is started, use the following command:

```
chsys -s rpc.lockd Parameters...
```

The status monitor maintains information on the location of connections as well as the status in the /etc/sm directory, the /etc/sm.bak file, and the /etc/state file. When restarted, the statd daemon queries these files and tries to reestablish the connection it had prior to termination. To restart the statd daemon, and subsequently the lockd daemon, without prior knowledge of existing locks or status, delete these files before restarting the statd daemon.

By default rpc.lockd establishes a dynamic socket port number for receiving requests. Entries may be added to the /etc/services file specifying the port that rpc.lock will listen for requests on. The service name is lockd and a unique port number should be specified. The following entries in /etc/services file would specify that port 16001 be used for both tcp and udp.

```
lockd 16001/tcp
lockd 16001/udp
```

### Flags

- **-a NumberOfThreads**
  Specifies the maximum number of rpc.lockd threads. This can be useful on an NFS file system because all the lockf() or flock() calls (and other file locking calls) must go through the rpc.lockd daemon. This can severely degrade system performance because the lock daemon might not be able to handle thousands of lock requests per second. When a large number of concurrent processes are forked from a single script, the rpc.lockd value increases to the default maximum of 33. After the maximum rpc.lockd value is reached, any other lock requests must wait for the lockd daemon to processing. This wait causes a recurring cycle of TCP timeouts thus allowing only one individual forked process to complete each cycle. The solution to this problem is to increase the maximum allowed number of rpc.lockd threads. This can be done with the following command:

```
startsrc -s rpc.lockd -a NumberOfThreads (511 max.)
```

or in the ODM with:

```
chsys -s rpc.lockd -a NumberOfThreads (511 max.)
```

- **-g GracePeriod**
  Uses the GracePeriod variable to specify the amount of time, in seconds, that the lockd daemon should wait for reclaim requests for previously granted locks. The default value of the GracePeriod variable is 45 seconds.

- **-t TimeOut**
  Uses the TimeOut variable to specify the interval between retransmitting lock requests to the remote server. The default value for the TimeOut variable is 15 seconds.

### Examples

1. To specify a grace period, enter:

```
/usr/sbin/rpc.lockd -g 60
```

   In this example, the grace period is set for 60 seconds.

2. To specify the amount of time the lockd daemon should wait before retransmitting a lock request, enter:

```
/usr/sbin/rpc.lockd -t 30
```

   In this example, the retransmissions occur after 30 seconds.
Related Information

The `chssys` command, `nfso` command, `no` command.

The `statd` daemon.

The `fcntl` subroutine, `lockf` subroutine, `signal` subroutine.

List of NFS Commands


AIX 5L Version 5.2 Performance Management Guide.

locktrace Command

Purpose

Controls kernel lock tracing.

Syntax

`locktrace [-r ClassName] [-s ClassName] [-S] [-R] [-l]`

Description

The `locktrace` command controls which kernel locks are being traced by the `trace` subsystem. The default is to trace none. If the machine has been rebooted after running the `bosboot -L` command, kernel lock tracing can be turned on or off for one or more individual lock classes, or for all lock classes. If `bosboot -L` was not run, lock tracing can only be turned on for all locks or none. The `trace` events collected in this case when locks are taken or missed (hook id 112), and released (hook id 113) do not have the lock class name available.

Flags

- `-r classname`  
  Turn off lock tracing for all the kernel locks belonging to the specified class. This option always fails if `bosboot -L` was not run.

- `-s classname`  
  Turn on lock tracing for all the kernel locks belonging to the specified class. This option always fails if `bosboot -L` has not been executed.

- `-R`  
  Turn off all lock tracing.

- `-S`  
  Turn on lock tracing for all locks regardless of their class membership.

- `-l`  
  List kernel lock tracing current status.

Examples

1. To start tracing the SEM_LOCK_CLASS, type:
   
   `locktrace -s SEM_LOCK_CLASS`

2. To stop all lock tracing, type:
   
   `locktrace -R`
Related Information
The `bosboot` command, and the `trace` command.

**elogeevent Command, logevent Command**

**Purpose**
Logs event information generated by the event response resource manager (ERRM) to a specified log file.

**Syntax**

```
elogevent [-h] log_file
```

```
logevent [-h] log_file
```

**Description**

The `elogevent` script always return messages in English. The language in which the messages of the `logevent` script are returned depend on the locale settings.

These scripts capture event information that is posted by the event response resource manager (ERRM) in environment variables that are generated by the ERRM when an event occurs. These scripts can be used as actions that are run by an event response resource. They can also be used as templates to create other user-defined actions.

Event information is returned about the ERRM environment variables, and also includes the following:

**Local Time**
Time when the event or rearm event is observed. The actual environment variable supplied by ERRM is ERRM_TIME. This value is localized and converted to readable form before being displayed.

These scripts use the `alog` command to write event information to and read event information from the specified `log_file`.

**Flags**

- `-h` Writes the script's usage statement to standard output.

**Parameters**

`log_file`

Specifies the name of the file where event information is logged. An absolute path for the `log_file` parameter should be specified.

The `log_file` is treated as a circular log and has a fixed size of 64KB. When `log_file` is full, new entries are written over the oldest existing entries.

If `log_file` already exists, event information is appended to it. If `log_file` does not exist, it is created so that event information can be written to it.
Exit Status

0   The script has run successfully.
1   A required log_file is not specified.
2   The log_file path is not valid.

Restrictions

- These scripts must be run on the node where the ERRM is running.
- The user who runs these scripts must have write permission for the log_file where the event information is logged.

Standard Output

When the -h flag is specified, the script's usage statement is written to standard output.

Examples

1. To log information, specify /tmp/event.log in the Web-based System Manager interface. ERRM runs this command:
   
   /usr/sbin/rsct/bin/logevent/tmp/event.log

   The /tmp/event.log file does not need to exist when the command is run.

2. To see the contents of the /tmp/event.log file, run this command:
   
   alog -f /tmp/event.log -o

   The following sample output shows a warning event for the /var file system (a file system resource):

   ==============================================================
   Event reported at Mon Mar 27 16:38:03 2002
   ==============================================================
   Condition Name: /var space used
   Severity: Warning
   Event Type: Event
   Expression: PercentTotUsed>90
   Resource Name: /var
   Resource Class Name: IBM.FileSystem
   Data Type: CT_UINT32
   Data Value: 91

Location

/usr/sbin/rsct/bin/elogevent   Contains the elogevent script
/usr/sbin/rsct/bin/logevent   Contains the logevent script

Related Information

Commands: alog

logform Command

Purpose

Initializes a logical volume for use as a Journaled File System (JFS) log. Initializes an Enhanced Journaled File System (JFS2) outline log. Reformats an inline log for an existing JFS2 file system using an inline log.
**Syntax**

```
logform [-V vfstype] LogName
```

**Description**

The `logform` command initializes a logical volume for use as a JFS or JFS2 log device. Running the `logform` command on any JFS log device or JFS2 outline or inline log device will destroy all log records on the log device. This may cause the file system to lose its recovery capability and therefore to lose the file system data.

When you run the `logform` command on an outline log for a JFS2 file system that is already using an outline log, the device type for the outline log must be `jfs2log`. Otherwise, the `logform` command will exit with an error.

To reuse an existing logical volume as an outline log device for a JFS2 file system, you must delete the logical volume and then recreate it as device type `jfs2log`.

When you run the `logform` command on a device with logical volume type `jfs2`, if the device has a file system with an inline log, then the inline log will be reformatted. If the device has a file system with an outline log, then an error will be reported.

When the `logform` command is used to format an inline log for an existing JFS2 file system, the file system data will not be affected; only the log records are destroyed. The logical volume type for an inline log is the same as for the file system. For a JFS2 file system, the inline log logical volume type is `jfs2`.

**Flags**

```
-V vfstype [jfslog | jfs2log]
```

If specified, `vfstype` indicates what type of file system the log should be formatted for. If this option is not specified, then the type is obtained from the logical volume’s type. If there is no type specified for the logical volume, then the default vfs type from `/etc/vfs` is used.

**Parameters**

`LogName` The `LogName` parameter specifies the absolute path to the logical volume to be initialized (for example, `/dev/jfslog1`). When the `logform` command is run on an inline log, `LogName` is the device name of the file system.

**Examples**

1. To create a JFS logging device on a newly created volume group, first create a logical volume of type `jfslog`:
   ```bash
   mklv -t jfslog -y jfslog1 newvg1
   ```
   
   This command creates a `jfslog` logical volume named `jfslog1` in the volume group `newvg1`. The size of the logical volume is 1 logical partition.

2. To format the `jfslog1` logical volume once it has been created, enter:
   ```bash
   logform /dev/jfslog1
   ```
   
   The `jfslog1` logical volume is now ready to be used as a JFS log device.

3. To format the inline log for an existing file system called `/j2` which is on the file system device `/dev/fs1v00`, type:
   ```bash
   logform /dev/fs1v00
   ```
   
   This formats the inline log for file system `/j2`, but does not touch the data in the file system.

210 Commands Reference, Volume 3
Files

/etc/filesystems

Lists the known file systems and defines their characteristics, including the log device.

Related Information

The crfs command, mkfs command, mkiv command.


logger Command

Purpose

Makes entries in the system log.

Syntax

logger [\(-f\) File] [\(-i\)] [\(-p\) Priority] [\(-t\) Tag] [Message]

Description

The logger command provides an interface to the syslog subroutine, which writes entries to the system log. A Message variable can be specified on the command line, which is logged immediately, or a File variable is read and each line of the File variable is logged. If you specify no flags or variables, the logger command will wait for you to enter a message from standard input. The messages returned by the LOG_KERN facility cannot be logged by this command.

Flags

-\(f\) File

Logs the specified File variable. If the Message variable is specified, this flag is ignored.

-\(i\)

Logs the process ID of the logger process with each line.

-\(p\) Priority

Enters the message with the specified priority. The Priority parameter may be a number or a facility.level priority specifier.

-\(t\) Tag

Marks every line in the log with the specified Tag parameter.

Message

Indicates the message to log. If this variable is not specified, the logger command logs either standard input or the file specified with the \(f\) File flag.

Examples

1. To log a message indicating a system reboot, enter:
   
   logger System rebooted

2. To log a message contained in the /tmp/msg1 file, enter:
   
   logger -f /tmp/msg1

3. To log the daemon facility critical level messages, enter:
   
   logger -pdaemon.crit
Exit Status
This command returns the following exit values:

0 Successful completion.
>0 An error occurred.

Files
/usr/bin/logger Contains the logger command.

Related Information
The syslog daemon.
The syslog subroutine.

login Command
Purpose
Initiates a user session.

Syntax
login [-h HostName] [-p] [-f User] [-k] [ User [ Environment ] ]

Description
The login command (part of the tsm command) initiates sessions on the system for the user specified by the User parameter. You can also specify environment variables to be added to the user’s environment. These are strings of the form Variable=Value. The login command is not normally entered on the command line.

Notes:
1. The PATH, IFS, HOME, and SHELL environment variables may not be initialized from the command line.
2. The login command supports multibyte user names. It is recommended that the system administrator restrict the user names to characters within the portable character set to remove any ambiguity.
3. If the /etc/nologin file exists, the system prevents the user from logging in and displays the contents of the /etc/nologin file. The system does allow the root user to log in if this file exists. The /etc/nologin file is removed when you reboot the system.

The login command can handle Distributed Computing Environment (DCE) user names of up to 1024 characters. DCE user names are stored in the LOGIN environment variable. Because DCE user names do not conform to standard operating system requirements, the first 8 characters of the DCE user name are stored in all standard operating system files and environments.

The login command performs the following functions:

Checks accounts The login command validates the user’s account, ensuring authentication, logins enabled properly, and correct capacity for the port used for the login.
**Authenticates users**

The `login` command verifies the user’s identity by using the system defined authentication methods for each user. If a password has expired, the user must supply a new password. If secondary authentication methods are defined, these methods are invoked but need not be successful in order to log in to the system.

**Establishes credentials**

The `login` command establishes the initial credentials for the user from the user database. These credentials define the user’s access rights and accountability on the system.

**Initiates a session**

The `login` command initializes the user environment from the user database, from the command line, and from the `/etc/environment` configuration file; changes the current directory to the user’s home directory (normally); and runs the user’s initial program.

These functions are performed in the order given; if one fails, the functions that follow are not performed.

When a user logs in successfully, the `login` command makes entries in the `/etc/utmp` file that tracks current user logins and the `/var/adm/wtmp` file that is used for accounting purposes. The `login` command also sets the `LOGIN` and `LOGNAME` environment variables.

Information pertaining to each unsuccessful login is recorded in the `/etc/security/failedlogin` file. The information stored is the same as that in the `/etc/utmp` file, except that unrecognizable user names are logged as `UNKNOWN_USER`. This ensures that a password accidentally entered as a user name, for example, is not allowed into the system unencrypted.

After a successful login, the `login` command displays the message of the day, the date and time of the last successful and unsuccessful login attempts for this account, and the total number of unsuccessful login attempts for this account since the last successful login. These messages are suppressed if there is a `.hushlogin` file in your home directory.

The `login` command also changes the ownership of the login port to the user. This includes any ports noted as synonyms in the `/etc/security/login.cfg` file.

In order to preserve the integrity of the system, only one session at a time is allowed to be logged in to a port. This means that the `login` command entered from the shell prompt cannot succeed, as both the original session and the new login session would be on the same port. However, the `exec` `login` command succeeds because a new shell replaces the current one. The `login` command is typically a built-in shell command, causing the shell to replace itself.

**Note:** Unless your terminal displays only uppercase letters, your user name should not consist of uppercase letters exclusively.

To log in with multibyte user names, you must first open a Japanese window (aixterm) and initiate a new login from the Japanese window.

**Flags**

- `-f User`
  Identifies a user who has already been authenticated. If the real ID of the login process is root (0), then the user is not authenticated.

- `-h HostName`
  Identifies the login as a remote login and specifies with the `HostName` variable the name of the machine requesting the login. This form of the login is used only by the `telnetd` and `rlogind` daemons.

- `-k`
  Identifies the login as using Kerberos authentication and causes login to pass control to `/usr/bin/k5dcelogin` to handle authentication. This form of login is only used by the `krshd` daemon.

- `-p`
  Preserves the current terminal type by setting it the value of the `STERM` environment variable instead of the type contained in the `CuAt/PdAt` object classes database.
Security
Access Control: This command sets the setuid permission to the root user, grants executable (x) permission to any user, and is in the Trusted Computing Base.

Examples
To log in to the system as user jamesd, enter the following at the login prompt:
login: jamesd

If a password is defined, the password prompt appears. Enter your password at this prompt.

Files

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/sbin/login</td>
<td>Contains the login command.</td>
</tr>
<tr>
<td>/etc/utmp</td>
<td>Contains accounting information.</td>
</tr>
<tr>
<td>/var/adm/wtmp</td>
<td>Contains accounting information.</td>
</tr>
<tr>
<td>/etc/motd</td>
<td>Contains the message of the day.</td>
</tr>
<tr>
<td>/etc/passwd</td>
<td>Contains passwords.</td>
</tr>
<tr>
<td>$HOME/.hushlogin</td>
<td>Suppresses login messages.</td>
</tr>
<tr>
<td>/etc/environment</td>
<td>Contains user environment configuration information.</td>
</tr>
<tr>
<td>/etc/security/login.cfg</td>
<td>Contains port synonyms.</td>
</tr>
<tr>
<td>/etc/security/lastlog</td>
<td>Contains information pertaining to the most recent successful and</td>
</tr>
<tr>
<td></td>
<td>unsuccessful login attempts.</td>
</tr>
<tr>
<td>/etc/security/faillogin</td>
<td>Contains information pertaining to each unsuccessful login.</td>
</tr>
</tbody>
</table>

Related Information
The getty command, setgroups command, setserv command, su command tsm command.

The utmp, wtmp, failedlogin file format, lastlog file format.

The authenticate subroutine, setuid subroutine.


Shells Overview in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices describes what shells are, the different types of shells, and how shells affect the way commands are interpreted.

logins Command

Purpose
Displays user and system login information.

Syntax
logins [ -a ] [ -m ] [ -o ] [ -p ] [ -s ] [ -t ] [ -u ] [ -x ] [ -g Groups ] [ -h Logins ]

Description
The logins command displays information about user and system logins. By default, the logins command prints the following:
• User ID
• primary group name
• primary group ID
• the /etc/passwd account field on user information.

The output is sorted by user ID, displaying system logins followed by user logins.

Depending on the options chosen, the following fields can also be displayed:
• user or system login
• user ID number
• multiple group names
• multiple group IDs
• home directory
• login shell
• four password aging parameters
• /etc/passwd account field value (user name or other information)
• primary group name
• primary group ID

Flags

-a In addition to the default output, the -a flag adds two password expiration fields to the display. These fields show how many days a password can remain unused before it automatically becomes inactive and the date that the password will expire.

-g Groups Displays all users belonging to group, sorted by user ID. Multiple groups can be specified as a comma separated list. Groups must specify valid group names on the system. Comma separate names when specifying more than one group.

-I Logins Displays the requested login. Multiple logins can be specified as a comma-separated list.

-logins must specify valid user names on the system.

-m Displays multiple group membership information.

-o Formats output into one line of colon separated fields.

-p Displays users without passwords.

-s Displays all system logins.

-t Sorts output by user name instead of by user ID.

-u Displays all user logins.

-x Prints an extended set of information about each selected user. Information for each user is printed on a separate line containing the home directory, login shell, and password aging information. The extended information includes the following:
• The password status
• The date on which the password was last changed
• The number of days required between changes
• The number of days allowed before a change is needed
• The number of days that the user will receive a password expiration warning message before the password expires

The password status is displayed in an abbreviated form as PS for logins with password, NP for no password or LK for locked.

Exit Status

0 The command completed successfully.
Examples

1. To list all the logins with no passwords, enter:
   ```
   logins -p
   ```
   The output looks similar to the following:
   ```
   pwdless  204  staff  1
   nopwd   208  staff  1
   ```
   The -p option ensures that only logins with no passwords are listed.

2. To list all the system logins sorted by alphabetical order enter:
   ```
   logins -st
   ```
   The output looks similar to the following:
   ```
   adm        4  adm        4
   bin        2  bin        2
   daemon     1  staff      1
   lp         11 lp         11
   lpd        9  nobody     -2
   root       0  system     0
   sys        3  sys        3
   uucp       5  uucp       5
   ```
   The -t option prints out the logins sorted alphabetically and not by uid.

3. To list the login details of users "root" and "admin", enter:
   ```
   logins -l root,adm
   ```
   The output looks similar to the following:
   ```
   root      0  system     0
   adm       4  adm        4
   ```

4. To list the password aging details of users "root" and "admin" enter:
   ```
   logins -xl root,adm
   ```
   The output looks similar to the following:
   ```
   root      0  system     0
              /usr/bin/ksh
              PS 021102 0 0 0
   adm       4  adm        4
              /var/adm
              /sbin/sh
              PS 000000 0 0 0
   ```
   The -x option ensures that extended password information for these logins are retrieved and printed in the output.

5. To display the multiple group information of a particular user in a colon separated format enter:
   ```
   logins -mol root,adm
   ```
   The output looks similar to the following:
   ```
   root:0:system:0::bin:2::sys:3::security:7::cron:8::audit:10::lp:11
   adm:4::adm:4:
   ```
   The -m option is used here to retrieve the multiple group information of a particular login (user). The -o option ensures that the output is displayed in colon separated format.
6. To display the users of the "staff" and "sys" groups in a colon separated format, sorted by user name, enter:

```
logins -tsog staff,sys
```

The output looks similar to the following:

```
bin:2:bin:2:
daemon:1:staff:1:
invscout:200:staff:1:
root:0:system:0:
sys:3:sys:3:
```

**Files**

- `/usr/bin/logins` Contains the `logins` command.
- `/etc/passwd` Contains the password file.
- `/etc/group` Contains the group file.

**Related Information**

The `lsuser` command, the `lsgroup` command.

---

**logname Command**

**Purpose**
Displays login name.

**Syntax**

```
logname
```

**Description**

The `logname` command displays the login name of the current process. This is the name that the user logged in with and corresponds to the `LOGNAME` variable in the system-state environment. This variable is only set when the user logs into the system.

**Security**

Access Control: This program is installed as a normal user program in the Trusted Computing Base.

**Exit Status**

This command returns the following exit values:

- 0   Successful completion.
- >0  An error occurred.

**Examples**

To display your login name to standard output, enter:

```
logname
```

**Files**

- `/usr/bin/logname` Contains the `logname` command.
Related Information
The getty command, the login command, the setgroups command, the setsenv command, the su command, the tsm command.

logout Command

Purpose
Stops all processes on a port.

Syntax
logout

Description
The logout command terminates all processes either with the same controlling terminal as the present process or with all processes which have this terminal open. Processes that are not children of the present process are terminated upon access to the terminal. The present process is also terminated. If the login command user and the logout command user do not match, the logout command permission is denied, and the command stops.

Examples
From a shell started by the ksh or bsh command, enter:

logout

Files
/usr/bin/logout Contains the logout command.
/etc/utmp Contains a record of logged-in users.

Related Information
The bsh command, getty command, init command, ksh command, login command, setgroups command, shell command, su command, tsh command tsm command.

The setuid subroutine.

Shells Overview in AIX 5L Version 5.2 System User’s Guide: Operating System and Devices describes what shells are, the different types of shells, and how shells affect the way commands are interpreted.

look Command

Purpose
Finds lines in a sorted file.

Syntax
look \([-d] [\-f] \) String \([\ File \ ... \]"

Description
The look command searches sorted files specified by the File parameter and prints all lines that begin with the string specified by the String parameter. The look command uses a binary search, therefore files specified by the File parameter must be sorted in the C locale collating sequence.
The -d and -f flags affect comparisons as in the sort command. This means a file must be sorted using the -f flag in the sort command before using the look command with the -f flag.

If the File parameter is not specified, the /usr/share/dict/words file is assumed with the collating sequence specified by the -df flags. The sort is completed using the current collating sequence. This should match the collating sequence used to produce the dictionary file. The look command limits the length of a word search to 256 characters.

Flags

- **d** Specifies dictionary order. Only letters, digits, tabs, and spaces are considered in comparisons.
- **f** Compares uppercase and lowercase letters as equivalent values. Case is not considered in the sorting so that initial-capital and all-capital words are not grouped together at the beginning of the output.

  **Note:** To use the look -f command, the input file must be sorted with the sort -f command.

Example

To search for all lines in the sortfile file that begin with the letter a, enter:

```
look a sortfile
```

File

```
/usr/share/dict/words
```
Contains the default dictionary.

Related Information

The [grep](#) command, sort command.


lookbib Command

Purpose

Finds references in a bibliography.

Syntax

```
lookbib [n] [ Database ... ]
```

Description

The lookbib command uses an inverted index made by the idxbib command to find sets of bibliographic references. The lookbib command reads keywords typed after the > prompt on the terminal, and retrieves records containing all these keywords. If nothing matches, nothing is returned except another > prompt.

The lookbib command asks if you need instructions and prints some brief information if you type a user-defined affirmative answer.

The Database parameter specifies files that contain bibliographic references, indexes, or similar types of information. It is possible to search multiple databases as long as they have a common index made by the idxbib command. In that case, only the first database name given to the idxbib command is specified to the lookbib command.
If the `lookbib` command does not find the index files (the `.i[abc]` files), it looks for a reference file with the same name as the first database, but without the suffixes. It creates a file with a `.ig` suffix, suitable for use with the `fgrep` command. It then uses this `fgrep` command file to find references. Using the `.ig` file is simpler but slower than using the `.i[abc]` files, and does not allow the use of multiple reference files.

**Flags**

- `-n` Turns off the prompt for instructions.

**Files**

- `Database.ia` Contains the entry file.
- `Database.ib` Contains the posting file.
- `Database.ic` Contains the tag file.
- `Database.ig` Contains the output file.

**Related Information**

The `addbib` command, `indxbib` command, `refer` command, `roffbib` command, `sortbib` command.

### Iorder Command

**Purpose**

Finds the best order for member files in an object library.

**Syntax**

```
Iorder [-X {32|64|32_64}] File ...
```

**Description**

The `Iorder` command reads one or more object or library archive files, looking for external references and writing a list of paired file names to standard output. The first pair of files contains references to identifiers that are defined in the second file.

If object files do not end with `.o`, the `Iorder` command overlooks them and attributes their global symbols and references to some other file.

**Flags**

- `-X mode` Specifies the type of object file `Iorder` should examine. The `mode` must be one of the following:
  - `32` Processes only 32-bit object files
  - `64` Processes only 64-bit object files
  - `32_64` Processes both 32-bit and 64-bit object files

  The default is to process 32-bit object files (ignore 64-bit objects). The `mode` can also be set with the `OBJECT_MODE` environment variable. For example, `OBJECT_MODE=64` causes `Iorder` to process any 64-bit objects and ignore 32-bit objects. The `-X` flag overrides the `OBJECT_MODE` variable.
Files

/tmp/sym* Contains temporary files.

Related Information
The **ar** command, **ld** command, **tsort** command, **xargs** command.

The **ar** file.

Subroutines Overview in AIX 5L Version 5.2 General Programming Concepts.

Ip Command

The **Ip** command includes information for the AIX Print Subsystem **lp** and the System V Print Subsystem **lp**

AIX Print Subsystem Ip Command

Purpose
Sends requests to a line printer.

Syntax
```
Ip [ [-c] [-d Queue] [-m] [-n Number] [-o Option] [-s] [-t Title] [-w] ] [ Files ]
```

Description
The **Ip** command arranges for the files specified by the **Files** parameter and their associated information (called a request) to be printed by a line printer. If you do not specify a value for the **Files** parameter, the **Ip** command accepts standard input. The file name - (dash) represents standard input and can be specified on the command line in addition to files. The **Ip** command sends the requests in the order specified. If the job is submitted to a local print queue, the **Ip** command displays the following to standard output:

```
Job number is: nnn
```

where **nnn** is the assigned job number. To suppress the job number use the **-s** flag.

Flags

- **-c** Copies the files to be printed immediately when the **Ip** command is run. The **Ip** command copies files only when requested. No links are created. If you specify the **-c** flag, be careful not to remove any of the files before they are printed. If you do not specify the **-c** flag, changes made to the files after the request is made appear in the printed output.

- **-d Queue** Specifies the print queue to which a job is sent.

- **-m** Sends mail (see the **mail** command) after the files are printed. By default, no mail is sent upon normal completion of the print request.

- **-n Number** Prints the number of copies of printed output. The default number of copies is 1.

- **-o Options** Specifies that flags specific to the backend be passed to the backend. Thus, for each queue, other flags not described in this article can be included with the **Ip** command. See the **piobe** command for a list of these flags. Specifying this flag is the same as specifying the **-o** flag for the **piobe** command.

- **-s** Suppresses the automatic return of job numbers. The **Ip** command reports the job number as the default, the **-s** flag overrides the default.

- **-t Title** Specifies printing the title of the file on the banner page of the output.
-w  Writes a message on the print requesters terminal after the files are printed. If the requester is not logged in, the mail command sends the message. If the user is logged in on multiple windows or terminals, the message may not be sent to the LFT where the command was issued. The message is sent to the first terminal on which the writesrv daemon sees the user to be logged in.

Note: If the -w flag is used in conjunction with the -m flag, the print requester will only receive mail and will not get a message on the terminal.

Examples
1. To print the /etc/motd file on printer lp0 attached to device dlp0, enter:
   lp /etc/motd
2. To print 30 copies of the /etc/motd file using a copy of the file, and to notify the user that the job is completed using mail, enter:
   lp -c -m -n30 -dlp0:lpd0 /etc/motd
3. To print the /etc/motd file using backend flags -f and -a, with a job title of blah, enter:
   lp -t "blah" -o -f -o -a /etc/motd
4. To queue the MyFile file and return the job number, enter:
   lp myfile
5. To queue the MyFile file and suppress the job number, enter:
   lp -s myfile

Exit Status
This command returns the following exit values:
0  All input files processed successfully.
>0  No output device is available, or an error occurred.

Files
/usr/sbin/qdaemon  Contains the queuing daemon.
/var/spool/lpd/qdir/*  Contains the queue requests.
/var/spool/lpd/stat/*  Contains information on the status of the devices.
/var/spool/qdaemon/*  Contains temporary copies of enqueued files.
/etc/qconfig  Contains the queue configuration file.
/etc/qconfig.bin  Contains digested, binary version of the /etc/qconfig file.

Related Information
The cancel command, enable command, lpr command, lpstat command, mail command.
The writesrv daemon.
The /etc/qconfig file.

Printer Overview for System Management in the AIX 5L Version 5.2 Guide to Printers and Printing.
System V Print Subsystem Ip Command

Purpose
Sends print requests

Syntax
Ip [print-options] [files]

Ip -i request-ID print-options

Description
The first form of the Ip command arranges for the named files and associated information (collectively called a request) to be printed. If filenames are not specified on the command line, the standard input is assumed. The standard input may be specified along with named files on the command line by listing the filenames and specifying - for the standard input. The files will be printed in the order in which they appear on the command line.

The LP print service associates a unique request-ID with each request and displays it on the standard output. This request-ID can be used later when canceling or changing a request, or when determining its status. See the cancel command for details about canceling a request, and Ipstat for information about checking the status of a print request.

The second form of Ip is used to change the options for a request submitted previously. The print request identified by the request-ID is changed according to the print-options specified with this command. The print-options available are the same as those with the first form of the Ip command. If the request has finished printing, the change is rejected. If the request is already printing, it will be stopped and restarted from the beginning (unless the -P flag has been given).

Sending a print request
The first form of the Ip command is used to send a print request either to a particular printer or to any printer capable of meeting all requirements of the print request.

Flags must always precede filenames, but may be specified in any order.

Printers for which requests are not being accepted will not be considered when the destination is any. (Use the Ipstat -a command to see which printers are accepting requests.) However, if a request is destined for a class of printers and the class itself is accepting requests, then all printers in the class will be considered, regardless of their acceptance status.

For printers that take mountable print wheels or font cartridges, if you do not specify a particular print wheel or font with the -S flag, whichever one happens to be mounted at the time your request is printed will be used. The Ipstat -p printer -i command is used to see which print wheels are available on a particular printer. The Ipstat -S -I command is used to see what print wheels are available and on which printers. Without the -S flag, the standard character set is used for printers that have selectable character sets.

If you experience problems with jobs that usually print but on occasion do not print, check the physical connections between the printer and your computer. If you are using an automatic data switch or an A/B switch, try removing it and see if the problem clears.
Flags

-\c 
Make copies of the files to be printed immediately when \texttt{lp} is invoked. Normally files will not be copied, but will be linked whenever possible. If the -c flag is not specified, the user should be careful not to remove any of the files before the request has been printed in its entirety. It should also be noted that if the -c flag is not specified, any changes made to the named files after the request is made but before it is printed will be reflected in the printed output.

-d dest
Choose dest as the printer or class of printers that is to do the printing. If dest is a printer, then the request will be printed only on that specific printer. If dest is a class of printers, then the request will be printed on the first available printer that is a member of the class. If dest is any, then the request will be printed on any printer that can handle it. Under certain conditions (unavailability of printers, file space limitations, and so on) requests for specific destinations may not be accepted (see \texttt{lpstat}). By default, dest is taken from the environment variable \texttt{LPDEST}. If \texttt{LPDEST} is not set, then dest is taken from the environment variable \texttt{PRINTER}. If \texttt{PRINTER} is not set, a default destination (if one exists) for the computer system is used. If no system default is set and -T is used, dest will be selected on the basis of content-type specified with the -T flag [see the description of -T]. Destination names vary between systems (see \texttt{lpstat}).

-f form-name [-d any]
Print the request on the form form-name. The LP print service ensures that the form is mounted on the printer. If form-name is requested with a printer destination that cannot support the form, the request is rejected. If form-name has not been defined for the system, or if the user is not allowed to use the form, the request is rejected. (see \texttt{pforms}). When the -d any flag is given, the request is printed on any printer that has the requested form mounted and can handle all other needs of the print request.

-H special-handling
Print the request according to the value of special-handling. Acceptable values for special-handling are defined below:

\texttt{hold}  
Do not print the request until notified. If printing has already begun, stop it. Other print requests will go ahead of a held request until it is resumed.

\texttt{resume}  
Resume a held request. If it had been printing when held, it will be the next request printed, unless subsequently bumped by an immediate request. The -i flag (followed by a request-ID) must be used whenever this argument is specified.

\texttt{immediate}  
(Available only to LP administrators) Print the request next. If more than one request is assigned immediate, the most recent request will be printed first. If another request is currently printing, it must be put on hold to allow this immediate request to print.

-L locale-name
Specify locale-name as the locale to use with this print request. By default, locale-name is set to the value of \texttt{LC_CTYPE}. If \texttt{LC_CTYPE} is not set, locale-name defaults to the C locale.

-m
Send mail after the files have been printed. By default, mail is not sent upon normal completion of the print request.

-n number
Print number copies of the output. The default is one copy.

-o options
Specify printer-dependent options. Several such options may be collected by specifying the -o keyletter more than once (that is, -o option[1] -o option[2] ... -o option[n]), or by specifying a list of options with one -o keyletter enclosed in double quotes and separated by spaces (that is, -o "option[1] option[2] ... option[n]").
nobanner
Do not print a banner page with this request. The administrator can disallow this option at any time.

nofilebreak
Do not insert a form feed between the files given, if submitting a job to print more than one file. This option is not supported by printers configured to use the PS (PostScript) interface.

length=scaled-decimal-number
Print this request with pages scaled-decimal-number long. A scaled-decimal-number is an optionally scaled decimal number that gives a size in lines, characters, inches, or centimeters, as appropriate. The scale is indicated by appending the letter l for inches, or the letter c for centimeters. For length or width settings, an unscaled number indicates lines or characters; for line pitch or character pitch settings, an unscaled number indicates lines per inch or characters per inch (the same as a number scaled with l). For example, length=66 indicates a page length of 66 lines, length=11i indicates a page length of 11 inches, and length=27.94c indicates a page length of 27.94 centimeters. This option may not be used with the -f option and is not supported by the PS (PostScript).

width=scaled-decimal-number
Print this request with pages scaled-decimal-number wide. (See the explanation of scaled-decimal-numbers in the discussion of length, above.) This option may not be used with the -f option and is not supported by the PS (PostScript).

lpi=scaled-decimal-number
Print this request with the line pitch set to scaled-decimal-number. (See the explanation of scaled-decimal-numbers in the discussion of length, above.) This option may not be used with the -f flag and is not supported by the PS (PostScript).

cpi=pica|elite|compressed
Print this request with the character pitch set to pica (representing 10 characters per inch), elite (representing 12 characters per inch), or compressed (representing as many characters per inch as a printer can handle). There is not a standard number of characters per inch for all printers; see the [terminfo] database for the default character pitch for your printer. This option may not be used with the -f flag and is not supported by the PS (PostScript).

stty=stty-option-list
A list of options valid for the stty command; enclose the list with single quotes if it contains blanks.

-P page-list
Print the pages specified in page-list. This flag can be used only if there is a filter available to handle it; otherwise, the print request will be rejected. The page-list may consist of ranges of numbers, single page numbers, or a combination of both. The pages will be printed in ascending order.

-q priority-level
Assign this request priority-level in the printing queue. The values of priority-level range from 0 (highest priority) to 39 (lowest priority). If a priority is not specified, the default for the print service is used, as assigned by the system administrator. A priority limit may be assigned to individual users by the system administrator.

-R
Remove file(s) after submitting the print request. Use this flag with caution.

-r
See -T content-type [-r] below.

-s
Suppress the “request id is ...” message.
-S character-set [-d any]

-S print-wheel [-d any]
Print this request using the specified character-set or print-wheel. If a form was requested and it requires a character set or print wheel other than the one specified with the -S flag, the request is rejected.

For printers that take print wheels: if the print wheel specified is not one listed by the administrator as acceptable for the printer specified in this request, the request is rejected unless the print wheel is already mounted on the printer.

For printers that use selectable or programmable character sets: if the character-set specified is not one defined in the Terminfo database for the printer (see [terminfo](#)), or is not an alias defined by the administrator, the request is rejected.

When the -d any flag is used, the request is printed on any printer that has the print wheel mounted or any printer that can select the character set, and that can handle all other needs of the request.

-t title Print title on the banner page of the output. The default is no title. Enclose title in quotes if it contains blanks.

-T content-type [-r]
Print the request on a printer that can support the specified content-type. If no printer accepts this type directly, a filter will be used to convert the content into an acceptable type. If the -r flag is specified, a filter will not be used. If -r is specified but no printer accepts the content-type directly, the request is rejected. If the content-type is not acceptable to any printer, either directly or with a filter, the request is rejected.

In addition to ensuring that no filters will be used, the -r flag will force the equivalent of the -o 'stty=opost' flag.

-w Write a message on the user’s terminal after the files have been printed. If the user is not logged in, or if the printer resides on a remote system, then mail will be sent instead. Be aware that messages may be sent to a window other than the one in which the command was originally entered.

-y mode-list
Print this request according to the printing modes listed in mode-list. The allowed values for mode-list are locally defined. This option may be used only if there is a filter available to handle it; otherwise, the print request will be rejected.

The following list describes the mode-list options:

-y reverse
Reverse the order in which pages are printed. This filter option is not supported by the LP Print Service.

-y landscape
Change the orientation of a physical page from portrait to landscape.

-y x=number,y=number
Change the default position of a logical page on a physical page by moving the origin.

-y group=number
Group multiple logical pages on a single physical page.

-y magnify=number
Change the logical size of each page in a document.

-o length=number
Select the number of lines in each page of the document.
-P number
Select, by page numbers, a subset of a document to be printed.

-n number
Print multiple copies of a document.

Files
/var/spool/lp/*

Related Information
The `lpstat` command, and the `mail` command.

**lp.cat, lp.set, lp.tell Command**

*Note*: This is a System V Print Subsystem command.

**Purpose**
Default printer filter used by interface programs.

**Syntax**
/usr/lib/lp/bin/lp.cat [-r] [ Rate ]

/usr/lib/lp/bin/lp.set H_pitch V_pitch Width Length Charset

/usr/lib/lp/bin/lp.tell Printer

**Description**
The `lp.cat` command is the default printer filter called by the interface programs. `lp.cat` reads the file to be printed on its standard input and writes it to the device to be printed on.

`lp.cat` handles the following signals:
- normal termination (SIGTERM)
- serial line hangup (SIGHUP due to DCD, Data Carrier Detect, dropping)
- interrupts (SIGINT and SIGQUIT)
- early pipe termination (SIGPIPE)

The `lp.cat` command aborts a printing job if it has to wait too long on output. The default maximum time to wait is calculated as twice the output buffer size (2 * 1024 bytes) divided by the smallest of the values of the transmission rate, print rate, or the specified `Rate` (all rates are in characters per second, CPS). A new maximum delay time may be defined by the `Rate` parameter provided that this increases the delay time. If `Rate` is 0 then the delay allowed is effectively infinite.

When the `-r` flag is specified, `lp.cat` reports the effective throughput in characters per second. This statistic is printed on the standard error after the transmission of every 1024 bytes.

Interface programs may call the `lp.set` command to set the character pitch (`H_pitch`), line pitch (`V_pitch`), page `Width`, page `Length`, and character set (`Charset`) on the printer.

The default units are characters per inch for the character and line pitch, number of columns for width, and number of lines for length. Units may be stated explicitly by appending the values with `c` for centimeters, and `i` for inches.
If it cannot set a particular characteristic, **lp.set** exits with a value of 1 and prints a one letter character code followed by newline to the standard error. The character codes are shown in the following table:

<table>
<thead>
<tr>
<th>Character code</th>
<th>Printer characteristic not set</th>
</tr>
</thead>
<tbody>
<tr>
<td>H</td>
<td>character pitch</td>
</tr>
<tr>
<td>L</td>
<td>page length</td>
</tr>
<tr>
<td>S</td>
<td>character set</td>
</tr>
<tr>
<td>V</td>
<td>line pitch</td>
</tr>
<tr>
<td>W</td>
<td>page width</td>
</tr>
</tbody>
</table>

Interface scripts use **lp.tell** to forward descriptions of printer faults to the print service. **lp.tell** sends everything that it reads on its standard input to the print service. The print service forwards the message as an alert to the print administrator.

**Flags**
- **-r** Specifies reports the effective throughput in characters per second for the **lp.cat** command. This statistic is printed on the standard error after the transmission of every 1024 bytes.

**Exit Status**
The following table shows the possible exit values for **lp.cat**:

<table>
<thead>
<tr>
<th>Exit status</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Normal exit; no error</td>
</tr>
<tr>
<td>1</td>
<td>Standard input not defined</td>
</tr>
<tr>
<td>2</td>
<td>Standard output not defined</td>
</tr>
<tr>
<td>3</td>
<td>Printer type (TERM) not defined or not in terminfo database</td>
</tr>
<tr>
<td>4</td>
<td>Standard input and output are identical</td>
</tr>
<tr>
<td>5</td>
<td>Write failed; printer may be off-line</td>
</tr>
<tr>
<td>6</td>
<td>Excessive delay caused timeout; printer may be off-line</td>
</tr>
<tr>
<td>7</td>
<td>Hangup (SIGHUP) detected; loss of carrier</td>
</tr>
<tr>
<td>8</td>
<td>Termination (SIGINT or SIGQUIT), or pipe closed early (SIGPIPE)</td>
</tr>
</tbody>
</table>

**lp.set** returns 0 if successful; otherwise it returns 1 on error.

**lp.tell** returns:

<table>
<thead>
<tr>
<th>Exit status</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Normal exit; no error</td>
</tr>
<tr>
<td>1</td>
<td>Cannot open message queue to the print service</td>
</tr>
<tr>
<td>90</td>
<td>Cannot get printer name or key from print service</td>
</tr>
<tr>
<td>91</td>
<td>Cannot send message to print service</td>
</tr>
<tr>
<td>92</td>
<td>Cannot receive acknowledgment from print service</td>
</tr>
<tr>
<td>93</td>
<td>Corrupted acknowledgment received from print service</td>
</tr>
<tr>
<td>94</td>
<td>Print service reports message corrupted in transmission</td>
</tr>
</tbody>
</table>

**Files**

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/lib/lp/bin/lp.cat</td>
<td>Full pathname of <strong>lp.cat</strong></td>
</tr>
<tr>
<td>/usr/lib/lp/bin/lp.set</td>
<td>Full pathname of <strong>lp.set</strong></td>
</tr>
<tr>
<td>/usr/lib/lp/bin/lp.tell</td>
<td>Full pathname of <strong>lp.tell</strong></td>
</tr>
<tr>
<td>/etc/lp/model</td>
<td>Printer interface programs directory</td>
</tr>
<tr>
<td>/etc/lp/interfaces</td>
<td>Printer interface programs directory</td>
</tr>
</tbody>
</table>
References
The `lp` command.
The `signal` subroutine.
The `terminfo` file.

Ipadmin Command

Note: This is a System V Print Subsystem command.

Purpose
Configures the LP print service.

Syntax
Adding or Changing the Configuration of a Local Printer

```
```

Adding or Changing the Configuration of a Remote Printer

```
```

Removing a Printer Destination

```
Ipadmin [-x] Destination
```

Setting or Changing the System Default Destination

```
Ipadmin [-d] [ Destination ]
```

Setting an Alert for a Print Wheel

```
Ipadmin [-S Print-Wheel] [-A AlertType [-W Minutes] [-Q Requests]
```

Description
The Ipadmin command configures the LP print service by defining printers and devices. It is used to:
- Add and change printers
- Remove printers from the service
- Set or change the system default destination
- Define alerts for printer faults
- Mount print wheels
- Define printers for remote printing services
Printer and class names may be no longer than the maximum length filename allowed for the filesystem type you are using, and may consist of all printable characters except the space, slash, backslash, colon, semicolon, comma, asterisk, question mark, and tilde. The dash can be used in any position except the first position in a printer name.

For additional information about network printers, see the `psystem` command.

**Adding or changing a printer**
The `p Printer` flag is used to configure a new printer or to change the configuration of an existing printer. When you use this form of the `ipadmin` command, you must select one of the following:

- `V Device`, required to configure a local printer
- `S ServerName [ !ServerPrinterName ]`, required to configure a remote printer

**Removing a printer destination**
The `-x dest` flag removes the destination `dest` (a printer or a class), from the LP print service. If `dest` is a printer and is the only member of a class, then the class is deleted. If `dest` is `all`, all printers and classes are removed. No other parameters are allowed with `-x`.

**Setting/changing the system default destination**
The `-d [ dest ]` flag makes `dest`, an existing printer or class, the new system default destination. If `dest` is not supplied, then there is no system default destination. No other parameters are allowed with `-d`. To unset the system default printer, the user can enter the keyword `none`.

**Setting an alert for a print wheel**
The `-p Print-Wheel` flag is used with the `-A Alert-Type` flag to define an alert to mount the print wheel when there are jobs queued for it. If this command is not used to arrange alerting for a print wheel, no alert will be sent for the print wheel. See the other use of `-A` flag, with `-p`.

The `Alert-Types` are the same as those available with the `-A` flag: `mail`, `write`, `quiet`, `none`, `shell-command`, and `list`. See the description of `-A`, for details about each.

The message sent appears as follows:

```
The print wheel Print-Wheel needs to be mounted
on the printer(s):
printer (integer1 requests)
integer2 print requests await this print wheel.
```

The printers listed are those that the administrator had earlier specified were candidates for this print wheel. The number `integer1` listed next to each printer is the number of requests eligible for the printer. The number `integer2` shown after the printer list is the total number of requests awaiting the print wheel. It will be less than the sum of the other numbers if some requests can be handled by more than one printer.

If the `Print-Wheel` is `all`, the alerting defined in this command applies to all print wheels already defined to have an alert.

If the `-W` flag is not given, the default procedure is that only one message will be sent per need to mount the print wheel. Not specifying the `-W` flag is equivalent to specifying `-W once` or `-W 0`. If `Minutes` is a number greater than zero, an alert is sent at intervals specified by minutes.

If the `-Q` flag is also given, the alert is sent when a certain number (specified by the argument `requests`) of print requests that need the print wheel are waiting. If the `-Q` flag is not given, or `requests` is 1 or the word `any` (which are both the default), a message is sent as soon as anyone submits a print request for the print wheel when it is not mounted.

**Flags**

**Notes:**
When creating a new printer, you must specify the -v, or -s flag. In addition, only one of the following may be supplied: -e, -i, or -m; if none of these three flags is supplied, the model standard is used.

If -s and/or -R flags are specified, the following flags are not valid: -A, -e, -F, -h, -i, -I, -M, -m, -o, -v, and -W.

- **A AlertType [-W minutes]**  
  The -A flag defines an alert to inform the administrator when a printer fault is detected, and periodically thereafter, until the printer fault is cleared by the administrator. If an alert is not defined for a particular printer, mail is sent to user lp by default. The AlertTypes are:

  - mail  
    Send the alert message via mail (see [mail](#)) to the administrator.
  - write  
    Write the message to the terminal on which the administrator is logged in. If the administrator is logged in on several terminals, one is chosen arbitrarily.
  - quiet  
    Do not send messages for the current condition. An administrator can use this option to temporarily stop receiving further messages about a known problem. Once the fault has been cleared and printing resumes, messages will again be sent when another fault occurs with the printer.
  - none  
    Do not send messages; any existing alert definition for the printer is removed. No alert is sent when the printer faults until a different alert-type (except quiet) is used.

- **shell-command**  
  Run the shell-command each time the alert needs to be sent. The shell command should expect the message in standard input. If there are blanks embedded in the command, enclose the command in quotes. The mail and write values for this option are equivalent to the values mail login-ID and write login-ID respectively, where login-ID is the current name for the administrator. This will be the login ID of the person submitting this command unless he or she has used the su command to change to another login ID. If the su command has been used to change the login ID, then the login-ID for the new login is used.

- **list**  
  Display the type of the alert for the printer fault. No change is made to the alert.

The message sent appears as follows:

The printer Printer has stopped printing for the reason given below. Fix the problem and bring the printer back on line.

Printing has stopped, but will be restarted in a few minutes; issue an enable command if you want to restart sooner.

Unless someone issues a change request

```
lp -i request-id -P . . .
```

to change the page list to print, the current request will be reprinted from the beginning.

The reason(s) it stopped (multiple reasons indicate reprinted attempts):

- **reason**  
  The LP print service can detect printer faults only through an adequate fast filter and only when the standard interface program or a suitable customized interface program is used. Furthermore, the level of recovery after a fault depends on the capabilities of the filter.

If the Printer is all, the alerting defined in this command applies to all existing printers. If the -W flag is not used to arrange fault alerting for Printer, the default procedure is to mail one message to the administrator of Printer per fault. This is equivalent to specifying -W once or -W 0. If minutes is a number greater than zero, an alert is sent at intervals specified by minutes.

- **c Class**  
  Insert Printer into the specified Class. Class is created if it does not already exist.

- **d [ Dest ]**  
  Makes dest, an existing printer or class, the new system default destination.

- **D Comment**  
  Saves the Comment for display whenever a user asks for a full description of Printer (see [lpstat](#)). The LP print service does not interpret this comment.
-e *Printer1*  Copies the interface program of an existing *Printer1* to be the interface program for *Printer*.

**Note:** Do not specify the -i and -m flags may not be specified with the -e flag.

-F *FaultRecovery*  Specifies the recovery to be used for any print request that is stopped because of a printer fault, according to the value of *FaultRecovery*:

*continue*  Continue printing on the top of the page where printing stopped. This requires a filter to wait for the fault to clear before automatically continuing.

*beginning*  Start printing the request again from the beginning.

*wait*  Disable printing on *Printer* and wait for the administrator or a user to enable printing again.

During the wait the administrator or the user who submitted the stopped print request can issue a change request that specifies where printing should resume. (See the -i flag of the lpd command.) If no change request is made before printing is enabled, printing will resume at the top of the page where stopped, if the filter allows; otherwise, the request will be printed from the beginning.

The default value of *FaultRecovery* is *beginning*.

- *f allow: FormList | -f deny: FormList*  Allows or denies the forms in *FormList* to be printed on *Printer*. By default no forms are allowed on a new printer.

For each printer, the LP print service keeps two lists of forms: an *allow-list* of forms that may be used with the printer, and a *deny-list* of forms that may not be used with the printer. With the *-f allow* flag, the forms listed are added to the allow-list and removed from the deny-list.

With the *-f deny* flag, the forms listed are added to the deny-list and removed from the allow-list.

If the allow-list is not empty, only the forms in the list may be used on the printer, regardless of the contents of the deny-list. If the allow-list is empty, but the deny-list is not, the forms in the deny-list may not be used with the printer. All forms can be excluded from a printer by specifying *-f deny: all*. All forms can be used on a printer (provided the printer can handle all the characteristics of each form) by specifying *-f allow: all*.

The LP print service uses this information as a set of guidelines for determining where a form can be mounted. Administrators, however, are not restricted from mounting a form on any printer. If mounting a form on a particular printer is in disagreement with the information in the allow-list or deny-list, the administrator is warned but the mount is accepted. Nonetheless, if a user attempts to issue a print or change request for a form and printer combination that is in disagreement with the information, the request is accepted only if the form is currently mounted on the printer. If the form is later unmounted before the request can print, the request is canceled and the user is notified by mail.

If the administrator tries to specify a form as acceptable for use on a printer that doesn’t have the capabilities needed by the form, the command is rejected.

The *lpadmin* command issues a warning when an invalid (nonexistent) form name is submitted with the *-f deny*: flag.

See the other use of *-f*, with the *M* flag.

-h  Indicates that the device associated with the printer is hardwired. If neither of the mutually exclusive flags, -h and -l, is specified, this flag is assumed.
-l Content-Type-List

Allow Printer to handle print requests with the content types listed in a Content-Type-List. If the list includes names of more than one type, the names must be separated by commas or blank spaces. If they are separated by blank spaces, the entire list must be enclosed in double quotes.

The type simple is recognized as the default content type. A simple type of file is a data stream containing only printable ASCII characters and the following control characters.

<table>
<thead>
<tr>
<th>Control Character</th>
<th>Octal Value</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>backspace</td>
<td>10</td>
<td>move back one character, except at beginning of line</td>
</tr>
<tr>
<td>tab</td>
<td>11</td>
<td>move to next tab stop</td>
</tr>
<tr>
<td>linefeed (newline)</td>
<td>12</td>
<td>move to beginning of next line</td>
</tr>
<tr>
<td>form feed</td>
<td>14</td>
<td>move to beginning of next page</td>
</tr>
<tr>
<td>carriage return</td>
<td>15</td>
<td>move to beginning of current line</td>
</tr>
</tbody>
</table>

To prevent the print service from considering simple a valid type for the printer, specify either an explicit value (such as the printer type) in the content-type-list, or an empty list. If you do want simple included along with other types, you must include simple in the content-type-list.

Except for simple, each content-type name is freely determined by the administrator. If the printer type is specified by the -T option, then the printer type is implicitly considered to be also a valid content type.

-i Interface

Establish a new interface program for Printer. The Interface is the pathname of the new program. Do not specify the -e and -m flags with this flag.

-l

Indicates that the device associated with Printer is a login terminal. The LP scheduler (lpsched) disables all login terminals automatically each time it is started. The -h flag may not be specified with this flag.

-M -f Form-Name [-a [-o filebreak]]

Mounts the form Form-Name on Printer. Print requests that need the pre-printed form Form-Name are printed on Printer. If more than one printer has the form mounted and the user has specified any with the -d flag of the lp command as the printer destination, then the print request is printed on the one printer that also meets the other needs of the request.

The page length and width, and character and line pitches needed by the form are compared with those allowed for the printer, by checking the capabilities in the terminfo database for the type of printer. If the form requires attributes that are not available with the printer, the administrator is warned but the mount is accepted. If the form lists a print wheel as mandatory, but the print wheel mounted on the printer is different, the administrator is also warned but the mount is accepted.

If the -a flag is given, an alignment pattern is printed, preceded by the same initialization of the physical printer that precedes a normal print request. Printing is assumed to start at the top of the first page of the form. After the pattern is printed, the administrator can adjust the mounted form in the printer and press return for another alignment pattern (no initialization this time), and can continue printing as many alignment patterns as desired. The administrator can quit the printing of alignment patterns by typing q.

If the -o filebreak flag is given, a form feed is inserted between each copy of the alignment pattern. By default, the alignment pattern is assumed to correctly fill a form, so no form feed is added.

A form is unmounted either by mounting a new form in its place or by using the -f none flag. By default, a new printer has no form mounted.

See the other use of -f without the -M.

-M -S Print-Wheel

Mount the Print-Wheel on Printer. Print requests that need the Print-Wheel will be printed on Printer. If more than one printer has Print-Wheel mounted and the user has specified any with the -d flag of the lp command as the printer destination, then the print request is printed on the one printer that also meets the other needs of the request.

If the Print-Wheel is not listed as acceptable for the printer, the administrator is warned but the mount is accepted. If the printer does not take print wheels, the command is rejected.

A print wheel is unmounted either by mounting a new print wheel in its place or by using the -S none flag. By default, a new printer has no print wheel mounted.

See the other uses of the -S flag without the -M.
**-m Model**
Select Model interface program, provided with the LP print service, for the printer. Do not use the -e and -i flags with this flag. The following interface programs are available:

**standard**
generic printer interface

**PS** interface for PostScript printers only
By default, the **standard** interface is used.

**-O Copy-Option**
The -O controls whether or not **lp** makes a copy of the user's file(s) when a print job is submitted. The copy-option can be either **copy** or **nocopy**. If -O copy is specified, the LP system always copies the user's source files to the spool area when a print job is submitted. If -O nocopy is specified, the files are copied only if the user specifies the -c flag of **lp** when submitting the job.

This flag sets the value of the **copy-files** parameter in the /etc/default/lp file. The value, which can be either on or off, is checked every time a print job is submitted.

**-o Printing-Option**
Specifies the, in the list below, the default given to an interface program if the option is not taken from a preprinted form description or is not explicitly given by the user submitting a request (see **lp**). The only -o options that can have defaults defined are listed below.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>length</td>
<td>scaled-decimal-number</td>
</tr>
<tr>
<td>width</td>
<td>scaled-decimal-number</td>
</tr>
<tr>
<td>cpi</td>
<td>scaled-decimal-number</td>
</tr>
<tr>
<td>lpi</td>
<td>scaled-decimal-number</td>
</tr>
<tr>
<td>stty</td>
<td>'stty-option-list'</td>
</tr>
</tbody>
</table>

**scaled-decimal-number** refers to a non-negative number used to indicate a unit of size. The type of unit is shown by a trailing letter attached to the number. Three types of scaled decimal numbers can be used with the LP print service:

- numbers that show sizes in centimeters, marked with a trailing c
- numbers that show sizes in inches, marked with a trailing i
- numbers that show sizes in units appropriate to use, without a trailing letter that is, lines, characters, lines per inch, or characters per inch.

The first four default option values must agree with the capabilities of the type of physical printer, as defined in the **terminfo** database for the printer type. If they do not, the command is rejected.

The **stty-option-list** is not checked for allowed values, but is passed directly to the **stty** program by the standard interface program. Any error messages produced by **stty** when a request is processed (by the standard interface program) are mailed to the user submitting the request.

For each printing option not specified, the defaults for the following attributes are defined in the **terminfo** entry for the specified printer type.

- **length**
- **width**
- **cpi**
- **lpi**

The default for **stty** is

```
stty=’9600 cs8 -cstopb -parenbixon
-ixany opost -olcuc onlcr -ocrel -onocr
-onlret -ofill nl0 cr0 tab0 bs0 vt0 ff0’
```

You can set any of the -o flags to the default values (which vary for different types of printers), by typing them without assigned values, as follows:

<table>
<thead>
<tr>
<th>Flag</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-o</td>
<td>nobanner</td>
</tr>
<tr>
<td>-o</td>
<td>banner</td>
</tr>
</tbody>
</table>

**-o nobanner** Allows a user to submit a print request specifying that no banner page be printed.

**-o banner** Forces a banner page to be printed with every print request, even when a user asks for no banner page. This is the default; you must specify -o nobanner if you want to allow users to be able to specify -o nobanner with the **lp** command.
-p Printer
Configures a new printer changes the configuration of an existing printer.

-Q Requests
Specifies that an alert be sent when a certain number of print Requests that need the print wheel are waiting.

-r Class
Remove Printer from the specified Class. If Printer is the last member of Class, then Class is removed.

-S List
Allows either the print wheels or aliases for character sets named in List to be used on the printer. The -S flag does not let you add items to a List specified with an earlier invocation of -S; instead, it replaces an existing List with a new one. Thus -S differs from the -f, -u, allow, and deny options, which allow you to modify existing lists of available forms and authorized users. Once you’ve run the -S flag, the print wheels and character sets specified, in List, on the current command line are the only ones available.

If the printer is a type that takes print wheels, then List is a comma or space separated list of print wheel names. Enclose the list with quotes if it contains blanks. These are the only print wheels considered mountable on the printer. You can always force a different print wheel to be mounted, however. Until the flag is used to specify a list, no print wheels is considered mountable on the printer, and print requests that ask for a particular print wheel with this printer is rejected.

If the printer is a type that has selectable character sets, then List is a comma or blank separated list of character set name mappings or aliases. Enclose the list with quotes if it contains blanks. Each mapping is of the form:

known-name=alias

The known-name is a character set number preceded by cs, such as cs3 for character set three, or a character set name from the Terminfo database entry csnm. See [terminfo]. If this flag is not used to specify a list, only the names already known from the Terminfo database or numbers with a prefix of cs are acceptable for the printer.

If List is the word none, any existing print wheel lists or character set aliases is removed.

See the other uses of the -S with the -M flag.

-s Server-Name [ !Server-Printer-Name ]
Specifies that you are configuring a remote printer. It makes a server printer accessible to users on your system. Server-Name is the name of the system on which the printer is located.

It must be listed in the LP systems table. See [ipsystem]. Server-Printer-Name is the name used on the server system for that printer. For example, if you want to access Printer1 on Server1 and you want it called Printer2 on your system, enter -p Printer2 -s Server1!Printer1.

If Server-Name is a Netware server, defined as -t nuc using the ipsystem command, then Server-Printer-Name can be the name of a Netware queue or Netware printer.

-T Printer-Type-List
Identify the printer as being of one or more Printer-Types. Each Printer-Type is used to extract data from the terminfo database; this information is used to initialize the printer before printing each user’s request. Some filters may also use a Printer-Type to convert content for the printer.

If this flag is not used, the default Printer-Type is unknown; no information will be extracted from terminfo so each user request is printed without first initializing the printer. Also, this flag must be used if the following are to work: -o cpi, -o lpi, -o width, and -o length flags of the Ipadmin and Ip commands, and the -S and -f flags of the Ipadmin command.

If the Printer-Type-List contains more than one type, then the content-type-list of the -I option must either be specified as simple, as empty (-I ""), or not specified at all.

-u allow:Login-ID-List
-u deny: Login-ID-List  Allows or denies the users in Login-ID-List access to the printer. By default all users on the local system are allowed on a new printer. The Login-ID-List parameter may include any or all of the following constructs:

  login-ID
  a user on the local system

  system-name!login-ID
  a user on system system-name

  system-name!all
  all users on system system-name

  all!login-ID
  a user on all systems

  all
  all users on the local system

  all!all
  all users on all systems

For each printer the LP print service keeps two lists of users:

- An allow-list of people allowed to use the printer
- A deny-list of people denied access to the printer.

With the -u allow flag, the users listed are added to the allow-list and removed from the deny-list. With the -u deny flag, the users listed are added to the deny-list and removed from the allow-list.

If the allow-list is not empty, only the users in the list may use the printer, regardless of the contents of the deny-list. If the allow-list is empty, but the deny-list is not, the users in the deny-list may not use the printer. All users can be denied access to the printer by specifying -u deny:all. All users may use the printer by specifying -u allow:all.

-v Device  Specifies you are configuring a local printer. It associates a Device with Printer. Device is the pathname of a file that is writable by lp. The same Device can be associated with more than one printer.

-x Dest  Removes the destination dest (a printer or a class), from the LP print service.

Files
/var/spool/lp/*
/etc/lp

Related Information

Ipc Command

  Note: This is a System V Print Subsystem command.

Purpose
Provides (BSD) line printer control.

Syntax
/usr/ucb/lpc [ Command [ Parameter . . . ] ]
Description

The `lpc` command controls the operation of the printer or of multiple printers. The `lpc` command can be used to start or stop a printer, disable or enable a printer’s spooling queue, rearrange the order of jobs in a queue, or display the status of each printer, along with its spooling queue and printer daemon.

With no parameters, the `lpc` command runs interactively, prompting with `lpc>`. If parameters are supplied, the `lpc` command interprets the first as a Command to execute; each subsequent parameter is taken as a Parameter for that command. The standard input can be redirected so that the `lpc` command reads Commands from a file.

Commands may be abbreviated to an unambiguous substring.

**Note:** The `printer` parameter is specified just by the name of the printer (as `lw`), not as you would specify it to `lpr` or `lpq` (not as `-Plw`).

```
? [ Command. . . ]
help [ Command . . . ]

abort [ all | [ Printer . . . ] ]
        Displays a short description of each command specified in
        the parameter list or, if no parameters are given, a list of
        the recognized commands.

        Terminates an active spooling daemon on the local host
        immediately and then disables printing (preventing new
demons from being started by `lpr`) for the specified
printers. The `abort` command can only be used by a
privileged user.

clean [ all | [ Printer . . . ] ]
        Removes all files created in the spool directory by the
d daemon from the specified printer queues on the local
machine. The `clean` command can only be used by a
privileged user.

disable [ all | [ Printer . . . ] ]
        Turns the specified printer queues off. This prevents new
printer jobs from being entered into the queue by `lpr`. The
disable command can only be used by a privileged user.

down [ all | [ Printer. . ] ] [ Message ]
        Turns the specified printer queue off, disables printing,
and puts `Message` in the printer status file. The message
does not need to be quoted. The remaining parameters
are treated like `echo`. This is normally used to take a
printer down and let others know why `lpq` indicates that
the printer is down, as does the `status` command.

enable [ all | [ Printer. . . ] ]
        Enables spooling on the local queue for the listed printers
so that `lpr` can put new jobs in the spool queue. The
`enable` command can only be used by a privileged user.

exit
        Exits from `lpc`.

quit
        Quits from `lpc`.

restart [ all | [ Printer,... ] ]
        Attempts to start a new printer daemon. This is useful
when some abnormal condition causes the daemon to die
unexpectedly leaving jobs in the queue. This command
can be run by any user.

start [ all | [ Printer... ] ]
        Enables printing and starts a spooling daemon for the
listed printers. The `start` command can only be used by a
privileged user.

status [ all | [ Printer... ] ]
        Displays the status of daemons and queues on the local
machine. This command can be run by any user.

stop [ all | [ Printer... ] ]
        Stops a spooling daemon after the current job completes
and disable printing. The `stop` command can only be used
by a privileged user.

topq Printer [ Job#... ] [ User... ]
        Moves the print jobs specified by `Job#` or those jobs
belonging to `User` to the top (head) of the printer queue.
The `topq` command can only be used by a privileged user.
```
up [ all ] [ Printer... ]

Enables everything and starts a new printer daemon. Undoes the effects of down.

Files
/var/spool/lp/*
/var/spool/lp/system/pstatus

Error Codes

?Ambiguous command
The abbreviation matches more than one command.

?Invalid command
A command or abbreviation is not recognized.

?Privileged command
The command can be executed only by the privileged user.

Ipc: printer: unknown printer to the print service
The printer was not found in the System V LP database. Usually this is a typing mistake; however, it may indicate that the printer does not exist on the system. Use /usr/lib/lpsched to find the reason.

Ipc: error on opening queue to spooler
The connection to lpsched on the local machine failed. This usually means the printer server started at boot time has died or is hung. Check if the printer spooler daemon /usr/lib/lpsched is running.

Ipc: Can't send message to LP print service
Ipc: Can't receive message from LP print service
Ipc: Received unexpected message from LP print service
Indicates that the LP print service has been stopped. Get help from the system administrator.

It is likely there is an error in this software. Get help from system administrator.

Related Information
The lpq command, lpr command, lprm command, lpsched command.

Ipd Command

Purpose
Provides the remote print server on a network.

Syntax
Ipd [-d] [-l] [-D DebugOutputFile]

Description
The Ipd daemon is the remote print server. It monitors port 515 for print requests. Each request is placed in a directory named /var/spool/lpd.

A computer on a network (host) that can create a Transmission Control Protocol/Internet Protocol (TCP/IP) data stream and use the Ipd protocol can print remotely or act as a print server. As a security feature, the Ipd daemon accepts print requests only from remote hosts that are listed in the local /etc/hosts.equiv or /etc/hosts.lpd file.
The `lpd` daemon can run on any host in the network; its function is to accept print requests from foreign hosts (on port 515). The `lpd` daemon handles each request by forking a child process. Remote requests are first checked against the `/etc/host.equiv` and `/etc/host.lpd` files for permission to print on the local host.

Changes can be made to the `/etc/host.equiv` and `/etc/host.lpd` files without restarting the system. To put changes to these files into effect without restarting the system, use the System Resource Controller (SRC) `refresh` command. This command causes the `/etc/host.equiv` and `/etc/host.lpd` database files to be reloaded and the changes implemented.

**Note:** The queuing system does not support multibyte host names.

The `/etc/locks/lpd` file contains the process ID of the currently running instance of the `lpd` daemon. If the current machine becomes inoperable, you may need to remove the ID for the `lpd` daemon when the system starts up again. The error message displayed is `lpd: lock file or duplicate daemon`.

### Manipulating the `lpd` Daemon with the System Resource Controller

The `lpd` daemon is a subsystem controlled by the System Resource Controller (SRC). The `lpd` daemon is a member of the TCP/IP system group.

Use the following SRC commands to manipulate the `lpd` daemon:

- `starts`rc
  - Starts a subsystem, group of subsystems, or a subserver.

- `stops`rc
  - Stops a subsystem, group of subsystems, or a subserver.

- `refresh`
  - Causes the subsystem or group of subsystems to reread the appropriate configuration file.

- `traces`on
  - Enables tracing of a subsystem, group of subsystems, or a subserver.

- `tracesoff`
  - Disables tracing of a subsystem, group of subsystems, or a subserver.

- `lssrc`
  - Gets the status of a subsystem, group of subsystems, or a subserver.

### Flags

- `-d`
  - Sends a status of Inactive to be logged with the SRC controller and sends error messages during socket communication setup failures to the user display.

- `-l`
  - Sends a status of Active to be logged with the SRC controller and sends valid or invalid job request messages to the user display.

- `-D DebugOutputFile`
  - Sends extensive debugging output used for problem determination to the file specified by `DebugOutputFile`. This should only be used during problem determination as the `DebugOutputFile` can grow large rapidly. If the output file specified already exists, new debugging output is appended to the end of it. If there are any problems creating or writing to the output file, the debugging option is ignored.

### Examples

1. To start the `lpd` server daemon, enter:
   ```shell
   startsrc -s lpd
   ```
2. To start the `lpd` server daemon while enabling the display of certain error messages, enter:
   ```shell
   startsrc -s lpd -a "-d"
   ```
3. To send logging information to the `stderr` daemon, enter:
   ```shell
   startsrc -s lpd -a "-l"
   ```
4. To start the `lpd` server daemon in debugging mode with output going to `/tmp/dbglpd.out`, enter:
   ```shell
   startsrc -s lpd -a "-D /tmp/dbglpd.out"
   ```
Files

/usr/sbin/lpd Specifies the path to the lpd daemon.
/dev/lp* Contains the names of print devices.
/etc/hosts.equiv Contains the names of hosts allowed to execute commands and print.
/etc/hosts.lpd Contains the names of hosts allowed to print only.
/var/spool/lpd Contains the spool directory for control, status, and data files.
/etc/locks/lpd Contains the PID of the currently running lpd daemon. After a system crash, this PID may need to be deleted. The following error message indicates the problem:
lpd: lock file or duplicate daemon

Related Information

Ipfilter Command

Note: This is a System V Print Subsystem command.

Purpose
Administers filters used with the LP print service.

Syntax
Ipfilter [f FilterName [F PathName]
Ipfilter [F]
Ipfilter [i]
Ipfilter [x]
Ipfilter [l]

Description
The ipfilter command is used to add, change, delete, and list a filter used with the LP print service. These filters are used to convert the content type of a file to a content type acceptable to a printer.

Flags
- (hyphen) Adds or changes a filter as specified from standard input.
-f FilterName Specifies the name of the filter to be added, changed, deleted, or listed.
-F PathName Add or changes a filter as specified by the contents of the file pathname.
-i Resets an original filter to its original settings.
-l Lists a filter description.
-x Deletes a filter.

The parameter all can be used instead of a FilterName with any of these flags. When all is specified with the -F or - flag, the requested change is made to all filters. Using all with the -i flag has the effect of restoring to their original settings all filters for which predefined settings were initially available. Using the all parameter with the -x flag results in all filters being deleted, and using it with the -l flag produces a list of all filters.
Adding or changing a filter
The filter named in the -f flag is added to the filter table. If the filter already exists, its description is changed to reflect the new information in the input.

The filter description is taken from the PathName if the -F flag is given or from the standard input if the -f flag is specified. One of the two must be given to define or change a filter. If the filter named is one originally delivered with the LPprint service, the -i flag restores the original filter description.

When an existing filter is changed with the -F flag or the -f flag, items that are not specified in the new information are left as they were. When a new filter is added with this command, unspecified items are given default values.

Filters are used to convert the content of a request into a data stream acceptable to a printer. For a given print request, the LP print service knows the following:

- Content in the request
- Name of the printer
- Type of the printer
- Types of content acceptable to the printer
- Modes of printing asked for by the originator of the request

It uses this information to find a filter or a pipeline of filters that converts the content into a type acceptable to the printer.

A list of items that provide input to this command and a description of each item follows. All lists are comma or space separated.

- Input types: content-type-list
- Output types: content-type-list
- Printer types: printer-type-list
- Printers: printer-list
- Filter type: filter-type
- Command: shell-command
- Flags: template-list

| Input types | Gives the types of content that can be accepted by the filter. (The default is any.) |
| Output types | Gives the types of content that the filter can produce from any of the input content types. (The default is any.) |
| Printer types | Gives the type of printers for which the filter can be used. The LP print service restricts the use of the filter to these types of printers. (The default is any.) |
| Printers | Gives the names of the printers for which the filter can be used. The LP print service restricts the use of the filter to just the printers named. (The default is any.) |
| Filter type | Marks the filter as a slow filter or a fast filter. Slow filters are generally those that take a long time to convert their input. They are run unconnected to a printer to keep the printers from being tied up while the filter is running. If a listed printer is on a remote system, the filter type for it must have the value slow. Fast filters are generally those that convert their input quickly or those that must be connected to the printer when run. These are given to the interface program IP to run connected to the physical printer. |
| Command | Specifies the program to run to invoke the filter. The full program pathname as well as fixed flags must be included in the shell-command; additional flags are constructed, based on the characteristics of each print request and on the “flags” field. A command must be given for each filter. |

The command must accept a data stream as standard input and produce the converted data stream on its standard output. This allows filter pipelines to be constructed to convert data not handled by a single filter.
Flags

Specifies the comma-separated list of templates used by the LP print service to construct flags to
the filter from the characteristics of each print request listed in the table later.

In general, each template is of the following form:

\texttt{keyword-pattern=replacement}

The \texttt{keyword} names the characteristic that the template attempts to map into a filter-specific flag; each valid \texttt{keyword} is listed in the table below. A \texttt{pattern} is one of the following: a literal pattern of one of the forms listed in the table, a single asterisk (*), or a regular expression. If \texttt{pattern} matches the value of the characteristic, the template fits and is used to generate a filter-specific flag. The \texttt{replacement} is what is used as the flag.

Regular expressions are the same as those found in the \texttt{ed} or \texttt{vi} commands. This includes the \texttt{	extbackslash( . . . \textbackslash)} and \texttt{	extbackslash n} constructions, which can be used to extract portions of the \texttt{pattern} for copying into the \texttt{replacement}, and the \texttt{``\&''}, which can be used to copy the entire \texttt{pattern} into the \texttt{replacement}.

The \texttt{replacement} can also contain a \texttt{``\*''}. It too, is replaced with the entire \texttt{pattern}, just like the \texttt{``\&''} of \texttt{ed} command.

<table>
<thead>
<tr>
<th>\texttt{lp flag}</th>
<th>Characteristic</th>
<th>\texttt{keyword}</th>
<th>Possible patterns</th>
</tr>
</thead>
<tbody>
<tr>
<td>-T</td>
<td>Content type (input)</td>
<td>INPUT</td>
<td>content-type</td>
</tr>
<tr>
<td>N/A</td>
<td>Content type (output)</td>
<td>OUTPUT</td>
<td>content-type</td>
</tr>
<tr>
<td>N/A</td>
<td>Printer type</td>
<td>TERM</td>
<td>printer-type</td>
</tr>
<tr>
<td>-d</td>
<td>Printer name</td>
<td>PRINTER</td>
<td>printer-name</td>
</tr>
<tr>
<td>-f, -o cpi=</td>
<td>Character pitch</td>
<td>CPI</td>
<td>integer</td>
</tr>
<tr>
<td>-f, -o lpi=</td>
<td>Line pitch</td>
<td>LPI</td>
<td>integer</td>
</tr>
<tr>
<td>-f, -o length=</td>
<td>Page length</td>
<td>LENGTH</td>
<td>integer</td>
</tr>
<tr>
<td>-f, -o width=</td>
<td>Page width</td>
<td>WIDTH</td>
<td>integer</td>
</tr>
<tr>
<td>-P</td>
<td>Pages to print</td>
<td>PAGES</td>
<td>page-list</td>
</tr>
<tr>
<td>-S</td>
<td>Character set Print wheel</td>
<td>CHARSET</td>
<td>character-set-name print-wheel-name</td>
</tr>
<tr>
<td>-f</td>
<td>Form name</td>
<td>FORM</td>
<td>form-name</td>
</tr>
<tr>
<td>-y</td>
<td>Modes</td>
<td>MODES</td>
<td>mode</td>
</tr>
<tr>
<td>-n</td>
<td>Number of copies</td>
<td>COPIES</td>
<td>integer</td>
</tr>
</tbody>
</table>

For example, the template \texttt{MODES landscape = -1} shows that if a print request is submitted with the \texttt{-y landscape} flag, the filter is given the flag \texttt{-I}. As another example, the template \texttt{TERM \* = -T \*} shows that the filter is given the flag \texttt{-T printer-type} for whichever \texttt{printer-type} is associated with a print request using the filter.

As a last example, consider the template \texttt{MODES prwidth\=\((.*\)\) = \-w\1}. Suppose a user gives the command \texttt{lp \-y prwidth=10}

From the table above, the LP print service determines that the \texttt{-y} flag is handled by a \texttt{MODES} template. The \texttt{MODES} template here works because the pattern \texttt{``prwidth\=\((\.*\)\)''} matches the \texttt{prwidth=10} given by the user. The \texttt{replacement -w10} causes the LP print service to generate the filter flag \texttt{-w10}. 
If necessary, the LP print service constructs a filter pipeline by concatenating several filters to handle the user’s file and all the print flags. If the print service constructs a filter pipeline, the INPUT and OUTPUT values used for each filter in the pipeline are the types of the input and output for that filter, not for the entire pipeline.

**Deleting a filter**
The `x` flag is used to delete the filter specified in `FilterName` from the LP filter table.

**Listing a filter description**
The `-l` flag is used to list the description of the filter named in `FilterName`. If the command is successful, the following message is sent to standard output:

```
Input types: content-type-list
Output types: content-type-list
Printer types: printer-type-list
Printers: printer-list
Filter type: filter-type
Command: shell-command
flags: template-list
```

If the command fails, an error message is sent to standard error.

**Related Information**
The `lpadmin` command.

---

**Ipforms Command**

**Note**: This is a System V Print Subsystem command.

**Purpose**
Administer forms used with the LP print service.

**Syntax**

```
lpforms -f FormName Options
lpforms -f FormName -A AlertType [-Q minutes] [-W requests]
```

**Description**
The `lpforms` command is used to administer the use of preprinted forms, such as company letterhead paper, with the LP print service. A form is specified by its `FormName`. Users may specify a form when submitting a print request (see the `lp` command). The parameter `all` can be used instead of `FormName` with either of the command lines shown above. The first command line allows the administrator to add, change, and delete forms, to list the attributes of an existing form, and to allow and deny users access to particular forms. The second command line is used to establish the method by which the administrator is alerted that the form `FormName` must be mounted on a printer.

With the first `lpforms` command line, one of the following flags must be used:

**Flags**

- `-` (hyphen) Adds or changes form `FormName`, as specified by the information from standard input.
- `-F pathname` Adds or changes form `FormName`, as specified by the information in `pathname`.
- `-l` Lists the attributes of form `FormName`.

---

Alphabetical Listing of Commands 243
Adding or changing a form

The -F pathname flag is used to add a new form, FormName, to the LP print service, or to change the attributes of an existing form. The form description is taken from pathname if the -F flag is given, or from the standard input if the - flag is used. One of these two flags must be used to define or change a form. pathname is the pathname of a file that contains all or any subset of the following information about the form:

- Page length: scaled-decimal-number1
- Page width: scaled-decimal-number2
- Number of pages: integer
- Line pitch: scaled-decimal-number3
- Character pitch: scaled-decimal-number4
- Character set choice: character-set/print-wheel [mandatory]
- Ribbon color: ribbon-color
- Comment: comment
- Alignment pattern: [content-type] content

The term "scaled-decimal-number" refers to a non-negative number used to indicate a unit of size. The type of unit is shown by a "trailing" letter attached to the number. Three types of scaled decimal numbers can be used with the LP print service: numbers that show sizes in centimeters (marked with a trailing "c"); numbers that show sizes in inches (marked with a trailing "i"); and numbers that show sizes in units appropriate to use (without a trailing letter), that is, lines, characters, lines per inch, or characters per inch.

Except for the last two lines, the above lines may appear in any order. The Comment and comment items must appear in consecutive order but may appear before the other items, and the "Alignment pattern" and the content items must appear in consecutive order at the end of the file. Also, the comment item may not contain a line that begins with any of the key phrases above, unless the key phrase is preceded with a ">". Any leading ">" sign found in the comment are removed when the comment is displayed. Case distinctions in the key phrases are ignored.

When this command is issued, the form specified by FormName is added to the list of forms. If the form already exists, its description is changed to reflect the new information. Once added, a form is available for use in a print request, except where access to the form has been restricted, as described under the -u flag. A form may also be allowed to be used on certain printers only.

A description of each form attribute is below:

Page length and Page width

Before printing the content of a print request needing this form, the generic interface program provided with the LP print service initializes the physical printer to handle pages scaled-decimal-number1 long, and scaled-decimal-number2 wide using the printer type as a key into the terminfo database.

The page length and page width are also passed, if possible, to each filter used in a request needing this form.

Number of pages

Each time the alignment pattern is printed, the LP print service attempts to truncate the content to a single form by, if possible, passing to each filter the page subset of 1-integer.
Line pitch and Character pitch

Before printing the content of a print request needing this form, the interface programs provided with the LP print service initializes the physical printer to handle these pitches, using the printer type as a key into the terminfo database. Also, the pitches are passed, if possible, to each filter used in a request needing this form.

*scaled-decimal-number3* is in lines per centimeter if a "c" is appended, and lines per inch otherwise; similarly, *scaled-decimal-number4* is in characters per centimeter if a "c" is appended, and characters per inch otherwise. The character pitch can also be given as *elite* (12 characters per inch), *pica* (10 characters per inch), or *compressed* (as many characters per inch as possible).

Character set choice

When the LP print service alerts an administrator to mount this form, it also mentions that the print wheel *print-wheel* should be used on those printers that take print wheels. If printing with this form is to be done on a printer that has selectable or loadable character sets instead of print wheels, the interface programs provided with the LP print service automatically selects or loads the correct character set. If *mandatory* is appended, a user is not allowed to select a different character set for use with the form; otherwise, the character set or print wheel named is a suggestion and a default only.

Ribbon color

When the LP print service alerts an administrator to mount this form, it also mentions that the color of the ribbon should be *ribbon-color*.

Comment

The LP print service displays the *comment* unaltered when a user asks about this form (see the *lpstat* command).

Alignment pattern

When mounting this form, an administrator can ask for the *content* to be printed repeatedly, as an aid in correctly positioning the preprinted form. The optional *content-type* defines the type of printer for which *content* had been generated. If *content-type* is not given, *simple* is assumed.

*Note:* The content is stored as given and is readable only by the user *lp*.

When an existing form is changed with this command, items missing in the new information are left as they were. When a new form is added with this command, missing items gets the following defaults:

- Page Length: 66
- Page Width: 80
- Number of Pages: 1
- Line Pitch: 6
- Character Pitch: 10
- Character Set Choice: any
- Ribbon Color: any

Deleting a form

The -x flag is used to delete the form *FormName* from the LP print service.

Listing form attributes

The -l flag is used to list the attributes of the existing form *FormName*. Because of the potentially sensitive nature of the alignment pattern, only the administrator can examine the form with this command. Other people may use the *lpstat* command to examine the non-sensitive part of the form description.
Allowing and denying access to a form

The `-u` flag, followed by the parameter **allow:**`login-ID-list` or **-u deny:**`login-ID-list` lets you determine which users are allowed to specify a particular form with a print request. This flag can be used with the `-F` or `-f` flag.

The `login-ID-list` parameter may include any or all of the following constructs:

- `login-ID` A user on the local system
- `system-name|login-ID` A user on system `system-name`
- `system-name|all` All users on system `system-name`
- `all|login-ID` A user on all systems
- `all` All users on the local system
- `all|all` All users on all systems

The default value of `login-ID-list` is `all`.

The LP print service keeps two lists of users for each form: an “allow-list” of people allowed to use the form, and a “deny-list” of people that may not use the form.

- If allow-list is present and `login-ID` is in it, access is allowed.
- If only deny-list is present and `login-ID` is not in it, access is allowed.
- If `login-ID` is in deny-list, access is denied.
- If neither allow-list or deny-list are present, access is denied.
- If both lists are present, and `login-ID` is in neither, access is denied.
- If only allow-list is present and `login-ID` is not in it, access is denied.

If the allow-list is not empty, only the users in the list are allowed access to the form, regardless of the contents of the deny-list. If the allow-list is empty but the deny-list is not, the users in the deny-list may not use the form (but all others may use it).

All users can be denied access to a form by specifying **-f deny:**`all`. All users can be allowed access to a form by specifying **-f allow:**`all`. (This is the default.)

Setting an alert to mount a form

The `-f FormName` flag is used with the `-A AlertType` flag to define an alert to mount the form when there are queued jobs which need it. If this flag is not used to arrange alerting for a form, no alert is sent for that form.

The method by which the alert is sent depends on the value of the `AlertType` parameter specified with the `-A` flag. The alert types are the same as those available with the `-A` flag to `lpadmin`: mail, write, quiet, none, shell-command, and list. See the description of `-A` on `lpadmin` for details about each.

The message sent appears as follows:

The form `FormName` needs to be mounted on the printer(s):

- `printer` (integer1 requests),
- integer2 print requests await this form.
- Use the `ribbon-color` ribbon.
- Use the `print-wheel` print wheel, if appropriate.

The printers listed are those that the administrator had earlier specified were candidates for this form. The number `integer1` listed next to each printer is the number of requests eligible for the printer. The number `integer2` shown after the list of printers is the total number of requests awaiting the form. It is less than the sum of the other numbers if some requests can be handled by more than one printer. The `ribbon-color` and
print-wheel are those specified in the form description. The last line in the message is always sent, even if none of the printers listed use print wheels, because the administrator may choose to mount the form on a printer that does use a print wheel.

Where any color ribbon or any print wheel can be used, the statements above read:

Use any ribbon.
Use any print-wheel.

If FormName is any, the alerting defined in this command applies to any form for which an alert has not yet been defined. If FormName is all, the alerting defined in this command applies to all forms.

If the -W flag is not given, the default procedure is that only one message is sent per need to mount the form. Not specifying the -W flag is equivalent to specifying -W once or -W 0. If minutes is a number greater than 0, an alert is sent at intervals specified by minutes.

If the -Q flag is also given, the alert is sent when a certain number (specified by the parameter requests) of print requests that need the form are waiting. If the -Q flag is not given, or the value of requests is 1 or any (which are both the default), a message is sent as soon as anyone submits a print request for the form when it is not mounted.

Listing the current alert
The -f flag, followed by the -A flag and the parameter list is used to list the type of alert that has been defined for the specified form FormName. No change is made to the alert. If FormName is recognized by the LP print service, one of the following lines is sent to the standard output, depending on the type of alert for the form.

When requests requests are queued:
alert with shell-command every minutes minutes

When requests requests are queued:
write to user-name every minutes minutes

When requests requests are queued:
mail to user-name every minutes minutes

No alert

The phrase "every minutes minutes" is replaced with "once" if minutes (-W minutes) is 0.

Terminating an active alert
The -A quiet flag is used to stop messages for the current condition. An administrator can use this flag to temporarily stop receiving further messages about a known problem. Once the form has been mounted and then unmounted, messages are again sent when the number of print requests reaches the threshold requests.

Removing an alert definition
No messages are sent after the -A none flag is used until the -A flag is given again with a different AlertType. This can be used to permanently stop further messages from being sent as any existing alert definition for the form is removed.

Related Information
The lpadmin command.
The terminfo file.

Ipmove Command

Note: This is a System V Print Subsystem command.
Purpose
Moves print requests.

Syntax
lpmove Requests Destination
lpmove Destination1 Destination2

Description
The lpmove command moves requests that were queued by lp between LP destinations. This command moves a specific Request to the specified Destination. Requests are request-IDs returned by lp. You can also attempt to move all requests for Destination1 to Destination2. This form of the lpmove command causes lp to reject any new requests for Destination1.

Note: When moving requests, lpmove never checks the acceptance status (see accept) of the new destination. Also, the request-IDs of the moved requests are not changed, so you can still find their requests. The lpmove command does not move requests that have options (such as content type and form required) that cannot be handled by the new destination.

If a request was originally queued for a class or the special destination any and the first form of lpmove was used, the destination of the request is changed to New-Destination. A request thus affected is printable only on New-Destination and not on other members of the class or other acceptable printers if the original destination was any.

Files
/var/spool/lp/*

Related Information
The accept command, enable command, lp command, lpadmin command, lpstat command.

Ippchk Command

Purpose
Verifies files of an installable software product.

Syntax
Ippchk [-c u] [-f] [-l] [m [1 2 3]] [-O [r s u]] [productName] [filelist ...]

Description
The Ippchk command verifies that files for an installable software product (fileset) match the Software Vital Product Data (SWVPD) database information for file sizes, checksum values, or symbolic links. A fileset is a separately installable option of a software package.

Flags
-c Performs a checksum operation on the FileList items and verifies that the checksum and the file size are consistent with the SWVPD database.
-f Checks that the FileList items are present and the file size matches the SWVPD database.
-l Verifies symbolic links for files as specified in the SWVPD database.
Displays three levels of information. The levels are as follows:

1. Error messages only (default).
2. Error messages and warnings.
3. Error messages, warnings and informational messages.

Verifies the specified parts of the program. This flag is not needed with standalone systems because without this option all parts are verified by default. The flags specify the following parts:

- Indicates the / (root) part is to be verified.
- Indicates the/usr/share part is to be verified.
- Indicates the /usr part is to be verified.

Updates the SWVPD with new checksum or size information from the system when the system information does not match the SWVPD database. This flag sets symbolic links that are found to be missing. This flag is valid with only the -c or -l flag.

Verifies that the / (root), /usr and /usr/share parts of the system are valid with each other. In other words, this flag verifies that all software products installed on the / (root) file system are also installed on the /usr file system and, conversely, all the software products installed in the /usr file system are also installed on the / (root) file system. You cannot specify FileList items with this flag. This flag also verifies requisites.

Note: Only one of the -c, -f, -l, and -v flags can be specified with each use of the lppchk command.

Parameters

FileList Specifies the file or files to check. This parameter is a list of file names separated by spaces. The file names can be a single name or a pair of names separated by a colon. The first form specifies a simple file and the second form specifies a member of an archive file, where the first name specifies the member and the second name specifies the archive file that contains the member. The full path name of the file or files must be specified. To specify multiple files you can use the pattern-matching characters * (asterisk) and ? (question mark), but they should be enclosed in a pair of ‘s (single quotes). Single quotes are recommended to prevent the korn shell wildcard expansion.

If this parameter is omitted, all files of a software product are checked. If this parameter is specified, it must be preceded by a software product name.

ProductName Specifies the name of the software product whose files are to be checked. If this parameter is omitted, all software products in the SWVPD are checked. To specify multiple software products you can use the pattern-matching characters * (asterisk) and ? (question mark), but they must be enclosed in a pair of ‘s (single quotes) to prevent the shell from expanding them.

Return Values

The lppchk command returns zero if no errors were found. Any other return value indicates an error was found.

Examples

1. To verify all files that comprise the X11.fnt package, type:
   `lppchk -c X11.fnt`
2. To verify the symbolic links of all software products whose names begin with X11, type:
   `lppchk -l 'X11*'`
3. To verify that all filesets have all required requisites and are completely installed, type:
   `lppchk -v`
Files

/etc/objrepos/lpp  Specifies installation information of all software products on the root.
/usr/lib/objrepos/lpp  Specifies installation information of all software products on the /usr file system.
/usr/share/lib/objrepos/lpp  Specifies installation information of all software products on the /usr/share file system.
/etc/objrepos/product  Specifies installation and update information of all software products on the root.
/usr/lib/objrepos/product  Specifies installation and update information of all software products on the /usr file system.
/usr/share/lib/objrepos/product  Specifies installation and update information of all the software products on the /usr file system.
/etc/objrepos/inventory  Specifies names and locations of files in a software product on the root.
/usr/lib/objrepos/inventory  Specifies names and locations of files in a software product on the /usr file system.
/usr/share/lib/objrepos/inventory  Specifies names and locations of files in a software product on the /usr/share file system.

Related Information

The `installp` command, `sum` command, `tcbck` command.

Ippmgr Command

Purpose

Manages an existing installp image source.

Syntax

```
Ippmgr [ DirectoryOrDevice ] { [ [ -r ] [ -m MoveDirectory ] ] [ [ -X ] [ -x ] [ -u ] [ -b ] [ -k LANG ] ] [ -p ] [ -t ] }
```

Description

`Ippmgr` is designed to perform the following functions on an existing installp image source (also known as an `lpp_source` in the NIM environment):

1. Remove duplicate updates (-u Flag).
2. Remove duplicate base levels (-b Flag).
3. Eliminating updates which are the same level as bases of the same fileset. Such updates can create conflicts that lead to installation failure (-u Flag).
4. Remove message and locale filesets other than the language you specify (-k Flag).
5. Remove superseded filesets (-x Flag).
6. Remove non-system images from a NIM lpp_source resource (-X Flag).

By default, `Ippmgr` will list all images filtered by the above routines. The "-r" flag can be used to remove the filtered images and the "-m" flag can be used to move the images to another location.

Note: `Ippmgr` is not intended to replace `bfccreate`, install anything, or work with installed filesets. It is also not intended to address any issues other than those aforementioned. Before using the "-X" flag, you should have a good understanding of NIM, system images (known as SIMAGES in NIM), and the workings of a NIM `lpp_source` resource.
Flags

-b Causes lppmgr to filter for base level duplicates.
-D Specifies debug mode. This flag is for debugging the lppmgr script.
   Note: Debug. This produces a large quantity of output and greatly reduces lppmgr
   performance. It is not useful for normal operations.
-d DeviceOrDirectory Specifies the device or directory where the installp images reside. Currently this can
   be any directory, NFS mount point, or cdrom device. If the directory is not writable,
   you should use the -t flag. If the target of your operation is a NIM lpp_source
   resource, you should specify the lpp_source location (see the lsnim command).
   This flag is required for all operations.
-k LANG Keeps only the message and locale images for the language specified by LANG. All
   other languages are filtered.
-i Lists filtered images only. By default, lppmgr will only list all filtered image files
   unless the "-r" or "-m" flag is specified. The "-i" flag will override the "-r" or "-m"
   flag.
-m Directory Moves filtered files to Directory. The location specified by Directory can be any
   writable directory path. This flag cannot be used with the "-r" flag.
-p Specifies prompt mode. Prompt when moving or removing files.
-r Removes files that have been filtered by lppmgr.
   Note: If the prompt flag is not specified (-p), lppmgr will remove all filtered files
   without further user interaction. This flag cannot be used with the "-m" flag.
-s Prints space usage info. This will print the amount of space a particular fileset is
   using and the total amount of space in question. Some buffer space is added for file
   metadata.
-t Specifies that lppmgr will not rebuild the .toc file. This can be useful for having a
   quick look without having to rebuild the entire .toc file, which can take some time.
   Also, this is required for read-only devices.
-u Causes lppmgr to filter for duplicate updates and conflicting updates which are the
   same level as bases of the same fileset.
-V Specifies verbose mode. lppmgr will give more output in certain situations.
-x Causes lppmgr to filter for superseded updates.
-X Filters non-system images from a NIM lpp_source resource.

Exit Status

0   All lppmgr related operations completed successfully.
>0   An error occurred.

Security

Only the root user can execute lppmgr.

Examples

1. To list all duplicate and conflicting updates in image source directory /myimages, enter:
   lppmgr -d /myimages -u
2. To remove all duplicate and conflicting updates in image source directory /myimages, enter:
   lppmgr -d /myimages -u -r
3. To remove all duplicate and conflicting updates, duplicate base levels, and all message/locale filesets
   other than "en_US" in prompted mode, enter:
   lppmgr -d /images -purb -k en_US
4. To move all superseded update images and non SIMAGES from NIM lpp_source location /lpps/433 to
   directory /backups, enter:
   lppmgr -d /lpps/433 -x -X -m /backups
Files

/usr/lib/instdl/lppmgr             Contains the lppmgr command.

Related Information

The installp command, lspp command, lsnim command.

Ipq Command

The ipq command includes information for the AIX Print Subsystem [lpq] and the System V Print Subsystem [lpq].

AIX Print Subsystem Ipq Command

Purpose

Examines the spool queue.

Syntax

Ipq [ 2 [ Number ] ] [ J | W ] [ P Printer ] [ JobNumber ] [ UserName ]

Description

The ipq command reports the status of the specified job or all jobs associated with the specified UserName and JobNumber variables. JobNumber variable specifies the number of the job in the spool queue that you want to view. A UserName variable specifies viewing the jobs for the name of the person who submitted the job to that queue.

The ipq command reports on any jobs currently in the default queue when invoked without any options. Parameters supplied that are not recognized as parameters are interpreted as user names or job numbers to filter out only those jobs of interest.

For each job submitted (each job called by the lpr command), the ipq command reports the user’s name, current rank in the queue, the name of the job, the job identifier (a number that can be supplied to the lprm command for removing a specific job), and the total size in blocks. Normally, only as much information as will fit on one line is displayed. Job ordering depends on the algorithm used to scan the spooling directory and is supposed to be FIFO (first-in-first-out). File names making up a job may be unavailable (when the lpr command is used as a sink in a pipeline). In this case, the file is indicated as - (standard input).

The display generated by the ipq command contains two entries for remote queues. The first entry contains the client’s local queue and local device name and its status information. The second entry follows immediately; it contains the client’s local queue name (again), followed by the remote queue name. Any jobs submitted to a remote queue are displayed first on the local side and are moved to the remote device as the job is processed on the remote machine.

Since the status commands communicate with remote machines, the status display may occasionally appear to hang while waiting for a response from the remote machine. The command will eventually time out if a connection cannot be established between the two machines.

Flags

-I       Generates the long output format.
+ [ Number ] Displays the spool queue until it empties. A Number variable is the time in seconds before the display regenerates.

-P Printer Displays the spool queue for the printer specified by the Printer variable.

Note: Any destination command line options override both the LPDEST and the PRINTER environment variables.

-W Displays a wide version of status information with longer queue names, device names, and job numbers. Longer job number information is available on AIX 4.3.2 and later. This flag cannot be used with the -i flag. If the -i flag and the -W flag are used simultaneously, the first one specified takes precedence.

Examples
1. To display a job number in the print queue lp0, enter:

   lpq -P lp0

   This command displays a list similar to the following:

   Queue  Dev Status  Job Files  User  PP %  Blks  CP  Rnk
   lp0   dlp0 running 39  motd  guest 10  83 12 1 1

2. To display the status of the default queue in wide format for AIX 4.3.2 or later, enter:

   lpq -W

Files
/usr/bin/lpq Contains the lpq command.
/usr/sbin/qdaemon Contains the queuing daemon.
/etc/qconfig Contains the queue configuration file.
/etc/qconfig.bin Contains the digested, binary version of the /etc/qconfig file.
/var/spool/lpd/qdir/*/ Contains queue requests.
/var/spool/lpd/stat/* Contains information on the status of the devices.
/var/spool/qdaemon/* Contains temporary copies of enqueued files.

Related Information
The lpr command, lprm command, lpstat command, qchk command.

The qconfig file.


Spooler Overview for System Management in the AIX 5L Version 5.2 Guide to Printers and Printing.

System V Print Subsystem lpq Command

Purpose
(BSD) Displays the queue of printer jobs

Syntax
/usr/bin/lpq [-Pprinter] [-l] [+ [interval] ] [job# ... ] [username ... ]
Description

The `lpq` command displays the contents of a printer queue. It reports the status of jobs specified by `job#`, or all jobs owned by the user specified by `username`. `lpq` reports on all jobs in the default printer queue when invoked with no arguments.

For each print job in the queue, `lpq` reports the user’s name, current position, the names of input files comprising the job, the job number (by which it is referred to when using `lprm`) and the total size in bytes. Normally, only as much information as will fit on one line is displayed. Jobs are normally queued on a first-in-first-out basis. Filenames comprising a job may be unavailable, such as when `lpr` is used at the end of a pipeline; in such cases the filename field indicates the standard input.

If `lpq` warns that there is no daemon present (that is, due to some malfunction), the `lpc` command can be used to restart a printer daemon.

Output formatting is sensitive to the line length of the terminal; this can result in widely-spaced columns.

Flags

-P `printer`
Display information about the queue for the specified `printer`. In the absence of the -P flag, the queue to the printer specified by the `PRINTER` variable in the environment is used. If the `PRINTER` variable is not set, the queue for the default printer is used.

-l Display queue information in long format; includes the name of the host from which the job originated.

+[interval ]
Display the spool queue periodically until it empties. This option clears the terminal screen before reporting on the queue. If an `interval` is supplied, `lpq` sleeps that number of seconds in between reports.

Files

/var/spool/lp
spooling directory.

/var/spool/lp/tmp/system_name/*-0
request files specifying jobs

Error Codes

lpq: `printer` is printing
The `lpq` program queries the spooler `LPSCHED` about the status of the printer. If the printer is disabled, the system administrator can restart the spooler using `lpc`.

lpq: `printer` waiting for auto-retry (offline ?)
The daemon could not open the printer device. The printer may be turned off-line. This message can also occur if a printer is out of paper, the paper is jammed, and so on. Another possible cause is that a process, such as an output filter, has exclusive use of the device. The only recourse in this case is to kill the offending process and restart the printer with `lpc`.

lpq: waiting for host to come up
A daemon is trying to connect to the remote machine named `host`, in order to send the files in the local queue. If the remote machine is up, `lpd` on the remote machine is probably dead or hung and should be restarted using `lpc`.

lpq: sending to `host`
The files are being transferred to the remote `host`, or else the local daemon has hung while trying to transfer the files.
lpq: printer disabled reason:
The printer has been marked as being unavailable with \texttt{lpc}.

lpq: The LP print service isn't running or can't be reached.
The \texttt{lpsched} process overseeing the spooling queue does not exist. You can restart the printer daemon with \texttt{lpc}.

lpq: \texttt{printer: unknown printer}
The printer was not found in the System V LP database. Usually this is a typing mistake; however, it may indicate that the printer does not exist on the system. Use \texttt{lpsched -p} to find the reason.

lpq: error on opening queue to spooler
The connection to \texttt{lpsched} on the local machine failed. This usually means the printer server started at boot time has died or is hung. Check if the printer spooler daemon /usr/lib/lp/lpsched is running.

lpq: Can't send message to LP print service
lpq: Can't establish contact with LP print service
These indicate that the LP print service has been stopped. Get help from the system administrator.

lpq: Received unexpected message from LP print service
It is likely there is an error in this software. Get help from system administrator.

Related Information
The \texttt{lpc} command, \texttt{lpr} command, and \texttt{lprm} command.

Lpr Command

The \texttt{lpr} command includes information for the AIX Print Subsystem \texttt{lpr} and the System V Print Subsystem \texttt{lpr}.

AIX Print Subsystem lpr Command

Purpose
Enqueues print jobs.

Syntax

Description
The \texttt{lpr} command uses a spooling daemon to print the named \textit{File} parameter when facilities become available. If no files are specified, the \texttt{lpr} command reads from standard input.

Flags
-# \textit{Number} Produces multiple copies of output, using the \textit{Number} variable as the number of copies for each file named.

-C \textit{Class} Specifies the print \textit{Class} as the job classification on the burst page.

-f Uses a filter that interprets the first character of each line as a standard FORTRAN carriage control character.

-g The files are assumed to contain standard plot data.

-h Suppresses printing of the burst page.

\textbf{Note:} The default is to print a header page and not a trailer page.
-i [Number] Indents output Number spaces. If the Number variable is not given, eight spaces are used as the default.

-j Specifies that the message Job number is: nnn, where nnn is the assigned job number, be displayed to standard output. This occurs only if the job is submitted to a local print queue.

-J Job Prints the Job variable as the job name on the burst page. Usually, the lpr command uses the name of the first file.

-l (Lowercase L) Uses a filter that allows control characters to be printed.

-m Sends mail upon completion of spooling.

-n Uses a filter that formats files containing ditroff (device-independent troff) data.

-P Printer Forces output to the Printer variable. If this flag is not specified, the following conditions occur:

- If a default exists, the lpr command uses the default printer.
- If the LPDEST environment variable is set, then lpr uses the value specified by the LPDEST variable. If set, this value is always used, even if the PRINTER variable is also set.
- If the PRINTER variable is set and no LPDEST variable is set, then lpr uses the value specified by the PRINTER environment variable.

Note: Any destination command line options override both the LPDEST and the PRINTER environment variables.

-p Uses the pr command to format the file (lpr -p is very much like pr | lpr).

-r Removes the file upon completion of spooling.

-s Prints from the files specified on the command line rather than trying to copy them (so large files can be printed). This means the data files should not be modified or removed until they have been printed. Note that this flag only works on the local host (files sent to remote printer hosts are copied anyway), and only with named data files. It does not work if the lpr command is at the end of a pipeline.

-T Title Uses the Title variable instead of the file name for the title used by the pr command.

-w Number Uses the Number variable as the page width for the pr command.

Examples

1. To print three copies of the files new.index.c, print.index.c, and more.c, enter:

   lpr -#3 new.index.c print.index.c more.c

   Prints three copies of the new.index.c file, three copies of the print.index.c file, and three copies of the more.c file.

2. To print three copies of the concatenation of three files new.index.c, print.index.c, and more.c, enter:

   cat new.index.c print.index.c more.c | lpr -#3

3. To print Operations on the burst page, followed by file new.index.c, enter:

   lpr -C Operations new.index.c

   This replaces the system name (the name returned by host name) with Operations on the burst page.

4. To queue the MyFile file and return the job number, enter:

   lpr -j MyFile

Files

/usr/sbin/qdaemon Queuing daemon.
/etc/qconfig Queue configuration file.
/etc/qconfig.bin Digested, binary version of the /etc/qconfig file.
/var/spool/lpd/qdir/* Queue requests.
/var/spool/lpd/stat/* Information on the status of the queues.
Related Information
The \texttt{lpd} command, \texttt{lpq} command, \texttt{lprm} command, \texttt{pr} command, \texttt{qdaemon} command.

The \texttt{qconfig} file.


Printer Overview for System Management in the \textit{AIX 5L Version 5.2 Guide to Printers and Printing}.

Spooler Overview for System Management in the \textit{AIX 5L Version 5.2 Guide to Printers and Printing}.

System V Print Subsystem \texttt{lpr} Command

Purpose
(BSD) Sends a job to the printer.

Syntax

\begin{verbatim}
/usr/bin/lpr [-P \texttt{printer}] [-# \texttt{copies}] [-C \texttt{class}] [-J \texttt{job}] [-T \texttt{title}] [-i \texttt{indent}] [-w \texttt{cols}] [-r] [-m] [-h] [-s] [filter_option] [file ...]
\end{verbatim}

Description
The \texttt{lpr} command forwards printer jobs to a spooling area for subsequent printing as facilities become available. Each printer job consists of copies of each \texttt{file} you specify. The spool area is managed by the line printer spooler, \texttt{lpsched}. \texttt{lpr} reads from the standard input if no files are specified.

\texttt{lp} is the preferred interface.

Command-line options cannot be combined into a single argument as with some other commands. The command:

\begin{verbatim}
lpr -fs
\end{verbatim}

is not equivalent to

\begin{verbatim}
lpr -f -s
\end{verbatim}

Placing the \texttt{-s} flag first, or writing each option as a separate argument, makes a link as expected.

\texttt{lpr -p} is not precisely equivalent to \texttt{pr} \texttt{lpr}. \texttt{lpr -p} puts the current date at the top of each page, rather than the date last modified.

Fonts for \texttt{roff} and \TeX reside on the printer host. It is not possible to use local font libraries.

\texttt{lpr} objects to printing binary files.

If userA uses \texttt{su} to become userB and uses \texttt{/usr/bin/lpr}, then the printer request will be entered as userB, not userA.
Flags

-P printer  
Send output to the named printer. Otherwise send output to the printer named in the PRINTER environment variable, or to the default printer, lp.

-# copies  
Produce the number of copies indicated for each named file. For example:

```
  lpr -#3 index.c lookup.c
```

produces three copies of index.c, followed by three copies of lookup.c. On the other hand,

```
  cat index.c lookup.c | lpr -#3
```

generates three copies of the concatenation of the files.

-C class  
Print class as the job classification on the burst page. For example,

```
  lpr -C Operations new.index.c
```

replaces the system name (the name returned by ```hostname``") with Operations on the burst page, and prints the file new.index.c.

-J job  
Print job as the job name on the burst page. Usually, lpr uses the first file's name.

-T title  
Use title instead of the file name for the title used by pr.

-i[indent]  
Indent output indent <Space> characters. Eight <Space> characters is the default.

-w cols  
Use cols as the page width for pr.

-r  
Remove the file upon completion of spooling, or upon completion of printing with the -s flag.

-m  
Send mail upon completion.

-h  
Suppress printing the burst page.

-s  
Use the full pathnames (not symbolic links) of the files to be printed rather than trying to copy them. This means the data files should not be modified or removed until they have been printed. This flag only prevents copies of local files from being made. Jobs from remote hosts are copied anyway. The -s flag only works with named data files; if the lpr command is at the end of a pipeline, the data is copied to the spool.

filter_option
The following single letter options notify the line printer spooler that the files are not standard text files. The spooling daemon will use the appropriate filters to print the data accordingly.

-p  
Use pr to format the files (lpr -p is very much like pr lpr).

-l  
Print control characters and suppress page breaks.

-t  
The files contain troff (cat phototypesetter) binary data.

-n  
The files contain data from ditroff (device independent troff).

-d  
The files contain data from tex (DVI format from Stanford).

-g  
The files contain standard plot data as produced by the routine plot for the filters used by the printer spooler.

-v  
The files contain a raster image. The printer must support an appropriate imaging model such as PostScript in order to print the image.

-c  
The files contain data produced by cifplot.
-f Interpret the first character of each line as a standard FORTRAN carriage control character.

If no filter_option is given (and the printer can interpret PostScript), the string `%!` as the first two characters of a file indicates that it contains PostScript commands.

These filter options offer a standard user interface, and all options may not be available for, nor applicable to, all printers.

Files

/usr/lib/lp/lpsched
System V line printer spooler

/var/spool/lp/tmp/*
directories used for spooling

/var/spool/lp/tmp/system/*-0
spooler control files

/var/spool/lp/tmp/system/*-N
(N is an integer and > 0) data files specified in `*-0` files

Error Codes

lpr: printer: unknown printer
The printer was not found in the LP database. Usually this is a typing mistake; however, it may indicate that the printer does not exist on the system. Use lpstat -p to find the reason.

lpr: error on opening queue to spooler
The connection to lpsched on the local machine failed. This usually means the printer server started at boot time has died or is hung. Check if the printer spooler daemon /usr/lib/lpsched is running.

lpr: printer: printer queue is disabled
This means the queue was turned off with
/usr/etc/lpc disable printer

to prevent lpr from putting files in the queue. This is usually done by the system manager when a printer is going to be down for a long time. The printer can be turned back on by a privileged user with lpc.

lpr: Can't send message to the LP print service

lpr: Can't establish contact with the LP print service
These indicate that the LP print service has been stopped. Get help from the system administrator.

lpr: Received unexpected message from LP print service
It is likely there is an error in this software. Get help from system administrator.

lpr: There is no filter to convert the file content
Use the lpstat -p -l command to find a printer that can handle the file type directly, or consult with your system administrator.

lpr: cannot access the file
Make sure file names are valid.

Related Information
The lpc command, lpq command, lprm command, and troff command.
Iprm Command

The `lprm` command includes information for the AIX Print Subsystem and the System V Print Subsystem.

AIX Print Subsystem Iprm Command

Purpose
Removes jobs from the line printer spooling queue.

Syntax
```
lprm [ -P Printer ] [ JobNumber ] [ UserName ... ] [ - ]
```

Description
The `lprm` command removes one or more jobs from the spool queue of a printer.

You cannot run the `lprm` command without specifying a job number, the - (minus sign) flag, or at least one user name.

Specifying a `UserName` parameter, or list of names, causes the `lprm` command to attempt to remove any jobs queued belonging to that user (or users).

You can remove an individual job from a queue by specifying its `JobNumber`. This job number is obtained by using the `lpq` command.

Flags
- Removes all jobs a user owns. Someone with root user authority can use this flag to remove all jobs from a queue. This flag is not valid for remote print.

- `P Printer` Specifies the queue associated with a specific `Printer` variable. If this flag is not specified, the following conditions occur:
  - If the `LPDEST` environment variable is set, then `lprm` uses the value specified by the `LPDEST` variable. If set, this value is always used, even if the `PRINTER` variable is also set.
  - If the `PRINTER` variable is set and no `LPDEST` variable is set, then `lprm` uses the value specified by the `PRINTER` environment variable.

If neither the `LPDEST` nor the `PRINTER` variable is set, the `lprm` command removes jobs from the default queue.

**Note:** Any destination command line options override both the `LPDEST` and the `PRINTER` environment variables.

Examples
1. To remove job number 13 from the default printer queue, enter:
```
lprm 13
```
2. To remove job number 13 from printer queue `lp0`, enter:
```
lprm -P lp0 13
```
3. To remove a job from the printer queue for a certain user, enter:
```
lprm guest
```
Files
/usr/bin/lprm Contains the lprm command.
/etc/qconfig Contains the configuration file.

Related Information
The lpq command, lpr command, qcan command.


System V Print Subsystem lprm Command

Purpose
(BSD) Remove jobs from the printer queue

Syntax
/usr/bin/lprm [-P printer] [-] [job #] [username...]

Description
The lprm command removes a job or jobs from a printer's spooling queue. Since the spool directory is protected from users, using lprm is normally the only method by which a user can remove a job.

Without any arguments, lprm deletes the job that is currently active, provided that the user who invoked lprm owns that job.

When the privileged user specifies a username, lprm removes all jobs belonging to that user.

You can remove a specific job by supplying its job number as an argument, which you can obtain using lpq. For example:

```
lpq -Phost
host is ready and printing
Rank  Owner  Job  Files  Total Size
active  wendy  385  standard input  35501 bytes
lprm -Phost 385
```

lprm reports the names of any files it removes, and is silent if there are no applicable jobs to remove.

lprm Sends the request to cancel a job to the print spooler, LPSCHED.

An active job may be incorrectly identified for removal by an lprm command issued with no arguments. During the interval between an lpq command and the execution of lprm, the next job in queue may have become active; that job may be removed unintentionally if it is owned by you. To avoid this, supply lprm with the job number to remove when a critical job that you own is next in line.

Only the privileged user can remove print jobs submitted from another host.

Flags

-P printer
 Specify the queue associated with a specific printer. Otherwise the value of the PRINTER variable in the environment is used. If this variable is unset, the queue for the default printer is used.
Remove all jobs owned by you. If invoked by the privileged user, all jobs in the spool are removed. Job ownership is determined by the user’s login name and host name on the machine where the lpr command was executed.

Files
/var/spool/lp/* spooling directories

Error Codes
lprm: printer: unknown printer
The printer was not found in the System V LP database. Usually this is a typing mistake; however, it may indicate that the printer does not exist on the system. Use lpstat -p to get the status of printers.

lprm: error on opening queue to spooler
The connection to lpsched on the local machine failed. This usually means the printer server started at boot time has died or is hung. Check if the printer spooler daemon /usr/lib/lp/lpsched is running.

lprm: Can’t send message to the LP print service
lprm: Can’t receive message from the LP print service
These indicate that the LP print service has been stopped. Get help from the system administrator.

lprm: Received unexpected message from the LP print service
It is likely there is an error in this software. Get help from system administrator.

lprm: Can’t cancel request
You are not allowed to remove another user’s print request.

Related Information
The lpr command, lpq command, lpr command, and lpsched command.

lpsched Command
Note: This is a System V Print Subsystem command.

Purpose
Starts/stops the print service.

Syntax
/usr/lib/lp/lpsched

lpshut

Description
The lpsched command starts the LP print service.

The lpshut command shuts down the print service. All printers that are printing at the time the lpshut command is invoked stop printing. When lpsched is started again, requests that were printing at the time a printer was shut down are reprinted from the beginning.

You must have the appropriate privilege to run these commands.
If the scheduler fails to run, check the `lpsched` log file, which contains all failed attempts to load print requests, printer descriptions, forms, filters, classes, alerts, and systems. The log files are located in `/var/lp/logs`. Useful information on the networked print service can also be found in the `/var/lp/logs/lpNet` log file.

**Files**

`/var/spool/lp/*`

**Related Information**

The `accept` command, `enable` command, `lpadmin` command.

**Ipstat Command**

The `lpstat` command includes information for the AIX Print Subsystem `lpstat` and the System V Print Subsystem `lpstat`.

**AIX Print Subsystem Ipstat Command**

**Purpose**

Displays line printer status information.

**Syntax**

`lpstat [-a List | -c List | -d | -e List | -p List | -r | -s | -t | -u List | -v List | -W]`

**Description**

The `lpstat` command displays information about the current status of the line printer.

If no flags are given, `lpstat` prints the status of all requests made by the `lp` command.

Flags can appear in any order and can be repeated. Some flags take an optional list as a parameter. Enter the list as either a list of items separated by commas, as in `lpstat -a Queue1,Queue2`, or as a list of items enclosed in single or double quotes and separated either by commas or one or more spaces, as in, for example, `lpstat -a"Queue1 Queue2"` or `lpstat -a"Queue1,Queue2"` or `lpstat -a'Queue1 Queue2'` or `lpstat -a'Queue1,Queue2'`.

If you specify a flag with no parameters, all information pertaining to that flag is printed.

The display generated by the `lpstat` command contains two entries for remote queues. The first entry contains the client's local queue and local device name and its status information. The second entry contains the client's local queue name followed by the remote queue name. The spooling subsystem first displays remote print requests on the local queue. When the remote machine begins to process the remote print job, the status display for the print job moves to the remote queue.

When a status command communicates with a remote host, the display occasionally appears to hang while the command waits for a response from the remote machine. The command eventually times out if no connection is established between the two machines.
Flags

-aList  Provides status and job information on queues. Specifying the lpstat command with this flag is the same as specifying the enq -q -PQueue1 -PQueue2 ... command (where Queue1, Queue2, etc., are items in List).
-cList  Provides status and job information on queues. Specifying the lpstat command with this flag is the same as specifying the enq -q -PQueue1 -PQueue2 ... command (where Queue1, Queue2, etc., are items in List).
-d      Prints the status information for the system default destination for the lp command. Specifying the lpstat command with this flag is the same as specifying the enq -q command.
-oList  Prints the status of print requests or print queues. List is a list of intermixed printer names and job numbers.
-pList  Prints the status of printers.
-r      Provides status and job information on queues. Specifying the lpstat command with this flag is the same as specifying the enq -A command.
-s      Displays a status summary, including a list of printers and their associated devices. Specifying the lpstat command with this flag is the same as specifying the enq -A command.
-t      Displays all status information, including a list of printers and their associated devices. Specifying the lpstat command with this flag is the same as specifying the enq -AL command.
-uList  Prints the status of all print requests for users specified in List. List is a list of login names. Specifying the lpstat command with this flag is the same as specifying the enq -u UserName command.
-vList  Prints the status of printers. The List variable is a list of printer names.
-W      Displays a wide version of status information with longer queue names, device names, and job numbers. Longer job number information is available on AIX 4.3.2 and later. This flag cannot be used with the -t flag. If the -t flag and the -W flag are used simultaneously, the first one specified takes precedence.

Exit Status
This command returns the following exit values:

0  Successful completion.
>0  An error occurred.

Examples
1. To display the status for all print queues, enter:
   lpstat
2. To display the long status for all printers, enter:
   lpstat -t
3. To display a job number in the print queue lp0, enter:
   lpstat -p lp0
   This command displays a list similar to the following:
   Queue  Dev  Status  Job   Files  User  PP  %  Blks  CP  Rnk
   lp0   dlp0  running  39  motd  guest  10  83  12  1  1
4. To display the status for users root, ghandi, and king, enter:
   lpstat -u root,ghandi,king
5. To display the status of all print queues in wide format for AIX 4.3.2 or later, enter:
   lpstat -W
Files
/var/spool/lpd/* Contains temporary copies of remote enqueued files.

Related Information
The disable command, enable command, enq command, lp command, lpr command, qchk command.


System V Print Subsystem Ipstat Command

Purpose
Prints information about the status of the LP print service.

Syntax
Ipstat [flags] [request-ID-list]

Description
The Ipstat command displays information about the current status of the LP print service. If no flags are given, Ipstat displays the status of all print requests made by you. (See lp for more information.)

The command Ipstat -o printername is used to list all the requests queued on the specified printer. If printername points to a remote printer (see ladmin), then Ipstat -o printername lists all the requests on the remote printer, not just those submitted locally.

Any arguments that are not flags are assumed to be request-IDs as returned by lp. The Ipstat command displays the status of such requests. The flags may appear in any order and may be repeated and intermixed with other arguments. Some of the keyletters below may be followed by an optional list that can be in one of two forms:
- a list of items separated by commas, for example, -p printer1,printer2
- a list of items separated by spaces and enclosed in quotes, for example, -u "user1 user2 user3"

Specifying all after any keyletter that takes list as an argument causes all information relevant to the keyletter to be displayed. For example, the command Ipstat -a all lists the accepting status of all print destinations.

The omission of a list following such keyletters causes all information relevant to the keyletter to be displayed. For example, the command Ipstat -a is equivalent to Ipstat -a all.

There are two exceptions to the behavior of the all keyword. The first is when it is used in conjunction with the -o flag: Ipstat -o all only lists requests submitted locally to remote printers. The second is when it is used with directory-enabled print queues. Use of the all keyword will only return non-directory-enabled print queues. Ipstat -a list will report whether the both directory-enabled and non-directory-enabled print queues in list are accepting requests. For the -a and -b flags, Ipsched will remember the directory-enabled print queues specified until it is restarted. Subsequent calls to Ipstat -a and Ipstat -p will report the status of all non-directory-enabled print queues as well as the directory-enabled print queues previously specified. Once Ipsched has been restarted, the use of the all keyword with the Ipstat
command will once again only display non-directory-enabled print queues. The \texttt{dslpsearch} command should be used to search for defined directory-enabled print queues.

**Flags**

-\texttt{a} [\texttt{list}]
  Report whether print destinations are accepting requests. \texttt{list} is a list of intermixed printer names and class names.

-\texttt{c} [\texttt{list}]
  Report names of all classes and their members. \texttt{list} is a list of class names.

-\texttt{d}
  Report what the system default destination is (if any).

-\texttt{f} [\texttt{list}] [-\texttt{l}]
  Verify that the forms in \texttt{list} are recognized by the LP print service. \texttt{list} is a list of forms; the default is \texttt{all}. The -\texttt{l} option will list the form parameters.

-\texttt{o} [\texttt{list}] [-\texttt{l}]
  Report the status of print requests. \texttt{list} is a list of intermixed printer names, class names, and \texttt{request-ID}s. The keyletter -\texttt{o} may be omitted. The -\texttt{l} option lists for each request whether it is queued for, assigned to, or being printed on a local printer, the form required (if any), and the character set or print wheel required (if any). Note that required forms (if any) are not listed for remote printers.

-\texttt{p} [\texttt{list}] [-D] [-\texttt{l}]
  If the -\texttt{D} flag is given, a brief description is printed for each printer in \texttt{list}. If the -\texttt{l} flag is given, a full description of each printer's configuration is given, including the form mounted, the acceptable content and printer types, a printer description, the interface used, and so on.

  In order to maintain system security access information, the information needed to produce the printer status given by \texttt{lpstat -p} is available only if the LP scheduler is running.

-\texttt{r}
  Report the status of the LP request scheduler (whether it is running).

-\texttt{R}
  Report a number showing the rank order of jobs in the print queue for each printer.

-\texttt{s} [-\texttt{l}]
  Display a status summary, including the status of the LP scheduler, the system default destination, a list of class names and their members, a list of printers and their associated devices, a list of the systems sharing print services, a list of all forms and their availability, and a list of all recognized character sets and print wheels. The -\texttt{l} flag displays all parameters for each form and the printer name where each character set or print wheel is available.

-\texttt{S} [\texttt{list}] [-\texttt{l}]
  Verify that the character sets or the print wheels specified in \texttt{list} are recognized by the LP print service. Items in \texttt{list} can be character sets or print wheels; the default for \texttt{list} is \texttt{all}. If the -\texttt{l} flag is given, each line is appended by a list of printers that can handle the print wheel or character set. The list also shows whether the print wheel or character set is mounted or specifies the built-in character set into which it maps.

-\texttt{t} [-\texttt{l}]
  Display all status information: all the information obtained with the -\texttt{s} flag, plus the acceptance and idle/busy status of all printers and status of all requests. The -\texttt{l} flag displays more detail as described for the -\texttt{f}, -\texttt{o}, -\texttt{p}, and -\texttt{s} flag.

-\texttt{u} [\texttt{login-ID-list}]
  Display the status of output requests for users. The \texttt{login-ID-list} argument may include any or all of the following constructs:

  \texttt{login-ID}
  a user on the local system

  \texttt{system-name/login-ID}
  a user on system \texttt{system-name}
system-name
all
   all users on system system-name
all!login-ID
   a user on all systems
all
   all users on the local system
all!all
   all users on all systems

The default value of login-ID-list is all.

Ipsystem Command

Note: This is a System V Print Subsystem command.

Purpose
Registers remote systems with the print service.

Syntax
Ipsystem [T Timeout] [R Retry] [Y Comment] SystemName [ SystemName . . . ]
Ipsystem [R] [ SystemName . . . ]
Ipsystem [r] SystemName [SystemName . . . ]
Ipsystem [A]

Description
The Ipsystem command defines parameters for the LP print service, with respect to communication (via a high-speed network such as TCP/IP) with remote systems.

Specifically, the Ipsystem command defines remote systems with which the local LP print service can exchange print requests. These remote systems are described to the local LP print service in terms of several parameters that control communication: type, retry, and timeout. These parameters are defined in /etc/lp/environments. You can edit this file with a text editor (such as vi), but editing is not recommended. By using Ipsystem, you can ensure that Ipsched is notified of any changes to the Systems file.

The Timeout parameter specifies the length of time (in minutes) that the print service should allow a network connection to be idle. If the connection to the remote system is idle (that is, there is no network traffic) for N minutes, then drop the connection. (When there is more work, the connection is re-established.) Legal values are n, 0, and N, where N is an integer greater than 0. If a decimal number is used for N, it is truncated to the whole number. The value n means never time out; 0 means as soon as the connection is idle, drop it. The default is n.

The Retry parameter specifies the length of time (in minutes) to wait before trying to re-establish a connection to the remote system, when the connection was dropped abnormally (that is, a network error). Legal values are n, 0, and N, where N is an integer greater than 0. It means wait N minutes before trying to reconnect. If a decimal number is used for N, it is truncated to the whole number. (The default is 10 minutes.) The value n means do not retry dropped connections until there is more work; 0 means try to reconnect immediately.

The Comment parameter allows you to associate a free form comment with the system entry. This is visible when Ipsystem -I is used.
The *SystemName* is the name of the remote system from which you want to be able to receive jobs and to which you want to be able to send jobs. A special entry is provided with the `/etc/lp/Systems` file by default, which allows all connections to *bsd* systems. That entry uses the asterisk (`*`) as the *SystemName*.

The command **lpsystem** `-l [ SystemName ]` prints out a description of the parameters associated with *SystemName* (if a system has been specified) or with all the systems in its database (if *SystemName* has not been specified).

The command **lpsystem** `-r SystemName` removes the entry associated with *SystemName*. The print service no longer accepts jobs from that system or send jobs to it, even if the remote printer is still defined on the local system. The scheduler must be running when the removal of a systems file entry occurs, because the scheduler checks whether the system entry is currently used by a printer destination. If currently used, the system entry cannot be removed.

If you use **lpsystem** `-r SystemName` to remove a system and you have active printers for that system, you will not be allowed to remove the system from the system file. **lpsystem** `-r SystemName` only works if no printers for that system exist.

With respect to the semantics of the *Timeout* and *Retry* values, the print service uses one process for each remote system with which it communicates, and it communicates with a remote system only when there is work to be done on that system or work is being sent from that system.

The system initiating the connection is the master process, and the system accepting the connection is the secondary process. This designation serves only to determine which process dies (the secondary) when a connection is dropped. This helps prevent more than one process communicating with a remote system. All connections are bi-directional, regardless of the master/secondary designation. You cannot control a system's master/secondary designation. Typically, a client machine has the master child, and the server machine has the secondary child. If a master process times out, then both the secondary and master exit. If a secondary process times out, then it is possible that the master may still live and retry the connection after the retry interval. Therefore, one system's resource management strategy can affect another system's strategy.

All forms of the **lpsystem** command accept `*` (asterisk enclosed in double quotes) for *SystemName*.

Depending upon the configuration of the name server, you may need to change the entry in the *SystemName* field in `/etc/lp/Systems` to a full domain name.

**Flags**

- **-A**
  - **-l [ SystemName ]**
  - **-r SystemName**
  - **-R Retry**
  - **-T Timeout**
  - **-y Comment**

  *-A* Prints out the TCP/IP address in a format.
  *-l [ SystemName ]* Prints out a description of the parameters associated with *SystemName* or with all the systems in its database.
  *-r SystemName* Removes the entry associated with *SystemName*.
  *-R Retry* Specifies time to wait before trying to reestablish a connection for a remote system.
  *-T Timeout* Specifies the time allowed for a network connection to be idle. *Timeout* is in minutes. Default is to never time out.
  *-y Comment* The *Comment* parameter allows you to associate a free-form comment with the system entry.

**Security**

Only a user with appropriate privileges may execute the **lpsystem** command.
Files
/etc/lp/*
/var/spool/lp/*

Related Information
The \texttt{lpsched} command.

\textbf{Iptest Command}

\textbf{Purpose}
Generates the line printer ripple pattern.

\textbf{Syntax}
\texttt{lptest [ ;Length Count ]}

\textbf{Description}
The \texttt{lptest} command writes the traditional “ripple” test pattern on a standard output device such as a terminal or a printer. In 96 lines, this pattern will print all 96 printable ASCII characters in each position. While originally created to test printers, the ripple pattern is quite useful for testing terminals, driving terminal ports for debug purposes, or any other task where a quick supply of random data is needed.

Using the \texttt{lptest} command, you can specify the output line length if the default length of 79 is not appropriate. You can also specify the number of output lines to be generated if the default \textit{Count} parameter of 200 is not appropriate. Note that if \textit{Count} parameter is specified, \textit{Length} must also be specified.

\textbf{Examples}
To display or print 100 lines of 80-column test output to standard output, enter:
\texttt{lptest 80 100}

\textbf{Related Information}
The \texttt{cancel} command, \texttt{disable} command, \texttt{enable} command, \texttt{lp} command, \texttt{lpstat} command.

\textbf{Ipusers Command}

\textit{Note:} This is a System V Print Subsystem command.

\textbf{Purpose}
Set printing queue priorities.

\textbf{Syntax}
\texttt{ipusers \texttt{-d} PriorityLevel}
\texttt{ipusers \texttt{-q} PriorityLimit \texttt{-u} LoginIDList}
\texttt{ipusers \texttt{-u} LoginIDList}
\texttt{ipusers \texttt{-q} PriorityLimit}
**lpusers [-l]**

**Description**
The `lpusers` command sets limits to the queue priority level that can be assigned to jobs submitted by users of the LP print service.

The first form of the command (with `-d`) sets the system-wide priority default to `PriorityLevel`, where `PriorityLevel` is a value of 0 to 39, with 0 being the highest priority. If a user does not specify a priority level with a print request (see the `lp` command), the default priority is used. Initially, the default priority level is 20.

The second form of the command (with `-q` and `-u`) sets the default `PriorityLimit` (from 0 to 39) that the users in the `LoginIDList` can request when submitting a print request. The `LoginIDList` parameter may include any or all of the following constructs:

Users that have been given a limit cannot submit a print request with a higher priority level than the one assigned, nor can they change a request already submitted to have a higher priority. Any print requests submitted with priority levels higher than allowed will be given the highest priority allowed.

The third form of the command (with `-u`) removes any explicit priority limit for the specified users.

The fourth form of the command (with `-q`) sets the default priority limit for all users not explicitly covered by the use of the second form of this command.

The last form of the command (with `-l`) lists the default priority level and the priority limits assigned to users.

**Parameters**

- **LoginID**
  - `system_name!login-ID`
  - `system_name!all`
  - `all!login-ID`
  - `all`
  Specifies a user on the local system.
  User on the system `system_name`
  Users on system `system_name`
  User on all systems
  Users on the local system

**Flags**

- `-d PriorityLevel`
  Sets the system-wide priority default to `PriorityLevel`.

- `-l`
  Lists the default priority level and the priority limits assigned to users.

- `-q PriorityLimit`
  Sets the default highest priority level for all users not explicitly covered.

- `-q PriorityLimit -u LoginIDList`
  Sets the default highest priority level users in `LoginIDList` can request when submitting a print request.

- `-u LoginIDList`
  Removes any explicit priority level for the specified users.

**Related Information**
The `lp` command.
Is Command

Purpose
Displays the contents of a directory.

Syntax

**To Display Contents of Directory or Name of File**

```
ls [ -1 ] [ A ] [ C ] [ F ] [ H ] [ L ] [ N ] [ R ] [ a ] [ b ] [ c ] [ d ] [ e ] [ f ] [ g ] [ j ] [ l ] [ m ] [ n ] [ o ] [ p ] [ q ] [ r ] [ s ] [ u ] [ x ] [ File ... ]
```

**To Display Contents of Directory**

```
ls [ C ] [ d ] [ f ] [ m ] [ s ] [ x ] [ t ] [ Directory ... ]
```

Description

The `ls` command writes to standard output the contents of each specified Directory parameter or the name of each specified File parameter, along with any other information you ask for with the flags. If you do not specify a File or Directory parameter, the `ls` command displays the contents of the current directory.

Specifying more than one of the options in the mutually exclusive pairs is not considered an error. The last option specified in each pair determines the output format.

By default, the `ls` command displays all information in alphabetic order by file name. The collating sequence is determined by the LANG or LC_COLLATE environment variable. The "National Language Support Overview for Programming" in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs contains more details.

When the `ls` command displays the contents of a directory, it does not show entries for files whose names begin with a . (dot) unless you use the -a or -A flag. If the command is executed by root, it uses the -A flag by default.

There are three main ways to format the output:

- List one entry per line.
- List entries in multiple columns by specifying either the C or x flag. The C flag is the default format when output is to a tty. The `ls` command displays single column output if file or directory names are too long.
- List entries in a comma-separated series by specifying the m flag.

To determine the number of character positions in the output line, the `ls` command uses the COLUMNS environment variable. If this variable is not set, the command gets the current column value of the display. If the `ls` command cannot determine the number of character positions by either of these methods, it uses a default value of 80.

The mode displayed with the e and f flags is interpreted as follows:

If the first character is:

**d**  The entry is a directory.
**b**  The entry is a block special file.
**c**  The entry is a character special file.
**l**  The entry is a symbolic link, and either the N flag was specified or the symbolic link did not point to an existing file.
**p**  The entry is a first-in, first-out (FIFO) special file.

Alphabetical Listing of Commands 271
The next nine characters are divided into three sets of three characters each. The first set of three characters show the owner’s permission. The next set of three characters show the permission of the other users in the group. The last set of three characters shows the permission of anyone else with access to the file. The three characters in each set indicate, respectively, read, write, and execute permission of the file. Execute permission of a directory lets you search a directory for a specified file.

Permissions are indicated as follows:

r Read
w Write (edit)
x Execute (search)
- Corresponding permission not granted

The group-execute permission character is s if the file has set-group-ID mode. The user-execute permission character is s if the file has set-user-ID mode. The last character of the mode (usually x or -) is T if the 01000 (octal) bit of the mode is set (see the chmod command for the meaning of this mode). The indications of set-ID and 01000 bit of the mode are capitalized (S and T, respectively) if the corresponding execute permission is not set. The mode t indicates that the sticky bit is on for the file or the directory.

The mode displayed with the -e flag is the same as with the -l flag, except for the addition of an 11th character interpreted as follows:

+ Indicates a file has extended security information. For example, the file may have extended ACL, TCB, or TP attributes in the mode.

The access control information (ACL) of a file is displayed by using the aclget command. The value of the TCB and TP attributes are displayed by using the chtcb command.

- Indicates a file does not have extended security information.

When the size of the files in a directory are listed, the ls command displays a total count of blocks, including indirect blocks.

Flags

-A Lists all entries except . (dot) and .. (dot-dot).
-a Lists all entries in the directory, including the entries that begin with a . (dot).
-b Displays nonprintable characters in an octal (\nnn) notation.
-c Uses the time of last modification of the i-node for either sorting (when used with the -r flag) or for displaying (when used with the -l flag). This flag must be used with either the -r or -l flag, or both.
-C Sorts output vertically in a multicolunm format. This is the default method when output is to a terminal.
-d Displays only the information for the directory named. Directories are treated like files, which is helpful when using the -l flag to get the status of a directory.
-e Displays the mode (including security information), number of links, owner, group, size (in bytes), time of last modification, and name of each file. If the file is a special file, the size field contains the major and minor device numbers. If the file is a symbolic link, the path name of the linked-to file is printed preceded by a -r (minus, greater than) sign. The attributes of the symbolic link are displayed.
-E Lists space reservation, fixed extent size, and extent allocation flag information for a file. -l must be specified with this flag.
-f Lists the name in each slot for each directory specified in the Directory parameter. This flag turns off the -r, -s, and -l flags, and turns on the -a flag. The order of the listing is the order in which entries appear in the directory.

272 Commands Reference, Volume 3
-F Puts a / (slash) after each file name if the file is a directory, an * (asterisk) if the file can be executed, an = (equal sign) if the file is a socket, a | (pipe) sign if the file is a FIFO, and an @ for a symbolic link.

**Note:** Symbolic links are displayed with the trailing -> only if the -N flag is used or if the link points to a nonexistent file. Otherwise, information about the target file is displayed. You can also invoke this option by entering the ls[-l] command.

-g Displays the same information as the [-l] flag, except the -g flag suppresses display of the owner and symbolic link information.

-H If a symbolic link referencing a file of type directory is specified on the command line, the ls command shall evaluate the file information and file type to be those of the file referenced by the link, and not the link itself; however, the ls command shall write the name of the link itself and not the file referenced by the link.

-i Displays the i-node number in the first column of the report for each file.

-L Lists the file or directory contents that the link references. This is the default action. Symbolic links are followed. If the -I option is used, the -N option becomes the default, and no symbolic links are followed. When the -I option is used, only the -L option can override the -N default.

-l (Lower case L) Displays the mode, number of links, owner, group, size (in bytes), and time of last modification for each file. If the file is a special file, the size field contains the major and minor device numbers. If the time of last modification is greater than six months ago, the time field is shown in the format month date year where as files modified within six months the time field is shown as month date time format.

If the file is a symbolic link, the path name of the linked-to file is printed preceded by a ->. The attributes of the symbolic link are displayed. The -m[-l] and -o flag overrides the -I flag.

**Notes:**

1. A symbolically linked file is followed by an arrow and the contents of the symbolic link.
2. The performance of the ls command when used with the -l option can be improved by executing the mkpasswd command. This is helpful when a directory contains files owned by different users, such as the /tmp directory.

-m Uses stream output format (a comma-separated series).

-n Displays the same information as the [-l] flag, except that the -n flag displays the user and the group IDs instead of the user and group names.

-N Does not follow symbolic links when determining the status of a file.

**Note:** If both the -L and -N options are used, the last one will dominate. Also, any time a symbolic link is given that includes a / (slash) as the final character, the link will automatically be followed regardless of any options used.

-o Displays the same information as the [-l] flag, except the -o flag suppresses display of the group and symbolic link information.

-p Puts a slash after each file name if that file is a directory. This is useful when you pipe the output of the ls command to the pr command, as follows:

```
ls -p | pr -5 -t -w80
```

-q Displays nonprintable characters in file names as a ? (question mark).

-r Reverses the order of the sort, giving reverse alphabetic or the oldest first, as appropriate.

-R Lists all subdirectories recursively.

-s Gives size in kilobytes (including indirect blocks) for each entry.

-t Sorts by time of last modification (latest first) instead of by name.

-u Uses the time of the last access, instead of the time of the last modification, for either sorting (when used with the [-l] flag) or for displaying (when used with the [-l] flag). This flag has no effect if it is not used with either the [-l] or [-r] flag, or both.

-x Sorts output horizontally in a multicolumn format.

-1 Forces output into one-entry-per-line format. This is the default when the output is not directed to a terminal.
Exit Status
This command returns the following exit values:

0    All files were written successfully.
>0   An error occurred.

Examples
1. To list all files in the current directory, type:
   
   ls

   This lists all files, including . (dot), .. (dot-dot), and other files with names beginning with a dot.

2. To display detailed information, type:
   
   ls -l chap1 .profile

   This displays a long listing with detailed information about chap1 and .profile.

3. To display detailed information about a directory, type:
   
   ls -l -d . manual manual/chap1

   This displays a long listing for the directories . and manual, and for the file manual/chap1. Without the
   -d flag, this would list the files in the . and manual directories instead of the detailed information about
   the directories themselves.

4. To list the files in order of modification time, type:
   
   ls -l -t

   This displays a long listing of the files that were modified most recently, followed by the older files.

Files

/usr/bin/ls
/etc/passwd
/etc/group
/usr/share/lib/terminfo/*

Contains the ls command.
Contains user IDs.
Contains group IDs.
Contains terminal information.

Related Information

The aclget, chmod, chtcb, find, mkpasswd, qprt command.

the structure and characteristics of directories in the file system.

Files and Directories Overview in AIX 5L Version 5.2 System User's Guide: Operating System and
Devices.

Files Overview in AIX 5L Version 5.2 System User's Guide: Operating System and Devices describes files,
file types, and how to name files.

Linking Files and Directories explains the concept of file linking.
Is-secldapclntd Command

Purpose
The **is-secldapclntd** command lists the status of the **secldapclntd** daemon process.

Syntax
```
/usr/sbin/is-secldapclntd
```

Description
The **is-secldapclntd** command lists the **secldapclntd** daemon status. The information returned includes the following:
- The LDAP server the **secldapclntd** daemon is talking to
- The LDAP server port number
- The version of the LDAP protocol used
- User base DN
- Group base DN
- System (id) base DN
- User cache size
- User cache size used
- Group cache size
- Group cache size used
- Cache time out (time to live) value
- **secldapclntd** to LDAP server heart beat interval
- Number of thread used by **secldapclntd** daemon
- User objectclass used in the LDAP server
- Group objectclass used in the LDAP server

Example
1. To list the status of the **secldapclntd** daemon, type:
   ```
   /usr/sbin/is-secldapclntd
   ```

Files
- **/etc/security/ldap/ldap.cfg** Contains information needed by the **secldapclntd** daemon to connect to the server.

Related Information
- The **secldapclntd** daemon
The `mkseclap`, `stop-seclapclntd`, `start-seclapclntd`, `restart-seclapclntd` and `flush-seclapclntd` commands.

The `/etc/security/ldap/ldap.cfg` file.

### Isactdef Command

#### Purpose
Displays the action definitions of a resource or a resource class.

#### Syntax
To display the action definitions of a resource:

```
```

To display the action definitions of a resource class:

```
```

To display all resource class names:

```
isactdef
```

#### Description
The `isactdef` command displays a list of the action definitions of a resource or a resource class. By default, this command displays the action definitions of a resource. To see the action definitions of a resource class, specify the `-c` flag.

If you do not specify any actions on the command line, this command only displays actions that are defined as `public`. To override this default, use the `-p` flag or specify on the command line the names of the actions that have definitions you want to display.

To see the structured data definition that is required as input when this action is invoked, specify the `-s i` flag. To see the structured data definition linked with the output that results from invoking this action, specify the `-s o` flag.

By default, this command does not display action descriptions. To display action definitions and descriptions, specify the `-e` flag.

#### Flags
- `-a` Specifies that this command applies to all nodes in the cluster. The cluster scope is determined by the `CT_MANAGEMENT_SCOPE` environment variable. If it is not set, first the management domain scope is chosen if it exists, then the peer domain scope is chosen if it exists, and then local scope is chosen, until the scope is valid for the command. The command will run once for the first valid scope found. For example, if both a management and peer domain exist, `isactdef -a` with `CT_MANAGEMENT_SCOPE` not set will list the management domain. In this case, to list the peer domain, set `CT_MANAGEMENT_SCOPE` to 2.

- `-c` Displays the action definitions for `resource_class`.

- `-d` Specifies delimiter-formatted output. The default delimiter is a colon (:). Use the `-D` flag if you want to change the default delimiter.
-D delimiter
   Specifies delimiter-formatted output that uses the specified delimiter. Use this flag to specify a
delimiter other than the default colon (:) . An example is when the data to be displayed contains
colons. Use this flag to specify a delimiter of one or more characters.

-e
   Specifies expanded format. Displays descriptions along with the action definitions.

-i
   Specifies input format. Generates a template of the Resource_Data_Input file. The output is
displayed in long (stanza) format. The attribute’s SD element data types are displayed as the
value in the attr=value pairs. It is suggested that when you use this flag, the output of the lsactdef
command be directed to a file. This flag overrides the -s o flag.

-l
   Specifies "long" format — one entry per line. This is the default display format. If the lsactdef
command is issued with the -l flag, but without a resource class name, the -l flag is ignored when
the command returns the list of defined resource class names.

-p property
   Displays actions with the specified property. By default, only the definitions for public actions are
displayed. To display all action definitions regardless of the action property, use the -p 0 flag.

   **Action properties:**
   
   0x0001    long_running
   0x0002    public
   
   A decimal or hexadecimal value can be specified for the property. To request the action definitions
for all actions that have one or more properties, “OR” the properties of interest together and then
specify the "OR"ed value with the -p flag. For example, to request the action definitions for all
actions that are long_running or public, enter:
   
   -p 0x03

-s i l o
   Displays the structured data definition for the action input or action response.
   
   i    Displays the action input structured data definitions. This is the default.
   
   o    Displays the action response (output) structured data definitions.

-t
   Specifies table format. Each attribute is displayed in a separate column, with one resource per
line.

-x
   Suppresses header printing.

-h
   Writes the command’s usage statement to standard output.

-T
   Writes the command’s trace messages to standard error. For your software-service organization’s
use only.

-V
   Writes the command’s verbose messages to standard output.

**Parameters**

*resource_class*
   Specifies the name of the resource class with the action definitions that you want to
display. If *resource_class* is not specified, a list of all of the resource class names is
displayed.

*action1 [action2...]*
   Specifies one or more actions. If *resource_class* is specified, zero or more action names
   can be specified. If no actions are specified, all of the action definitions for *resource_class*
   are displayed. Enter specific action names to control which actions are displayed and in
what order. Use blank spaces to separate action names.
Security
The user needs read permission for the resource_class specified in lsactdef to run lsactdef. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status
0  The command has run successfully.
1  An error occurred with RMC.
2  An error occurred with the command-line interface (CLI) script.
3  An incorrect flag was specified on the command line.
4  An incorrect parameter was specified on the command line.
5  An error occurred with RMC that was based on incorrect command-line input.

Environment Variables
CT_CONTACT
When the CT_CONTACT environment variable is set to a host name or IP address, the command contacts the Resource Monitoring and Control (RMC) daemon on the specified host. If the environment variable is not set, the command contacts the RMC daemon on the local system where the command is being run. The resource class or resources that are displayed or modified by the command are located on the system to which the connection is established.

CT_MANAGEMENT_SCOPE
Determines the management scope that is used for the session with the RMC daemon to monitor and control the resources and resource classes. The management scope determines the set of possible target nodes where the resources and resource classes can be monitored and controlled. The valid values are:
0  Specifies local scope.
1  Specifies local scope.
2  Specifies peer domain scope.
3  Specifies management domain scope.

If this environment variable is not set, local scope is used.

Standard Output
When the -h flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
1. To list the names of all of the resource classes, enter:
   lsactdef

   The output will look like this:
   class_name
   "IBM.Association"
   "IBM.AuditLog"
   "IBM.AuditLogTemplate"
   "IBM.Condition"
"IBM.EventResponse"
"IBM.Host"
"IBM.Program"
"IBM.Sensor"
"IBM.ManagedNode"
...

2. To list the public resource action definitions for resource class IBM®.AuditLog, enter:

   lsactdef IBM.AuditLog

   The output will look like this:

   Resource Action Definitions for
   class_name: IBM.AuditLog
   action 1:
     action_name = "GetRecords"
     display_name = ""
     description = ""
     properties = {"public"}
     confirm_prompt = ""
     action_id = 0
     variety_list = {{1..1}}
     variety_count = 1
     timeout = 0
   action 2:
     action_name = "DeleteRecords"
     display_name = ""
     description = ""
     properties = {"public"}
     confirm_prompt = ""
     action_id = 1
     variety_list = {{1..1}}
     variety_count = 1
     timeout = 0
   ....

3. To list the structured data definition required for invoking the action on resources in resource class IBM.AuditLog, action GetRecords, enter:

   lsactdef -s -i IBM.AuditLog GetRecords

   The output will look like this:

   Resource Action Input for: IBM.AuditLog
   action_name GetRecords:
   sd_element 1:
     element_name = "MatchCriteria"
     display_name = ""
     description = ""
     element_data_type = "char_ptr"
     element_index = 0
   sd_element 2:
     element_name = "IncludeDetail"
     display_name = ""
     description = ""
     element_data_type = "uint32"
     element_index = 1

Location

/usr/sbin/rsct/bin/lsactdef Contains the lsactdef command

Related Information

Commands: lsrsrdef

Files: rmccli
Isallq Command

Purpose
Lists the names of all configured queues.

Syntax
Isallq [-c]

Description
The Isallq command lists the names of all configured queues contained in the /etc/qconfig file. By specifying the -c flag, this listing is displayed in colon format. This flag is used mainly by SMIT.

You can use the Printer Queues application in Web-based System Manager (wsm) to change print queue characteristics. You could also use the System Management Interface Tool (SMIT) smit Isallq fast path to run this command.

Flag
-c Causes colon format output for use by SMIT.

Examples
1. To list all of the queue names in the /etc/qconfig file, enter:
   Isallq

   A listing similar to the following is displayed:
   lp0
   lp1
   lp2

2. To list all configured queues in colon format, enter:
   Isallq -c

   A listing similar to the following is displayed:
   lp0
   lp0:queue1
   lp0:queue2
   lp1

Files
/usr/bin/isallq Contains the Isallq command.
/etc/qconfig Configuration file.

Related Information
The chque command, lsq command, lsallqdev command, mkque command, rmque command.

The qconfig file.

Listing Print Queues and Print Queue Devices in the AIX 5L Version 5.2 Guide to Printers and Printing.
For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

Isallqdev Command

Purpose
Lists all configured printer and plotter queue device names within a specified queue.

Syntax
Isallqdev [-c] [-q Name]

Description
The Isallqdev command lists all configured device names within a specified queue in the /etc/qconfig file.

You can use the Printer Queues application in Web-based System Manager (wsm) to change print queue characteristics. You could also use the System Management Interface Tool (SMIT) smit Isallqdev fast path to run this command.

Flags
-q Name Specifies the queue name.
-c Causes colon format output for use by SMIT.

Examples
1. To list the names all of the configured queue devices within the lp0 queue in the /etc/qconfig file, enter:
   Isallqdev -q lp0

   A listing similar to the following is displayed:
   lpd0
   lpd1
   lpd2

2. To list the names of all of the configured queue device within the lp0 queue in the /etc/qconfig file in colon format, enter:
   Isallqdev -q lp0 -c

   A listing similar to the following is displayed:
   lp0:lpd1
   lp0:lpd2

Files
/usr/bin/lsallqdev Contains the lsallqdev command.
/etc/qconfig Configuration file.

Related Information
The chquedev command, lsquedev command, lsallq command, mkquedev command, rmquedev command.
The `qconfig` file.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

Listing Print Queues and Print Queue Devices in the AIX 5L Version 5.2 Guide to Printers and Printing.

---

## Isarm command

### Purpose

Displays Application Response Measurement (ARM) application and process usage information.

### Syntax

```bash
lsarm [-a] [ -g ] [ -u ] [ ApplicationName ... ]
```

or

```bash
lsarm [-p] [ -a ] [ -g ] [ -u ] [ ProcessID ... ]
```

### Description

The `lsarm` command displays information about applications registered with the operating system using the Application Response Measurement (ARM) APIs. The `-a` flag displays information about applications by application name. The `-p` option displays information about the applications used by a process.

### Flags

- `-a`  Displays application names.
- `-g`  Displays group names associated with the application.
- `-p`  Displays transaction names associated with the application.
- `-u`  Displays process numbers using the application.
- `-t`  Displays the applications used by a process.

### Parameters

- **ApplicationName**  Specifies a list of one or more applications for which the `lsarm` command should display information.
- **ProcessID**  Specifies a list of one or more process IDs for which the `lsarm` command should display information.

### Exit Status

- **0**  The command completed successfully.
- **>0**  An error occurred.

### Examples

1. To display all application names, type:
   ```bash
   lsarm -a
   ```
2. To display group names, transaction class names, and process numbers using the 'database' application, type:
   ```bash
   lsarm -a [ 'database' ]
   ```
lsarm -a -g -t -u database

3. To display the process IDs for all processes using ARM applications, type:
   lsarm -p

4. To display the application names, group names, and transaction names used by process 25038, type:
   lsarm -p -a -g -t 25038

Location
/usr/ewlm/sbin/lsarm

Related Information

Isattr Command

Purpose
Displays attribute characteristics and possible values of attributes for devices in the system.

Syntax
Isattr [-R] Name [ -c Class ] [ -s Subclass ] [ -t Type ] [ -a Attribute ] [-f] [ -h ] [ -H ]
Isattr [ -c Class ] [ -s Subclass ] [ -t Type ] [ -o operation [ ... ] ] [-f] Format [-Z Character ] ] [ -f ] File ] [ -h ] [ -H ]

Description
The Isattr command displays information about the attributes of a given device or kind of device. If you do not specify the device logical name (-I Name), you must use a combination of one or all of the -c Class, -s Subclass, and -t Type flags to uniquely identify the predefined device.

You must specify one of the following flags with the Isattr command:

- **-D** Displays default values.
- **-E** Displays effective values (valid only for customized devices specified with the -I flag).
- **-F Format** Specifies the user-defined format.
- **-R** Displays the range of legal values.

When displaying the effective values of the attributes for a customized device, the information is obtained from the Configuration database, not the device. Generally the database values reflect how the device is configured, unless it is reconfigured with the chdev command using the -P or -T flag. If this has occurred, the information displayed by the Isattr command might not correctly indicate the current device configuration until after the next system boot.

If you use the -D or -E flag, the output defaults to the values for the attribute’s name, value, description, and user-settable strings, unless also used with the -O flag. The -O flag displays the names of all attributes specified, separated by colons. On the next line, the -O flag displays all the corresponding attribute values, separated by colons. The -H flag can be used with either the -D, -E, or -F flag to display headers above the column names. You can define the format of the output with a user-specified format.
where the Format parameter is a quoted list of column names separated by nonalphanumeric characters or white space using the -F Format flag. If the -F Format flag is specified, the -Z Character flag may also be specified to change the default record separator from a 'newline' character to the indicated 'Character'.

The lsattr command can display "operation" information from the Extended Predefined Attribute (PdAtXtd) object class. This information is used by the web-based System Manager. The operation information is accessed through the -o operation flag. The -o operation flag and the -a attribute flag cannot be specified in the same invocation of the lsattr command. The -o operation flag is also not valid with the -R flag. When the -o operation flag is specified, only fields from the PdAtXtd object class can be specified with the -F Format flag.

You can supply the flags either on the command line or from the specified File parameter.

**Flags**

- **-a Attribute** Displays information for the specified attributes of a specific device or kind of device. You can use one -a flag for each attribute name or multiple attribute names. If you use one -a flag for multiple attribute names, the list of attribute names must be enclosed in quotes with spaces between the names. Using the -R flag, you must specify only one -a flag with only one attribute name. If you do not specify either the -a or -R flag, the lsattr command displays all information for all attributes of the specified device. The -a Attribute flag cannot be used in conjunction with the -o Operation flag. This combination of flags causes the lsattr command to exit with an error message.

- **-c Class** Specifies a device class name. This flag can be used to restrict the output to that for devices of a specified class. This flag cannot be used with the -E or -R flag.

- **-D** Displays the attribute names, default values, descriptions, and user-settable flag values for a specific device when not used with the -O flag. The -D flag displays only the attribute name and default value in colon format when used with the -O flag. This flag can be used with any combination of the -c, -s, and -I flags that uniquely identifies a device from the Predefined Devices object class or with the -H flag. This flag cannot be used with the -E, -F, or -R flag.

- **-E** Displays the attribute names, current values, descriptions, and user-settable flag values for a specific device when not used with the -O flag. The -E flag displays only the attribute name and current value in colon format when used with the -O flag. This flag cannot be used with the -c, -F, -R, -s, or -I flag.

- **-I File** Reads the needed flags from the File parameter.

- **-F Format** Displays the output in a user-specified format, where the Format parameter is a quoted list of column names separated by nonalphanumeric characters or white space. Using white space as the separator, the lsattr command displays the output in aligned columns. Only column names from the Predefined Attributes (PdAt), Customized Attributes (CuAt), and the Extended Predefined Attributes (PdAtXtd) object classes can be specified. In addition to the column names, there are two special purpose names that can be used. The name description can be used to obtain a display of attribute descriptions and user-settable can be used to obtain an indication as to whether or not an attribute can be changed. This flag cannot be used with the -c, -D, -O, or -R flag.

- **-H** Displays headers above the column output. To use the -H flag with the -O flag is meaningless, the -O flag prevails. To use the -H flag with the -R flag is meaningless, the -R flag prevails.

- **-h** Displays the command usage message.

- **-l Name** Specifies the device logical name in the Customized Devices object class whose attribute names or values are to be displayed.
-o Operation Displays information for the specified operations of a specific device or kind of device. You can use one -o flag for each operation name or multiple operation names. If you use one -o flag for multiple operation names, the list of attribute names must be enclosed in quotes with spaces between the names. Wildcard characters may also be used for the operation name. The valid set of wildcard characters are the same set used by the odmget command. All operations associated with a specific device, or kind of device, can be displayed by using an operation value of "?". The -o Operation flag cannot be used in conjunction with the -a attribute flag or the -R flag. Any combination of these flags causes the lsattr command to exit with an error message.

-O Displays all attribute names separated by colons and, on the second line, displays all the corresponding attribute values separated by colons. The attribute values are current values when the -E flag is also specified and default values when the -D flag is specified. This flag cannot be used with the -F and -R flags.

-R Displays the legal values for an attribute name. The -R flag cannot be used with the -D or -E flags, but can be used with any combination of the -C, -S and -R flags that uniquely identifies a device from the Predefined Devices object class or with the -F flag. The -R flag displays the list attribute values in a vertical column as follows:

Value1
Value2
...
ValueN

The -R flag displays the range attribute values as x...n(+i) where x is the start of the range, n is the end of the range, and i is the increment.

-s Subclass Specifies a device subclass name. This flag can be used to restrict the output to that for devices of a specified subclass. This flag cannot be used with the -E or -F flag.

-t Type Specifies a device type name. This flag can be used to restrict the output to that for devices of a specified class. This flag cannot be used with the -E or -F flag.

-Z Character The -Z Character flag is meant to be used for programs that need to deal with ODM fields that may have embedded new line characters. An example of such a program is the Web-based System Manager. The -Z Character flag is used to change the record separator character for each record (line) of output generated. The new record separator is the ‘Character’ argument to this flag. The -Z Character flag is only relevant when the -F Format flag is specified. The -Z Character flag cannot be used with the -D, -E, -O, or the -R flags.

Examples

1. To list the current attribute values for the tape device rmt0, type:
   
   lsattr -l rmt0 -E
   
   The system displays a message similar to the following:
   
   mode yes Use DEVICE BUFFERS during writes True
   block_size 1024 BLOCK size (0=variable length) True
   extfm yes Use EXTENDED file marks True
   ret no RETENTION on tape change or reset True
   density_set_1 37 DENSITY setting #1 True
   density_set_2 36 DENSITY setting #2 True
   compress yes Use data COMPRESSION True
   size_in_mb 12000 Size in Megabytes False
   ret_error no RETURN error on tape change or reset True

2. To list the default attribute values for the tape device rmt0, type:
   
   lsattr -l rmt0 -D
   
   The system displays a message similar to the following:
   
   mode yes Use DEVICE BUFFERS during writes True
   block_size 1024 BLOCK size (0=variable length) True
   extfm yes Use EXTENDED file marks True
   ret no RETENTION on tape change or reset True
density_set_1 37  DENSITY setting #1    True
density_set_2 36  DENSITY setting #2    True
compress    yes  Use data COMPRESSION    True
size_in_mb  12000  Size in Megabytes    False
ret_error    no   RETURN error on tape change or reset    True

3. To list the current value of the **bus_intr_lvl** attribute for the SCSI adapter scsi0, type:
   `lsattr -l scsi0 -a bus_intr_lvl -E`

The system displays a message similar to the following:
```
bmu_intr_lvl 1  Bus interrupt level False
```

4. To list the possible values of the login attribute for the tty device tty0, type:
   `lsattr -l tty0 -a login -R`

The system displays a message similar to the following:
```
enable
disable
share
delay
```

5. To list the default attribute values for an IBM 4340 parallel printer, type:
   `lsattr -c printer -s parallel -t ibm4340 -D`

The system displays a message similar to the following:
```
ptop  600  Printer TIME OUT period    True
line  60  Number of LINES per page    True
col   80  Number of COLUMNS per page   True
ind   0   Number of columns to INDENT True
plot  no  Send all characters to printer UNMODIFIED True
backspace yes  Send BACKSPACES        True
cr    yes  Send CARRIAGE RETURNS     True
form  yes  Send FORM FEEDS           True
lf    yes  Send LINE FEEDS           True
addcr yes  Add CARRIAGE RETURNS to LINE FEEDS True
case no  Convert lowercase to UPPERCASE True
tabs yes  EXPAND TABS on eight position boundaries True
wrap no  WRAP CHARACTERS beyond the specified width True
mode no  Return on ERROR             True
interface standard  Type of PARALLEL INTERFACE True
autoconfig available STATE to be configured at boot time True
busy_delay 0  Microseconds to delay between characters True
```

6. To list the possible values of the **ptop** attribute for an IBM 4340 parallel printer, type:
   `lsattr -c printer -s parallel -t ibm430 -a ptop -R`

The system displays a message similar to the following:
```
1...1000 (+1)
```

7. To list the current attribute values for the tape device rmt0 in colon-separated format, type:
   `lsattr -l rmt0 -E -O`

The system displays a message similar to the following:
```
```

8. To display system attributes, type:
   `lsattr -E -l sys0`

The system displays output similar to the following:
```
keylock  normal  State of system keylock at boot time   False
maxbuf  20  Maximum number of pages in block I/O BUFFER CACHE True
maxmbuf  0  Maximum Kbytes of real memory allowed for MBUFS True
```
maxuproc  128  Maximum number of PROCESSES allowed per user  True
autosrestart  false  Automatically REBOOT system after a crash  True
iostat  false  Continuously maintain DISK I/O history  True
realmem  4194304  Amount of usable physical memory in Kbytes  False
conslogin  enable  System Console Login  False
fwversion  IBM,SPH00221  Firmware version and revision levels  False
maxpout  0  HIGH water mark for pending write I/Os per file  True
minpout  0  LOW water mark for pending write I/Os per file  True
fullcore  false  Enable full CORE dump  True
pre430core  false  Use pre-430 style CORE dump  True
ncargs  6  ARG/ENV list size in 4K byte blocks  True
autorestart  false  Automatically REBOOT system after a crash  True
realmem  4194304  Amount of usable physical memory in Kbytes  False
conslogin  enable  System Console Login  False
fwversion  IBM,SPH00221  Firmware version and revision levels  False
maxpout  0  HIGH water mark for pending write I/Os per file  True
minpout  0  LOW water mark for pending write I/Os per file  True
fullcore  false  Enable full CORE dump  True
pre430core  false  Use pre-430 style CORE dump  True
ncargs  6  ARG/ENV list size in 4K byte blocks  True
rtasversion  1  Open Firmware RTAS version  False
modelname  IBM,7044-270  Machine name  False
systemid  IBM,011037D1F  Hardware system identifier  False
boottype  disk  N/A  False
SW_dist_intr  false  Enable SW distribution of interrupts  True
cpuguard  disable  CPU Guard  True
frequency  93750000  System Bus Frequency  False

Note: The same information is available in a more readable format using SMIT (System Environments -> Change / Show Characteristics of Operating Systems).

Files

/usr/sbin/lsattr  Contains the lsattr command.

Related Information

The chdev command, Isconn command, lsdev command, lsparent command, mkdev command, rmdev command.

Summary of Tunable Parameters in AIX 5L Version 5.2 Performance Management Guide


Isaudrec Command

Purpose

Lists records from the audit log.

Syntax

lsaudrec [–l] [–a | –n node_name1,node_name2] [–S subsystem_name] [–s selection_string] [–x] [–h] [field_name1 field_name2...]

Description

The lsaudrec command is used to list records in the audit log.

The audit log is a facility for recording information about the system's operation. It can include information about the normal operation of the system as well as failures and other errors. It augments the error log functionality by conveying the relationship of the error relative to other system activities. All detailed information about failures is still written to the AIX® error log.
Records are created in the audit log by subsystems that have been instrumented to do that. For example, the event response subsystem runs in the background to monitor administrator-defined conditions and then invokes one or more actions when a condition becomes true. Because this subsystem runs in the background, it is difficult for the operator or administrator to understand the total set of events that occurred and the results of any actions that were taken in response to an event. Because the event response subsystem records its activity in the audit log, the administrator can easily view its activity as well as that of other subsystems using this command.

Each record in the audit log contains named fields. Each field contains a value that provides information about the situation corresponding to the record. For example, the field named **Time** indicates the time at which the situation occurred. Each record has a set of common fields and a set of subsystem-specific fields. The common fields are present in every record in the audit log. The subsystem-specific fields vary from record to record. Their names are only significant when used with a subsystem name because they may not be unique across all subsystems. Each record is derived from a template that defines which subsystem-specific fields are present in the record and defines a format string that is used to generate a message describing the situation. The format string may use record fields as inserts. A subsystem typically has many templates.

The field names can be used as variables in a *selection string* to choose which records are displayed. A selection string is an expression that is made up of field names, constants, and operators. The syntax of a selection string is similar to an expression in the C programming language or the SQL “where” clause. The selection string is matched against each record using the referenced fields of each record to perform the match. Any records that match are displayed. The selection string is specified with the `-s` flag.

You can also specify field names as parameters to this command to choose which fields are displayed and the order in which they are displayed. The common field names are:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time</td>
<td>The time when the situation occurred that the record corresponds to. The value is a 64-bit integer and represents the number of microseconds since Unix Epoch (00:00:00 GMT January 1, 1970). See the constants below for specifying the time in more user-friendly formats.</td>
</tr>
<tr>
<td>Subsystem</td>
<td>The subsystem that generated the record. This is a string.</td>
</tr>
<tr>
<td>Category</td>
<td>Indicates the importance of the situation corresponding to the audit record, as determined by the subsystem that generated the record. The valid values are: 0 (informational) and 1 (error).</td>
</tr>
<tr>
<td>SequenceNumber</td>
<td>The unique 64-bit integer that is assigned to the record. No other record in the audit log will have the same sequence number.</td>
</tr>
<tr>
<td>TemplateId</td>
<td>The subsystem-dependent identifier that is assigned to records that have the same content and format string. This value is a 32-bit unsigned integer.</td>
</tr>
<tr>
<td>NodeName</td>
<td>The name of the node from which the record was obtained. This field name cannot be used in a selection string.</td>
</tr>
</tbody>
</table>

In addition to the constants in expressions, you can use the following syntax for dates and times with this command:

```
#mmddhmmmyyyy
```

This format consists of a sequence of decimal characters that are interpreted according to the pattern shown. The fields in the pattern are, from left to right: `mm` = month, `dd` = day, `hh` = hour, `mm` = minutes, `yyyy` = year. For example, `#010523042004` corresponds to January 5, 11:04 PM, 2004. The fields can be omitted from right to left. If not present, the following defaults are used: `year` = the current year, `minutes` = 0, `hour` = 0, `day` = 1, and `month` = the current month.
The format is similar to the previous one, but is relative to the current time and date. For example, the value 
-001 corresponds to one day ago and the value -01001 corresponds to one month and one hour ago. Fields can be omitted starting from the right and are replaced by 0.

The audit records considered for display and matched against the selection string can be restricted to a specific subsystem by using the -S flag. If this flag is specified, the subsystem-specific field names can be used in the selection string in addition to the common field names.

The nodes from which audit log records are considered for display and matched against the selection string can be restricted to a set of specific nodes by using the -n flag. If this flag is specified, the search is limited to the set of nodes listed. Otherwise, the search is performed for all nodes defined within the current management scope, as determined by the CT_MANAGEMENT_SCOPE environment variable.

The audit records are displayed in a table. Field names specified as parameters control which fields are displayed and the order in which they appear on each line. By default, the columns displayed are: the date and time, the subsystem name that generated the record, the severity of the situation, and the subsystem-specific message that describes the situation. If the management scope is not local, the node name is displayed in the first column.

Flags

-1 Indicates that long output should be produced. Long output includes subsystem-specific fields that are not included in the formatted message text.

-a Specifies that records from all nodes in the domain are to be displayed. If both the -n and the -a flags are omitted, records from the local node only are displayed.

-n node_name1[,node_name2]...
Specifies the list of nodes containing audit log records that will be examined and displayed if they meet the other criteria, such as matching the specified selection string. Node group names can also be specified, which are expanded into a list of node names. If both the -n and the -a flags are omitted, records from the local node only are displayed.

-S subsystem_name
Specifies a subsystem name. If this flag is present, only records identified by subsystem_name are considered for display. The records displayed can be further restricted by the -S flag. If the subsystem name contains any spaces, it must be enclosed in single or double quotation marks.

For backward compatibility, the subsystem name can be specified using the -n flag only if the -a and the -S flags are not specified.

-s selection_string
Specifies a selection string. This string is evaluated against each record in the audit log. All records that match the selection string will be displayed. If the selection string contains any spaces, it must be enclosed in single or double quotation marks.

The names of fields in the record can be used in the expression. If the -S flag is not specified, only the names of common fields can be used. See the Description for a list of the common field names and their data types. If the -S flag is specified, the name of any field for the specified subsystem as well as the common field names can be used.

If this flag is omitted, the records that are displayed will depend on the -S flag. If the -S flag is omitted, all records from the audit log are displayed. Otherwise, all records for the subsystem identified by the -S flag are displayed.

-x Excludes the header (suppresses header printing).

-h Writes the command’s usage statement to standard output.
Parameters

[field_name1 [field_name2...]]
   Specifies one or more fields in the audit log records to be displayed. The order of the field names on the command line corresponds to the order in which they are displayed. If no field names are specified, Time, Subsystem, Severity, and Message are displayed by default. If the management scope is not local, NodeName is displayed as the first column by default. See the Description for information about these and other fields.

Security

In order to list records from an audit log when the -S flag is omitted, a user must have read access to the target resource class on each node from which records are to be listed. When the -S flag is specified, the user must have read access to the audit log resource corresponding to the subsystem identified by the -S flag on each node from which records are to be listed.

Authorization is controlled by the RMC access control list (ACL) file that exists on each node.

Exit Status

0   The command ran successfully.
1   An error occurred with RMC.
2   An error occurred with a command-line interface script.
3   An incorrect flag was entered on the command line.
4   An incorrect parameter was entered on the command line.
5   An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT
   Determines the system where the session with the resource monitoring and control (RMC) daemon is established. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that can be affected by this command.

CT_MANAGEMENT_SCOPE
   Determines (in conjunction with the -a and -n flags) the management scope that is used for the session with the RMC daemon. The management scope determines the set of possible target nodes where audit log records can be listed. If the -a and -n flags are not specified, local scope is used. When either of these flags is specified, CT_MANAGEMENT_SCOPE is used to determine the management scope directly. The valid values are:

0    Specifies local scope.
1    Specifies local scope.
2    Specifies peer domain scope.
3    Specifies management domain scope.

If this environment variable is not set, local scope is used.

Standard Output

When the -h flag is specified, this command’s usage statement is written to standard output.
Examples
1. To list all records in the audit log on every node in the current management scope as determined by the CT_MANAGEMENT_SCOPE environment variable, enter:
   `lsaudrec`
2. To list all records that were logged in the last hour on every node in the current management scope as determined by the CT_MANAGEMENT_SCOPE environment variable, enter:
   `lsaudrec -s "Time > #000001"`
3. To list the time and sequence number of every record in the audit log for the subsystem abc on nodes mynode and yournode, enter:
   `lsaudrec -n mynode,yournode -S abc Time SequenceNumber`

Location
/usr/sbin/rsct/bin/lsaudrec Contains the lsaudrec command

Related Information
Commands: rmaudrec

lsauthent Command

Purpose
Lists the authentication methods currently configured on the system.

Syntax
lsauthent

Description
The lsauthent command calls the get_auth_method subroutine in the libauthm.a library, translates a list of authentication methods returned, and prints the authentication methods configured to stdout. Each authentication method is outputted on a separate line.

The authentication methods are listed in the order in which they are configured. If none of the authentication methods are configured, lsauthent returns without printing anything.

The lsauthent command writes an error message to stderr and returns a -1 if get_auth_method fails.

Examples
If all of the authentication methods are configured as:
   `lsauthent`
the output would consist of:
   Kerberos 5
   Kerberos 4
   Standard AIX

Related Information
The chauthent command, ftp command, rcp command, rlogin command, rsh command, telnet, tn, or tn3270 command.

The get_auth_method and set_auth_method subroutines.
IsC2admin Command

Purpose
Display the name of the current C2 System Administrative Host.

Syntax
IsC2admin

Description
The IsC2admin command displays the name of the administrative host. An administrative host must have been defined, and the system must have been installed in C2 mode for this command to operate successfully.

Exit Status
0  The administrative host information has been successfully displayed.
1  This system was not installed with C2 security.
2  This system has not been initialized to operate in C2 mode.
3  An error occurred while displaying the name of the administrative host.

Files
/usr/sbin/IsC2admin
Contains the IsC2admin command.

Related Information
The chC2admin command, IsC2host command, mkC2admin command, rmC2admin command.

IsCCadmin Command

Purpose
Display the name of the current Common Criteria enabled System Administrative Host.

Syntax
IsCCadmin

Description
The IsCCadmin command displays the name of the administrative host. An administrative host must have been defined, and the system must have been installed in Common Criteria enabled mode for this command to operate successfully.

Exit Status
0  The administrative host information has been successfully displayed.
1  This system was not installed with Common Criteria enabled security.
This system has not been initialized to operate in Common Criteria enabled mode.

An error occurred while displaying the name of the administrative host.

Files
/usr/sbin/lsCCadmin
Contains the lsCCadmin command.

Related Information
The chCCadmin command, isCChost command, mkCCadmin command, rmCCadmin command.

Iscfg Command

Purpose
Displays configuration, diagnostic, and vital product data (VPD) information about the system.

Syntax
To Display Specific Data on all Systems
lscfg [-v] [-p] [-s] [-I Name]

Description
If you run the Iscfg command without any flags, it displays the name, location and description of each device found in the current Customized VPD object class that is a child device of the sys0 object. It will not display any device that has been marked missing in the Customized Device Object Class. The list is sorted by parent, child, and device location. Information on a specific device can be displayed with the -I flag.

Use the Iscfg command to display vital product data (VPD) such as part numbers, serial numbers, and engineering change levels from either the Customized VPD object class or platform specific areas. Not all devices contain VPD data.

VPD data that is preceded by ME signifies that the VPD data was entered manually using a diagnostic service aid. For some devices, the vital product data is collected automatically from the devices through methods and added to the Customized VPD object class.

If you run the Iscfg command with the -p flag, it displays device information stored in the platform specific data areas. When used with the -v flag, VPD data stored for these devices is also displayed. This information is obtained on a Common Hardware Reference Platform (CHRP) system from the open firmware device tree.

-I Name Displays device information for the named device.
-p Displays the platform-specific device information. This flag only applies to AIX 4.2.1 or later.
-v Displays the VPD found in the Customized VPD object class. Also, on AIX 4.2.1 or later, displays platform specific VPD when used with the -p flag.
-s Displays the device description on a separate line from the name and location.

Examples
1. To display the system configuration, enter:
   lscfg
   The system displays a message similar to the following:
   INSTALLED RESOURCE LIST
The following resources are installed on the machine:

+/− = Added or deleted from Resource List.
* = Diagnostic support not available.

Model Architecture: chrp
Model Implementation: Multiple Processor, PCI bus

+ sys0  System Object
+ sysplanar0  System Planar
+ mem0  Memory
+ L2cache0  L2 Cache
+ proc0  U1.1-P1-C1  Processor
* pci3  U0.2-P1  PCI Bus
+ scsi0  U0.1-P1/Z1  Wide/Ultra-2 SCSI I/O Controller
+ rmt0  U1.1-P1/Z1-A3  SCSI 4mm Tape Drive (12000 MB)
+ cd0  U1.1-P1/Z1-A5  SCSI Multimedia CD-ROM Drive (650 MB)
+ hdisk0  U1.1-P1/Z1-A9  16 Bit LVD SCSI Disk Drive (4500 MB)
+ fd0  U0.1-P1-D1  Diskette Drive
...

2. To display the system configuration with the device description on a separate line, enter:
   ```
   lscfg -s
   ```
   The system displays a message similar to the following:
   
   INSTALLED RESOURCE LIST

   The following resources are installed on the machine:
   
   +/- = Added or deleted from Resource List.
   * = Diagnostic support not available.

   Model Architecture: chrp
   Model Implementation: Multiple Processor, PCI bus

   + sys0  System Object
   + sysplanar0  System Planar
   + mem0  Memory
   + L2cache0  L2 Cache
   + proc0  U1.1-P1-C1  Processor
   proc1  U5734.100.1234567-P1-C2  Processor
   proc2  U5734.100.1234567-P1-C2  Processor
   proc3  U5734.100.1234567-P1-C2  Processor
   proc4  U5734.100.1234567-P1-C2  Processor
   proc5  U5734.100.1234567-P1-C2  Processor
   proc6  U5734.100.1234567-P1-C2  Processor
   proc7  U5734.100.1234567-P1-C2  Processor
   +hdisk0  U5734.100.1234567-P1-D9  16 Bit LVD SCSI Disk Drive (4500 MB)
   +fd0  U5734.100.1234567-P1-D1  Diskette Drive
   ...

3. To display the name, location, and description for devices specified by the logical name proc without VPD, enter:
   ```
   lscfg -lproc\*
   ```
   The system displays information for all devices with logical names beginning with proc, as follows:

   proc0  U1.1-P1-C1  Processor
   proc1  U1.1-P1-C1  Processor
   proc2  U1.1-P1-C1  Processor
   proc3  U1.1-P1-C1  Processor
   proc4  U1.1-P1-C1  Processor
   proc5  U1.1-P1-C1  Processor
   proc6  U1.1-P1-C1  Processor
   proc7  U1.1-P1-C1  Processor
4. To display the VPD for a specific device specified by the logical name ent0, enter:

```
lsconf -v -l ent0
```

The system displays the following:
```
ent0 U0.1-P1-I2/E1 Gigabit Ethernet-SX PCI Adapter (14100401)
```
```
Network Address..............0004AC7C00C4
Displayable Message...........Gigabit Ethernet-SX PCI Adapter (14100401)
```
```
EC Level....................E77998
Part Number..................07L8916
FRU Number...................07L8918
Device Specific.(YL).........U0.1-P1-I2/E1
```

5. To display the VPD in the open firmware device tree for the corresponding node to the ent0 device, enter:

```
lsconf -vp -lent0
```

The following is displayed:
```
ent0 U0.1-P1-I2/E1 Gigabit Ethernet-SX PCI Adapter (14100401)
```
```
Network Address..............0004AC7C00C4
Displayable Message...........Gigabit Ethernet-SX PCI Adapter (14100401)
```
```
EC Level....................E77998
Part Number..................07L8916
FRU Number...................07L8918
Device Specific.(YL).........U0.1-P1-I2/E1
```

PLATFORM SPECIFIC

Name: ethernet
Model: Galaxy, EtherLink 1000-SX-IBM
Node: ethernet01
Device Type: network
Physical Location: U0.1-P1-I2/E1

Files

```
/usr/sbin/lsconf
```
Contains the `lsconf` command.

Related Information

The `lsattr` command, `lsconn` command, `lsdev` command, `lsparent` command.

Isclass Command

Purpose
List Workload Management classes and their limits.

Syntax

```
```

Description

The `lsclass` command, with no argument, returns the list of superclasses, one per line. With a class name as argument, it prints the class. The subclasses can be displayed with the `-r` (recursive) flag, or with the `-S Superclass` flag.
When WLM is started, if an empty string is passed as the name of the configuration with the -d flag, lsclass lists the classes defined in the in-core WLM data structures.

The lsclass command does not require any special level of privilege and is accessible for all users.

**Note:** If this command is given a set of time-based configurations (either specified with the -d flag, or because the current configuration is a set), the lsclass command returns the classes of the regular configuration which applies (or would apply) at the time the command is issued.

**Flags**

- **-C** Displays the class attributes and limits in colon-separated records, as follows:
  ```
  lsclass -C myclass
  CPUshares:CPUm:CPUso:max:CPUhard:max:memoryshares:memorymin:memoryso:max:mem:
totalCPUunit:totalDiskIohar:max:totalDiskIounit:totalConnecttimehar:max:totalConne:
  ctimeun:totalProcesseshar:max:totalThreadshar:max:totalLoginshar:max:localshm:
  ttimeun:totalProcesseshar:max:totalThreadshar:max:totalLoginshar:max:localshm:
  class::0:no:root:system:root:system:::-:0:50:100:4:10:50:75:6:0:100:100:-:s:-:K
  B:-:s:-:-:-:no
  ```

- **-d Config_Dir** Use /etc/wlm/Config_Dir as alternate directory for the definition files. If an empty string is passed (for example, -d ""), lsclass lists the classes defined in the in-core WLM data structures. If this flag is not present, the current configuration files in the directory pointed to by /etc/wlm/current are used.

- **-D** Displays the default values for the class attributes and limits in colon-separated records. Any other flag or argument used in conjunction with -D is ignored. For example:
  ```
  lsclass -D
  CPUshares:CPUm:CPUso:max:CPUhard:max:memoryshares:memorymin:memoryso:max:mem:
totalCPUunit:totalDiskIohar:max:totalDiskIounit:totalConnecttimehar:max:totalConne:
  ctimeun:totalProcesseshar:max:totalThreadshar:max:totalLoginshar:max:localshm:
  ttimeun:totalProcesseshar:max:totalThreadshar:max:totalLoginshar:max:localshm:
  class::0:no:root:system:root:system:::-:0:50:100:4:10:50:75:6:0:100:100:-:s:-:K
  B:-:s:-:-:-:no
  ```

- **-f** Displays the output in stanzas, with each stanza identified by a class name. Each Attribute=Value pair is listed on a separate line:
  ```
  Class:
  attribute1=value
  attribute2=value
  attribute3=value
  ```

- **-r** Displays, recursively, the superclasses with all their subclasses. When specifying -r:
  - If Class is not specified, lsclass shows all the superclasses with all their subclasses.
  - If the name of a superclass is specified, lsclass displays the superclass with all its subclasses.
  - If the name of a subclass is specified, -r is ineffective (displays only the subclass).

- **-S SuperClass** Restrictions the scope of the command to the subclasses of the specified superclass. Only subclasses are shown with the -S flag.

**Files**

- **classes** Contains the names and definitions of the classes.
- **limits** Contains the limits enforced on the classes.
- **shares** Contains the resource shares attributes for each class.
Related Information
The \texttt{wlmcntr} command, \texttt{chclass} command, \texttt{mkclass} command, and \texttt{rmclass} command.

\section*{lscomg Command}

\subsection*{Purpose}
Displays information about the communication groups of a peer domain.

\subsection*{Syntax}
\texttt{lscomg [-l | -t | -d | -D delimiter] [-x] [-i] [-h] [-TV] [communication\_group]}

\subsection*{Description}
The \texttt{lscomg} command displays information about the communication groups that are defined to the online peer domain on which the command runs. If you specify the name of a communication group, the \texttt{lscomg} command displays information about that communication group only.

Some of the communication group information that is displayed follows:

\begin{tabular}{|l|l|}
\hline
\textbf{Field} & \textbf{Description} \\
\hline
Name & The name of the communication group \\
Sensitivity & The number of missed heartbeats that constitute a failure \\
Period & The number of seconds between heartbeats \\
Priority & The relative priority of the communication group \\
Broadcast & Indicates whether broadcast should be used if it is supported by the underlying media \\
SourceRouting & Indicates whether source routing should be used if it is supported by the underlying media \\
NIMPath & The path to the Network Interface Module (NIM) that supports the adapter types in the communication group \\
NIMParameters & The NIM start parameters \\
\hline
\end{tabular}

\subsection*{Interface resources}
Use the \texttt{-i} flag to display information about the interface resources that refer to \texttt{communication\_group}. If you specify the \texttt{-i} flag, \texttt{lscomg} displays the following information:

\begin{tabular}{|l|l|}
\hline
\textbf{Field} & \textbf{Description} \\
\hline
Name & The name of the interface resource that refers to \texttt{communication\_group} \\
NodeName & The host name of the interface resource that refers to \texttt{communication\_group} \\
IPAddress & The IP address of the interface resource that refers to \texttt{communication\_group} \\
SubnetMask & The subnet mask of the interface resource that refers to \texttt{communication\_group} \\
Subnet & The subnet of the interface resource that refers to \texttt{communication\_group} \\
\hline
\end{tabular}

\subsection*{Flags}
\texttt{-l} Displays the information on separate lines (long format).
-t Displays the information in separate columns (table format). This is the default format.
-d Displays the information using delimiters. The default delimiter is a colon (:). Use the -D flag if you want to change the default delimiter.
-D delimiter Displays the information using the specified delimiter. Use this flag to specify a delimiter other than the default colon (:) — when the information you want to display contains colons, for example. You can use this flag to specify a delimiter of one or more characters.
-x Excludes the header (suppresses header printing).
-i Displays information about the interface resource that refers to communication_group.
-h Writes the command’s usage statement to standard output.
-T Writes the command’s trace messages to standard error. For your software service organization’s use only.
-V Writes the command’s verbose messages to standard output.

Parameters

communication_group

Specifies the name of the communication group about which you want to display information. You can specify a communication group name or a substring of a communication group name for this parameter. If you specify a substring, the command displays information about any defined communication group with a name that contains the substring.

Security

The user of the lscomg command needs read permission for the IBM.CommunicationGroup resource class. Read permission for the IBM.NetworkInterface resource class is required to display the network interface information. By default, root on any node in the peer domain has read and write access to these resource classes through the configuration resource manager.

Exit Status

<table>
<thead>
<tr>
<th>Status Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>The command ran successfully.</td>
</tr>
<tr>
<td>1</td>
<td>An error occurred with RMC.</td>
</tr>
<tr>
<td>2</td>
<td>An error occurred with a command-line interface script.</td>
</tr>
<tr>
<td>3</td>
<td>An incorrect flag was entered on the command line.</td>
</tr>
<tr>
<td>4</td>
<td>An incorrect parameter was entered on the command line.</td>
</tr>
<tr>
<td>5</td>
<td>An error occurred that was based on incorrect command-line input.</td>
</tr>
<tr>
<td>6</td>
<td>The communication group definition does not exist.</td>
</tr>
</tbody>
</table>

Environment Variables

CT_CONTACT

Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

Restrictions

This command must be run on a node that is defined and online to the peer domain on which the communication group exists.
Standard Input
When the -f "-" flag is specified, this command reads one or more node names from standard input.

Standard Output
When the -h flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
In these examples, nodeA is defined and online to peer domain ApplDomain.
1. To display general information about the communication groups for ApplDomain, run this command on nodeA:
   
   lscomg

   The output will look like this:

   Name  Sensitivity  Period  Priority  Broadcast  SourceRouting  NIMPath  NIMParameters
   ComG1  2           2       1        no         yes          /usr/sbin/rsct/bin/hats_nim  -l 5

2. To display information about the interface resources that refer to the communication group ComGrp1 for the peer domain ApplDomain, run this command on nodeA:
   
   lscomg -i ComGrp1

   The output will look like this:

   Name  NodeName  IPAddr    SubnetMask  Subnet
   eth0  n24      9.234.32.45 255.255.255.2 9.235.345.34
   eth0  n25      9.234.32.46 255.255.255.2 9.235.345.34

Location
/usr/sbin/rsct/bin/lscomg     Contains the lscomg command

Related Information
Commands: chcomg, lsrpnode, lsrpdomain, mkcomg, preprpnode, rmcomg

Files: rmccli, for general information about RMC-related commands

lscondition Command

Purpose
Lists information about one or more conditions.

Syntax
lscondition [-a] [ -m | -n | -e ] [ -C | -l | -t | -d | -D delimiter ] [ -A ] [ -q ] [ -U ] [ -x ] [ -h ] [ -TV ]
[condition1 [condition2,...]:node_name]

Description
The lscondition command lists the following information about defined conditions:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>The name of the condition</td>
</tr>
<tr>
<td>Node</td>
<td>The location of the condition (for management domain scope or peer domain scope)</td>
</tr>
</tbody>
</table>
MonitorStatus The status of the condition
ResourceClass The resource class that is monitored by this condition
EventExpression The expression that is used in monitoring this condition
EventDescription A description of the EventExpression field
RearmExpression The expression used in determining when monitoring should restart for this condition after an event has occurred
RearmDescription A description of the RearmExpression field
SelectionString The selection string that is applied to the attributes of ResourceClass to determine which resources are included in the monitoring of this condition
Severity The severity of the condition: critical, warning, or informational.
NodeNames The host names of the nodes where the condition is registered
MgtScope The RMC scope in which the condition is monitored
Locked Specifies whether the resource is locked or unlocked

For a list of all conditions, enter the lscondition command without any condition names specified. A list of all the condition names is returned with the monitoring status for each condition. The default format in this case is tabular. Specifying a node name following the condition names limits the display to the conditions defined on that node. You can list all of the conditions on a node by specifying a colon (:) followed by the node name. The node name is a node within the management scope, which is determined by the CT_MANAGEMENT_SCOPE environment variable. The management scope determines the list of nodes from which the conditions are listed. For local scope, only conditions on the local node are listed. Otherwise, the conditions from all nodes within the domain are listed.

For all of the information about all condition names, specify the -A flag with the lscondition command. The -A flag causes all information about a condition to be listed when no condition names are specified. When all the information about all conditions is listed, the default format is long. If a monitoring-status flag (-e, -m, or -n) is specified, the conditions with that status are listed.

When more than one condition is specified, the condition information is listed in the order in which the condition names are entered.

By default, when a condition name is specified with the lscondition command, all of the condition’s attributes are displayed.

Flags

-a Specifies that this command applies to all nodes in the cluster. The cluster scope is determined by the CT_MANAGEMENT_SCOPE environment variable. If it is not set, first the management domain scope is chosen if it exists, then the peer domain scope is chosen if it exists, and then local scope is chosen, until the scope is valid for the command. The command will run once for the first valid scope found. For example, if both a management and peer domain exist, lscondition -a with CT_MANAGEMENT_SCOPE not set will list the management domain. In this case, to list the peer domain, set CT_MANAGEMENT_SCOPE to 2.

-m Lists only those conditions that are being monitored without error.

-n Lists only those conditions that are not being monitored.

-e Lists only those conditions that are monitored in error.

-C Displays the mkcondition command that could be used to create the condition. If more than one condition is specified, each mkcondition command appears on a separate line. This flag is ignored when no conditions are specified. This flag overrides the –I flag.
-l Produces long-formatted output. Displays the condition information on separate lines.
- t Displays the condition information in separate columns (table format).
- d Produces delimiter-formatted output. The default delimiter is a colon (:). Use the –D flag if you want to change the default delimiter.

- D delimiter
  Produces delimiter-formatted output that uses the specified delimiter. Use this flag to specify something other than the default, colon (:). An example is when the data to be displayed contains colons. Use this flag to specify a delimiter of one or more characters.

- A Displays all of the attributes of the condition.
- q Does not return an error when the condition does not exist.
- U Indicates whether the resource is locked.
- x Suppresses header printing.
- h Writes the command’s usage statement to standard output.
- T Writes the command’s trace messages to standard error. For your software service organization’s use only.
- V Writes the command’s verbose messages to standard output.

Parameters

condition1 [,condition2,...]  
Specifies the name of an existing condition that is defined on the host name node_name. You can specify more than one condition name. This parameter can be a condition name or a substring of a condition name. When it is a substring, any defined condition name that contains the substring will be listed.

node_name  
Specifies the node where the condition is defined. If node_name is not specified, the local node is used. node_name is a node within the scope determined by the CT_MANAGEMENT SCOPE environment variable.

Security

The user needs read permission for the IBM.Condition resource class to run lscondition. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status

0   The command ran successfully.
1   An error occurred with RMC.
2   An error occurred with a command-line interface script.
3   An incorrect flag was entered on the command line.
4   An incorrect parameter was entered on the command line.
5   An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT
  Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the
RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

**CT_MANAGEMENT_SCOPE**
Determines the management scope that is used for the session with the RMC daemon in processing the resources of the event-response resource manager (ERRM). The management scope determines the set of possible target nodes where the resources can be processed. The valid values are:

- **0** Specifies local scope.
- **1** Specifies local scope.
- **2** Specifies peer domain scope.
- **3** Specifies management domain scope.

If this environment variable is not set, local scope is used.

**Standard Output**
When the **-h** flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

**Standard Error**
All trace messages are written to standard error.

**Examples**
These examples apply to standalone systems:

1. To list all conditions and their monitoring status, run this command:
   
   `lscondition`
   
   The output will look like this:

<table>
<thead>
<tr>
<th>Name</th>
<th>Node</th>
<th>MonitorStatus</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Monitored&quot;</td>
</tr>
<tr>
<td>&quot;tmp space used&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Not monitored&quot;</td>
</tr>
<tr>
<td>&quot;var space used&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Error&quot;</td>
</tr>
</tbody>
</table>

2. To list general information about the condition "FileSystem space used" in long form, run this command:

   `lscondition "FileSystem space used"`
   
   The output will look like this:

   ```
   Name           = "FileSystem space used"
   Node           = "nodeA"
   MonitorStatus  = "Monitored"
   ResourceClass  = "IBM.FileSystem"
   EventExpression= "PercentTotUsed > 99"
   EventDescription= "Generate event when space used is greater than 99 percent full"
   RearmExpression= "PercentTotUsed < 85"
   RearmDescription= "Start monitoring again after it is less than 85 percent"
   SelectionString= ""
   Severity       = "w"
   NodeNames      = "{}"
   MgtScope       = "l"
   ```

3. To list the command that would create the condition "FileSystem space used", run this command:

   `lscondition -C "FileSystem space used"`
The output will look like this:

```bash
mkcondition -r IBM.FileSystem -a PercentTotUsed -e "PercentTotUsed > 99" -E "PercentTotUsed < 85" -d "Generate event when space used is greater than 99 percent full" -D "Start monitoring after it is less than 85 percent" -S w "FileSystem space used"
```

4. To list all conditions that have the string `space` in their names, run this command:

```
lscondition space
```

The output will look like this:

```
Name = "FileSystem space used"
MonitorStatus = "Monitored"

Name = "tmp space used"
MonitorStatus = "Not Monitored"

Name = "var space used"
MonitorStatus = "Monitored"
```

5. To list the conditions that are in error, run this command:

```
lscondition -e
```

The output will look like this:

```
Name MonitorStatus
"var space used" "Error"
```

This example applies to clustered systems:

1. To list all conditions and their monitoring status, run this command:

```
lscondition -a
```

The output will look like this:

```
Name Node MonitorStatus
"FileSystem space used" "nodeA" "Monitored"
"tmp space used" "nodeB" "Not monitored"
"var space used" "nodeC" "Error"
```

## Location

`/usr/sbin/rsct/bin/lscondition` Contains the `lscondition` command

## Related Information

Commands: `chcondition`, `lscondresp`, `mkcondition`, `rmcondition`

Files: `rmccli`

### Iscondresp Command

## Purpose

Lists information about a condition and any of its condition/response associations.

## Syntax

To list the link between a condition and one or more responses:
Iscondresp [-a | -n] [-l | -t | -d | -D delimiter] [-q] [-U] [-x] [-z] [-h] [-TV] [condition[:node_name] [response1 [response2...]]]

To list all of the links to one or more responses:

response f[:node_name] [response2...]

Description
The Iscondresp command lists information about a condition and its linked responses. A link between a condition and a response is called a condition/response association. The information shows which responses are linked with a condition and whether monitoring is active for a condition and its linked response. The following information is listed:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Condition</td>
<td>The name of the condition linked with a response.</td>
</tr>
<tr>
<td>Response</td>
<td>The name of the response linked with the condition.</td>
</tr>
<tr>
<td>State</td>
<td>The state of the response for the condition. The state indicates whether a specified response is active or not.</td>
</tr>
<tr>
<td>Node</td>
<td>The location of the condition and the response.</td>
</tr>
<tr>
<td>Locked</td>
<td>Indicates whether the resource is locked or unlocked.</td>
</tr>
</tbody>
</table>

To list a particular condition and response, specify both the condition and the response. To list all responses to a condition, specify the condition only. To list all conditions to which a response is linked, specify the response and the -r flag. To list all conditions and their linked responses, do not specify any condition or response parameters.

Specifying a node name limits the display to the condition/response associations that are defined on that node. List all of the condition/response associations on a node by specifying a colon (:) followed by the node name. The node name is a node within the management scope determined by the CT_MANAGEMENT_SCOPE environment variable. The management scope determines the list of nodes from which the condition/response associations are listed. For local scope, only condition/response associations on the local node are listed. For management domain scope and peer domain scope, the condition/response associations from all nodes within the domain are listed.

When neither the -a flag nor the -n flag is specified, all selected conditions for the responses are listed. Tabular format is the default.

Flags
- **-a** Lists only those responses that are active for the condition.
- **-n** Lists only those responses that are not active for the condition.
- **-l** Displays the condition information and response information on separate lines (long format).
- **-t** Displays the condition information and response information in separate columns (table format).
- **-d** Specifies delimiter-formatted output. The default delimiter is a colon (:). Use the -D flag if you want to change the default delimiter.
- **-D delimiter** Specifies delimiter-formatted output that uses delimiter. Use this flag to specify something other than the default colon (:). For example, when the data to be displayed contains colons, use this flag to specify another delimiter of one or more characters.
- **-q** Does not return an error if either the condition or the response does not exist.

304 Commands Reference, Volume 3
-U Indicates whether the resource is locked.
-x Suppresses header printing.
-z Specifies that this command applies to all nodes in the cluster. The cluster scope is determined by the CT_MANAGEMENT_SCOPE environment variable. If it is not set, first the management domain scope is chosen if it exists, then the peer domain scope is chosen if it exists, and then local scope is chosen, until the scope is valid for the command. The command will run once for the first valid scope found. For example, if both a management and peer domain exist, lscondresp -z with CT_MANAGEMENT_SCOPE not set will list the management domain. In this case, to list the peer domain, set CT_MANAGEMENT_SCOPE to 2.
-r Lists information about all of the condition/response associations for the specified responses. Use this flag to indicate that all command parameters specified are responses, not conditions.
-h Writes the command’s usage statement to standard output.
-T Writes the command’s trace messages to standard error. For your software service organization’s use only.
-V Writes the command’s verbose messages to standard output.

Parameters

condition The condition can be a condition name or a substring of a condition name. When it is a substring, any defined condition name that contains the substring and is linked to the response will be listed.

response1 [response2...]
This parameter can be a response name or a substring of a response name. You can specify more than one response name. When it is a substring, any defined response name that contains the substring and is linked to the condition will be listed.

node_name Specifies the node where the condition or response is defined. If node_name is not specified, the local node is used. node_name is a node within the scope determined by the CT_MANAGEMENT_SCOPE environment variable.

Security
The user needs read permission for the IBM.Association resource class to run lscondresp. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status
0 The command ran successfully.
1 An error occurred with RMC.
2 An error occurred with a command-line interface script.
3 An incorrect flag was entered on the command line.
4 An incorrect parameter was entered on the command line.
5 An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the
RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

**CT_MANAGEMENT_SCOPE**

Determines the management scope that is used for the session with the RMC daemon in processing the resources of the event-response resource manager (ERRM). The management scope determines the set of possible target nodes where the resources can be processed. The valid values are:

0  Specifies *local* scope.
1  Specifies *local* scope.
2  Specifies *peer domain* scope.
3  Specifies *management domain* scope.

If this environment variable is not set, *local* scope is used.

**Standard Output**

When the `-h` flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

**Standard Error**

All trace messages are written to standard error.

**Examples**

To see which resources are locked, run this command:

```
lscondresp -U
```

The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
<th>Locked</th>
</tr>
</thead>
<tbody>
<tr>
<td>/tmp space used</td>
<td>E-mail root off-shift</td>
<td>nodeA</td>
<td>Not active</td>
<td>Yes</td>
</tr>
<tr>
<td>Page space in rate</td>
<td>E-mail root anytime</td>
<td>nodeA</td>
<td>Not active</td>
<td>No</td>
</tr>
</tbody>
</table>

These examples apply to standalone systems:

1. To list all conditions with their linked responses, run this command:

```
lscondresp
```

The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>FileSystem space used</td>
<td>Broadcast event on-shift</td>
<td>nodeA</td>
<td>Active</td>
</tr>
<tr>
<td>FileSystem space used</td>
<td>E-mail root anytime</td>
<td>nodeA</td>
<td>Not Active</td>
</tr>
<tr>
<td>Page in Rate</td>
<td>Log event anytime</td>
<td>nodeA</td>
<td>Active</td>
</tr>
</tbody>
</table>

2. To list information about the condition "FileSystem space used", run this command:

```
lscondresp "FileSystem space used"
```

The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>FileSystem space used</td>
<td>Broadcast event on-shift</td>
<td>nodeA</td>
<td>Active</td>
</tr>
<tr>
<td>FileSystem space used</td>
<td>E-mail root anytime</td>
<td>nodeA</td>
<td>Not Active</td>
</tr>
</tbody>
</table>

3. To list information about the condition "FileSystem space used" for responses that are active, run this command:

```
lscondresp -a "FileSystem space used"
```
The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
</tbody>
</table>

4. To list information about the condition "FileSystem space used" with the linked response "Broadcast event on-shift", run this command:

   `lscondresp "FileSystem space used" "Broadcast event on-shift"`

   The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
</tbody>
</table>

5. To list all conditions that have the string **space** in their names with their linked responses, run this command:

   `lscondresp space`

   The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;E-mail root anytime&quot;</td>
<td>&quot;nodeA&quot;</td>
<td>&quot;Not Active&quot;</td>
</tr>
</tbody>
</table>

These examples apply to management domains:

1. In this example, the condition "FileSystem space used" is defined on the management server. To list information about "FileSystem space used", run this command on the management server:

   `lscondresp "FileSystem space used"`

   The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeB&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;E-mail root anytime&quot;</td>
<td>&quot;nodeB&quot;</td>
<td>&quot;Not Active&quot;</td>
</tr>
</tbody>
</table>

2. In this example, the condition "FileSystem space used" is defined on the managed node **nodeC**. To list information about "FileSystem space used", run this command on the management server:

   `lscondresp "FileSystem space used":nodeC`

   The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeC&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;E-mail root anytime&quot;</td>
<td>&quot;nodeC&quot;</td>
<td>&quot;Not Active&quot;</td>
</tr>
</tbody>
</table>

This example applies to a peer domain:

1. In this example, the condition "FileSystem space used" is defined in the domain. To list information about "FileSystem space used", run this command on one of the nodes in the domain:

   `lscondresp "FileSystem space used"`

   The output will look like this:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Response</th>
<th>Node</th>
<th>State</th>
</tr>
</thead>
<tbody>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeD&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;E-mail root anytime&quot;</td>
<td>&quot;nodeD&quot;</td>
<td>&quot;Not Active&quot;</td>
</tr>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;Broadcast event on-shift&quot;</td>
<td>&quot;nodeE&quot;</td>
<td>&quot;Active&quot;</td>
</tr>
<tr>
<td>&quot;FileSystem space used&quot;</td>
<td>&quot;E-mail root anytime&quot;</td>
<td>&quot;nodeE&quot;</td>
<td>&quot;Not Active&quot;</td>
</tr>
</tbody>
</table>
Location
/usr/sbin/rsct/bin/lscondresp Contains the lscondresp command

Related Information
Commands: mkcondition, mkcondresp, mkresponse, rmcondresp, startcondresp, stopcondresp

Files: rmccli

Isconn Command

Purpose
Displays the connections a given device, or kind of device, can accept.

Syntax

Description
The lsconn command, when used with the -p ParentName flag, displays the connection locations on the parent device to which the device specified by the -I ChildName flag can be connected, or to which devices of the connection type specified by the -k ChildConnectionKey flag can be connected. If the -k and -I flags are not used, the Isconn command displays information as to where a child device can be connected on the specified parent.

If the -p ParentName flag is not used, you must use a combination of one or all of the -c ParentClass, -s ParentSubclass, and -t ParentType flags to uniquely identify the predefined parent device.

You can display the default output, which is the connection location (or connection location and connection key if no child is specified), from the Predefined Connection object class. If you do not display the default, you can display the output in a user-specified format where the Format parameter is a quoted list of column names separated by nonalphanumeric characters or white space using the -F Format flag. You can insert headers above the columns using the -H flag.

You can supply the flags either on the command line or from the specified File parameter.

Flags

- **-c ParentClass**
  Specifies the class name of a possible parent device in the Predefined Devices object class. This flag cannot be used with the **-p** flag.

- **-f File**
  Reads the needed flags from the File parameter.

- **-F Format**
  Formats the output in a user-specified format where the Format parameter is a quoted list of column names from the Predefined Connection object class separated and possibly terminated by nonalphanumeric characters or white space. Using white space as the separator, the lsconn command displays the output in aligned columns.

- **-h**
  Displays headers above the column output.

- **-H**
  Displays the command usage message.

- **-k ChildConnectionKey**
  Specifies the connection key that identifies the subclass of the child device. This flag cannot be used with the **-h** flag.

- **-l ChildName**
  Specifies the logical name of a possible child device. This flag cannot be used with the **-k** flag.

- **-p ParentName**
  Specifies the parent device’s logical name from the Customized Devices object class. This flag cannot be used with the **-c** or **-s** flag.
-s ParentSubclass  Specifies the subclass of a possible parent device in the Predefined Devices object class. This flag cannot be used with the -p flag.

-t ParentType     Specifies the device type of a possible parent device from the Predefined Devices object class. This flag cannot be used with the -p flag.

Examples
1. To list all possible connection locations on the IBM 8-Port EIA-232/RS-422A (PCI) Adapter sa2 that will accept an RS-232 device connection, type:

   lsconn -p sa2 -k rs232

   The system displays a message similar to the following:
   
   0
   1
   2
   3
   4
   5
   6
   7

2. To list all possible connection locations and connection types on the IBM 8-Port EIA-232/RS-422A (PCI) Adapter sa2, enter:

   lsconn -p sa2

   The system displays a message similar to the following:
   
   0  rs232
   1  rs232
   2  rs232
   3  rs232
   4  rs232
   5  rs232
   6  rs232
   7  rs232
   0  rs422
   1  rs422
   2  rs422
   3  rs422
   4  rs422
   5  rs422
   6  rs422
   7  rs422

Files

/usr/sbin/lsconn  Specifies the command file.

Related Information

The chdev command, lsattr command, lsdev command, lsparent command, mkdev command, rmdev command.

Iscons Command

Purpose
Writes the name of the current console device to standard output.

Syntax
lscons [-s] [-a] [-O]
lscons [-b] [-s] [-a] [-O]
lscons [-d] [-s]

Description
The lscons command writes the name of the current console device to standard output. This command is also used to write the name of the device that is to be the console on the next start of the system to standard output. You can change the current console device using the swcons command. You can change the device to be the system console on the next start of the system using the chcons command.

Flags

-a   Displays a list of attribute name = attribute value pairs for the console device and console logging and tagging attributes. When used with the -b flag, the values are retrieved from the ODM. Without the -b flag, the values are retrieved from the console device driver. For additional information about console output logging and tagging, see the console Special File in the AIX 5L Version 5.2 Files Reference book.
   Note: This flag is not valid with the -O flag or the -d flag.
-b   Displays the full path name of the system console selected for the next startup of the system.
-d   Displays the full path name of the system console selected on the current startup of the system.
   Note: This flag is not valid with the -O flag or the -a flag.
-O   Similar to the -a flag but outputs the attribute names and values in a format suitable for use by SMIT. This flag is NOT valid with the -d flag.
   Note: This flag is not valid with the -d flag or the -a flag.
-s   Suppresses reporting of the path name.

Exit Status
This command returns the following exit values:

0   The device you are using is the current system console.
1   The device you are using is not the current system console.
2   The device you are using is the console device selected at system start but is not currently the device supporting console message output.
3   Flags specified are not valid.
4   System error occurred.

Examples
1. To display the full path name of the current system console, type:
   lscons
2. To display the full path name of the system console effective on the next startup of the system, type:
   lscons [-b]
3. To display the full path name of the system console selected on the current startup of the system, type:
   lscons [-d]
4. To test whether or not the current system console is directed to your display, type:
Files

/usr/sbin/lsdev

Contains the `lsdev` command.

Related Information

The `chcons` command, `swcons` command.

The `console` special file.

Isdev Command

Purpose

Displays devices in the system and their characteristics.

Syntax

```
```

```
```

Description

The `lsdev` command displays information about devices in the Device Configuration database. You can display information about all devices in the Customized Devices object class using the `-C` flag. Any combination of the `-c Class`, `-s Subclass`, `-t Type`, `-l Name`, `-p Parent`, and `-S State` flags selects a subset of the customized devices. You can display information about all devices in the Predefined Devices object class using the `-P` flag. Any combination of the `-c Class`, `-s Subclass`, and `-t Type` flags selects a subset of the predefined devices.

You can display the default output one of two ways. You can either display the default output from the Customized Devices object class using the `-C` flag, or display the default output from the Predefined Devices object class using the `-P` flag. To override these two default outputs, you can use the `-f Format` flag. The `-F Format` flag displays the output in a user-specified format where the `Format` parameter is a quoted list of column names separated and possibly ended by nonalphanumeric characters or white space.

The `-r ColumnName` flag displays the range of values for a particular column over the specified set of devices.

Note that the `lsdev` command only shows information about devices based upon information in the Customized Devices (Cudv) object class or the Predefined Devices (PdDv) object class. Other object classes (such as the Customized Path (CuPath) object class) are not examined. This situation means that there may be conditions where a device may not be displayed. For example, if the `-p Parent` flag is used, but the parent identified in the Customized Devices object for a device does not match the `Parent` specified through the `-p` flag, the device will not be displayed. However, the device may have a path to the given `Parent` that is defined in the Customized Paths object class. The `lspath` command should be used to show all MPIO capable child devices of the given parent.
You can use the Devices application in Web-based System Manager to change device characteristics. You could also use the System Management Interface Tool (SMIT) smit lsdev fast path to run this command.

**Flags**

- `-C` Lists information about a device that is in the Customized Devices object class. The default information displayed is name, status, location, and description. The `-C` flag is not required. It is maintained for compatibility reasons. The `-C` flag cannot be specified with the `-P` flag. If neither are specified, the lsdev command behaves as if the `-C` flag were specified.

- `-c Class` Specifies a device class name. This flag can be used to restrict output to devices in a specified class.

- `-f File` Reads the needed flags from the `File` parameter.

- `-F Format` Displays the output in a user-specified format, where the `Format` parameter is a quoted list of column names from the Predefined or Customized Devices object class, separated and possibly ended by nonalphanumeric characters or white space. Using white space as the separator, the lsdev command displays the output in aligned columns.

If you specify the `-F Format` flag with the `-C` flag, you can specify column names from both the Customized and Predefined Devices object classes. If you specify the `-F Format` flag with the `-P` flag, you can only specify column names from the Predefined Devices object class. In addition to the column names, the special purpose name `description` can be used to obtain a display of device descriptions. This flag cannot be used with the `-C` `ColumnName` flag. Also, the `physloc` special purpose name can be used to display a device’s physical location code.

- `-H` Displays headers above the column output.

- `-h` Displays the command usage message.

- `-l Name` Specifies the device logical name from the Customized Devices object class for the device for which information is listed. The `Name` argument to the `-l` flag may contain the same wildcard characters that can be used with the `odmget` command. If the `Name` argument is a dash, names will be read from STDIN. Names on STDIN must be separated by a comma, a tab, a space, or a “newline” character. The names cannot contain wildcard characters. This flag cannot be used with the `-P` flag.

- `-p Parent` Specifies the device logical name from the Customized Devices object class for the parent of devices to be displayed. The `-p Parent` flag can be used to show the child devices of the given `Parent`. The `Parent` argument to the `-p` flag may contain the same wildcard characters that can be used with the `odmget` command. This flag cannot be used with the `-P` flag.

- `-P` Lists information about a device that is in the Predefined Devices object class. The default information displayed is class, type, subclass, description. This flag cannot be used with the `-C`, `-r`, or `-S` flags.

- `-r ColumnName` Displays the set of values in a column. For example, the `ColumnName` parameter takes the value of the `Class` parameter to list all the classes. If you specify the `-r ColumnName` flag with the `-P` flag, you can specify column names from both the Customized and Predefined Devices object classes. If you specify the `-r ColumnName` flag with the `-S` flag, you can only specify column names from the Predefined Devices object class. This flag cannot be used with the `-F Format` flag.

- `-S State` Lists all devices in a specified state as named by the `State` parameter. This flag cannot be used with the `-P` flag. The `State` parameter can either be a value of d, D, 0 or defined for the Defined state; a, A, 1, or available for the Available state; or s, S, 2, or stopped for the Stopped state. This flag can be used to restrict output to devices in a specified state.

- `-s Subclass` Specifies a device subclass name. This flag can be used to restrict output to devices in a specified subclass.

- `-t Type` Specifies a device type name. This flag can be used to restrict output to devices of a specified type.

**Examples**

1. To list all devices in the Predefined Devices object class with column headers, type:

   ```
   lsdev -P -H
   ```
The system displays a message similar to the following:

class type subclass description
logical_volume vgtype vgsubclass Volume group
logical_volume lvtype lvsubclass Logical volume
lvm lvd node LVM Device Driver
posix_aio posix_aio node Posix Asynchronous I/O
aio aio node Asynchronous I/O (Legacy)
pseudo tty node Asynchronous Pseudo-Terminal
mouse 030102 usbd0 USB mouse
keyboard 030101 usbd0 USB keyboard
driver tmssar node Target Mode SSA Router
tmssar tmssar Target Mode SSA Device
disk hdisk ssar SSA Logical Disk Drive
pdisk 1000mbc ssar 1GB SSA C Physical Disk Drive
pdisk 2000mbc ssar 2GB SSA C Physical Disk Drive
bus pcic pci PCI Bus
bus isa pci ISA Bus
adapter df1000f9 pc1 FC Adapter
driver efscsi 10cb FC SCSI I/O Controller Protocol Device
adapter c1110358 pc1 USB OHCI Adapter (c1110358)
adapter ad100501 pc1 ATA/IDE Controller Device
adapter 4f11100 pc1 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
adapter ccm pc1 Name of the Common Character Mode device driver
driver hd1c 33112b9 IBM HDLC Network Device Driver
adapter 33112b9 pc1 IBM 2-Port Multiprotocol Adapter (33112b9)
adapter 2b102005 pc1 GXT130P Graphics Adapter
adapter 2b101a05 pc1 GXT120P Graphics Adapter
adapter 23100020 pc1 IBM 10/100 Mbps Ethernet PCI Adapter (23100020)

if tr TR Token Ring Network Interface
if vi VI Virtual IP Address Network Interface
if xt XT X.25 Network Interface
tcpip inet TCPIP Internet Network Extension
swap paging nfs NFS Swap DEVICE
driver mediamedia SCSI Device Drawer
driver scsi1 dasd SCSI DASD Drawer
adapter 4f111b00 pc1 IBM 128-Port Async (PCI) Adapter
concentrator 16c232 sync_pci 16-Port RAN EIA-232 for 128-Port Adapter
concentrator 16e232 sync_pci 16-Port Enhanced RAN EIA-232 for 128-Port Adapter
concentrator 16e422 sync_pci 16-Port Enhanced RAN RS-422 for 128-Port Adapter
if at AT ATM Network Interface
adapter 14105300 pc1 IBM PCI 25MBPS ATM Adapter (14105300)

2. To list all the devices in the Customized Devices object class, type:
   lsdev -C

The system displays a message similar to the following:

sys0 Available System Object
sysplanar0 Available System Planar
mem0 Available Memory
L2cache0 Available L2 Cache
proc0 Available 00-00 Processor
pci0 Available PCI Bus

Alphabetical Listing of Commands 313
The system displays a message similar to the following:

```
sa0 Available 01-S1 Standard I/O Serial Port
sa1 Available 01-S2 Standard I/O Serial Port
siokma0 Available 01-K1 Keyboard/Mouse Adapter
fda0 Available 01-D1 Standard I/O Diskette Adapter
scsi0 Available 10-60 Wide/Ultra-2 SCSI I/O Controller
scsi1 Available 10-61 Wide/Ultra-2 SCSI I/O Controller
sa2 Available 10-68 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
sa3 Available 10-70 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
sa4 Available 10-78 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
```

4. To list all tape devices in the Predefined Devices object class, type:

```
lsdev -P -c tape
```

The system displays a message similar to the following:

```
tape ost  fcp  Other FC SCSI Tape Drive
tape scsd  fcp  FC SCSI Tape Drive
tape ost  iscsi  Other iSCSI Tape Drive
tape scsd iscsi iSCSI Tape Drive
tape 1200mb-c scsi 1.2 GB 1/4-Inch Tape Drive
tape 150mb scsi 150 MB 1/4-Inch Tape Drive
tape 3490e scsi 3490E Autoloading Tape Drive
tape 4mm2gb scsi 2.0 GB 4mm Tape Drive
tape 4mm4gb scsi 4.0 GB 4mm Tape Drive
tape 525mb scsi 525 MB 1/4-Inch Tape Drive
tape 8mm scsi 2.3 GB 8mm Tape Drive
```
5. To list the supported device classes from the Predefined Devices object class, type:

`lsdev -P -r class`

The system displays a message similar to the following:

```
PCM
adapter
aio
array
bus
cdrom
concentrator
container
dial
disk
diskette
drawer
driver
gxme
if
keyboard
lft
logical_volume
lpfk
lvm
memory
mouse
pdisk
planar
port
posix_aio
printer
processor
pseudo
pty
rcm
rwoptical
swap
sys
tablet
tape
tcpiptm
scsd
tmssa
tty
```

6. To list the supported subclasses in the Predefined Devices object class for the `disk` class, type:

`lsdev -P -c disk -r subclass`

The system displays a message similar to the following:

```
dar
fcp
fdar
ide
iscsi
iscsi
scr
scsi
ssar
```
7. To list the name, class, subclass, and type of every device in the Available state in the Customized Devices object class with column headers, type:

```bash
lsdev -C -H -S a -F 'name class subclass type'
```

The system displays a message similar to the following:

```
name  class  subclass  type
sys0  sys    node     chrp
sysplanar0 planar sys sysplanar_rspc
mem0  memory sys      totmem
L2cache0 memory sys    L2cache_rspc
proc0 processor sys    proc_rspc
pci0  bus    chrp     pci
pci1  bus    chrp     pci
isa0  bus    pci      isa
siota0 adapter isa_sio isa_tablet
ppa0  adapter isa_sio chrp_ecp
sa0   adapter isa_sio pnp501
sal   adapter isa_sio pnp501
paud0 adapter isa_sio baud4232
siokma0 adapter isa_sio kma_chrp
fda0  adapter isa_sio pnp700
scsi0 adapter pci     sym96
scsi1 adapter pci     sym96
sa2   adapter pci     4f111100
sa3   adapter pci     4f111100
sa4   adapter pci     4f111100
ent0  adapter pci     23100020
mg20  adapter pci     2b102005
sa5   adapter pci     4f111100
sioka0 adapter kma_chrp keyboard
sioma0 adapter kma_chrp mouse
fd0   diskette siofd   fd
cd0   cdrom scsi      scsd
hdisk0 disk scsi      scsd
kbd0  keyboard std_k  ps2
mouse0 mouse std_m    mse_3b
lvdd  lvm lvm  lvdd
1ft0  lft node  1ft
inet0 tcpip TCP/IP inet
en0   if EN    en
lo0   if LO    lo
pty0  pty pty    pty
rcm0  rcm node  rcm
tty0  tty rs232    tty
tty1  tty rs232    tty
```

8. To list the name, class, location, and physloc of all adapter devices in the Customized Devices object class with column headers, type:

```bash
lsdev -C -c adapter -F 'name class location physloc'
```

The system displays a message similar to the following:

```
siota0 adapter 01-Q1 P1/Q1
ppa0   adapter 01-R1 P1/R1
sa0    adapter 01-S1 P1/S1
sal    adapter 01-S2 P1/S2
paud0  adapter 01-Q2 P1/Q2
siokma0 adapter 01-K1 P1/K1
fda0   adapter 01-D1 P1/D1
scsi0  adapter 10-60 P1/Z1
scsi1  adapter 10-61 P1/Z2
sa2    adapter 10-68 P1-13/Q1
sa3    adapter 10-70 P1-14/Q1
sa4    adapter 10-78 P1-15/Q1
ent0   adapter 10-80 P1/E1
```

316 Commands Reference, Volume 3
To list the all the devices whose names begin with the letters 'hdi', enter:

```
lsdev -l hdi*
```

The system displays a message similar to the following:

```
hdisk0 Available 10-60-00-8,0 16 Bit SCSI Disk Drive
hdisk1 Available 10-60-00-9,0 16 Bit SCSI Disk Drive
```

To list the all the children of the pci0 bus, type:

```
lsdev -p pci0
```

The system displays a message similar to the following:

```
ent0 Available 10-80 IBM 10/100 Mbps Ethernet PCI Adapter (23100020)
isa0 Available 10-58 ISA Bus
scsi0 Available 10-60 Wide/Fast-20 SCSI I/O Controller
scsi1 Available 10-88 Wide/2 SCSI I/O Controller
tok0 Available 10-70 IBM PCI Tokenring Adapter (14103e00)
```

To list the devices whose names are contained in the file /tmp/f, type:

```
cat /tmp/f | lsdev -l -
```

The system displays a message similar to the following:

```
pci0 Available PCI Bus
scsi0 Available 10-60 Wide/Fast-20 SCSI I/O Controller
hdisk0 Available 10-60-00-8,0 16 Bit SCSI Disk Drive
```

### Files

```
/usr/sbin/lsdev
```

Contains the `lsdev` command.

### Related Information

The `chdev`, `lsattr`, `lsconn`, `lsparent`, `mkdev`, `rmdev` command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

System Management Interface Tool (SMIT) Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices tells you about the SMIT application.

### lsdisp Command

#### Purpose

Lists the displays available on the system.

#### Syntax

```
lsdisp [ -H ]
```

Description

The `lsdisp` command lists the displays currently available on the system, displaying a logical name of the display, a physical slot number of a display adapter, the type of bus to which a graphics display is attached, a display name and a description of each of the displays. This command also lists the default display.

Flags

- `l` Specifies the removal of all header information and `Default display` from format.

Examples

To list all available displays, enter:

```
lsdisp
```

The following output of the `lsdisp` command lists three available displays for AIX 5.1 and earlier:

<table>
<thead>
<tr>
<th>DEV_NAME</th>
<th>SLOT</th>
<th>BUS</th>
<th>ADPT_NAME</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>ppr0</td>
<td>00-01</td>
<td>mca</td>
<td>POWER_G4</td>
<td>Midrange Graphics Adapter</td>
</tr>
<tr>
<td>gda0</td>
<td>00-03</td>
<td>mca</td>
<td>colordga</td>
<td>Color Graphics Display Adapter</td>
</tr>
<tr>
<td>ppr1</td>
<td>00-04</td>
<td>mca</td>
<td>POWER_Gt3</td>
<td>Midrange Entry Graphics Adapter</td>
</tr>
</tbody>
</table>

Default display = gda0

Files

`bin/lsdisp` Contains the `lsdisp` command.

Related Information

The `chdisp` command.


Isfilt Command

Purpose

Lists filter rules from either the filter table or the IP Security subsystem.

Syntax

```
isfilt -v 4|6 [-n fid_list] [-a] [-d]
```

Description

Use the `isfilt` command to list filter rules and their status.

Note: Filter description fields are not listed in the kernel. No filter description text will be displayed when active or dynamic filter rules are listed.

Flags

- `a` List only the active filter rules. The active filter rules are the rules being used by the filter kernel currently. If omitted, all the filter rules in the filter rule table will be listed.
Lists the dynamic filter rules used for Internet Key Exchange (IKE) tunnels. This table is built dynamically as IKE negotiations start creating IP Security tunnels and their corresponding filter rules are added to the dynamic IKE filter table.

Specifies the ID(s) of filter rule(s) that are displayed. The fid_list is a list of filter IDs separated by a space or "," or "-". The -n is not for active filter rules. This flag cannot be used with the -a flag.

IP version of the filter rule you want to list. Valid values for this flag are 4 and 6. If this flag is not used, both IP version 4 and IP version 6 are listed.

Isfont Command

Purpose
Lists the fonts available to the display.

Syntax
Isfont [-l]

Description
The Isfont command displays a list of the fonts available to the display. The font identifier can help you change fonts using the chfont command.

You can use a Web-based System Manager Devices application (wsm devices fast path) to run this command. You could also use the System Management Interface Tool (SMIT) smit Isfont fast path to run this command.

Flags
- Specifies the removal of all header information from format of data.

Examples
To list all fonts available to the display, enter:
lsfont

The following example displays the font identifier, font name, glyph size, and font encoding for each available font:

<table>
<thead>
<tr>
<th>FNT</th>
<th>FILE</th>
<th>GLYPH</th>
<th>FNT</th>
<th>ID</th>
<th>NAME</th>
<th>SIZE</th>
<th>ENCODING</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Erg22.isol.snf</td>
<td>12x30</td>
<td>ISO8859-1</td>
<td>0</td>
<td>Erg22.isol.snf</td>
<td>12x30</td>
<td>ISO8859-1</td>
</tr>
<tr>
<td>1</td>
<td>Erg11.isol.snf</td>
<td>8x15</td>
<td>ISO8859-1</td>
<td>1</td>
<td>Erg11.isol.snf</td>
<td>8x15</td>
<td>ISO8859-1</td>
</tr>
</tbody>
</table>

For further details about the fonts available, see Text Fonts for the AIX System

Files
/bin/isfont Contains the Isfont command.
/usr/lpp/fonts Contains fonts directory.

Related Information
The chfont command, mkfont command.
Isfs Command

Purpose
Displays the characteristics of file systems.

Syntax
```
lsfs [-q] [-c | -l] [-a | -v] VfsType [-u MountGroup] [FileSystem...] 
```

Description
The `lsfs` command displays characteristics of file systems, such as mount points, automatic mounts, permissions, and file system size. The `FileSystem` parameter reports on a specific file system. The following subsets can be queried for a listing of characteristics:

- All file systems
- All file systems of a certain mount group
- All file systems of a certain virtual file system type
- One or more individual file systems

The `lsfs` command displays additional Journaled File System (JFS) or Enhanced Journaled File System (JFS2) characteristics if the `-q` flag is specified.

You can use a Web-based System Manager File Systems application (`wsm fs` fast path) to run this command. You could also use the System Management Interface Tool (SMIT) `smit lsfs` fast path to run this command.

Flags

- `-a`
  Lists all file systems (default).
- `-c`
  Specifies that the output should be in colon format.
- `-l`
  Specifies that the output should be in list format.
- `-q`
  Queries the logical volume manager (LVM) for the logical volume size (in 512-byte blocks). Additional information is provided for JFS and JFS2 file systems. Queries the JFS superblock for the file system size, the fragment size, the compression algorithm (if any), and the number of bytes per i-node (nbpi). Queries the JFS2 superblock for the file system size and the file system block size.

  This information is not reported for other virtual file system types. It is displayed in addition to other file system characteristics reported by the `lsfs` command.

- `-u MountGroup`
  Reports on all file systems of a specified mount group.
- `-v VfsType`
  Reports on all file systems of a specified type.

Examples

1. To show all file systems in the `/etc/filesystems` file, enter:
   
   ```
   lsfs
   ```

2. To show all file systems of vfs type jfs, enter:
   
   ```
   lsfs -v jfs
   ```

320  Commands Reference, Volume 3
3. To show the file system size, the fragment size, the compression algorithm (if any), and the number of bytes per i-node as recorded in the superblock of the root file system, enter:

```
lsfs -q /
```

### Files

#### /etc/filesystems

Lists the known file systems and defines their characteristics.

### Related Information

The **chfs** command, **crfs** command, **rmfs** command.


For information on installing the Web-based System Manager, see [Chapter 2: Installing Web-based System Manager](https://www.aixdocs.com) in *AIX 5L Version 5.2 Web-based System Manager Administration Guide*.


### lsgroup Command

#### Purpose

Displays group attributes.

#### Syntax

```
lsgroup [ -R load_module ] [ -c | -f ] [ -a List ] { ALL | Group [ ,Group ] ...}
```

#### Description

The **lsgroup** command displays group attributes. You can use this command to list all the system groups and their attributes or you can list all the attributes of individual groups. Since there is no default parameter, you must enter the **ALL** keyword to list all the system groups and their attributes. All the attributes described in the **chgroup** command appear. If the **lsgroup** command cannot read one or more attributes, it lists as much information as possible. To view a selected attribute, use the **-a List** flag.

**Note:** If you have a Network Information Service (NIS) database installed on your system, some user information may not appear when you use the **lsgroup** command.

By default, the **lsgroup** command lists each group on one line. It displays attribute information as **Attribute=Value** definitions, each separated by a blank space. To list the group attributes in stanza format, use the **-f** flag. To list the information in colon-separated records, use the **-c** flag.

You can use a Web-based System Manager Users application (**wsm users** fast path) to run this command. You could also use the System Management Interface Tool (SMIT) **smit lsgroup** fast path to run this command.
Flags

-a List Specifies the attributes to display. The List parameter can include any attribute defined in the chgroup command, and requires a blank space between attributes. If you specify an empty list, only the group names are listed.

-c Displays the attributes for each group in colon-separated records, as follows:
#name: attribute1: attribute2: ...
Group: value1: value2: ...

-f Displays the group attributes in stanzas. Each stanza is identified by a group name. Each Attribute=Value pair is listed on a separate line:
group:
    attribute1=value
    attribute2=value
    attribute3=value

-R load_module Specifies the loadable I&A module used to get the group attribute list.

Security

Access Control: This command should be a general user program with execute (x) access for all users. Attributes are read with the access rights of the invoker, so all users may not be able to access all the information. This depends on the access policy of your system. This command should have the trusted computing base attribute.

Files Accessed:

Mode File
r /etc/group
r /etc/security/group
r /etc/passwd

Limitations

Listing a group may not be supported by all loadable I&A modules. If the loadable I&A module does not support listing a group, then an error is returned.

Examples

1. To display the attributes of the finance group in the default format, type:
   lsgroup finance

2. To display the id, members (users), and administrators (adms) of the finance group in stanza format, type:
   lsgroup -f -a id users adms finance

3. To display the attributes of all the groups in colon-separated format, type:
   lsgroup -c ALL

   All the attribute information appears, with each attribute separated by a blank space.

4. To display the attributes of the LDAP I&A loadable module group monsters, type:
   lsgroup -R LDAP monsters

Files

/usr/sbin/lsgroup Contains the lsgroup command.

322 Commands Reference, Volume 3
/etc/group
Contains the basic attributes of groups.
/etc/security/group
Contains the extended attributes of groups.
/etc/passwd
Contains user IDs, user names, home directories, login shell, and finger information.

Related Information
The `chfn` command, `chgrp` command, `chgrpmem` command, `chsh` command, `chuser` command, `lsuser` command, `mkgroup` command, `mkuser` command, `passwd` command, `pwdadm` command, `chgrpmem` command, `setgroups` command, `setsenv` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the `AIX 5L Version 5.2 Security Guide`.

For information on installing the Web-based System Manager, see `Chapter 2: Installing Web-based System Manager` in `AIX 5L Version 5.2 Web-based System Manager Administration Guide`.

Isitab Command

Purpose
Lists records in the `/etc/inittab` file.

Syntax
`lsitab { [-a] | Identifier }`

Description
The `lsitab` command displays a record in the `/etc/inittab` file. You can display all of the records in the `/etc/inittab` file, or use the `Identifier` parameter to display a specific record. The `Identifier` parameter is a 14-character field that uniquely identifies an object.

Flags
- `-a` Specifies that all records in the `/etc/inittab` file are listed.

Examples
1. To display the record for `tty2`, enter:
   ```
   lsitab "tty002"
   ```
   The output is similar to: `tty002:2:respawn:/usr/sbin/getty /dev/tty2`

2. To display all of the records in the `/etc/inittab` file, enter:
   ```
   lsitab -a
   ```
   All of the records in the `/etc/inittab` file are displayed.

Files
`/etc/inittab` Indicates which processes the `init` command starts.

Related Information
The `chitab` command, `init` command, `mkittab` command, `rmitab` command.
### lskbd Command

#### Purpose
List the current software keyboard map loaded into the system.

#### Syntax
lskbd

#### Description
The **lskbd** command displays the absolute pathname of the current software keyboard map loaded into the system.

To list the current software keyboard map enter:
```
lskbd
```

You can use the Devices application in Web-based System Manager (wsm) to change device characteristics. You could also use the System Management Interface Tool (SMIT) **smit lskbd** fast path to run this command.

**Note:** This command can be used only on an LFT display.

#### Example
Following is an example of the listing displayed by the lskbd command:

```
The current software keyboard map = /usr/lib/nls/loc/C.lftkeymap
```

#### Files

<table>
<thead>
<tr>
<th>Path</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>/usr/bin/lskbd</code></td>
<td>Contains the <code>lskbd</code> command.</td>
</tr>
<tr>
<td><code>/usr/lib/nls/loc</code></td>
<td>Software keyboard map directory.</td>
</tr>
</tbody>
</table>

#### Related Information
The **chkbd** command, **smit** command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in **AIX 5L Version 5.2 Web-based System Manager Administration Guide**.

### lslicense Command

#### Purpose
Displays the number of fixed licenses and the status of the floating licensing.

#### Syntax
lslicense [A] [C]

#### Description
The **lslicense** command displays the number of fixed licenses and the status of the floating licensing.
Flags

-A  The -A flag causes the lslicense command to report the current number of available fixed licenses. When the -A flag is not specified, the maximum number of fixed licenses and license status is reported.

-c  Displays the output in : (colon) form.

Examples

1. To display the number of fixed licenses and the floating license status, enter:
   
   lslicense
   
   Example output would be:
   
   Maximum number of fixed licenses is 10.
   Floating licensing is enabled.

2. To display the number of fixed licenses and the floating license status in a colon format, enter:
   
   lslicense -c
   
   Example output would be:
   
   #fixed:floating
   10:on

3. To display license information including the number of available fixed licenses, enter:
   
   lslicense -A
   
   Output similar to the following will display:
   
   Maximum number of fixed licenses is 2.
   Floating licensing is disabled.
   Number of available fixed licenses is 2.

Related Information

The chlicense command and monitor daemon.

llspp Command

Purpose

Lists installed software products.

Syntax

llspp {-d | -f | -h | -i | -L | -p} { -a | -c | -J | -q | -I | -O { -r | -s | -u}} [ FilesetName ... | FixID ... | all ]

llspp -w { -c | -q | -O { -r | -s | -u }} [ FileName ... | all ]

llspp -L -c [-v]

llspp -S [A|O]

llspp -e
Description

The `lslpp` command displays information about installed filesets or fileset updates. The `FilesetName` parameter is the name of a software product. The `FixID` (also known as PTF or program temporary fix ID) parameter specifies the identifier of an update to a formatted fileset.

When only the `-i` (lowercase L) flag is entered, the `lslpp` command displays the latest installed level of the fileset specified for formatted filesets. The base level fileset is displayed for formatted filesets. When the `-a` flag is entered along with the `-i` flag, the `lslpp` command displays information about all installed filesets for the `FilesetName` specified. The `-I` (uppercase i) flag combined with the `-i` (lowercase L) flag specifies that the output from the `lslpp` command should be limited to base level filesets.

The `-d`, `-f`, `-h`, `-i`, `-l` (lowercase L), `-L`, and `-p` flags request different types of output reports.

The `-a`, `-c`, `-J`, and `-q` flags specify the amount and format of the information that is displayed in the report.

The `-O` flag specifies that data is to come from a specified part of the fileset. The part may be the root part, `-Or`, the share part, `-Os`, or the `usr` part, `-Ou`.

The default value for the `FilesetName` parameter is `all`, which displays information about all installed software products. Pattern matching characters, such as `*` (asterisk) and `?` (question mark), are valid in the `ProductName` and `FixID` parameters. You don’t have to enclose these characters in “” (single quotation marks). However, using single quotation marks prevents you from searching the contents of your present directory.

Output Values

Much of the output from the `lslpp` command is understandable without an explanation. Other fields contain data that needs to be defined. The following sections define terms used in several of the output fields.

State Values

The `state` field in the `lslpp` output gives the state of the fileset on your system. It can have the following values:

<table>
<thead>
<tr>
<th>State</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>APPLIED</td>
<td>The specified fileset is installed on the system. The <code>APPLIED</code> state means</td>
</tr>
<tr>
<td></td>
<td>that the fileset can be rejected with the <code>installp</code> command and the</td>
</tr>
<tr>
<td></td>
<td>previous level of the fileset restored. This state is only valid for</td>
</tr>
<tr>
<td></td>
<td>Version 4 fileset updates and 3.2 migrated filesets.</td>
</tr>
<tr>
<td>APPLYING</td>
<td>An attempt was made to apply the specified fileset, but it did not</td>
</tr>
<tr>
<td></td>
<td>complete successfully, and cleanup was not performed.</td>
</tr>
<tr>
<td>BROKEN</td>
<td>The specified fileset or fileset update is broken and should be</td>
</tr>
<tr>
<td></td>
<td>reinstalled before being used.</td>
</tr>
<tr>
<td>COMMITTED</td>
<td>The specified fileset is installed on the system. The <code>COMMITTED</code> state</td>
</tr>
<tr>
<td></td>
<td>means that a commitment has been made to this level of the software. A</td>
</tr>
<tr>
<td></td>
<td>committed fileset update cannot be rejected, but a committed fileset base</td>
</tr>
<tr>
<td></td>
<td>level and its updates (regardless of state) can be removed or deinstalled</td>
</tr>
<tr>
<td></td>
<td>by the <code>installp</code> command.</td>
</tr>
<tr>
<td>EFIX LOCKED</td>
<td>The specified fileset was installed successfully and locked by the</td>
</tr>
<tr>
<td></td>
<td>emergency fix (efix) manager.</td>
</tr>
<tr>
<td>OBSOLETE</td>
<td>The specified fileset was installed with an earlier version of the</td>
</tr>
<tr>
<td></td>
<td>operating system but has been replaced by a repackaged (renamed) newer</td>
</tr>
<tr>
<td></td>
<td>version. Some of the files that belonged to this fileset have been</td>
</tr>
<tr>
<td></td>
<td>replaced by versions from the repackaged fileset.</td>
</tr>
<tr>
<td>COMMITTING</td>
<td>An attempt was made to commit the specified fileset, but it did not</td>
</tr>
<tr>
<td></td>
<td>complete successfully, and cleanup was not performed.</td>
</tr>
<tr>
<td>REJECTING</td>
<td>An attempt was made to reject the specified fileset, but it did not</td>
</tr>
<tr>
<td></td>
<td>complete successfully, and cleanup was not performed.</td>
</tr>
</tbody>
</table>

326  Commands Reference, Volume 3
**Action Values**

The action field in the `lspp` output identifies the installation action that was taken for the fileset. The following values may be found in this field:

<table>
<thead>
<tr>
<th>Action</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>APPLY</td>
<td>An attempt was made to apply the specified fileset.</td>
</tr>
<tr>
<td>CLEANUP</td>
<td>An attempt was made to perform cleanup for the specified fileset.</td>
</tr>
<tr>
<td>COMMIT</td>
<td>An attempt was made to commit the specified fileset.</td>
</tr>
<tr>
<td>REJECT</td>
<td>An attempt was made to reject the specified fileset.</td>
</tr>
</tbody>
</table>

**Status Values**

The status field in the `lspp` output identifies the resultant status in the history of installation actions. The following values may be found in this field:

<table>
<thead>
<tr>
<th>Status</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>BROKEN</td>
<td>The fileset was left in a broken state after the specified action.</td>
</tr>
<tr>
<td>CANCELED</td>
<td>The specified action was canceled before it completed.</td>
</tr>
<tr>
<td>COMPLETE</td>
<td>The commitment of the fileset has completed successfully.</td>
</tr>
</tbody>
</table>

**Flags**

- **-a** Displays all the information about filesets specified when combined with other flags. This flag shows all updates when combined with the `-i` flag and all history when combined with the `-h` flag. This flag cannot be specified with the `-f` flag.
- **-c** Displays information as a list separated by colons. This flag cannot be specified with the `-J` flag.
- **-d** Displays filesets that are dependents of the specified software. A dependent fileset is one that has the specified software as a prerequisite, corequisite, ifrequisite, or installed requisite.
- **-e** Displays every efix installed on the system.
- **-E** Lists license agreements.
- **-f** Displays the names of the files added to the system during installation of the specified fileset. This flag cannot be specified with the `-a` flag.
- **-h** Displays the installation and update history information for the specified fileset. You cannot use this flag with the `-J` flag.
- **-I** (uppercase i) Limits the inputs to software products.
- **-i** Displays the product information for the specified fileset.
- **-J** Generates output in a form suitable for the System Management Interface Tool (SMIT) command to list output. This flag can only be specified with the `-I` (lowercase L) and `-L` flags.
- **-l** (lowercase L) Displays the name, most recent level, state, and description of the specified fileset.
- **-L** Displays the name, most recent level, state, and description of the specified fileset. Part information (usr, root, and share) is consolidated into the same listing. For formatted filesets, it displays the most recent maintenance level for the specified filesets. In addition, this flag lists any subsystem selective fixes that were installed on top of the maintenance level. RPM and ISMP images are also listed.
- **-O** Lists information for the specified part of the fileset. When the `-O` flag is not specified information is listed for all parts. This option is designed for use by the nim command to list software product information for diskless or databaseless workstations. You can use the following flags with this flag:
  - **-r** Indicates to list information for the root part.
  - **-s** Indicates to list information for the /usr/share part.
  - **-u** Indicates to list information for the /usr part.
- **-p** Displays requisite information for the specified fileset.
- **-q** Suppresses the display of column headings.
- **-S [A/O]** Displays a list of automatically installed filesets and a list of optionally installed filesets. If the `-S` flag is followed by `A`, then only the automatically installed filesets are listed. If the `-S` flag is followed by `O`, then only the optionally installed filesets are listed.
-v Displays additional information from the vendor database. This flag is only valid when used with both the -L and the -c flag.

-w Lists fileset that owns this file. This flag applies to AIX 4.2 or later.

You must specify one of the mutually exclusive flags: -d, -e, -E, -f, -h, -i, -l, -L, -p, -s, and -w.

**Examples**

1. To list the installation state for the most recent level of installed filesets for all of the bos.rte filesets, type:
   ```
   lslpp -l "bos.rte.*"
   ```

2. To list the installation state for the base level and updates for the fileset bos.rte.filesystem, type:
   ```
   lslpp -la bos.rte.filesystem
   ```

3. To list the installation history information of all the filesets in the bos.net software package, type:
   ```
   lslpp -ha 'bos.net.*'
   ```

4. To list the names of all the file of the bos.rte.lvm fileset, type:
   ```
   lslpp -f bos.rte.lvm
   ```

5. To list the fileset that owns installp, type:
   ```
   lslpp -w /usr/sbin/installp
   ```

   Output similar to the following displays:

   ```
   File Fileset Type
   /usr/sbin/installp bos.rte.install File
   ```

6. To list the fileset that owns all file names that contain installp, type:
   ```
   lslpp -w **installp*"
   ```

   Output similar to the following displays:

   ```
   File Fileset Type
   /usr/sbin/installp bos.rte.install File
   /usr/clvm/sbin/linstallpv prpq.clvm File
   /usr/lpp/bos.sysmgt/nim/methods/c_installp
      --bos.sysmgt.nim.client File
   ```

7. To display all files in the inventory database, type:
   ```
   lslpp -w
   ```

8. To display the installation state for the RPM cdrecord image, type:
   ```
   lslpp -L cdrecord
   ```

**Files**

- `/etc/objrepos/history` Specifies installation and update history information of all software products on the root.

- `/usr/lib/objrepos/history` Specifies installation and update history information of all software products on the `/usr` file system.

- `/usr/share/lib/objrepos/history` Specifies installation and update history information of all software products on the `/usr/share` file system.

- `/etc/objrepos/lpp` Specifies installation information of all software products on the root.

- `/usr/lib/objrepos/lpp` Specifies installation information of all software products on the `/usr` file system.
/usr/share/lib/objrepos/lpp
Specifies installation information of all software products on the /usr/share file system.

/etc/objrepos/product
Specifies installation and update information of all software products on the root.

/usr/lib/objrepos/product
Specifies installation and update information of all software products on the /usr file system.

/usr/share/lib/objrepos/product
Specifies installation and update information of all software products on the /usr/share file system.

/etc/objrepos/inventory
Specifies names and locations of files in a software product on the root.

/usr/lib/objrepos/inventory
Specifies names and locations of files in a software product on the /usr file system.

/usr/share/lib/objrepos/inventory
Specifies names and locations of files in a software product on the /usr/share file system.

Related Information
The emgr command, installp command, inulag command, nim command.

islv Command
Purpose
Displays information about a logical volume.

Syntax
To Display Logical Volume Information
islv [-L] [ -l ] [ -m ] [ -n PhysicalVolume ] LogicalVolume

To Display Logical Volume Allocation Map
islv [-L] [ -n PhysicalVolume ] [-P PhysicalVolume ] LogicalVolume

Description
The islv command displays the characteristics and status of the LogicalVolume or lists the logical volume allocation map for the physical partitions on the PhysicalVolume. The logical volume can be a name or identifier.

Note: If the islv command cannot find information for a field in the Device Configuration Database, it will insert a question mark (?) in the value field. As an example, if there is no information for the LABEL field, the following is displayed:

LABEL: ?

The command attempts to obtain as much information as possible from the description area when it is given a logical volume identifier.

You can use the Volumes application in Web-based System Manager to change volume characteristics. You could also use the System Management Interface Tool (SMIT) smit islv fast path to run this command.

Flags
-L
Specifies no waiting to obtain a lock on the Volume group.

Note: If the volume group is being changed, using the -L flag gives unreliable date.
-l

Lists the following fields for each physical volume in the logical volume:

PV       Physical volume name.

Copies   The following three fields:
          - The number of logical partitions containing at least one physical partition
            (no copies) on the physical volume
          - The number of logical partitions containing at least two physical
            partitions (one copy) on the physical volume
          - The number of logical partitions containing three physical partitions (two
            copies) on the physical volume

In band  The percentage of physical partitions on the physical volume that belong to
          the logical volume and were allocated within the physical volume region
          specified by Intra-physical allocation policy.

Distribution
          The number of physical partitions allocated within each section of the
          physical volume: outer edge, outer middle, center, inner middle, and inner
          edge of the physical volume.

-m

Lists the following fields for each logical partition:

LPs      Logical partition number.

PV1      Physical volume name where the logical partition’s first physical partition is
          located.

PP1      First physical partition number allocated to the logical partition.

PV2      Physical volume name where the logical partition’s second physical
          partition (first copy) is located.

PP2      Second physical partition number allocated to the logical partition.

PV3      Physical volume name where the logical partition’s third physical partition
          (second copy) is located.

PP3      Third physical partition number allocated to the logical partition.

-n PhysicalVolume

Accesses information from the specific descriptor area of PhysicalVolume variable.
The information may not be current since the information accessed with the -n flag
has not been validated for the logical volumes. If you do not use the -n flag, the
descriptor area from the physical volume that holds the validated information is
accessed and therefore the information that is displayed is current. The volume
need not be active when you use this flag.

-p PhysicalVolume

Displays the logical volume allocation map for the PhysicalVolume variable. If you
use the LogicalVolume parameter, any partition allocated to that logical volume is
listed by logical partition number. Otherwise, the state of the partition is listed as
one of the following:

used     Indicates that the partition is allocated to another logical volume.

free     Indicates that the specified partition is not being used on the system.

stale    Indicates that the specified partition is no longer consistent with other
          partitions. The computer lists the logical partitions number with a question
          mark if the partition is stale.

If no flags are specified, the following status is displayed:

Logical volume    Name of the logical volume. Logical volume names must be unique
                  systemwide and can range from 1 to 15 characters.

Volume group      Name of the volume group. Volume group names must be unique
                  systemwide and can range from 1 to 15 characters.
### Logical volume identifier (LV identifier)
Identifier of the logical volume.

### Permission
Access permission; read-only or read-write.

### Volume group state (VG state)
State of the volume group. If the volume group is activated with the `varyonvg` command, the state is either active/complete (indicating all physical volumes are active) or active/partial (indicating all physical volumes are not active). If the volume group is not activated with the `varyonvg` command, the state is inactive.

### Logical volume state (LV state)
State of the logical volume. The Opened/stale status indicates the logical volume is open but contains physical partitions that are not current. Opened/syncd indicates the logical volume is open and synchronized. Closed indicates the logical volume has not been opened.

#### Type
Logical volume type.

#### Write verify
Write verify state of on or off.

#### Mirror write consistency
Mirror write consistency state of on or off.

#### Max LPs
Maximum number of logical partitions the logical volume can hold.

#### PP size
Size of each physical partition.

#### Copies
Number of physical partitions created for each logical partition when allocating.

#### Schedule policy (Sched policy)
Sequential or parallel scheduling policy.

#### LPs
Number of logical partitions currently in the logical volume.

#### PPs
Number of physical partitions currently in the logical volume.

#### Stale partitions
Number of physical partitions in the logical volume that are not current.

#### BB policy
Bad block relocation policy.

#### Inter-policy
Inter-physical allocation policy.

#### Intra-policy
Intra-physical allocation policy.

#### Upper bound
If the logical volume is super strict, upper bound is the maximum number of disks in a mirror copy.

#### Relocatable
Indicates whether the partitions can be relocated if a reorganization of partition allocation takes place.

#### Mount point
File system mount point for the logical volume, if applicable.

#### Label
Specifies the label field for the logical volume.

#### Each LP copy on a separate PV?
The strictness value. Current state of allocation, strict, nonstrict, or superstrict. A strict allocation states that no copies for a logical partition are allocated on the same physical volume. If the allocation does not follow the strict criteria, it is called nonstrict. A nonstrict allocation states that at least one occurrence of two physical partitions belong to the same logical partition. A superstrict allocation states that no partition from one mirror copy may reside the same disk as another mirror copy.

#### Serialize IO?
Serialization of overlapping IOs state of yes or no. If serialization is turned on (yes), then overlapping IOs are not allowed on a block range, and only a single IO in a block range is processed at any one time. Most applications, such as file systems and databases, perform serialization; therefore, serialization should be turned off (no). The default setting for new logical volumes is no.

#### Stripe width
The number of physical volumes being striped across.

#### Stripe size
The number of bytes per stripe.

### Examples

1. To display information about logical volume lv03, enter:
   ```
   ls LV lv03
   ```
   Information about logical volume lv03, its logical and physical partitions, and the volume group to which it belongs is displayed.
2. To display the logical volume allocation map for hdisk2, enter:
   
   ```
   ls1v -p hdisk2
   ```

   An allocation map for hdisk2 is displayed, showing the state of each partition. Since no **LogicalVolume** parameter was included, the map does not contain logical partition numbers specific to any logical volume.

3. To display information about logical volume lv03 by physical volume, enter:
   
   ```
   ls1v -l lv03
   ```

   The characteristics and status of lv03 are displayed, with the output arranged by physical volume.

4. To display information about physical volume hdisk3 gathered from the descriptor area on hdisk2, enter:
   
   ```
   ls1v -n hdisk2 -p hdisk3 lv02
   ```

   An allocation map, using the descriptor area on hdisk2, is displayed. Because the **LogicalVolume** parameter is included, the number of each logical partition allocated to that logical volume is displayed on the map.

5. To display information about a specific logical volume, using the identifier, enter:
   
   ```
   ls1v 00000256a81634bc.2
   ```

   All available characteristics and status of this logical volume are displayed.

**File**

/`usr/sbin` Contains the **ls1v** command.

**Related Information**

The `chlv` command, `lspv` command, `lsvg` command, `mklv` command, `reorgvg` command, `varyonvg` command.

Monitoring and Tuning Disk I/O in AIX 5L Version 5.2 Performance Management Guide

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

System Management Interface Tool (SMIT) Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices explains the structure, main menus, and tasks that are done with SMIT.

**Ismaster Command**

**Purpose**

Displays the characteristics for the configuration of an NIS master server.

**Syntax**

/`usr/sbin/ismaster [ `-c` | `-l` ]`
Description
The `lsmaster` command displays the characteristics of an NIS master server. The host names of the slave servers are listed along with the currently served domains.

You can use the Devices application in Web-based System Manager (wsm) to change device characteristics. You could also use the System Management Interface Tool (SMIT) `smit lsmaster` fast path to run this command.

Flags
- `-c` Specifies that the output should be in colon format. This flag is the default.
- `-l` Specifies that the output should be in list format.

Examples
To list the NIS master server characteristics in colon format, enter:

```
lsmaster -c
```

Files
```
/var/yp/domainname directory
```
Contains the NIS maps for the NIS domain.

Related Information
The `chmaster` command, `mkmaster` command, `rmyp` command, `smit` command.


NIS Reference in AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


Ismcode Command

Purpose
Displays microcode and firmware levels of the system and adapters and devices.

Syntax
```
lscode [ -A ] [ -d ] [ -r ] [ -c ]
```
Description
The **lsmcode** command when run without any flags, displays the platform system firmware microcode level and the service processor microcode levels, if supported. Not all systems contain a service processor, nor do all systems support displaying the system processor level. Information on a specific device is displayed with the `-d` flag.

If you run the **lsmcode** command with the `-r` or `-c` flag, it displays the microcode levels in a **printf** format; that is, not a menu. This method is preferred if running **lsmcode** from a script.

Flags

- **-A**
  Displays microcode level information for all supported devices. Using this flag assumes the `-r` flag.

- **-c**
  Displays the microcode/firmware levels without using menus.

- **-d Name**
  Displays microcode level information for the named device.

- **-r**
  Displays the microcode/firmware levels in a tabular format. The microcode level is preceded by a Type if supported or required.

Current supported Types are as follows:

- **system**
  System Firmware

- **service**
  Service Processor

- **adapter**
  Adapter Functional microcode

- **adapter-boot**
  Adapter Boot Microcode

- **raid-dasd**
  DASD Microcode in a RAID array

- **backplane**
  Backplane Microcode in a RAID subsystem

Examples

1. To display the system firmware level and service processor (if present), type:

   `lsmcode -c`

   The system displays a message similar to the following:

   ```
   System Firmware level is TCP99256
   ```

2. To display the system firmware level and service processor (if present) in raw mode, type:

   `lsmcode -r`

   The system displays a message similar to the following:

   ```
   system:TCP99256
   ```

3. To display the adapter microcode levels for a RAID adapter **scraid0**, type:

   `lsmcode -r -d scraid0`

   The system displays a message similar to the following:

   ```
   adapter:4.20.18|adapter-boot:4.00.26
   raid-dasd:22:FFC #:DDYS-T0.52403031.53393446
   raid-dasd:26:FFC #:DDYS-T0.52403031.53393446
   raid-dasd:2e:FFC #:DDYS-T0.528A3034.53393243
   ```

4. To display the microcode level for a tape drive **rmt0**, type:

   `lsmcode -r -d rmt0`
The system displays a message similar to the following:

```
C009
```

5. To display the microcode level for all supported devices, type:
```
lsmcode -A
```

The system displays a message similar to the following:

```
sys0!system:TCP99256
rmt0!C009
scraid0!adapter:4.20.18|adapter-boot:4.00.26
raid-dasd:22:FFC #:DDYS-T0.524D3031.53393446
raid-dasd:26:FFC #:DDYS-T0.524D3031.53393446
raid-dasd:2e:FFC #:DDYS-T0.525A3034.53393243
.....
```

**Files**

```
/usr/sbin/lsmcode
```

Contains the `lsmcode` command.

**Related Information**

The `diag` command.

### Ismksysb Command

**Purpose**

Lists or restores the contents of a volume group backup on a specified media.

**Syntax**

```
lsmksysb [ b blocks ] [ f device ] [ a ] [ c ] [ l ] [ r ] [ s ] [ d path ] [ B ] [ D ] [ L ] [ V ] [ file_list ]
```

**Description**

The `lsmksysb` command lists the contents of a volume group backup from tape, file, CD-ROM, or other source and can be used to restore files from a valid backup source. The `lsmksysb` command also works for multi-volume backups such as multiple CDs, DVDs, or tapes.

The `lsmksysb -r` and `restorevgfiles` commands perform identical operations and should be considered interchangeable.

**Flags**

- `a`
  
  Verifies the physical block size of the tape backup, as specified by the `-b block` flag. You may need to alter the block size if necessary to read the backup. The `-a` flag is valid only when a tape backup is used.

- `b blocks`
  
  Specifies the number of 512-byte blocks to read in a single input operation, as defined by the `blocks` parameter. If the `blocks` parameter is not specified, the number of blocks read will default to 100.

- `B`
  
  Prints the volume group backup log to `stdout`.

  This flag will display the past 256 backups (roughly). The log is in alog format and is kept in `/var/adm/ras/vgbackuplog`. Each line of the log is a semicolon-separated list of the file or device name, the command used to make backup, date, shrink size, full size of the backup, and recommended maintenance level (if any).

  **Note:** The shrink size is the size of the data on all filesystems. The full size is total size of each filesystem (unused + data).

- `c`
  
  Produces colon-separated output. This flag only works with the `-l` and `-L` flags.
-d path Specifies the directory path to which the files will be restored, as defined by the path parameter. If the -d parameter is not used, the current working directory is used. This can be a problem if the current working directory is root. We recommend writing to a temporary folder instead of to root.

-D Produces debug output.

-I Displays useful information about a volume group backup.

This flag requires the -f device flag. This flag causes lsmksysb to display information such as volume group, date and time backup was made, uname output from backed up system, oslevel, recommended maintenance level, backup size in megabytes, and backup shrink size in megabytes. The shrink size is the size of the data on all file systems. The full size is the total size of each file system (unused + data). The -f flag also displays the logical volume and file system information of the backed up volume group, equivalent to running "lsvg -I vname".

-L Displays lpp filset information about a mksysb backup only.

This flag requires the -f device flag and displays the equivalent information to that produced by invoking "lspp -I" on the running backed up system. This flag does not produce output about any volume group backup other than that produced by mksysb.

-f device Specifies the type of device containing the backup (file, tape, CD-ROM, or other source) as defined by the device parameter. When -f is not specified, device will default to /dev/rmt0.

-r Specifies to restore the backup files, as defined by the file-list parameter. If the file-list parameter is not specified, then all files in the backup will be restored. If the -r flag is not used, then executing the lsmksysb command only lists the files in the specified backup.

-s Specifies that the backup source is a user volume group and not rootvg.

-V Verifies a tape backup.

This flag requires the -f device flag and works for tape devices only. The -V flag causes lsmksysb to verify the readability of the header of each file on the volume group backup and print any errors that occur to stderr.

Parameters

file_list Identifies the list of files to be restored. This parameter is used only when the -r flag is specified. The full path of the files relative to the current directory should be specified in the space-separated list. All files in the specified directory will be restored unless otherwise directed. If you are restoring all files in a directory, we recommend writing to a temporary folder instead of to root.

Examples

1. To list the contents of the system backup located on the default device /dev/rmt0, enter:
   lsmksysb
2. To list the contents of the system backup located on device /dev/cd1, enter:
   lsmksysb -f /dev/cd1
3. To list the contents of the system backup located on device /dev/cd1, which is a user volume group that is not rootvg, enter:
   lsmksysb -f /dev/cd1 -s
4. To restore /etc/filesystems from the system backup located on device /dev/cd1, enter:
   lsmksysb -f /dev/cd1 -r . /etc/filesystems
5. To restore all files in the /myfs/test directory of the non-rootvg backup, which is located on device /dev/cd1, and write the restored files to /data/myfiles, enter:
   lsmksysb -f /dev/cd1 -r -s -d /data/myfiles . /myfs/test
6. To display colon separated lpp information about a mksysb backup tape located on /dev/rmt0, enter the following:
   lsmksysb -Lc -f /dev/rmt0
7. To display the volume group backup log to `stdout`, enter:
   
   `lssavevg -B`

8. To list volume group and general backup data about a backup located at `/tmp/mybackup`, enter:
   
   `lsmksysb -l -f /tmp/mybackup`

9. To verify the readability of each header on a volume group backup tape in `/dev/rmt0`, enter:
   
   `lsmksysb -V -f /dev/rmt0`

**Files**

`/usr/bin/lsmksysb`  
Contains the `lsmksysb` command

**Related Information**

The `restorevgfiles` command.

---

**Lsnamsv Command**

**Purpose**

Shows name service information stored in the database.

**Syntax**

`lsnamsv { -C | -S "AttributeList ..." } [ -Z ]`

**Description**

The `lsnamsv` high-level command shows customized, TCP/IP-based name service information from the `/etc/resolv.conf` file only. No information from the name server database is shown. The command can extract all customized name service information or selected name service attribute information from the configuration database.

You can use the Network application in Web-based System Manager (wsm) to change network characteristics. You could also use the System Management Interface Tool (SMIT) `smit lsnamerslv` fast path to run this command.

**Flags**

- `-C`
  - Extracts all customized name service configuration information.

- `-S "AttributeList ..."
  - Specifies a selected set of attributes to be extracted from the system configuration database. Attributes can be the following:

    - `domain` Domain name
    - `nameserver` Internet address of name server in dotted decimal format

- `-Z`
  - Specifies that the output be in colon format. This flag is used when the `lsnamsv` command is invoked from the SMIT usability interface.

**Examples**

1. To list all customized name service configuration information in dotted decimal format, enter the following command:
   
   `lsnamsv -C`

2. To list selected attributes, enter the following command:
   
   `lsnamsv -S "domain nameserver"`
The -S flag indicates that the quoted list that follows contains a list of attributes to display.

Related Information

The `namerslv` command.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide. Naming and Understanding the SMIT Interface for TCP/IP in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

lsnfsexp Command

Purpose
Displays the characteristics of directories that are exported with the Network File System (NFS).

Syntax
`/usr/sbin/lsnfsexp [ -c ] [ -l ] [ Directory ] [ -f Exports_file ]`

Description
The `lsnfsexp` command displays the characteristics of NFS-exported directories. The `Directory` parameter specifies the directory to be displayed. If no directory is specified, all directories exported with NFS will be displayed.

Flags

- `-c` Specifies that the output should be in colon format.
- `-l` (Lowercase L) Specifies that the output should be in list format. This flag is the default.
- `Directory` Specifies the directory to be displayed. If no directory is specified, all directories exported with NFS will be displayed.
- `-f Exports_file` Specifies the full path name of the export file to use if other than `/etc/exports`.

Examples
1. To list all of the directories currently exported with NFS in the colon format, enter:
   `lsnfsexp -c`
2. To list all of the directories currently exported with NFS in the colon format and use a specified path name other than `/etc/exports` enter:
   `lsnfsexp -c -f /etc/exports.other`

File
`/etc/exports` Lists the directories the server can export.

Related Information

The `chnfsexp` command, `exports` command, `mknfsexp` command, `rmnfsexp` command, `smit` command.

lsnfsmnt Command

Purpose
Displays the characteristics of NFS mountable file systems.

Syntax
/usr/sbin/lsnfsmnt [-c | -l] [FileSystem]

Description
The lsnfsmnt command displays the current characteristics of NFS mountable file systems. The FileSystem parameter specifies the file system to be displayed in the output. If no file system is specified, all of the file systems that are NFS mountable will be displayed.

Flags
-c Specifies that the output should be in colon format.
-l (Lowercase L) Specifies that the output should be in list format. This flag is the default.

Examples
To list all of the NFS mounted file systems in the colon format, enter:
lsnfsmnt -c

Files
/etc/filesystems Centralizes file system characteristics.

Related Information
The chnfsmnt command, mknfsmnt command, mount command, rmnfsmnt command, smit command.

Isnim Command

Purpose
Displays information about the Network Installation Management (NIM) environment.
Syntax

To Display a List of Supported NIM Classes, Subclasses, or Types

\texttt{lsnim} \{-p\} \{-P\} \{-c\ Class\} \{-s\}

To Display Predefined NIM Information

\texttt{lsnim} \{-p\} \{-P\} \{-c\ Class\} \{-S\} \{-s\ Subclass\} \{-t\ Type\} \{-I\} \{-O\} \{-Z\}

OR

\texttt{lsnim} \{-p\} \{-P\} \{-a\ Attribute\} \ldots \{-Z\}

To Display Attributes Required for an Operation

\texttt{lsnim} \{-Type\} \{-q\ Operation\}

To Display Information about All Customized NIM Objects

\texttt{lsnim} \{-c\ Class\} \{-s\ Subclass\} \{-t\ Type\} \{-I\} \{-O\} \{-Z\}

OR

\texttt{lsnim} \{-a\ Attribute\} \ldots \{-Z\}

To Display Information about a Specific NIM Object

\texttt{lsnim} \{-l\} \{-O\} \{-a\ Attribute\} \ldots \{-Z\} ObjectName

OR

\texttt{lsnim} \{-q\ Operation\} ObjectName

To Display Information about Resources Available to a Specific NIM Machine

\texttt{lsnim} \{-L\} \{-s\ Subclass\} \{-t\ Type\} ObjectName

To Display Information about NIM Groups

\texttt{lsnim} \{-g\} \{-m\} \{-a\ Attribute\} \{-c\ Class\} \{-L\} \{-I\} \{-s\ Subclass\} \{-t\ Type\} GroupObjectName

Description

The \texttt{lsnim} command displays information about the NIM environment. This information is divided into two basic categories: predefined and customized.

Predefined information consists of values that are preset by NIM and cannot be modified by the user. Examples of predefined information include:

- The types of objects supported by NIM
- The classes and subclasses into which NIM organizes objects
- The operations that can be performed on NIM objects
- The attributes that can be entered by the user

In general, NIM uses this information to make decisions during operations. Predefined information can be displayed by using the \texttt{-p} or \texttt{-P} flag. The \texttt{-p} flag displays default values while the \texttt{-P} flag displays help information.

Customized information consists of values that you enter or modify. This information represents the physical environment in which NIM operates. Related pieces of customized information are grouped together to form objects, which are organized in the NIM database by object type and class. Some examples of object types include diskless, paging, and standalone. Two examples of object classes are machines and network.
For example, a standalone workstation that is part of the NIM environment is represented by a unique object. This object is classified by NIM as a standalonemachines object, where standalone represents the object type and machines represents the object class. Entering the lsnim command on the command line without any flags displays information on all customized objects.

You can also use the lsnim command to display relationships between customized objects. Choose an object to anchor on (specified by the Objectname parameter) and then select the desired relationship with the -c, -s, or -t flag. The information displayed then depends upon the type and class of the anchored object. For example, if you select an object of type spot, the type of relationships that can be displayed are:

- Machines that use the Shared Product Object Tree (SPOT) resource.
- Networks that can access the SPOT resource.

When not displaying relationships, the lsnim command provides flags that can be used to filter the output that it would normally display. The -a, -c, -O, -s, or -t flag can be used to restrict the amount of information which is displayed.

**Flags**

- **-a Attribute** Filters displayed information based on the specified attribute name. The possible attributes are:
  - Operation
  - subclass
  - type
  - class
- **-c Class** Specifies a NIM object class. When this flag is used without the Objectname parameter, it filters the displayed information so only information about objects in that class is displayed.
- **-g** Displays long listing of group object with state information for individual members.
- **-I** Displays detailed information.
- **-L** Displays information about resources that can be accessed by a client machine.
- **-m** Applies other flags specified to group members.
- **-O** Lists the operations NIM supports.
- **-p** Displays predefined information using default values.
- **-P** Displays help information for predefined data.
- **-q Operation** Lists the attributes required for the specified operation.
- **-S** Displays a list of NIM subclasses.
- **-s Subclass** Specifies a NIM subclass. When this flag is used without the Objectname parameter, it filters the displayed information so only information about objects in that subclass is displayed.
- **-t Type** Specifies a NIM object type. When this flag is used without the Objectname parameter, it filters the displayed information so only information about objects of that type is displayed.
- **-Z** Displays information in colon-separated format.

**Security**

Access Control: You must have root authority to run the lsnim command.

**Examples**

1. To display a list of NIM object classes, enter:

   lsnim -p

2. To display a list of NIM subclasses, enter:

   lsnim -p -S
3. To display the list of NIM object types for the `machines` object class, enter:
   
   \texttt{lsnim -p -c machines} 

4. To display help information about NIM object types for the `machines` object class, enter:

   \texttt{lsnim -P -c machines} 

5. To display detailed information about the NIM attributes named `lpp_source` and `Rstate`, enter:

   \texttt{lsnim -p -a lpp_source -a Rstate} 

6. To display the operations which can be performed on the `paging` object type, enter:

   \texttt{lsnim -p -t paging -O} 

7. To display the information required to perform a `bos_inst` operation on an object of the `standalone` object type, enter:

   \texttt{lsnim -t standalone -q bos_inst} 

8. To display information about all customized objects of the `diskless` object type, enter:

   \texttt{lsnim -t diskless} 

9. To display all customized objects in the `networks` object class, enter:

   \texttt{lsnim -c networks} 

10. To display detailed information about a NIM object named `altoid`, enter:

    \texttt{lsnim -l altoid} 

11. To display the relationship between an object named `altoid` and all NIM resources, enter:

    \texttt{lsnim -c resources altoid} 

12. To display a list of operations that can be applied to `altoid`, enter:

    \texttt{lsnim -O altoid} 

13. To display a list of resources available to `altoid`, enter:

    \texttt{lsnim -L altoid} 

14. To display the members of the `machines` object class, enter:

    \texttt{lsnim -g MacGrp1} 

15. To display basic information about the members of the `resources` object class, enter:

    \texttt{lsnim -m ResGrp1} 

16. To display a long listing of members of the `machines` object class, enter:

    \texttt{lsnim -m -Fl MacGrp1} 

17. To display all members of the `machines` object class which has a spot allocated, enter:

    \texttt{lsnim -ma spot MacGrp1} 

Files

\texttt{/etc/niminfo} Contains variables used by NIM.

Related Information

The \texttt{nim} command, \texttt{nimclient} command, \texttt{nimconfig} command, \texttt{niminit} command.

The \texttt{.info} file.

\underline{Lsnlspath Command}

**Purpose**

Shows the value of the secure NLSPATH system configuration variable.
Syntax
lsnlspath

Description
The lsnlspath command outputs the current value of the secure NLSPATH variable.

Related Information
The chnlspath command.

Isparent Command

Purpose
Displays the possible parent devices that accept a specified connection type or device.

Syntax
lsparent { -C | -P } { -k ChildConnectionKey | -l ChildName } [ -F Format ] [ -h ] [ -H ]

Description
The lsparent command lists devices from the Device Configuration database that can accept a given device as a child device, specified by the -l ChildName flag, or a given type of child device connection, specified by the -k ChildConnectionKey flag.

You can display the default output one of two ways. You can either display the default output information for a device from the Customized Devices object class, which is name, state, location, and description, using the -C flag, or display the default output information for a device from the Predefined Devices object class, which is class, type, subclass, and description, using the -P flag. To override these two default outputs, you can use the -F Format flag to display the output as designated by a user-formatted string. The Format parameter is a quoted list of column names separated and possibly terminated by nonalphanumeric characters.

You can supply the flags either on the command line or from the specified File parameter.

Flags

- C
  Lists information about a device that is in the Customized Devices object class. The information displayed can be from both the Customized and Predefined Devices object classes. This flag cannot be used with the -P flag.

- f File
  Reads the needed flags from the File variable.

- F Format
  Displays the output in a user-specified format, where the Format variable is a quoted list of column names from the Predefined Devices object class or the Customized Devices object class separated and possibly terminated by nonalphanumeric characters. Using white space as the separator, the lsparent command displays the output in aligned columns. In addition to the column names in the two object classes, the special name description can be used to display a text description of the device.

- H
  Displays headers above the column output.

- h
  Displays the command usage message.

- k ChildConnectionKey
  Specifies the connection key that identifies the device subclass name of the child device. This flag cannot be used with the -l flag.

- l ChildName
  Specifies the logical name of a possible child device. This flag cannot be used with the -k flag.
-P

Lists information about a device that is in the Predefined Devices object class. The information displayed can be from both the Customized and Predefined Devices object classes. This flag cannot be used with the -C flag.

Examples

1. To list possible parent devices in the Customized Devices object class that accept an RS-232 device, type:
lsparent -C -k rs232

The system displays a message similar to the following:

sa0 Available 01-S1 Standard I/O Serial Port
sa1 Available 01-S2 Standard I/O Serial Port
sa2 Available 10-68 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
sa3 Available 10-70 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
sa4 Available 10-78 IBM 8-Port EIA-232/RS-422A (PCI) Adapter
sa5 Available 20-58 IBM 8-Port EIA-232/RS-422A (PCI) Adapter

2. To list possible types of parent devices in the Predefined Devices object class that accept an RS-232 device, type:
lsparent -P -k rs232

The system displays a message similar to the following:

adapter pnp501 isa_sio Standard I/O Serial Port
adapter 4f111100 pci IBM 8-Port EIA-232/RS-422A (PCI) Adapter
concentrator 16c232 sync_pci 16-Port RAN EIA-232 for 128-Port Adapter
concentrator 16e232 sync_pci 16-Port Enhanced RAN EIA-232 for 128-Port Adapter

3. To list possible parent devices in the Customized Devices object class that accept the tape device rmt0 as a child device, type:
lsparent -C -l rmt0

The system displays a message similar to the following:

scsi2 Available 20-60 Wide/Ultra-2 SCSI I/O Controller
scsi3 Available 20-61 Wide/Ultra-2 SCSI I/O Controller
scsi1 Available 10-88 Wide/Ultra-2 SCSI I/O Controller
scsi0 Available 10-60 Wide/Fast-20 SCSI I/O Controller

4. To list possible types of parent devices in the Predefined Devices object class that accept the tape device rmt0 as a child device, type:
lsparent -P -l rmt0

The system displays a message similar to the following:

adapter sym896 pci Wide/Ultra-2 SCSI I/O Controller
adapter sym895 pci Wide/Ultra-2 SCSI I/O Controller
adapter sym875 pci Wide/Fast-20 SCSI I/O Controller

Files

/usr/sbin/lsparent Contains the lsparent command.

Related Information

The chdev command, lsattr command, lsconn command, lsdev command, mkdev command, rmdev command.
Ispath Command

Purpose
Displays information about paths to an MultiPath I/O (MPIO) capable device.

Syntax
```
```
```
```
```
```
```
Ispath -h
```

Description
The Ispath command displays one of two types of information about paths to an MPIO capable device. It either displays the operational status for one or more paths to a single device, or it displays one or more attributes for a single path to a single MPIO capable device. The first syntax shown above displays the operational status for one or more paths to a particular MPIO capable device. In this instance, the Ispath command is similar to the lsdev command. The second syntax (keyed by the presence of the -A flag) displays one or more attributes for a single path to a particular MPIO capable device. In this instance, the Ispath command is similar to the lsattr command. In fact, all of the flags for the lsattr command are supported on the Ispath command when displaying path attributes.

Displaying Path Status with the Ispath Command
When displaying path status, the set of paths to display is obtained by searching the device configuration database for paths that match the following criteria:

- The target device name matches the device specified with the -l flag. If the -l flag is not present, then the target device is not used in the criteria.
- The parent device name matches the device specified with the -p flag. If the -p flag is not present, then parent is not used in the criteria.
- The connection matches the connection specified with the -w flag. If the -w flag is not present, then connection is not used in the criteria.
- The path status matches status specified with the -s flag. If the -s flag is not present, the path status is not used in the criteria.

If none of the -l, -p, -w, -s flags are specified, then all paths known to the system are displayed.

By default, this command will display the information in columnar form. When no flags are specified that qualify the paths to display, the format of the output is:
```
status device parent
```

The default display format can be overridden by using the -F Format flag. The -F Format flag displays the output in a user-specified format where the Format parameter is a quoted list of field names separated by,
and possibly ended by, non-alphanumeric characters or white space. The field names are the fields defined in the CuPath class or one of the column heading defined above.

**Note:** The column names above are not translated into other languages (either when output as column headings or when input as part of the Format of the -F flag).

Possible values that can appear for the status column are:

**enabled** Indicates that the path is configured and operational. It will be considered when paths are selected for IO.

**disabled** Indicates that the path is configured, but not currently operational. It has been manually disabled and will not be considered when paths are selected for IO.

**failed** Indicates that the path is configured, but it has had IO failures that have rendered it unusable. It will not be considered when paths are selected for IO.

**defined** Indicates that the path has not been configured into the device driver.

**missing** Indicates that the path was defined in a previous boot, but it was not detected in the most recent boot of the system.

**detected** Indicates that the path was detected in the most recent boot of the system, but for some reason it was not configured. A path should only have this status during boot and so this status should never appear as a result of the lspath command.

**Displaying Path Attributes with the lspath Command**

When displaying attributes for a path, the path must be fully qualified. Multiple attributes for a path can be displayed, but attributes belonging to multiple paths cannot be displayed in a single invocation of the lspath command. Therefore, in addition to the -A flag, the -l, -p, or -w flags are required to uniquely identify a single path. For example:

- if only one path exists to a device, the -l flag is required
- if only one path between a device and a specific parent, the -l and -p flags are required
- if there are multiple paths between a device and a specific parent, the -l, -p, and -w flags are required

Furthermore, the -s flag is not allowed.

The same rules used by the lsattr command for displaying device attributes applies to the lspath command for displaying path attributes.

By default, this command will display the information in columnar form. The format of the output is the same as the lsattr command:

```
attribute  value  description  user_settable
```

All fields are shown by default. The default display format can be overridden by using the -F Format flag. The -F Format flag displays the output in a user-specified format where the Format parameter is a quoted list of column names separated by, and possibly ended by, non-alphanumeric characters or white space. The column names allowed are the field names from the CuPathAt, PdPathAt, and PdAtXtd object classes plus the columns listed above. The PdAtXtd based information is used by the Web-based System Manager.

**Note:** The column names above are not translated into other languages (either when output as column headings or when input as part of the Format of the -F flag).
Flags

-a Attribute
Identifies the specific attribute to list. The 'Attribute' is the name of a path specific attribute. When this flag is provided, only the identified attribute is displayed. Multiple instances of this flag may be used to list multiple attributes. If this flag is not specified at all, all attributes associated with the identified path will be listed.

-A
Indicates that attributes for a specific path are to be displayed. When the -A flag is present, the -s Status flag is not allowed. However, the -I Name, -p Parent, and -w Connection flags must be present to fully qualify the path.

-D
Displays the attribute names, default values, descriptions, and user-settable flag values for a specific path when not used with the -O flag. The -D flag displays only the attribute name and default value in colon format when used with the -O flag. This flag is only valid when displaying path attributes and it cannot be used with the -E, -F, or -R flag.

-E
Displays the attribute names, current values, descriptions, and user-settable flag values for a specific path when not used with the -O flag. The -E flag displays only the attribute name and current value in colon format when used with the -O flag. This flag is only valid when displaying path attributes and it cannot be used with the -D, -F, or -R flag.

-F File
Reads the needed flags from the File parameter.

-F Format
Displays the output in a user-specified format, where the Format parameter is a quoted list of column names separated by non-alphanumeric characters or white space. Using white space as the separator, the lsPath command displays the output in aligned columns. Valid column names depends upon the type of information requested. For path display, column names from the CuPath object class can be specified. For path attribute display (the -A flag is specified), column names from the PdPathAt and CuPathAt object classes can be specified. In addition to the column names, there are two special purpose names that can be used. The name description can be used to obtain a display of attribute descriptions and user-settable can be used to obtain an indication as to whether or not an attribute can be changed. This flag cannot be used with the -E, -D, -O or -R flag.

-h
Displays the command usage message.

-H
Displays headers above the column output. To use the -H flag with the -O flag is meaningless, the -O flag prevails. To use the -H flag with the -R flag is meaningless; the -R flag prevails.

-I Name
Specifies the logical device name of the target device whose path information is to be displayed. This flag is optional for displaying path status, but is required for displaying path attributes.

-O
Displays all attribute names separated by colons and, on the second line, displays all the corresponding attribute values separated by colons. The attribute values are current values when the -E flag is also specified and default values when the -D flag is specified. This flag is only valid when displaying path attributes and it cannot be used with the -F and -R flags.

-p Parent
Indicates the logical device name of the parent device of the path(s) to be displayed. This flag is optional for displaying path status, but is required for displaying path attributes.

-R
Displays the legal values for an attribute name. The -R flag cannot be used with the -D, -E, -F and -O flags. The -R flag displays the list attribute values in a vertical column as follows:

Value1
Value2
.
.
ValueN

The -R flag displays the range attribute values as x...n(+i) where x is the start of the range, n is the end of the range, and i is the increment.
-s Status

The -s Status flag indicates the status to use in qualifying the paths to be displayed. When displaying path information, the allowable values for this flag are:

**enabled**
Display paths that are enabled for MPIO path selection.

**disabled**
Display paths that are disabled from MPIO path selection.

**failed**
Display paths that are failed due to IO errors.

**available**
Display paths whose path_status is PATH_AVAILABLE (that is, paths that are configured in the system, includes enabled, disabled, and failed paths).

**defined**
Display paths whose path_status is PATH_DEFINED.

**missing**
Display paths whose path_status is PATH_MISSING.

-w Connection
Indicates the connection information to use in qualifying the paths to be displayed. This flag is optional for displaying path status, but is required for displaying path attributes.

-Z Character
The -Z Character flag is meant to be used for programs that need to deal with ODM fields that may have embedded new line characters. An example of such a program is the Web-based System Manager. The -Z Character flag is used to change the record separator character for each record (line) of output generated. The new record separator is the 'Character' argument to this flag. The -Z Character flag is only relevant when the -A and the -F Format flags are specified. The -Z Character flag cannot be used with the -D, -E, -O, or the -R flags.

### Security

**Privilege Control:** All users can execute this command.

**Auditing Events:** None.

### Examples

Examples of displaying path status:

1. To display the status of all paths to hdisk1 with column headers, enter:
   ```
   lspath -H -l hdisk1
   ```

   The system will display a message similar to the following:

   ```
   status    device    parent
   enabled   hdisk1    scsi0
   disabled  hdisk1    scsi1
   missing   hdisk1    scsi2
   ```

2. To display, without column headers, the set of paths whose operational status is disabled, enter:
   ```
   lspath -s disabled
   ```

   The system will display a message similar to the following:

   ```
   disabled hdisk1  scsi1
   disabled hdisk2  scsi1
   disabled hdisk23 scsi8
   disabled hdisk25 scsi8
   ```

3. To display the set of paths whose operational status is failed, enter:
   ```
   lspath -s failed
   ```

   The system will display a message similar to the following:
4. To display in a user-specified format, without column headers, the set of paths to hdisk1 whose path status is available enter:

`lspath -l hdisk1 -s available -F"connection:parent:path_status:status"`

The system will display a message similar to the following:

```
5,0:scsi0:available:enabled
6,0:scsi1:available:disabled
```

Note that this output shows both the path status and the operational status of the device. The path status simply indicates whether the path is configured or not. The operational status indicates how the path is being used with respect to path selection processing in the device driver. Only paths with a path status of available also have an operational status. If a path is not currently configured into the device driver, it does not have an operational status.

Examples of displaying path attributes:

1. If the target device is a SCSI disk, to display all attributes for the path to parent scsi0 at connection 5,0, use the command:

   `lspath -AHE -l hdisk10 -p scsi0 -w "5,0"

   The system will display a message similar to the following:

<table>
<thead>
<tr>
<th>attribute</th>
<th>value</th>
<th>description</th>
<th>user_settable</th>
</tr>
</thead>
<tbody>
<tr>
<td>weight</td>
<td>1</td>
<td>Order of path</td>
<td>true</td>
</tr>
<tr>
<td>failover</td>
<td></td>
<td>selection processing</td>
<td></td>
</tr>
</tbody>
</table>

**Related Information**

The `chpath` command, `mkpath` command, and `rmpath` command.

**Isprtsv Command**

**Purpose**

Shows print service information stored in the database.

**Syntax**

```
Isprtsv [-c | -p] [-h] [-q QEntry ...] [-Z]
```

**Description**

The `Isprtsv` high-level command shows predefined and customized TCP/IP-based print service information. Use the `Isprtsv` command to extract customized or predefined print service information.

The `Isprtsv` command can show the following information:

- A list of host names that have access rights to the print server
- Logical print queue information only

**Flags**

- `-c` Extracts customized configuration information.
- `-h` Shows a list of host names that can use the print server.
- `-p` Extracts predefined configuration information.
-q QEntry...
Shows the logical print queues specified and their attributes available on a host. The QEntry variable specifies the names of the queues to display.

-Z
Specifies that the output be produced in colon format. This flag is used if the lsprtsv command is invoked from the SMIT usability interface.

Examples
1. To show all host names who have access rights to a print server, enter:

   $ lsprtsv -c -h

2. To show which logical printers are available on a given client machine, enter:

   lsprtsv -c -q sahara

Related Information
The chque command, chquedev command, ruser command.

The lpd daemon, qdaemon daemon.

Understanding the SMIT Interface for TCP/IP in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

Isps Command

Purpose
Displays the characteristics of a paging space.

Syntax
lsps {[s] | [c] | [l] | [a] | [t] | lv | nfs} | PagingSpace }

Description
The lsps command displays the characteristics of a paging space. The lsps command displays characteristics such as the paging-space name, physical-volume name, volume-group name, size, percentage of the paging space used, whether the space is active or inactive, and whether the paging space is set to automatic. The PagingSpace parameter specifies the paging space whose characteristics are to be shown.

For NFS paging spaces, the physical-volume name and volume-group name will be replaced by the host name of the NFS server and the path name of the file that is used for paging.

You can use the File Systems application in Web-based System Manager (wsm) to change file system characteristics. You could also use the System Management Interface Tool (SMIT) smit lsps fast path to run this command.

Flags
-a Specifies that the characteristics of all paging spaces are to be given. The size is given in megabytes.
-c Specifies that the output should be in colon format. The colon format gives the paging space size in logical partitions.
-l Specifies that the output should be in list format.
-s    Specifies that the summary characteristics of all paging spaces are to be given. This information consists of the total paging space in megabytes and the percentage of paging space currently assigned (used). If the -s flag is specified, all other flags are ignored.
Note: Setting the environment variable PSALLOC=early causes the use of early paging space algorithm. In this case, the value the -s flag specifies is different from the value returned for a single paging space or when using the -a flag for all the paging spaces. The value the -s flag displays is the percentage of paging space allocated (reserved), whether the paging space has been assigned (used) or not. Therefore, the percentage reported by the -s flag is usually larger than that reported by the -a flag when PSALLOC is set to early.
-t    Specifies the characteristics of the paging space. One of the following variables is required:

   iv    Specifies that the characteristics of only logical volume paging spaces are to be given.
   nfs   Specifies that the characteristics of only NFS paging spaces are to be given. The heading of the output will be changed to display the host name of the NFS server and the path name of the file that resides on the server that is being used for NFS paging.

Examples
To list the characteristics of all paging spaces, enter:

lsps  -a

This displays the characteristics for all paging spaces and provides a listing similar to the following listing:

<table>
<thead>
<tr>
<th>Page Space</th>
<th>Physical Volume</th>
<th>Volume Group</th>
<th>Size</th>
<th>%Used</th>
<th>Active</th>
<th>Auto</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>hd6</td>
<td>hdisk0</td>
<td>rootvg</td>
<td>512MB</td>
<td>1</td>
<td>yes</td>
<td>yes</td>
<td>lv</td>
</tr>
</tbody>
</table>

Files
/etc/swapspaces    Specifies the paging space devices activated by the swapon -a command.

Related Information
The mkps command, chps command, rmgs command, swapon command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

Ispv Command

Purpose
Displays information about a physical volume within a volume group.
Syntax

lspv

OR


Description

The `lspv` command displays information about the physical volume if the specific physical volume name is specified. If you do not add flags to the `lspv` command, the default is to print every known physical volume in the system along with its physical disk name, physical volume identifiers (PVIDs), to which volume group (if any) the physical volume belongs, and the state of the volume group as active if the volume group is varied on or concurrent if it is varied on in concurrent mode.

Note: If the `lspv` command cannot find information for a field in the Device Configuration Database, it will insert a question mark (?) in the value field. As an example, if there is no information for the PP RANGE field, the following might be displayed:

PP RANGE: ?

The `lspv` command attempts to obtain as much information as possible from the description area when it is given a logical volume identifier.

When the `PhysicalVolume` parameter is used, the following characteristics of the specified physical volume are displayed:

- Physical volume: Name of the physical volume.
- Volume group: Name of volume group. Volume group names must be unique systemwide names and can be from 1 to 15 characters long.
- PV Identifier: The physical volume identifier for this physical disk.
- VG Identifier: The volume group identifier of which this physical disk is a member.
- PVstate: State of the physical volume. If the volume group that contains the physical volume is varied on with the `varyonvg` command, the state is active, missing, or removed. If the physical volume is varied off with the `varyoffvg` command, the state is varied off.
- Allocatable: Allocation permission for this physical volume.
- Logical volumes: Number of logical volumes using the physical volume.
- Stale PPs: Number of physical partitions on the physical volume that are not current.
- VG descriptors: Number of volume group descriptors on the physical volume.
- PP size: Size of physical partitions on the volume.
- Total PPs: Total number of physical partitions on the physical volume.
- Free PPs: Number of free physical partitions on the physical volume.
- Used PPs: Number of used physical partitions on the physical volume.
- Free distribution: Number of free partitions available in each intra-physical volume section.
- Used distribution: Number of used partitions in each intra-physical volume section.

You can use the Volumes application in Web-based System Manager (wsm) to change volume characteristics. You could also use the System Management Interface Tool (SMIT) `smit lspv` fast path to run this command.

Flags

- **-L**: Specifies no waiting to obtain a lock on the Volume group.

  **Note**: If the volume group is being changed, using the `-L` flag gives unreliable date.
Lists the following fields for each logical volume on the physical volume:

**LVname**  Name of the logical volume to which the physical partitions are allocated.

**LPs**  The number of logical partitions within the logical volume that are contained on this physical volume.

**PPs**  The number of physical partitions within the logical volume that are contained on this physical volume.

**Distribution**  The number of physical partitions, belonging to the logical volume, that are allocated within each of the following sections of the physical volume: outer edge, outer middle, center, inner middle and inner edge of the physical volume.

**Mount Point**  File system mount point for the logical volume, if applicable.

Lists the following fields for each logical volume on the physical volume:

**PVname:**PVnum [**LVname:** LPnum [:Copynum] [**PPstate**]]

**Where:**

**PVname**  Name of the physical volume as specified by the system.

**PPnum**  Physical partition number.

**LVname**  Name of the logical volume to which the physical partitions are allocated. Logical volume names must be system-wide unique names, and can range from 1 to 64 characters.

**LPnum**  Logical partition number. Logical partition numbers can range from 1 to 64,000.

**Copynum**  Mirror number.

**PPstate**  Only the physical partitions on the physical volume that are not current are shown as stale.

Accesses information from the variable descriptor area specified by the

**DescriptorPhysicalVolume** variable. The information may not be current, since the information accessed with the -n flag has not been validated for the logical volumes. If you do not use the -n flag, the descriptor area from the physical volume that holds the validated information is accessed, and therefore the information displayed is current. The volume group need not be active when you use this flag.

Lists the following fields for each physical partition on the physical volume:

**Range**  A range of consecutive physical partitions contained on a single region of the physical volume.

**State**  The current state of the physical partitions: free, used, stale, or vgda.

**Note:** If a volume group is converted to a big vg format, it may be necessary to use some data partitions for volume group descriptor area. These partitions will be marked vgda.

**Region**  The intra-physical volume region in which the partitions are located.

**LVname**  The name of the logical volume to which the physical partitions are allocated.

**Type**  The type of the logical volume to which the partitions are allocated.

**Mount point**  File system mount point for the logical volume, if applicable.

Accesses information based on the **VolumeGroupID** variable. This flag is needed only when the lspv command does not function due to incorrect information in the Device Configuration Database. The **VolumeGroupID** variable is the hexadecimal representation of the volume group identifier, which is generated by the **mkvg** command.
Examples

1. To display the status and characteristics of physical volume hdisk3, enter:
   
   `lspv hdisk3`

2. To display the status and characteristics of physical volume hdisk5 by physical partition number, enter:
   
   `lspv -p hdisk5`

3. To display the status and characteristics of physical volume hdisk5 using the volume group ID, enter:
   
   `lspv -v 00014A782B12655F hdisk5`

The following is an example of the output:

```
  lspv
  hdisk0  0000000012345678  rootvg  active
  hdisk1  10000BC876543258  vg00   concurrent
  hdisk2  ABCD000054C23486   None
```

The previous example shows that physical volume hdisk0 contains the volume group rootvg, and it is varied on. Physical volume hdisk1 contains the volume group vg00, and it is varied on in concurrent mode. Physical volume hdisk2 does not contain a varied on volume group.

Files

`/usr/sbin` Contains the `lspv` command.

Related Information

The `chpv` command, `lslv` command, `lsvg` command, `mklv` command, `varyonvg` command.

Logical Volume Storage Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices explains the Logical Volume Manager, physical volumes, logical volumes, volume groups, organization, ensuring data integrity, and allocation characteristics.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

System Management Interface Tool (SMIT) Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices explains the structure, main menus, and tasks that are done with SMIT.

lsque Command

Purpose

Displays the queue stanza name.

Syntax

`lsque [-c] [-q] Name`

Description

The `lsque` command uses the `printf` subroutine to display the name of the queue stanza and associated attributes from the `/etc/qconfig` file.
Flags
- `c` Causes colon output format for use by SMIT.
- `-q Name` Specifies the Name of the queue stanza that is sent to standard output.

Examples
1. To display the name of queue stanza `lp0`, enter:

   `lsque -q lp0`

   A list similar to the following is displayed:
   
   ```
   lp0:
   device = lpd0
   host = neptune
   rq = nlp0
   ```

2. To display the name of queue stanza `lp0` in colon format, enter:

   `lsque -c -q lp0`

   A list similar to the following is displayed:
   
   ```
   device:discipline:up:acctfile:host:s_statfilter:l_statfilter:rq
   lpd0:fcfs:true:false:neptune:::nlp0
   ```

Files
/`usr/bin/lsque` Contains the `lsque` command.
/`etc/qconfig` Contains the configuration file.

Related Information
The `chque` command, `lsquedev` command, `mkque` command, `rmque` command.

The `qconfig` file.

The `printf` subroutine.

Printer Specific Information, Printer Support, and Virtual Printer Definitions and Attribute Values in AIX 5L Version 5.2 Guide to Printers and Printing.


**lsquedev Command**

**Purpose**
Displays the device stanza name.

**Syntax**

```
lsquedev [-c] [-q Name -d Name]
```
Description
The `lsquedev` command displays the name of the queue stanza and associated attributes from the `/etc/qconfig` file.

Flags
- `-c` Specifies colon output format for use by SMIT.
- `-d Name` Specifies the Name variable of the device stanza that is displayed.
- `-q Name` Specifies the Name variable of the queue containing the device stanza that is displayed.

Examples
1. To display device stanza `dlp0` on the `lp0` queue, type:

   ```
   lsquedev -q lp0 -d dlp0
   ```

   A listing similar to the following is displayed:
   ```
   dlp0:
   FILE = /dev/lp0
   BACKEND = /usr/lib/lpd/piobe
   ```

2. To display device stanza `dlp0` on the `lp0` queue in colon format, type:

   ```
   lsquedev -c -q lp0 -d dlp0
   ```

   A listing similar to the following is displayed:
   ```
   file:access:feed:header:trailer:backend:align
   dlp0:/dev/lp0:read:never:never:never:/usr/lib/lpd/piobe:TRUE
   ```

Files
- `/usr/bin/lsquedev` Contains the `lsquedev` command.
- `/etc/qconfig` Contains the configuration file.

Related Information
- The `chquedev` command, `lsque` command, `mkquedev` command, `rmquedev` command.
- The `qconfig` file.
- The `printf` subroutine.
- Printer Overview for System Management in the AIX 5L Version 5.2 Guide to Printers and Printing.
- Printer Specific Information, Printer Support, and Virtual Printer Definitions and Attribute Values in the AIX 5L Version 5.2 Guide to Printers and Printing.
- Spooler Overview for System Management in the AIX 5L Version 5.2 Guide to Printers and Printing.

Isresource Command

Purpose
Displays bus resources for available devices in the system and recommends attribute values for bus resource resolution.
Syntax
isresource [ -a | -r ] [ -d ] [ -l ] Name

Description
The **isresource** command writes a list of assigned bus resources to standard out, or determines if the bus resources for devices resolve.

The **isresource** command lets you display the currently assigned values for the bus resource attributes for the device specified by the given device logical name (-l Name). Use the -a flag to display the currently assigned values for all bus resource attributes for all devices under the same parent bus as the specified device.

Use the -r flag to determine if the bus resources for the specified device are resolvable. In this case, the **isresource** command checks all devices under the same parent bus as the specified device, including defined but not available devices, to see if their bus resource attributes are resolvable. The **isresource** command produces no output if all attributes resolve. The **isresource** command provides information depending on the type of conflict detected if any device's bus resources are unresolvable. In some cases, the **isresource** command can provide you with information that leads to the resolution of the conflict.

The **isresource** command identifies the device name, attribute name, and a suggested value for the attribute if a conflict results from an attribute that only a user can change. Setting the attribute to the suggested value should resolve the conflict. This may aid in the configuration of devices with attributes that can only a user can change. Such devices include adapter cards which use jumpers or switches on the card to select values.

In some cases, a conflict may be due to an attribute which the system can normally adjust at boot time but is prevented from doing so at run time because the device is in the Available state. In these situations, the **isresource** command will indicate that the configuration will be resolved by rebooting the system.

It is possible that multiple user changeable attributes will be identified when unresolvable conflicts occur. These may be for the device specified by the given device logical name (-l Name) or for other devices in the system. All of the identified attributes will need to be changed to resolve the conflict. It may even be the case where user changeable attributes are identified and a reboot is indicated. In this case, all of the identified attributes will need to be changed and the system rebooted to resolve the conflicts.

Finally, **isresource** may determine that the set of devices currently defined in the devices configuration database can not be resolved regardless of attributes being changed or the system rebooted. In this case, a list of the devices which could not be resolved is written to standard out. If the problem has resulted from a new device just being defined, that device should be removed, or the devices listed by **isresource** should be removed. If the problem is not resolved by removing devices, there could be additional problems on the next reboot. This is because the order in which devices are resolved at boot time may differ from the order they are resolved by **isresource**, resulting in a different set of unresolvable devices at boot time. If the set of unresolvable devices at boot time should now include a device needed for booting, problems such as no console being configured or the system failing to boot could occur.

The following applies when **isresource** is used to list currently assigned bus resource values (the -r flag is not specified).

The **TYPE** field in the output listing contains the following symbols:

- **B** Bus Memory Address Values
- **M** Bus Memory Address Values
- **O** I/O Address Values
- **I** Bus Interrupt Levels
- **N** Non-sharable Bus Interrupt Levels
The S column denotes shared attributes. These are attributes which are required to be set to the same value. They are grouped by the number specified in the column. All attributes with a 1 by them must be set to the same value, all attributes with a 2 by them must be set to the same value, and so on. In some cases, two or more interrupt attributes may be set to the same value but have no numbers in the S column indicating that they are shared. This is because the values are not required to be the same but just happen to be set to the same value because they could not be assigned their own unique values.

The G column denotes attributes in a group. These are a set of attributes whose values depend on each other. If one is changed to the next possible value, the rest of the attributes in the group must also be changed to the next possible value. Their groupings are indicated by the number specified in the column. All attributes with a 1 by them are in the same group, all attributes with a 2 by them are same group, and so on.

On some models, the interrupt value displayed may be followed by a value enclosed in parenthesis. This is not part of the interrupt value but serves to identify the interrupt controller to which the interrupt is associated. The identifier consists of a letter followed by a number, such as A0. The letter indicates the type of interrupt controller and the number distinguishes between multiple instances of that type of controller. There are two types of interrupt controllers that may be identified:

A Indicates an AT interrupt controller.
B Indicates a non-AT interrupt controller.

Flags

-a Specifies that all allocated bus resource attributes for all devices connected to the same top parent bus as the device specified with the -l flag are to be displayed. This flag cannot be used with the `-r` flag.

-d Specifies that the attribute text descriptions are to be included in the output.

-l Name (Lowercase L) Specifies the logical name of the device attributes to display.

-r Specifies to attempt to resolve all bus resources of all devices connected to the same top parent bus as the device specified with the -l flag. This will include all devices that are in the DEFINED state. The `lsresource` command will display any conflicts and advise the user on changeable values. No changes to the ODM database are made. This flag cannot be used with the `-a` flag.

Security

Access Control: Any User

Auditing Events: N/A

Examples

1. To list bus attributes for the token ring device, enter:
   `lsresource -l tok0`

   The system displays a message similar to the following:

<table>
<thead>
<tr>
<th>TYPE</th>
<th>DEVICE</th>
<th>ATTRIBUTE</th>
<th>S</th>
<th>G CURRENT VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>M</td>
<td>dma_bus_mem</td>
<td></td>
<td>0x003b2000 - 0x003f1fff</td>
</tr>
<tr>
<td></td>
<td>O</td>
<td>bus_io_addr</td>
<td></td>
<td>0x0000086a0 - 0x0000086af</td>
</tr>
<tr>
<td></td>
<td>N</td>
<td>bus_intr_lvl</td>
<td></td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>A</td>
<td>dma_lvl</td>
<td></td>
<td>7</td>
</tr>
</tbody>
</table>

2. To list bus attributes for all devices, enter:
   `lsresource -a -l tok0`

358 Commands Reference, Volume 3
The system displays a message similar to the following:

```
<table>
<thead>
<tr>
<th>TYPE</th>
<th>DEVICE</th>
<th>ATTRIBUTE</th>
<th>S</th>
<th>G</th>
<th>CURRENT VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>bus0</td>
<td>bus_iocc_mem</td>
<td></td>
<td></td>
<td>0x00ffffff0 - 0x00ffffff</td>
</tr>
<tr>
<td>M</td>
<td>gda0</td>
<td>vram_start</td>
<td>1</td>
<td></td>
<td>0x00400000 - 0x007fffffff</td>
</tr>
<tr>
<td>M</td>
<td>gda0</td>
<td>bus_mem_start</td>
<td></td>
<td></td>
<td>0x000c0000 - 0x000c1fff</td>
</tr>
<tr>
<td>M</td>
<td>gda0</td>
<td>dma1_start</td>
<td></td>
<td></td>
<td>0x00800000 - 0x009fffffff</td>
</tr>
<tr>
<td>M</td>
<td>gda0</td>
<td>dma2_start</td>
<td></td>
<td></td>
<td>0x00a00000 - 0x00bfffffff</td>
</tr>
<tr>
<td>M</td>
<td>gda0</td>
<td>dma3_start</td>
<td></td>
<td></td>
<td>0x00c00000 - 0x00dfffffff</td>
</tr>
<tr>
<td>M</td>
<td>gda0</td>
<td>dma4_start</td>
<td></td>
<td></td>
<td>0x01000000 - 0x011fffffff</td>
</tr>
<tr>
<td>M</td>
<td>scsi0</td>
<td>bus_mem_addr</td>
<td></td>
<td></td>
<td>0x000e0000 - 0x000e0fff</td>
</tr>
<tr>
<td>M</td>
<td>scsi0</td>
<td>dma_bus_mem</td>
<td></td>
<td></td>
<td>0x00100000 - 0x00301fff</td>
</tr>
<tr>
<td>M</td>
<td>tok0</td>
<td>dma_bus_mem</td>
<td></td>
<td></td>
<td>0x003b2000 - 0x003f1fff</td>
</tr>
<tr>
<td>O</td>
<td>da0</td>
<td>bus_io_addr</td>
<td></td>
<td></td>
<td>0x00000060 - 0x0000006f</td>
</tr>
<tr>
<td>O</td>
<td>siokta0</td>
<td>bus_io_addr</td>
<td></td>
<td></td>
<td>0x00000050 - 0x00000051</td>
</tr>
<tr>
<td>O</td>
<td>sioma0</td>
<td>bus_io_addr</td>
<td></td>
<td></td>
<td>0x00000048 - 0x00000049</td>
</tr>
<tr>
<td>O</td>
<td>ppa0</td>
<td>bus_io_addr</td>
<td></td>
<td></td>
<td>0x00000078 - 0x0000007a</td>
</tr>
<tr>
<td>O</td>
<td>gda0</td>
<td>bus_addr_start</td>
<td>1</td>
<td></td>
<td>0x00002110 - 0x0000211f</td>
</tr>
<tr>
<td>O</td>
<td>tok0</td>
<td>bus_io_addr</td>
<td></td>
<td></td>
<td>0x000086a0 - 0x000086af</td>
</tr>
</tbody>
</table>
```

3. To report the outcome of a resolution of device attributes, enter:

```
lsresource -r -d -l tok0
```

Depending on the outcome of the resolution, different messages may be displayed. The output below signifies to a user that the resolution can be successful if changes are made, i.e., the attributes are changed to the suggested values.

lsresource: The attribute(s) for some device(s) in the system could not be resolved. To resolve conflicts, attribute(s) need to be modified. A suggested value for each attribute is provided.

```
<table>
<thead>
<tr>
<th>DEVICE</th>
<th>ATTRIBUTE</th>
<th>CURRENT</th>
<th>SUGGESTED</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>ent1</td>
<td>bus_intr_lvl</td>
<td>11</td>
<td>5</td>
<td>Bus interrupt level</td>
</tr>
<tr>
<td>ent1</td>
<td>bus_mem_addr</td>
<td>0xc0000</td>
<td>0xc4000</td>
<td>Bus memory address</td>
</tr>
<tr>
<td>ent2</td>
<td>bus_io_addr</td>
<td>0x300</td>
<td>0x320</td>
<td>Bus I/O address</td>
</tr>
<tr>
<td>ent2</td>
<td>bus_intr_lvl</td>
<td>11</td>
<td>7</td>
<td>Bus interrupt level</td>
</tr>
<tr>
<td>ent2</td>
<td>bus_mem_addr</td>
<td>0xc0000</td>
<td>0xc8000</td>
<td>Bus memory address</td>
</tr>
</tbody>
</table>
```

Files

/usr/sbin/lsresource Contains the lsresource command.

Isresource Command

Purpose

Lists information about one or more responses.
Syntax

[response1[,response2,...] :node_name]

Description

The lsresponse command lists the following information about defined responses:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ResponseName</td>
<td>The name of the response.</td>
</tr>
<tr>
<td>Node</td>
<td>The location of the response.</td>
</tr>
<tr>
<td>Action</td>
<td>The name of an action.</td>
</tr>
<tr>
<td>DaysOfWeek</td>
<td>The days of the week when the action can be run. The values for the days can be separated by plus signs (+) or displayed as a range of days separated by a hyphen (-). Multiple DaysOfWeek values are separated by commas (,). The number of DaysOfWeek values must match the number of TimeOfDay values. The values for each day follow: 1 Sunday 2 Monday 3 Tuesday 4 Wednesday 5 Thursday 6 Friday 7 Saturday</td>
</tr>
<tr>
<td>TimeOfDay</td>
<td>The time range when Action can be run, consisting of the start time followed by the end time separated by a hyphen. The time is in 24-hour format (HHMM), where the first two digits represent the hour and the last two digits represent the minutes. Multiple TimeOfDay values are separated by commas (,). The number of DaysOfWeek values must match the number of TimeOfDay values.</td>
</tr>
<tr>
<td>ActionScript</td>
<td>The script or command to run for the action.</td>
</tr>
<tr>
<td>ReturnCode</td>
<td>The expected return code for ActionScript.</td>
</tr>
<tr>
<td>CheckReturnCode</td>
<td>Indicates whether the actual return code for ActionScript is compared to its expected return code. The values are: y (yes) and n (no).</td>
</tr>
<tr>
<td>EventType</td>
<td>The type of event that causes the action to be run: event, rearm event, or both.</td>
</tr>
<tr>
<td>StandardOut</td>
<td>Indicates whether standard output is directed to the audit log. The values are: y (yes) and n (no).</td>
</tr>
<tr>
<td>EnvironmentVars</td>
<td>Indicates any environment variables that will be set before the action is run.</td>
</tr>
<tr>
<td>UndefRes</td>
<td>Indicates whether the action is to be run if a monitored resource becomes undefined. The values are: y (yes) and n (no).</td>
</tr>
<tr>
<td>Locked</td>
<td>Indicates whether the resource is locked or unlocked.</td>
</tr>
</tbody>
</table>
To get a list of all response names, run the **Isresponse** command alone without any response names specified. A list of all response names is returned. The default format in this case is tabular.

Specifying a node name after the response names limits the display to the responses defined on that node. List all of the responses on a node by specifying a colon (:) followed by the node name. The node name is a node within the management scope determined by the CT_MANAGEMENT_SCOPE environment variable. The management scope determines the list of nodes from which the responses are listed. For local scope, only responses on the local node are listed. Otherwise, the responses from all nodes within the domain are listed.

To see all the information about all response names, specify the -A flag with the Isresponse command. The -A flag causes all information about a response to be listed when no response names are specified. When all of the information about all responses is listed, the long format is the default.

When more than one response is specified, the response information is listed in the order in which the responses are entered.

**Flags**

- **–a** Specifies that this command applies to all nodes in the cluster. The cluster scope is determined by the CT_MANAGEMENT_SCOPE environment variable. If it is not set, first the management domain scope is chosen if it exists, then the peer domain scope is chosen if it exists, and then local scope is chosen, until the scope is valid for the command. The command will run once for the first valid scope found. For example, if both a management and peer domain exist, **lsresponse -a** with CT_MANAGEMENT_SCOPE not set will list the management domain. In this case, to list the peer domain, set CT_MANAGEMENT_SCOPE to 2.

- **–C** Displays the mkresponse command that can be used to create the response and one of its actions. If more than one response is specified, each mkresponse command appears on a separate line. This flag is ignored when no responses are specified. This flag overrides the –I flag.

- **–I** Displays the response information on separate lines (long form).

- **–t** Displays the response information in separate columns (table form).

- **–d** Specifies delimiter-formatted output. The default delimiter is a colon (:). Use the –D flag if you wish to change the default delimiter.

- **–D delimiter** Specifies delimiter-formatted output that uses the specified delimiter. Use this flag to specify something other than the default, colon (:). For example, when the data to be displayed contains colons, use this flag to specify another delimiter of one or more characters.

- **–A** Displays all of the attributes of the response.

- **–q** Does not return an error when response does not exist.

- **–U** Indicates whether the resource is locked.

- **–x** Suppresses headers when printing.

- **–h** Writes the command’s usage statement to standard output.

- **–T** Writes the command’s trace messages to standard error. For your software service organization’s use only.

- **–V** Writes the command’s verbose messages to standard output.
Parameters

$response1[,\ldots]$  
This parameter can be a response name or a substring of a response name. You can specify more than one response name. When it is a substring, any defined response name that contains the substring is listed.

$node\_name$  
Specifies the node where the response is defined. If $node\_name$ is not specified, the local node is used. $node\_name$ is a node within the scope determined by the CT_MANAGEMENT_SCOPE environment variable.

Security

The user needs read permission for the IBM.EventResponse resource class to run lsresponse. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status

0  The command ran successfully.
1  An error occurred with RMC.
2  An error occurred with a command-line interface script.
3  An incorrect flag was entered on the command line.
4  An incorrect parameter was entered on the command line.
5  An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT  
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

CT_MANAGEMENT_SCOPE  
Determines the management scope that is used for the session with the RMC daemon in processing the resources of the event-response resource manager (ERRM). The management scope determines the set of possible target nodes where the resources can be processed. The valid values are:

0  Specifies local scope.
1  Specifies local scope.
2  Specifies peer domain scope.
3  Specifies management domain scope.

If this environment variable is not set, local scope is used.

Standard Output

When the -h flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error

All trace messages are written to standard error.
Examples

1. To list all of the responses, run this command:

   `lsresponse`

   The output will look like this:

   ResponseName
   "E-mail root anytime"
   "E-mail root first shift"
   "Critical notifications"
   "Generate SNMP trap"

2. To see which resources are locked, run this command:

   `lsresponse -U`

   The output will look like this:

   ResponseName          Node   Locked
   "Broadcast event on-shift" "nodeA"  No
   "E-mail root off-shift" "nodeA"  No
   "E-mail root anytime"   "nodeA"  No
   "Log event anytime"    "nodeA"  No
   "Informational notifications" "nodeA"  No
   "Warning notifications" "nodeA"  No
   "Critical notifications" "nodeA"  No
   "Generate SNMP trap"    "nodeA"  No

3. To list general information about the response "Critical notifications", run this command:

   `lsresponse "Critical notifications"

   The output will look like this:

   ResponseName = "Critical notifications"
   Node = "nodeA"
   Action = "Log Critical Event"
   DaysOfWeek = 1+2+7
   TimeOfDay = 0000-2400
   ActionScript = "/usr/sbin/rsct/bin/logevent /tmp/criticalEvents"
   ReturnCode = 0
   CheckReturnCode = "y"
   EventType = "b"
   StandardOut = "y"
   EnvironmentVars = "Env1=5","Env=10"
   UndefRes = "n"

   ResponseName = "Critical notifications"
   Node = "nodeA"
   Action = "E-mail root"
   DaysOfWeek = 6+2,6+2,6+5
   TimeOfDay = 1700-2400,0000-0800,0000-2400
   ActionScript = "/usr/sbin/rsct/bin/notifyevent root"
   ReturnCode = 0
   CheckReturnCode = "y"
   EventType = "b"
   StandardOut = "y"
   EnvironmentVars = ""
   UndefRes = "n"

4. To list the command that would create the response "Critical notifications" along with one of its actions, run this command:

   `lsresponse -C "Critical notifications"

   The output will look like this:

   mkresponse -n "Log Critical Event" -d 1+2+7 -t 0000-2400
   -s "/usr/sbin/rsct/bin/logevent /tmp/criticalEvents"
   -e b -r 0 "Critical notifications"
5. To list all responses that have the string **E-mail** in their names, run this command:

   `lsresponse "E-mail"`

   The output will look like this:

   ResponseName = "E-mail root anytime"
   Action = "E-mail root"
   
   ResponseName = "E-mail root first shift"
   Action = "E-mail root"

**Location**

/`usr/sbin/rsct/bin/lsresponse` Contains the `lsresponse` command

**Related Information**

Commands: `chresponse`, `lscondresp`, `mkcondresp`, `mkresponse`, `rmresponse`

Files: `rmccli`

---

**Isrole Command**

**Purpose**

Displays role attributes. This command applies only to AIX 4.2.1 and later.

**Syntax**

`lsrole [-c|-f] [ -a List ] { ALL | Name [ ,Name ] ... }`

**Description**

The `lsrole` command displays the role attributes. You can use this command to list all attributes of all the roles or all the attributes of specific roles. Since there is no default parameter, you must enter the `ALL` keyword to see the attributes of all the roles. By default, the `lsrole` command displays all role attributes. To view selected attributes, use the `-a List` flag. If one or more attributes cannot be read, the `lsrole` command lists as much information as possible.

By default, the `lsrole` command lists each role’s attributes on one line. It displays attribute information as `Attribute=Value` definitions, each separated by a blank space. To list the role attributes in stanza format, use the `-f` flag. To list the information as colon-separated records, use the `-c` flag.

You can use the Users application in Web-based System Manager (wsm) to change user characteristics. You could also use the System Management Interface Tool (SMIT) to run this command.

**Flags**

- `-a List` Lists the attributes to display. The `List` variable can include any attribute defined in the `chrole` command and requires a blank space between attributes. If you specify an empty list, only the role names are displayed.

- `-c` Displays the role attributes in colon-separated records, as follows:

  # role: attribute1: attribute2: ...
  Role: value1: value2: ...

---

364 Commands Reference, Volume 3
-f Displays the output in stanzas, with each stanza identified by a role name. Each $Attribute=Value$ pair is listed on a separate line:

```plaintext
Role:
  attribute1=value
  attribute2=value
  attribute3=value
```

**Security**

Files Accessed:

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>/etc/security/roles</td>
</tr>
</tbody>
</table>

**Examples**

To display the role rolelist and groups of the role **ManageAllUsers** in colon format, enter:

```
lsrole -c -a rolelist groups ManageAllUsers
```

Information similar to the following appears:

```
# role: rolelist:groups
ManageAllUsers: ManagerBasicUser:security
```

**Files**

`/etc/security/roles`

Contains the attributes of roles.

**Related Information**

The `chrole` command, `chuser` command, `lsuser` command, `mkrole` command, `mkuser` command, `rmrole` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the [AIX 5L Version 5.2 Security Guide](#).

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

**lsrpdomain Command**

**Purpose**

Displays peer domain information for the node.

**Syntax**

```
lsrpdomain [-o | -O] [-I | -t | -d | -D delimiter] [-x] [-h] [-TV] [peer_domain]
```

**Description**

The `lsrpdomain` command displays information about the peer domains that the node where the command runs belongs to. Use the command’s flags and parameters to specify which information you want to display and how you want to display it. When you specify the name of a peer domain, the command displays information about that peer domain only. The `-o` and `-O` flags also limit the information this command displays. The `-o` flag displays information only about the online peer domain. The `-O` flag displays information only about peer domains that are offline.
By default, the `lsrpdomain` command displays information in table format (-t).

Some of the peer domain information that is displayed follows:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>The name of the peer domain</td>
</tr>
<tr>
<td>RSCTActiveVersion</td>
<td>The version of RSCT that is active in the peer domain</td>
</tr>
<tr>
<td>MixedVersions</td>
<td>Indicates whether more than one version of RSCT is active in the peer domain</td>
</tr>
<tr>
<td>TSPort</td>
<td>The topology services port number</td>
</tr>
<tr>
<td>GSPort</td>
<td>The group services port number</td>
</tr>
<tr>
<td>OpState</td>
<td>The current state of the peer domain</td>
</tr>
</tbody>
</table>

**Flags**

- `-o` Displays information about the node’s online peer domain.
- `-O` Displays information about peer domains that are offline for the node.
- `-l` Displays the information on separate lines (long format).
- `-t` Displays the information in separate columns (table format). This is the default.
- `-d` Displays the information using delimiters. The default delimiter is a colon (`:`). Use the `-D` flag if you want to change the default delimiter.
- `-D delimiter` Displays the information using the specified delimiter. Use this flag to specify a delimiter other than the default colon (`:`) — when the information you want to display contains colons, for example. You can use this flag to specify a delimiter of one or more characters.
- `-x` Excludes the header (suppresses header printing).
- `-h` Writes the command’s usage statement to standard output.
- `-T` Writes the command’s trace messages to standard error. For your software service organization’s use only.
- `-V` Writes the command’s verbose messages to standard output.

**Parameters**

`peer_domain` Specifies the name of the peer domain about which you want to display information. You can specify a peer domain name or a substring of a peer domain name for this parameter. If you specify a substring, the command displays information about any defined peer domain with a name that contains the substring.

**Security**
The user of the `lsrpdomain` command needs read permission for the `IBM.PeerDomain` resource class on the node on which the command runs. By default, `root` on any node in the peer domain has read and write access to this resource class through the configuration resource manager.

**Exit Status**

0  The command ran successfully.
1  An error occurred with RMC.
2  An error occurred with a command-line interface script.
An incorrect flag was entered on the command line.
An incorrect parameter was entered on the command line.
An error occurred that was based on incorrect command-line input.
The peer domain definition does not exist.

Environment Variables

CT_CONTACT
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

Restrictions
This command must be run on the node for which the peer domain information is requested.

Standard Input
When the -f "-" flag is specified, this command reads one or more node names from standard input.

Standard Output
When the -h flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
1. To display general information about the peer domains to which nodeA belongs, run this command on nodeA:
   lsrpdomain
   The output will look like this:
   Name   OpState  RSCTActiveVersion MixedVersions  TSPort  GSPort
   ApplDomain Online  2.3.3.0        No     12347  12348

2. To display general information about the peer domains to which nodeA belongs, with the default delimiter (but without the heading), run this command on nodeA:
   lsrpdomain -xd
   The output will look like this:
   ApplDomain:Online:2.3.3.0:No:12347:12348:

3. To display general information about the peer domains to which nodeA belongs, in long format, run this command on nodeA:
   lsrpdomain -l
   The output will look like this:
Name          =  App1Domain
OpState       =  Online
RSCTActiveVersion =  2.3.3.0
MixedVersions =  No
TSPort        =  12347
GSPort        =  12348

Location
/usr/sbin/rsct/bin/lsrpdomain  Contains the Lsrpdomain command

Related Information
Commands: lsrpnod, mkrpdomain, preprpnode, rmrpdomain, startrpdomain, stoprpdomain

Files: rmccli, for general information about RMC-related commands

Isrpnode Command

Purpose
Displays information about one or more of the nodes that are defined in the online peer domain.

Syntax
lsrpnode [ -o | -O | -L ] [ -l | -t | -d | -D delimiter ] [-x] [-h] [-TV] [node_name]
lsrpnode [ -l | -t | -d | -D delimiter ] [-x] -p peer_domain [-h] [-TV]

Description
The lsrpnode command displays information about one or more of the nodes that are defined in the online peer domain. Use the command’s flags and parameters to specify which information you want to display and how you want to display it. When you specify a node name, the command displays information about that node only. The -o, -O, and -L flags also limit the information this command displays. The -o flag displays information about nodes that are online. The -O flag displays information about nodes that are offline. The -L flag displays information about the local node, which is the node the command runs on.

By default, the lsrpnode command displays information in table format (-t).

Some of the node information that is displayed follows:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>The name of the node in the peer domain</td>
</tr>
<tr>
<td>OpState</td>
<td>The operational state of the node</td>
</tr>
<tr>
<td>RSCTVersion</td>
<td>The version of RSCT that is active in the node</td>
</tr>
</tbody>
</table>

Flags
- **-o** Displays information about the nodes that are online in the peer domain.
- **-O** Displays information about the nodes that are offline in the peer domain.
- **-L** Displays information about the local node only, which is the node that the command runs on.
- **-p peer_domain** Displays information about nodes defined in an offline peer domain that the local node belongs to.
  (By default, the lsrpnode command displays information about the nodes that are defined in the
domain where you are currently online.) However, this information might not reflect changes that are made to the domain after the local node is taken offline, because an offline node might not have the latest configuration.

The -p flag ignores the CT_CONTACT environment variable.

-\l Displays the information on separate lines (long format).
-\t Displays the information in separate columns (table format). This is the default format.
-\d Displays the information using delimiters. The default delimiter is a colon (:). Use the -D flag if you want to change the default delimiter.

-D delimiter
Displays the information using the specified delimiter. Use this flag to specify a delimiter other than the default colon (:) — when the information you want to display contains colons, for example. You can use this flag to specify a delimiter of one or more characters.

-x Excludes the header (suppresses header printing).
-h Writes the command’s usage statement to standard output.
-T Writes the command’s trace messages to standard error. For your software service organization’s use only.
-V Writes the command’s verbose messages to standard output.

**Parameters**

\node_name\nSpecifies the name of the node about which you want to display information. You can specify a node name or a substring of a node name for this parameter. If you specify a substring, the command displays information about any defined node with a name that contains the substring.

**Security**
The user of the lsrpnode command needs read permission for the IBM.PeerNode resource class on the node this command runs on. By default, root on any node in the peer domain has read and write access to this resource class through the configuration resource manager.

**Exit Status**

0 The command ran successfully.
1 An error occurred with RMC.
2 An error occurred with a command-line interface script.
3 An incorrect flag was entered on the command line.
4 An incorrect parameter was entered on the command line.
5 An error occurred that was based on incorrect command-line input.

**Environment Variables**

**CT_CONTACT**
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.
Restrictions
This command must be run on a node that is online in the peer domain.

Standard Input
When the -f "-" flag is specified, this command reads one or more node names from standard input.

Standard Output
When the -h flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
1. To display general information about the nodes in the online peer domain that nodeA belongs to, run this command on nodeA:

   lsrpnode

   The output will look like this:

   Name       OpState  RSCTVersion
   nodeA      online   2.3.3.0
   nodeB      online   2.3.3.0
   nodeC      offline  2.3.3.0

2. To display general information about the nodes in the online peer domain that nodeA belongs to, with the default delimiter (but without the heading), run this command on nodeA:

   lsrpnode -xd

   The output will look like this:

   nodeA:online:2.3.3.0:
   nodeB:online:2.3.3.0:
   nodeC:offline:2.3.3.0:

3. To display general information about the nodes in the online peer domain that nodeA belongs to, in long format, run this command on nodeA:

   lsrpnode -l

   The output will look like this:

   Name = nodeA
   OpState = online
   RSCTVersion = 2.3.3.0

   Name = nodeB
   OpState = online
   RSCTVersion = 2.3.3.0

   Name = nodeC
   OpState = offline
   RSCTVersion = 2.3.3.0

Location
/usr/sbin/rsct/bin/lsrpnode  Contains the lsrpnode command
Related Information
Commands: `addrpnode`, `preprpnode`, `startrpnode`, `stoprpnode`

Files: `rmccli`, for general information about RMC-related commands

lsrset Command

Purpose
Displays system rset contents.

Syntax
```bash
lsrset [ -f ] [ -v ] [ -o ] [ -r rsetname ] [ -n namespace ] [ -a ]
```

or
```bash
lsrset [ -P ] [ -v ] [ -o ] [ -p pid ]
```

Description
The `lsrset` command displays information contained in rsets stored in the system registry or rsets attached to a process.

Flags
- `-f` Displays rset owner, group, and mode data.
- `-v` Verbose mode. Displays resources contained in the rset, rset owner, group and mode data.
- `-o` Displays only the online resources contained in the rset. The default is to display all resources.
- `-p pid` Displays the effective rset attached to this process.
- `-r rsetname` Displays the rset with this name in the system registry. The name consists of a namespace and an `rsname` separated by a "/" (slash). Both the `namespace` and `rsname` may contain up to 255 characters. See the `rs_registername()` service for additional information about character set limits of rset names.
- `-n namespace` Displays all rsets in this namespace in the system registry.
- `-a` Displays all rsets in the system registry.
- `-P` Displays the partition rset attached to the specified process.

Examples
1. To display all resources for all rsets in the system registry, type:
   ```bash
   lsrset -v -a
   ```
2. To display a summary of the effective rset attached to pid 28026, type:
   ```bash
   lsrset -p 28026
   ```
3. To display the online resources in the effective rset attached to pid 28026, type:
   ```bash
   lsrset -o -p 28026
   ```
4. To display all the resources in the effective rset attached to pid 28026, type:
   ```bash
   lsrset -v -p 28026
   ```
5. To display online resources for all rsets in the system registry, type:
   ```bash
   lsrset -a -o
   ```

Files
`/etc/bin/lsrset` Contains the `lsrset` command
Related Information
The attachset, detachset, execrset, mkrset and rmrset commands.

lsrsrc Command

Purpose
Displays attributes and values for a resource or a resource class.

Syntax
To display the attributes and values for a resource:

```
lsrsrc [−s"selection_string"] [−A p | d | b] [−p property] [−l | −i | −t | −d | −D delimiter] [−x] [−a] [−h] [−TV] [resource_class] [attr...]
```

```
lsrsrc [−s"selection_string"] −r [−l | −i | −t | −d | −D delimiter] [−x] [−a] [−h] [−TV] [resource_class]
```

To display the attributes and values for a resource class:

```
lsrsrc −c [−A p | d | b] | −C rpdb_name_1 ... rpdb_name_n [−p property] [−l | −i | −t | −d | −D delimiter] [−x] [−a] [−h] [−TV] resource_class [attr...]
```

To display a list of all of the resource classes:

```
lsrsrc
```

Description
The lsrsrc command displays the persistent and dynamic attributes and their values for a resource or a resource class.

When no attribute names are specified:
- only attributes that are defined as public are displayed. Use the -p flag to override this default.
- the -A p | d | b flag controls whether persistent attributes or dynamic attributes or both — and their values — are displayed.

When one or more attribute names are specified, these names and their values are displayed in the order specified, provided that each of the specified attribute names is valid.

To display a list of the attributes and values for a resource class, specify the -c flag.

Specify the -r flag to display only the resource handles associated with the resources for the specified resource class.

By default, the resource attributes and values are displayed in long format. Use the -t, -d, or -D flag for the resources to be displayed in table format or delimiter-formatted output.

For best performance, specify either the -A p flag or only persistent attributes as parameters.

Note: Any attribute that has a data type defined as ct_none (for example, a Quantum) is not listed by the lsrsrc command. RMC does not return attribute values for attributes that are defined as Quantum.

To list attribute definitions, use the lsrsrcdef command.
Flags

- For 0x0001 with display the display flags.

- With display the display flags.

- Generates colons.

- Specifies an attribute type. By default only persistent attributes are displayed. This flag can be used only when no attribute names are specified on the command line.

- Displays only persistent attributes.

- Displays only dynamic attributes.

- Displays both persistent and dynamic attributes.

For best performance, specify the -A p flag.

- Displays the attributes for the resource class. This flag overrides the -r flag.

To display all RSCT peer domains in Distributed Management (DM) scope, specify the -c flag with the -a flag. The lsrsr -c -a command displays peer domain names at the end of the output, as a class attribute ActivePeerDomain.

- Displays the class attributes of globalized classes in RSCT peer domains for Distributed Management (DM) scope. The lsrsr -C command displays peer domain names at the end of the output, as a class attribute ActivePeerDomain.

- Specifies delimiter-formatted output. The default delimiter is a colon (:). Use the -D flag if you want to change the default delimiter.

- Specifies delimiter-formatted output that uses the specified delimiter. Use this flag to specify something other than the default colon (:). An example is when the data to be displayed contains colons. Use this flag to specify a delimiter of one or more characters.

- Generates a template of the Resource_Data_Input file that can then, after appropriate editing, be used as input to the mkrsrc command. The output is displayed in long (stanza) format. All required and optional attributes that can be used to define a resource are displayed. The attribute data type is displayed as the value in the attr=value pairs. It is suggested that when you use this flag, the output of the lsrsr command be directed to a file. This flag overrides the -s and -A d flags.

- Specifies long formatted output. Each attribute is displayed on a separate line. This is the default display format. If the lsrsr command is issued with the -l flag, but without a resource class name, the -l flag is ignored when the command returns the list of defined resource class names.

- Displays attributes with the specified property. By default, only public attributes are displayed. To display all of the attributes regardless of the property, use the -p 0 flag. Use this flag in conjunction with the -A flag when no attributes are specified on the command line.

**Persistent attribute properties:**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>0x0001</td>
<td>read_only</td>
</tr>
</tbody>
</table>
**Dynamic attribute properties:**

0x0020 public

A decimal or hexadecimal value can be specified for the property. To display attributes and their values for all attributes that have one or more properties, "OR" the properties of interest together and then specify the "OR"ed value with the -p flag. For example, to display attributes and their values for all persistent attributes that are either reqd_for_define or option_for_define, enter:

`lsrsrc -p 0x0a`

- `r` Displays the resource handles for the resources that match the specified selection string or all resources when no selection string is specified.

- `s "selection_string"` Specifies a selection string. All selection strings must be enclosed within either double or single quotation marks. If the selection string contains double quotation marks, enclose the entire selection string in single quotation marks. For example:

  - `-s 'Name == "testing"'
  - `-s 'Name ?= "test"'

Only persistent attributes may be listed in a selection string.

- `t` Specifies table format. Each attribute is displayed in a separate column, with one resource per line.

- `x` Suppresses header printing.

- `h` Writes the command’s usage statement to standard output.

- `T` Writes the command’s trace messages to standard error. For your software-service organization’s use only.

- `V` Writes the command’s verbose messages to standard output.

### Parameters

**resource_class**

Specifies the name of the resource class with the resources that you want to display.

**attr...** Specifies one or more attribute names. Both persistent and dynamic attribute names can be specified to control which attributes are displayed and their order. Zero or more attributes can be specified. Attributes must be separated by spaces. If no attribute names are specified, the -A p l d l b flag controls whether persistent attributes or dynamic attributes or both are displayed. When no attribute names are specified, only attributes that are defined as public are displayed. Use the -p flag to override this default.

### Security

The user needs read permission for the resource_class specified in lsrsr to run lsrsr. Permissions are specified in the access control list (ACL) file on the contacted system.
Exit Status

0  The command has run successfully.
1  An error occurred with RMC.
2  An error occurred with the command-line interface (CLI) script.
3  An incorrect flag was specified on the command line.
4  An incorrect parameter was specified on the command line.
5  An error occurred with RMC that was based on incorrect command-line input.

Environment Variables

CT_CONTACT
When the CT_CONTACT environment variable is set to a host name or IP address, the command contacts the Resource Monitoring and Control (RMC) daemon on the specified host. If the environment variable is not set, the command contacts the RMC daemon on the local system where the command is being run. The resource class or resources that are displayed or modified by the command are located on the system to which the connection is established.

CT_MANAGEMENT_SCOPE
Determines the management scope that is used for the session with the RMC daemon to monitor and control the resources and resource classes. The management scope determines the set of possible target nodes where the resources and resource classes can be monitored and controlled. The valid values are:

0  Specifies local scope.
1  Specifies local scope.
2  Specifies peer domain scope.
3  Specifies management domain scope.
If this environment variable is not set, local scope is used.

Standard Output
When the -h flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples

1. To list the names of all of the resource classes, enter:
   
   \texttt{lsrsrcc}
   
   The output will look like this:
   
   \begin{verbatim}
   class_name
   "IBM.Association"
   "IBM.Condition"
   "IBM.EventResponse"
   "IBM.Host"
   "IBM.Ethernet"
   "IBM.TokenRing"
   \end{verbatim}

2. To list the persistent attributes for resource IBM.Host that have 4 processors, enter:
   
   \texttt{lsrsrcc -s NumProcessors == 4 -A -p 0 IBM.Host}
The output will look like this:

```
Resource Persistent Attributes for: IBM.Host
resource 1:
  Name = "c175n05.ppd.pok.ibm.com"
  ResourceHandle = "0x4008 0x0001 0x00000000 0x0069684c 0x0d7f55d5 0x0c32fde3"
  Variety = 1
  NodeList = [1]
  NumProcessors = 4
  RealMemSize = 1073696768
```

3. To list the public dynamic attributes for resource IBM.Host on node 1, enter:

```
lsrsrc -s 'Name == "c175n05.ppd.pok.ibm.com"' -A d IBM.Host
```

The output will look like this:

```
Resource Dynamic Attributes for: IBM.Host
resource 1:
  ProcRunQueue = 1.03347987093142
  ProcSwapQueue = 1.00548852941929
  TotalPgSpSize = 65536
  TotalPgSpFree = 65131
  PctTotalPgSpUsed = 0.61798095703125
  PctTotalPgSpFree = 99.3820190429688
  PctTotalTimeIdle = 0
  PctTotalTimeWait = 51.5244382399734
  PctTotalTimeUser = 12.8246006482343
  PctTotalTimeKernel = 35.6509611117922
  PctRealMemFree = 66
  PctRealMemPinned = 4
  RealMemFramesFree = 173361
  VMPgInRate = 0
  VMPgOutRate = 0
  VMPgFaultRate = 0
  ...
```

4. To list the Name, Variety, and ProcessorType attributes for the IBM.Processor resource on all the online nodes, enter:

```
lsrsrc IBM.Processor Name Variety ProcessorType
```

The output will look like this:

```
Resource Persistent Attributes for: IBM.Processor
resource 1:
  Name = "proc3"
  Variety = 1
  ProcessorType = "PowerPC_604"
resource 2:
  Name = "proc2"
  Variety = 1
  ProcessorType = "PowerPC_604"
resource 3:
  Name = "proc1"
  Variety = 1
  ProcessorType = "PowerPC_604"
resource 4:
  Name = "proc0"
  Variety = 1
  ProcessorType = "PowerPC_604"
```

5. To list both the persistent and dynamic attributes for the resource class IBM.Condition, enter:

```
lsrsrc -c -A b -p 0 IBM.Condition
```

The output will look like this:
Resource Class Persistent and Dynamic Attributes for: IBM.Condition

resource 1:
  ResourceType = 0
  Variety    = 0

Location
/usr/sbin/rsct/bin/lsrsrc Contains the lsrsrc command

Related Information
Commands: lsrsrcdef, mksrsc

Files: rmccli

Isrsrcdef Command

Purpose
Displays definition information for a resource or a resource class.

Syntax
For a resource...

To display the definition:


To display the persistent attribute definitions:


To display the dynamic attribute definitions:


For a resource class...

To display the definition:


To display the persistent attribute definitions:


To display the dynamic attribute definitions:


To display a list of all of the resource class names:
Isrsrctdef

Description
The Isrsrctdef command displays the definition of a resource or a resource class or the persistent or dynamic attribute definitions of a resource or a resource class. By default:

- if no attr parameters are specified on the command line, this command displays the definitions for public attributes. To override this default, use the -p flag or specify the name of the attribute you want to display.
- this command does not display attribute descriptions. To display attribute definitions and descriptions, specify the -e flag.

Flags
-a Specifies that this command applies to all nodes in the cluster. The cluster scope is determined by the CT_MANAGEMENT_SCOPE environment variable. If it is not set, first the management domain scope is chosen if it exists, then the peer domain scope is chosen if it exists, and then local scope is chosen, until the scope is valid for the command. The command will run once for the first valid scope found. For example, if both a management and peer domain exist, Isrsrctdef -a with CT_MANAGEMENT_SCOPE not set will display the management domain. In this case, to display the peer domain, set CT_MANAGEMENT_SCOPE to 2.

-A p l d Specifies the attribute type. You can display either persistent or dynamic attribute definitions. Use this flag with the -c flag to display the persistent or dynamic attribute definitions of a resource class.
  - p Displays only persistent attributes
  - d Displays only dynamic attributes
  - c Displays the definition of a resource class definition. To display the persistent attribute definitions for a resource class, specify this flag with the -A p flag. To display the dynamic attribute definitions for a resource class, specify this flag with the -A d flag.
  - d Specifies delimiter-formatted output. The default delimiter is a colon (:). Use the -D flag to change the default delimiter.

-D delimiter
  Specifies delimiter-formatted output that uses the specified delimiter. Use this flag to specify something other than the default colon (:). An example is when the data to be displayed contains colons. Use this flag to specify a delimiter of one or more characters.

-e Specifies expanded format. By default, the descriptions of the definitions are not displayed. Specify this flag to display the definitions and the descriptions.

-i Generates a template of the Resource_Data_Input file that can then, after appropriate editing, be used as input to the mkrsrc command. The output is displayed in long (stanza) format. All required and optional attributes that can be used to define a resource are displayed. The attribute data type is displayed as the value in the attr=value pairs. It is suggested that when you use this flag, the output of the Isrsrctdef command be directed to a file. This flag overrides the -s and -A d flags.

-l Specifies “long” format — one entry per line. This is the default display format. If the Isrsrctdef -l command is issued without a resource class name, this flag is ignored when the command returns the list of defined resource class names.

-p property
  Displays attribute definitions for attributes with the specified property. By default, only the definitions for public attributes are displayed. To display all attribute definitions regardless of the property, use the -p 0 flag.
Persistent attribute properties:

<table>
<thead>
<tr>
<th>Value</th>
<th>Property</th>
</tr>
</thead>
<tbody>
<tr>
<td>0x0001</td>
<td>read_only</td>
</tr>
<tr>
<td>0x0002</td>
<td>reqd_for_define</td>
</tr>
<tr>
<td>0x0004</td>
<td>inval_for_define</td>
</tr>
<tr>
<td>0x0008</td>
<td>option_for_define</td>
</tr>
<tr>
<td>0x0010</td>
<td>selectable</td>
</tr>
<tr>
<td>0x0020</td>
<td>public</td>
</tr>
</tbody>
</table>

Dynamic attribute properties:

<table>
<thead>
<tr>
<th>Value</th>
<th>Property</th>
</tr>
</thead>
<tbody>
<tr>
<td>0x0020</td>
<td>public</td>
</tr>
</tbody>
</table>

A decimal or hexadecimal value can be specified for the property. To request the attribute definitions for all attributes that have one or more properties, “OR” the properties of interest together and then specify the “OR”ed value with the -p flag. For example, to request the attribute definitions for all persistent attributes that are either reqd_for_define or option_for_define, enter:

```
lsrsrcdef -p 0x0a
```

-s Displays the structured data definition. Specify this flag for the structured data definition to be expanded so that each element definition of the structured data attributes is displayed.

-t Specifies table format. Each attribute is displayed in a separate column, with one resource per line.

-x Suppresses header printing.

-h Writes the command’s usage statement to standard output.

-T Writes the command’s trace messages to standard error. For your software-service organization’s use only.

-V Writes the command’s verbose messages to standard output.

Parameters

resource_class

Specifies the name of the resource class with the attribute definitions you want to display.

attr

If a resource_class parameter is specified, zero or more attribute names can be specified. If no attr parameter is specified, the definition for all of the attributes for the resource are displayed. Specify individual attribute names to control which attributes are displayed and their order. Specify only persistent attribute names when the -A p flag is used. Specify only dynamic attribute names when the -A d flag is used. Attributes must be separated by spaces.

Security

The user needs write permission for the resource_class specified in lsrsrcdef to run lsrsrcdef. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status

0 The command has run successfully.
1 An error occurred with RMC.
2 An error occurred with the command-line interface (CLI) script.
3 An incorrect flag was specified on the command line.
4 An incorrect parameter was specified on the command line.
An error occurred with RMC that was based on incorrect command-line input.

Environment Variables

**CT_CONTACT**
When the CT_CONTACT environment variable is set to a host name or IP address, the command contacts the Resource Monitoring and Control (RMC) daemon on the specified host. If the environment variable is not set, the command contacts the RMC daemon on the local system where the command is being run. The resource class or resources that are displayed or modified by the command are located on the system to which the connection is established.

**CT_MANAGEMENT_SCOPE**
Determines the management scope that is used for the session with the RMC daemon to monitor and control the resources and resource classes. The management scope determines the set of possible target nodes where the resources and resource classes can be monitored and controlled. The valid values are:

- **0** Specifies *local* scope.
- **1** Specifies *local* scope.
- **2** Specifies *peer domain* scope.
- **3** Specifies *management domain* scope.

If this environment variable is *not* set, *local* scope is used.

Standard Output
When the **-h** flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples

1. To display the names of all of the resource classes defined on the system, enter:
   ```
   lsrsrcdef
   ```
   The output will look like this:
   ```
   class_name  "IBM.ATMDevice"
   "IBM.Association"
   "IBM.AuditLog"
   "IBM.AuditLogTemplate"
   "IBM.Condition"
   "IBM.EthernetDevice"
   "IBM.EventResponse"
   ...
   ```

2. To display the resource class definitions for resource IBM.Host, enter:
   ```
   lsrsrcdef -c IBM.Host
   ```
   The output will look like this:
   ```
   Resource Class Definition for: IBM.Host  
   resource class 1:  
   class_name = "IBM.Host"
   class_id = 8
   properties = {"has_rsrc_insts","mtype_subdivided"}
   display_name = ""
   description = ""
locator = "NodeList"
class_pattr_count = 1
class_dattr_count = 3
class_action_count = 0
pattr_count = 6
dattr_count = 47
action_count = 0
error_count = 0
rsrc_mgr_count = 1

rsrc_mgrs 1:
  mgr_name = "IBM.HostRM"
  first_key = 1
  last_key = 1

3. To display the resource class persistent attribute definitions for resource IBM.Host, enter:

lsrsrcrefl -c -A p -p 0 IBM.Host

The output will look like this:

Resource Class Persistent Attribute Definitions for: IBM.Host
attribute 1:
  program_name = "Variety"
  display_name = ""
  group_name = ""
  properties = {"read_only","inval_for_define")
  description = ""
  attribute_id = 0
  group_id = 255
  data_type = "uint32"
  variety_list = {{1..1}}
  variety_count = 1
  default_value = 0

4. To display the resource persistent attribute definitions and descriptions for resource IBM.Host, enter:

lsrsrcrefl -A -p 0 -e IBM.Host

The output will look like this:

Resource Persistent Attribute Definitions for: IBM.Host
attribute 1:
  program_name = "Name"
  display_name = "Name"
  group_name = "General"
  properties = {"reqd_for_define","public","selectable")
  description = "Identifies the current name of the host as returned by command."
  attribute_id = 0
  group_id = 0
  data_type = "char_ptr"
  variety_list = {{1..1}}
  variety_count = 1
  default_value = ""

attribute 2:
  program_name = "ResourceHandle"
  display_name = "Resource Handle"
  group_name = "Internal"
  properties = {"read_only","inval_for_define","selectable")
  description = "A globally unique handle that identifies the host. Every resource is assigned a resource handle, which is used internally for identifying and locating each resource. The resource handle is fixed in size and avoids the problems of name space collisions across different types of resources."
  attribute_id = 1
  group_id = 255
  data_type = "rsrc_handle_ptr"
  variety_list = {{1..1}}
5. To display the public dynamic attributes for resource IBM.Host, enter:

\texttt{lsrsrdef -A d IBM.Host}

The output will look like this:

Resource Dynamic Attribute Definitions for: IBM.Host
attribute 1:
\begin{verbatim}
program_name   = "ProcRunQueue"
display_name   = "" 
group_name     = "" 
properties      = \{"public"\} 
description    = "" 
attribute_id   = 1 
group_id       = 1 
data_type       = "float64" 
variable_type  = 0 
variety_list   = \{\{1..1\}\} 
variety_count  = 1 
init_value     = 0 
min_value      = 0 
max_value      = 100 
expression     = "(ProcRunQueue - ProcRunQueue@P) >= (ProcRunQueue@P * 0.5)"
expression_description = "" 
rearm_expression = "ProcRunQueue < 50" 
rearm_description = "" 
PTX_name       = "" 
\end{verbatim}

attribute 2:
\ ...

\textbf{Location}

\texttt{/usr/sbin/rsct/bin/lsrsrdef} Contains the \texttt{lsrsrdef} command

\textbf{Related Information}

Commands: \texttt{lsrsrc, mkrsrc}

Files: \texttt{Resource\_Data\_Input, rmccli}

\textbf{Issavevg Command}

\textbf{Purpose}

Lists or restores the contents of a volume group backup on a specified media.

\textbf{Syntax}


\textbf{Description}

The \texttt{Issavevg} command lists the contents of a volume group backup from tape, file, CD-ROM, or other source and can be used to restore files from a valid backup source. The \texttt{Issavevg} command also works for multi-volume backups such as multiple CDs, DVDs, or tapes.
The `issavevg` -r and `restorevfiles` commands perform identical operations and should be considered interchangeable.

### Flags

- `-a` verifies the physical block size of the tape backup, as specified by the `-b block` flag. You may need to alter the block size if necessary to read the backup. The `-a` flag is valid only when a tape backup is used.

- `-b blocks` specifies the number of 512-byte blocks to read in a single input operation, as defined by the `blocks` parameter. If the `blocks` parameter is not specified, the number of blocks read will default to 100.

- `-B` prints the volume group backup log to `stdout`.

  This flag will display the past 256 backups (roughly). The log is in alog format and is kept in `/var/adm/ras/vgbackuplog`. Each line of the log is a semicolon-separated list of the file or device name, the command used to make backup, date, shrink size, full size of the backup, and recommended maintenance level (if any).

  **Note:** The shrink size is the size of the data on all filesystems. The full size is total size of each filesystem (unused + data).

- `-c` produces colon-separated output. This flag only works with the `-i` and `-L` flags.

- `-d path` specifies the directory path to which the files will be restored, as defined by the `path` parameter. If the `-d` parameter is not used, the current working directory is used. This can be a problem if the current working directory is root. We recommend writing to a temporary folder instead of to root.

- `-D` produces debug output.

- `-I` displays useful information about a volume group backup.

  This flag requires the `-f device` flag. This flag causes `issavevg` to display information such as volume group, date and time backup was made, uname output from backed up system, oslevel, recommended maintenance level, backup size in megabytes, and backup shrink size in megabytes. The shrink size is the size of the data on all filesystems. The full size is the total size of each filesystem (unused + data). The `-I` flag also displays the logical volume and filesystem information of the backed up volume group, equivalent to running "`lsvg -i vgroupname`".

- `-L` displays lpp fileset information about a `mksysb` backup only.

  This flag requires the `-f device` flag and displays the equivalent information to that produced by invoking "`lspp -I`" on the running backed up system. This flag does not produce output about any volume group backup other than that produced by `mksysb`.

- `-f device` specifies the type of device containing the backup (file, tape, CD-ROM, or other source) as defined by the `device` parameter. When `-f` is not specified, `device` will default to `/dev/rmt0`.

- `-r` specifies to restore the backup files, as defined by the `file-list` parameter. If the `file-list` parameter is not specified, then all files in the backup will be restored. If the `-r` flag is not used, then executing the `issavevg` command only lists the files in the specified backup.

- `-s` specifies that the backup source is a user volume group and not rootvg.

- `-V` verifies a tape backup.

  This flag requires the `-f device` flag and works for tape devices only. The `-V` flag causes `issavevg` to verify the readability of the header of each file on the volume group backup and print any errors that occur to `stderr`.

### Parameters

- `file_list` identifies the list of files to be restored. This parameter is used only when the `-r` flag is specified. The full path of the files relative to the current directory should be specified in the space-separated list. All files in the specified directory will be restored unless otherwise directed. If you are restoring all files in a directory, we recommend writing to a temporary folder instead of to root.
Examples

1. To list the contents of the system backup located on the default device /dev/rmt0, enter:
   
   lssavevg

2. To list the contents of the system backup located on device /dev/cd1, enter:
   
   lssavevg -f /dev/cd1

3. To list the contents of the system backup located on device /dev/cd1, which is a user volume group that is not rootvg, enter:
   
   lssavevg -f /dev/cd1 -s

4. To restore /etc/filesystems from the system backup located on device /dev/cd1, enter:
   
   lssavevg -f /dev/cd1 -r . /etc/filesystems

5. To restore all files in the /myfs/test directory of the non-rootvg backup, which is located on device /dev/cd1, and write the restored files to /data/myfiles, enter:
   
   lssavevg -f /dev/cd1 -r -s -d /data/myfiles . /myfs/test

6. To display colon separated lpp information about a mksysb backup tape located on /dev/rmt0, enter the following:
   
   lsmksysb -Lc -f /dev/rmt0

7. To display the volume group backup log to stdout, enter:
   
   lssavevg -B

8. To list volume group and general backup data about a backup located at /tmp/mybackup, enter:
   
   lssavevg -l -f /tmp/mybackup

9. To verify the readability of each header on a volume group backup tape in /dev/rmt0, enter:
   
   lsmksysb -V -f /dev/rmt0

Files

/usr/bin/lssavevg

Contains the lssavevg command

Related Information

The restorevg files command.
• /etc/security/passwd
• /etc/security/portlog
• /etc/security/user

When listing attributes in the /etc/security/environ, /etc/security/lastlog, /etc/security/limits, /etc/security/passwd, and /etc/security/user files, the stanza name specified by the Stanza parameter must be either a valid user name or default. When listing attributes in the /etc/security/group file, the stanza name specified by the Stanza parameter must be either a valid group name or default. When listing attributes in the /etc/security/portlog file, the Stanza parameter must be a valid port name. When listing attributes in the /etc/security/login.cfg file, the Stanza parameter must be either a valid port name, a method name, or the usw attribute.

You cannot list the password attribute of the /etc/security/passwd file with the lssec command.

Only the root user or a user with PasswdAdmin authorization can list the lastupdate and flags attributes for administrative users.

Flags

-c Specifies that the output should be in colon-separated format.
-f File Specifies the name of the stanza file to list.
-s Stanza Specifies the name of the stanza to list.
-a Attribute Specifies the attribute to list.

Security

Access Control: This command grants execute access only to the root user and the security group. The command has the trusted computing base attribute and runs the setuid subroutine for the root user to access the security databases.

Files Accessed:

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>/etc/security/environ</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/group</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/lastlog</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/limits</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/login.cfg</td>
</tr>
<tr>
<td>r</td>
<td>/usr/lib/security/mkuser.default</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/passwd</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/portlog</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/user</td>
</tr>
</tbody>
</table>

Examples

1. To list the number of unsuccessful login attempts by the root user since the last successful login of the root user, enter:
   
   lssec -f /etc/security/lastlog -s root -a unsuccessful_login_count
   
   The system displays the result as follows:
   
   root unsuccessful_login_count=15

2. To list the times that logins are allowed on the /dev/tty2 port, enter:
   
   lssec -f /etc/security/login.cfg -s /dev/tty2 -a logintimes
The system displays the result as follows:

```
/dev/tty0 logintimes=!january1,!july4,!december25
```

3. To list the default setting for the **path** attribute and the **ttys** attribute in colon format,

4. Enter:

```
lssec -c -f /etc/security/user -s default -a tpath -a ttys
```

The system displays the result as follows:

```
#name:tpath:ttys
default:nosak:ALL
```

### Files

- `/usr/bin/lssec` Specifies the path to the `lssec` command.
- `/etc/security/environment` Contains the environment attributes of users.
- `/etc/security/group` Contains extended attributes of groups.
- `/etc/security/lastlog` Defines the last login attributes for users.
- `/etc/security/limits` Defines resource quotas and limits for each user.
- `/etc/security/login.cfg` Contains port configuration information.
- `/etc/security/user` Contains the defaults values for new users.
- `/usr/lib/security/mkuser.default` Contains password information.
- `/etc/security/portlog` Contains unsuccessful login attempt information for each port.
- `/etc/security/extended` Contains the extended attributes of users.

### Related Information

The commands `chgroup`, `chsec`, `chuser`, `grpck`, `login`, `lssec`, `lsuser`, `mkgroup`, `mkuser`, `passwd`, `pwdck`, `rmgroup`, `rmuser`, `su`, `usrck` command.

The subroutines `getgroupattr`, `getportattr`, `getuserattr`, `getuserpw`, `putgroupattr`, `putportattr`, `putuserattr`, `putuserpw` subroutine.

List of Security and Auditing Subroutines in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

### Issensor Command

**Purpose**
Displays the sensors that are defined to the resource monitoring and control (RMC) subsystem.

**Syntax**

```
issensor [-a | -n host1[,host2...]] [-h] [-v | -V] [-A | name1 [name2...]]
```

**Description**

The `issensor` command displays the attributes of the sensor identified by `name`. If `name` is omitted, the `issensor` command lists the names of all of the sensors. Use the `-A` flag to list all of the sensors and all of their attributes and values.

Using `issensor name` or `issensor -A` causes the sensor command to run. The sensor command is the command or script that is defined to set the sensor attribute values. It is specified using the `mksensor` command.
The `lssensor` command runs on any node. If you want `lssensor` to run on all of the nodes in a domain, use the the `-a` flag. If you want `lssensor` to run on a subset of nodes in a domain, use the the `-n` flag.

The `lssensor` command lists the following information about defined sensors:

<table>
<thead>
<tr>
<th>Field</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>The name of the sensor</td>
</tr>
<tr>
<td>Command</td>
<td>The command to run that updates the sensor attributes</td>
</tr>
<tr>
<td>ConfigChanged</td>
<td>Information about persistent attribute or access changes</td>
</tr>
<tr>
<td>Description</td>
<td>This field is not used</td>
</tr>
<tr>
<td>ErrorExitValue</td>
<td>Indicates how the exit value is interpreted by the sensor resource manager</td>
</tr>
<tr>
<td>ExitValue</td>
<td>The exit code from the command that is running</td>
</tr>
<tr>
<td>Float32</td>
<td>The type float32 attribute for this sensor resource</td>
</tr>
<tr>
<td>Float64</td>
<td>The type float64 attribute for this sensor resource</td>
</tr>
<tr>
<td>Int32</td>
<td>The type int32 attribute for this sensor resource</td>
</tr>
<tr>
<td>Int64</td>
<td>The type int64 attribute for this sensor resource</td>
</tr>
<tr>
<td>UInt32</td>
<td>The type uint32 attribute for this sensor resource</td>
</tr>
<tr>
<td>UInt64</td>
<td>The type uint64 attribute for this sensor resource</td>
</tr>
<tr>
<td>String</td>
<td>The type string attribute for this sensor resource</td>
</tr>
<tr>
<td>NodeNameList</td>
<td>The name of the node where the sensor resource is defined</td>
</tr>
<tr>
<td>RefreshInterval</td>
<td>The interval in seconds in which the sensor attribute values are updated by running the sensor command</td>
</tr>
<tr>
<td>UserName</td>
<td>The user ID to use when running the sensor command</td>
</tr>
</tbody>
</table>

A `sensor` is an RMC resource with attributes that can be monitored. The values of the attributes are set by the sensor resource manager that updates the values at defined intervals using commands or scripts as designated by the `mksensor` command. The sensor values can also be updated using the `refsensor` command.

**Flags**

- `-a` Lists sensors that match the specified name on all nodes in the domain. The CT_MANAGEMENT_SCOPE environment variable determines the cluster scope. If CT_MANAGEMENT_SCOPE is not set, first the management domain scope is chosen if it exists, then the peer domain scope is chosen if it exists, and then local scope is chosen, until the scope is valid for the command. The command will run once for the first valid scope found. For example, if both a management domain and a peer domain exist, `lssensor -a` with CT_MANAGEMENT_SCOPE not set will run in the management domain. In this case, to run in the peer domain, set CT_MANAGEMENT_SCOPE to 2.

- `-A` Displays all of the sensors with their attributes and values.

- `-n host1[,host2...]` Specifies the node from which the sensor should be listed. By default, the sensor is listed from the local node. This flag is only appropriate in a management domain or a peer domain.

- `-h` Writes the command’s usage statement to standard output.

- `-v | -V` Writes the command’s verbose messages to standard output.
Parameters

\texttt{name1 [name2...]}

Specifies the names of one or more sensors to display.

Security

The user needs read permission for the \texttt{IBM.Sensor} resource class in order to run \texttt{lssensor}. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status

0  The command has run successfully.
1  An incorrect combination of flags and parameters has been entered.
6  No sensor resources were found.
\(n\)  Based on other errors that can be returned by the RMC subsystem.

Environment Variables

\texttt{CT\_CONTACT}

When the \texttt{CT\_CONTACT} environment variable is set to a host name or IP address, the command contacts the resource monitoring and control (RMC) daemon on the specified host. If this environment variable is not set, the command contacts the RMC daemon on the local system where the command is being run. The resource class or resources that are displayed or modified by the command are located on the system to which the connection is established.

\texttt{CT\_MANAGEMENT\_SCOPE}

Determines the management scope that is used for the session with the RMC daemon to monitor and control the resources and resource classes. The management scope determines the set of possible target nodes where the resources and resource classes can be monitored and controlled.

The valid values are:

0  Specifies \textit{local} scope.
1  Specifies \textit{local} scope.
2  Specifies \textit{peer domain} scope.
3  Specifies \textit{management domain} scope.

If this environment variable is \textit{not} set, \textit{local} scope is used.

Examples

1. To list the names of all of the sensors, enter:
   \texttt{lssensor}

   The output will look like this:

   \texttt{sensor1}
   \texttt{sensor2}
   \texttt{sensor3}

2. To list the names and attributes of all sensors, enter:
   \texttt{lssensor -A}

   The output will look like this:

   Name = sensor1
   Command = /usr/local/bin/sensorcmd1
   ConfigChanged = 0
To list the names and attributes of all sensors, enter:

lssensor sensor2

The output will look like this:

Name = sensor2
Command = /usr/local/bin/sensorcmd2
ConfigChanged = 0
Description =
ErrorExitValue = 1
ExitValue = 127
Float32 = 0
Float64 = 0
Int32 = 0
Int64 = 0
NodeNameList = {somenode.pok.ibm.com}
RefreshInterval = 60
String =
Uint32 = 0
Uint64 = 0
UserName = root

Location
/usr/sbin/rsc/bin/lssensor       Contains the lssensor command

Related Information
Commands: chsensor, mksensor, refsensor, rmsensor
Isslot Command

Purpose
Displays dynamically reconfigurable slots, such as hot plug slots, and their characteristics.

Syntax
Isslot [c] ConnectorType [a | o | l] DeviceName [s] Slot [l F] Delimiter

Description
The Isslot command displays all the specified hot plug slots and their characteristics. Hot plug slots are the plug-in points for connecting entities that can be added and removed from the system without turning the system power off or rebooting the operating system. The -c flag is required. It specifies the type of hot plug connector, for example, pci for hot pluggable PCI adapters. You can display only the empty, that is, available, hot plug slots with the -a flag, the occupied slots with the -o flag, or a specific slot by using the -s flag. The -l flag can be used to locate the slot associated with specified DeviceName, as listed by the lsdev command.

The Isslot command is used to list the connectors which are connection points for either physical entities like PCI adapters or logical entities like PCI Host Bridges (PHBs). The connector types supported are pci, slot and phb, where pci is a physical connector and the slot and phb are the logical connectors. The -a and the -o flags will be ignored for the logical connectors. The Isslot command in the case of the logical connectors displays the logical entities that are currently assigned to the partition, depending upon the connector type specified.

The output of the Isslot command is dependent on the ConnectorType and the platform on which the command is executed. The characteristics of a slot may include the following:

- Slot name or identification
- Connector type or slot description, for example, a PCI hot plug slot
- Connected device name(s), for example, scsi0, ent0

When the PHBs are listed using the Isslot command, the Device(s) Connected column will display the ODM name of the PHB followed by the ODM names of the devices corresponding to the logical slots underneath the PHB, with all the ODM devices associated with each logical slot displayed on each separate line under the ODM name of the PHB. In case there is no ODM name for the PHB, a blank line will be displayed.

Flags

-a Displays available hot plug slots and their characteristics. Available slots are those slots that do not have a hot plug device connected. This flag is ignored for connector types of slot and phb.

-c ConnectorType Displays the slots of the specified ConnectorType. ConnectorType identifies the type of connector. For example, the ConnectorType for a hot plug PCI slot is pci, for logical slots, it is slot and for PHBs, it is phb. This flag is required.

-F Delimiter Specifies a single character to delimit the output. The heading is not displayed and the columns are delimited by the Delimiter character.

-I DeviceName Displays the characteristics of the slot to which DeviceName is associated. The DeviceName is the logical device name of the device connected to the slot, as listed by the lsdev command.

-o Displays the characteristics of the occupied slots. Occupied slots have a hot plug device connected. This flag is ignored for connector types of slot and phb.

-s Slot Displays characteristics for the specified Slot. The format of Slot is platform/connector_type dependent.


Examples

1. To list the available PCI hot plug slots, enter:
   
   lsslot -c pci -a

   The system displays a message similar to the following:

<table>
<thead>
<tr>
<th>Slot name</th>
<th>Description</th>
<th>Device(s) Connected</th>
</tr>
</thead>
<tbody>
<tr>
<td>U0.4-P1-i1</td>
<td>PCI 64 bit, 66MHz, 3.3 volt slot</td>
<td>empty</td>
</tr>
<tr>
<td>U0.4-P1-i2</td>
<td>PCI 64 bit, 66MHz, 3.3 volt slot</td>
<td>empty</td>
</tr>
<tr>
<td>U0.4-P1-i3</td>
<td>PCI 64 bit, 66MHz, 3.3 volt slot</td>
<td>empty</td>
</tr>
</tbody>
</table>

2. To list the PCI hot plug slot associated with a scsi adapter named scsi1, enter:
   
   lsslot -c pci -l scsi1

   The system displays a message similar to the following:

<table>
<thead>
<tr>
<th>Slot name</th>
<th>Description</th>
<th>Device(s) Connected</th>
</tr>
</thead>
<tbody>
<tr>
<td>U0.4-P1-i1</td>
<td>PCI 64 bit, 33MHz, 5 volt slot</td>
<td>scsi1</td>
</tr>
</tbody>
</table>

3. To list all the PCI hot plug slots, enter:
   
   lsslot -c pci

   The system displays a message similar to the following:

<table>
<thead>
<tr>
<th>Slot name</th>
<th>Description</th>
<th>Device(s) Connected</th>
</tr>
</thead>
<tbody>
<tr>
<td>U0.4-P1-i1</td>
<td>PCI 64 bit, 33MHz, 3.3 volt slot</td>
<td>empty</td>
</tr>
<tr>
<td>U0.4-P1-i2</td>
<td>PCI 64 bit, 33MHz, 3.3 volt slot</td>
<td>scsi0</td>
</tr>
<tr>
<td>U0.4-P1-i3</td>
<td>PCI 64 bit, 33MHz, 3.3 volt slot</td>
<td>unknown</td>
</tr>
<tr>
<td>U0.4-P1-i5</td>
<td>PCI 64 bit, 33MHz, 3.3 volt slot</td>
<td>empty</td>
</tr>
</tbody>
</table>

   Slots that have unknown in the Device(s) Connected column have a device connected to the slot, but the device isn't in the ODM customized device (CuDv) database. This can be due to the device having been newly added but not configured yet, deleted with the rmdev -d command, or the system may not be installed with the software packages associated with the device.

4. To list all the PCI Host Bridges that are assigned to the partition, enter:
   
   lsslot -c phb

   This displays output similar to the following:

<table>
<thead>
<tr>
<th>PHB Name</th>
<th>Description</th>
<th>Device(s) Connected</th>
</tr>
</thead>
<tbody>
<tr>
<td>PHB 1</td>
<td>Logical PCI Host Bridge</td>
<td>pci0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>pci2 scsi1</td>
</tr>
<tr>
<td>PHB 2</td>
<td>Logical PCI Host Bridge</td>
<td>pci1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>pci3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>pci4 scsi2</td>
</tr>
</tbody>
</table>

5. In case the PCI Host Bridge is assigned to the partition but has no ODM data, column will show blank as shown in this example. For example, when you enter:
   
   lsslot -c phb

   The output will look similar to the following:

<table>
<thead>
<tr>
<th>PHB Name</th>
<th>Description</th>
<th>Device(s) Connected</th>
</tr>
</thead>
<tbody>
<tr>
<td>PHB 4</td>
<td>Logical PCI Host Bridge</td>
<td></td>
</tr>
<tr>
<td>PHB 5</td>
<td>Logical PCI Host Bridge</td>
<td></td>
</tr>
</tbody>
</table>

Files

/usr/sbin/lsslot
Related Information
The `drslot` command, the `lsdev` command.

For information about Hot Plug Management and PCI Hot Plug Support for PCI Adapters, see PCI Hot Plug Management in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.

For information about using PCI Hot Plug Support for PCI Adapters, see Managing Hot Plug Connectors in AIX 5L Version 5.2 System Management Guide: Operating System and Devices.

## Issrc Command

### Purpose
Gets the status of a subsystem, a group of subsystems, or a subserver.

### Syntax

**To Get All Status**
```
issrc [ -H Host ] -a
```

**To Get Group Status**
```
issrc [ -H Host ] -g GroupName
```

**To Get Subsystem Status**
```
issrc [ -H Host ] [ -l ] -s Subsystem
```

**To Get Status by PID**
```
issrc [ -H Host ] [ -l ] -p SubsystemPID
```

**To Get Subserver Status**
```
issrc [ -H Host ] [ -l ] -t Type [ -p SubsystemPID ] [ -o Object ] [ -P SubserverPID ]
```

**To Get Subsystem Status in SMIT Format**
```
issrc [ -S ] [ -s Subsystem ] [ -d ]
```

**To Get Subserver Status in SMIT Format**
```
issrc [ -T ] [ -t Type ]
```

**To Get Notify in SMIT Format**
```
issrc [ -N ] [ -n NotifyName ]
```

### Description

The `issrc` command sends a request to the System Resource Controller to get status on a subsystem, a group of subsystems, or all subsystems. The `issrc` command sends a subsystem request packet to the daemon to be forwarded to the subsystem for a subserver status or a long subsystem status.

You can choose whether to request a short or long status for a subserver. When the `-l` flag is absent, the status request is assumed to be a short status. A short status of a subsystem, group of subsystems, or all subsystems is handled by the System Resource Controller.

When the `-l` flag is present for a subsystem, a status request is taken to the subsystem and the subsystem sends the status back. The `-l` flag is supported only for those subsystems not using signals as their communication method. For either a long or short status of a subserver, the subsystem is sent a status request packet, and the subsystem sends the status back.
Flags

-a Lists the current status of all defined subsystem.
-d Specifies that the default record is printed.
-g GroupName Specifies a group of subsystems to get status for. The command is unsuccessful if the GroupName variable is not contained in the subsystem object class.
-h Host Specifies the foreign host on which this status action is requested. The local user must be running as "root". The remote system must be configured to accept remote System Resource Controller requests. That is, the srcmstr daemon (see /etc/inettab) must be started with the -r flag and the /etc/hosts.equiv or .rhosts file must be configured to allow remote requests.
-I Requests that a subsystem send its current status in long form. Long status requires that a status request be sent to the subsystem; it is the responsibility of the subsystem to return the status.
-n NotifyName Specifies the name of a notify method.
-N Specifies that the Object Data Manager (ODM) records are output in SMIT format for the notify object class.
-o Object Specifies that a subserver Object variable is passed to the subsystem as a character string.
-p SubsystemPID Specifies a particular instance of the SubsystemPID variable to get status for, or a particular instance of the subsystem to which the status subserver request is to be taken.
-P SubserverPID Specifies that a SubserverPID variable is to be passed to the subsystem as a character string.
-s Subsystem Specifies a subsystem to get status for. The Subsystem variable can be the actual subsystem name or the synonym name for the subsystem. The command is unsuccessful if the Subsystem variable is not contained in the subsystem object class.
-S Specifies that the ODM records are output in SMIT format for the subsystem object class.
-t Type Requests that a subsystem send the current status of a subserver. The command is unsuccessful if the subserver Type variable is not contained in the subserver object class.
-T Specifies that the ODM records are output in SMIT format for the subserver object class.

Examples

1. To get the status of all subsystems on the local machine, enter:
   
   lssrc -a
   
   This gets the status of all subsystems known on the local machine.

2. To get the status of all subsystems on a foreign host, enter:
   
   lssrc -H zork -a
   
   This gets the status of all subsystems known on the zork machine.

3. To get the status of the srctest subsystem, enter:
   
   lssrc -S srctest
   
   This gets the status of all instances of the srctest subsystem on the local machine.

4. To get the status of the subsystem by PID, enter:
   
   lssrc -P 1234
This gets the status of the subsystem with the subsystem PID of 1234 on the local machine.

5. To get the status of the tcpip subsystem group, enter:

   `lssrc -g tcpip`

   This gets the status of all instances of subsystems in the tcpip group on the local machine.

6. To get the status of the tester subserver, enter:

   `lssrc -t tester -p 1234`

   This gets the status of tester subserver that belongs to the srctest subsystem with the subsystem PID of 1234 on the local machine.

7. To get the status of the subsystem by PID, enter:

   `lssrc -l -p 1234`

   This gets the long status of the subsystem with the PID of 1234.

Files

/etc/objrepos/SRCsubsys  Specifies the SRC Subsystem Configuration Object Class.
/etc/objrepos/SRCsubsvr  Specifies the SRC Subserver Configuration Object Class.
/etc/objrepos/SRCnotify  Specifies the SRC Notify Configuration Object Class.
/etc/services  Defines the sockets and protocols used for Internet services.
/dev/SRC  Specifies the AF_UNIX socket file.
/dev/.SRC-unix  Specifies the location for temporary socket files.

Related Information

The `mkssys` command, `rmssys` command.


Istun Command

Purpose

Lists tunnel definition(s).

Syntax

   `istun [-v 4|6 | -t tid_list] -p manual | -a`

Description

Use the `istun` command to list the tunnel definition(s) and their current status. This command can either list the tunnels in the tunnel database or in the active system.

Flags

   `-v`  This flag specifies the IP version. For listing IP version 4 tunnel only, use the value of 4. For listing IP version 6 tunnel only, use the value of 6. If this flag is not used, both the version 4 and version 6 tunnels will be listed.
-t Only list the tunnel definition and its current status for the tunnel whose tunnel ID is in tid_list. If this flag is not used, all the tunnel definitions and their current status will be listed.

-p Selects the type of the tunnel to be listed. Using the -p flag with the value of manual lists manual tunnels only. The -p flag is for listing tunnel definitions in the tunnel database only and thus is mutually exclusive with the -a flag.

-a Lists the tunnels active in the IP Security subsystem.

Related Information
The gentun command, chtun command, imptun command, exptun command, mktun command, and rmtun command.

Isuser Command

Purpose
Displays user account attributes.

Syntax
lsuser [-R load_module] [-c | -f] [-a List] [ ALL | Name [ ,Name ] ... ]

Description
The lsuser command displays the user account attributes. You can use this command to list all attributes of all the system users or all the attributes of specific users. Since there is no default parameter, you must enter the ALL keyword to see the attributes of all the users. By default, the lsuser command displays all user attributes. To view selected attributes, use the -a List flag. If one or more attributes cannot be read, the lsuser command lists as much information as possible.

Note: If you have a Network Information Service (NIS) database installed on your system, some user information may not appear when you use the lsuser command.

By default, the lsuser command lists each user’s attributes on one line. It displays attribute information as Attribute=Value definitions, each separated by a blank space. To list the user attributes in stanza format, use the -f flag. To list the information as colon-separated records, use the -c flag.

You can use the Users application in Web-based System Manager (wsm) to change user characteristics. You could also use the System Management Interface Tool (SMIT) smit lsusers fast path to run this command.

Flags

-a List Lists the attributes to display. The List variable can include any attribute defined in the chuser command and requires a blank space between attributes. If you specify an empty list, only the user names are displayed.

-c Displays the user attributes in colon-separated records, as follows:

# name: attribute1: attribute2: ...
User: value1: value2: ...

-f Displays the output in stanzas, with each stanza identified by a user name. Each Attribute=Value pair is listed on a separate line:

user:
attribute1=value
attribute2=value
attribute3=value

-R Specifies the loadable I&A module used to display the user account attributes.

load_module
Security
Access Control: This command should be a general user program with execute (x) access for all users. Since the attributes are read with the access rights of the user who invokes the command, some users may not be able to access all the information. This command should have the trusted computing base attribute.

Files Accessed:

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>/etc/passwd</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/user</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/user.roles</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/limits</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/environ</td>
</tr>
<tr>
<td>r</td>
<td>/etc/group</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/audit/config</td>
</tr>
</tbody>
</table>

Examples
1. To display the user id and group-related information about the smith account in stanza form, enter:
   
   lsuser -f -a id pgrp groups admgroups smith

   Information similar to the following appears:
   
   smith:
   ID=2457
   pgrp=sysadmin
   groups=sysadmin,finance,staff,accounting
   admgroups=finance,accounting

2. To display the user id, groups, and home directory of smith in colon format, enter:
   
   lsuser -c -a id home groups smith

   Information like the following appears:
   
   # name: ID:home:groups
   smith: 2457:/home/smith:system,finance,staff,accounting

3. To display all the attributes of user smith in the default format, enter:

   lsuser smith

   All the attribute information appears, with each attribute separated by a blank space.

4. To display all the attributes of all the users, enter:

   lsuser ALL

   All the attribute information appears, with each attribute separated by a blank space.

Files

<table>
<thead>
<tr>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/sbin/lsuser</td>
</tr>
<tr>
<td>/etc/passwd</td>
</tr>
<tr>
<td>/etc/security/limits</td>
</tr>
<tr>
<td>/etc/security/user</td>
</tr>
<tr>
<td>/etc/security/user.roles</td>
</tr>
<tr>
<td>/etc/security/environ</td>
</tr>
<tr>
<td>/etc/group</td>
</tr>
</tbody>
</table>

Contains the lsuser command.
Contains basic user information.
Defines resource quotas and limits for each user.
Contains the extended attributes of users.
Contains the administrative role attributes of users.
Contains the environment attributes of users.
Contains basic group attributes.
Related Information
The `chfn` command, `chgrp` command, `chgrp_mem` command, `chsh` command, `chuser` command, `lsgroup` command, `mkgroup` command, `mkuser` command, `passwd` command, `pwdadm` command, `rmgrp` command, `rmuser` command, `setgroups` command, `setserv` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the [AIX 5L Version 5.2 Security Guide](#).

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in [AIX 5L Version 5.2 Web-based System Manager Administration Guide](#).

For more information about administrative roles, refer to [Administrative Roles Overview](#) in [AIX 5L Version 5.2 Security Guide](#).

Isvfs Command

**Purpose**
Lists entries in the `/etc/vfs` file.

**Syntax**
```
isvfs [ -a ] VfsName 
```

**Description**
The `isvfs` command lists entries in the `/etc/vfs` file. You can display information about a specific Virtual File System (VFS) type or all known VFS types.

**Flag**
- `a` Lists all stanzas in the `/etc/vfs` file, including the default stanza.

**Parameter**

- `VfsName` Specifies the name of a virtual file system.

**Examples**
1. To list the vfs entry named newvfs, enter:
   ```bash
   isvfs newvfs
   ```
2. To list all vfs types, enter:
   ```bash
   isvfs -a
   ```

**Files**

- `/etc/vfs` Contains descriptions of virtual file system types.

**Related Information**
The `chvfs` command, `crvfs` command, `rmvfs` command, `mount` command.
**File Systems** in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices* explains file system types, management, structure, and maintenance.

**Mounting Overview** in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices* explains mounting files and directories, mount points, and automatic mounts.

---

**Isvg Command**

**Purpose**
Displays information about volume groups.

**Syntax**
```
Isvg [-L] [O] [n DescriptorPhysicalVolume] [i] [M] [p] VolumeGroup ...
```

**Description**
The `Isvg` command displays information about volume groups. If you use the `VolumeGroup` parameter, only the information for that volume group is displayed. If you do not use the `VolumeGroup` parameter, a list of the names of all defined volume groups is displayed.

When information from the Device Configuration database is unavailable, some of the fields will contain a question mark (?) in place of the missing data. The `Isvg` command attempts to obtain as much information as possible from the description area when the command is given a logical volume identifier.

**Note:** To determine a volume group's major number, use the `ls -al /dev/VGName` command. This command lists the special device file that represents the volume group. The volume group major number is the same as the major device number of the special device file. For example, for a volume group named `halvg`, enter the following command:
```
Isvg /dev/halvg
```

This command returns the following:
```
crw-rw---- 1 root system 52, 0 Aug 27 19:57 /dev/halvg
```

In this example, the volume group major number is 52.

You can use the Volumes application in Web-based System Manager (wsm) to change volume characteristics. You could also use the System Management Interface Tool (SMIT) `smit Isvg` fast path to run this command.

**Flags**

- **-L** Specifies no waiting to obtain a lock on the Volume group.

  **Note:** If the volume group is being changed, using the `-L` flag gives unreliable date.
-p

Lists the following information for each physical volume within the group specified by the `VolumeGroup` parameter:

**Physical volume**
A physical volume within the group.

**PVstate**
State of the physical volume.

**Total PPs**
Total number of physical partitions on the physical volume.

**Free PPs**
Number of free physical partitions on the physical volume.

**Distribution**
The number of physical partitions allocated within each section of the physical volume: outer edge, outer middle, center, inner middle, and inner edge of the physical volume.

-i

Lists the following information for each logical volume within the group specified by the `VolumeGroup` parameter:

**LV**
A logical volume within the volume group.

**Type**
Logical volume type.

**LPs**
Number of logical partitions in the logical volume.

**PPs**
Number of physical partitions used by the logical volume.

**PVs**
Number of physical volumes used by the logical volume.

**Logical volume state**
State of the logical volume. Opened/stale indicates the logical volume is open but contains partitions that are not current. Opened/syncd indicates the logical volume is open and synchronized. Closed indicates the logical volume has not been opened.

**Mount point**
File system mount point for the logical volume, if applicable.

-i

Reads volume group names from standard input.

-M

Lists the following fields for each logical volume on the physical volume:

`PVname:PPnum [LVname: LPnum [:Copynum] [PPstate]]`

**PVname**
Name of the physical volume as specified by the system.

**PPnum**
Physical partition number. Physical partition numbers can range from 1 to 1016.

**LVname**
Name of the logical volume to which the physical partitions are allocated. Logical volume names must be system-wide unique names, and can range from 1 to 64 characters.

**LPnum**
Logical partition number. Logical partition numbers can range from 1 to 64,000.

**Copynum**
Mirror number.

**PPstate**
Only the physical partitions on the physical volume that are not current are shown as stale.

-n **DescriptorPhysicalVolume**

Accesses information from the descriptor area specified by the `DescriptorPhysicalVolume` variable. The information may not be current, since the information accessed with the -n flag has not been validated for the logical volumes. If you do not use the -n flag, the descriptor area from the physical volume that holds the most validated information is accessed, and therefore the information displayed is current. The volume group need not be active when you use this flag.
-o Lists only the active volume groups (those that are varied on). An active volume group is one that is available for use.

Information displayed if you do not specify any flags:

Volume group Name of the volume group. Volume group names must be unique systemwide and can range from 1 to 15 characters.

Volume group state State of the volume group. If the volume group is activated with the `varyonvg` command, the state is either active/complete (indicating all physical volumes are active) or active/partial (indicating some physical volumes are not active). If the volume group is not activated with the `varyonvg` command, the state is inactive.

Permission Access permission: read-only or read-write.

Max LVs Maximum number of logical volumes allowed in the volume group.

LVs Number of logical volumes currently in the volume group.

Open LVs Number of logical volumes within the volume group that are currently open.

Total PVs Total number of physical volumes within the volume group.

Active PVs Number of physical volumes that are currently active.

VG identifier The volume group identifier.

PP size Size of each physical partition.

Total PPs Total number of physical partitions within the volume group.

Free PPs Number of physical partitions not allocated.

Alloc PPs Number of physical partitions currently allocated to logical volumes.

Quorum Number of physical volumes needed for a majority.

VGDS Number of volume group descriptor areas within the volume group.

Auto-on Automatic activation at IPL (yes or no).

Concurrent States whether the volume group is Concurrent Capable or Non-Concurrent Capable and whether a concurrent volume group is Enhanced-Capable or only Concurrent-Capable.

Auto-Concurrent States whether you should autovary the Concurrent Capable volume group in concurrent or non-concurrent mode. For volume groups that are Non-Concurrent Capable, this value defaults to Disabled.

VG Mode The vary on mode of the volume group: Concurrent or Non-Concurrent.

Node ID Node id of this node if volume group is varied on in concurrent node.

Active Nodes Node ids of other concurrent nodes that have this volume group varied on.

Max PPs Per PV Maximum number of physical partitions per physical volume allowed for this volume group.

Max PVs Maximum number of physical volumes allowed in this volume group.

LTG size Logical track group size, in number of kilobytes, of the volume group.

BB POLICY Bad block relocation policy of the volume group.

SNAPSHOT VG Snapshot volume group name if the snapshot volume group is active else snapshot volume group identifier.

PRIMARY VG Original volume group name of a snapshot volume group if the original volume group is active else original volume group identifier.

Examples

1. To display the names of all active volume groups, enter:

   `lsvg -o`

2. To display the names of all volume groups within the system, enter:

   `lsvg`

3. To display information about volume group vg02, enter:

   `lsvg vg02`
The characteristics and status of both the logical and physical partitions of volume group vg02 are displayed.

4. To display the names, characteristics, and status of all the logical volumes in volume group vg02, enter:

`lsvg -l vg02`

**Files**

`/usr/sbin` Contains the directory where the `lsvg` command resides.

**Related Information**

The `chvg` command, `lspv` command, `lsv` command, `varyonvg` command.

[Logical Volume Storage Overview](#) in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices* explains the Logical Volume Manager, physical volumes, logical volumes, volume groups, organization, ensuring data integrity, and allocation characteristics.

For information on installing the Web-based System Manager, see [Chapter 2: Installing Web-based System Manager](#) in *AIX 5L Version 5.2 Web-based System Manager Administration Guide*.

[System Management Interface Tool (SMIT): Overview](#) in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices* explains the structure, main menus, and tasks that are done with SMIT.

**lsvirprt Command**

**Purpose**

Displays the attribute values of a virtual printer.

**Syntax**

`lsvirprt [-q QueueName] [-d DeviceName] [ [-f Format] | -n ] [ [-a AttributeName] | [-s SectionName] ] ... [-i | -D ]`

**Description**

The `lsvirprt` command displays the attribute values for the virtual printer assigned to the `PrintQueueName` and `QueueDeviceName` variables.

The `lsvirprt` command becomes interactive if no flags are specified with the command. A list of print queue names is displayed, and a prompt appears requesting that the desired print queue name be selected. After a valid print queue name is selected, a prompt appears requesting that attribute names be entered. If an attribute name of * (asterisk) is entered, a list of all attributes is displayed.

**Note:** Attribute names for default values of the `qprt` command line flags can be specified by entering the flag letters. For example, to view the default value for the `-w` flag (page width), enter the `w` attribute name. All other attribute names must be 2 characters long.

You can use the Printer Queues application in Web-based System Manager (wsm) to change printer characteristics. You could also use the System Management Interface Tool (SMIT) `smit lsvirprt` fast path to run this command.
Flags

- **a AttributeName**
  Specifies the name of an attribute for which information is to be displayed. The flag cannot be used with the -s flag. The -a flag can be specified many times to list multiple attributes. The AttributeName value can be a single-character name (for example, j), a simple two-character name (for example, ci), or a regular expression that specifies multiple attributes (for example, ^i.*).

- **d QueueDeviceName**
  Specifies the name of the queue device to which the virtual printer is assigned. This flag is optional, but can be specified only if the -q flag is specified.

- **D**
  Displays data streams supported by a given queue and queue device name variable values. The -D flag displays the default data stream first and all other supported data streams in alphabetical order.

- **f Format**
  Specifies the display format for attribute information. Attribute information includes the attribute value, limits field, and attribute description. The Format value is specified in printf format. The -f Format option also supports the following predefined set of position arguments:

  `%1$[\^i.*]s`  
  Message catalog name

  `%2$[\^i.*]d`  
  Message number

  `%3$[\^i.*]s`  
  Attribute name

  `%4$[\^i.*]s`  
  Limits field

  `%5$[\^i.*]s`  
  Attribute value

  `%6$[\^i.*]s`  
  Attribute description

  `%7$sc`  
  Second character of attribute name.

- **i**
  Sets the command to interactive mode. The -q and -d flags must be specified with the -i flag. If values have been assigned to the QueueName and DeviceName variables, the command does not prompt for the queue and device names and accepts attribute names interactively.

- **n**
  Displays only the specified attributes that havenonnull values.

- **s SectionName**
  Specifies a section name in the virtual printer attribute database of the specified queue and queue device. The SectionName values begin with two underscores and contain three characters that identify a section. For example, the name of a section that contains all the flag attributes is __FLG. The -s flag can not be used with the -a flag. This option can be repeated to list multiple attributes. The SectionName variable value can be a regular expression.

- **q PrintQueueName**
  Specifies the name of the print queue to which the virtual printer is assigned. This flag is optional, but can be specified only if the -d flag is specified.

Examples

1. To show the attribute values for the w (default page width) and si (user to receive the "Intervention Required" messages) attributes for the virtual printer assigned to the mypro queue device on the proq print queue, enter:

   `lsvirprt -dmypro -qproq -a \w -a si`

   The output from this command is:

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>\w</td>
<td>COLUMNS per page</td>
<td>136</td>
</tr>
<tr>
<td>si</td>
<td>USERS to get intervention messages</td>
<td></td>
</tr>
</tbody>
</table>
2. To show the same attributes in Example 1, but the be prompted for the flag values, enter:
   lsvirpt

   The output from this command is:
   1   e4039c   @piobe   ibm4039 (PCL Emulation)
   2   e4039s   @piobe   ibm4039 (PostScript)
   3   f32hp4s  jzfile   hplj-4 (PostScript)
   4   hpc14    hp@pc15  hplj-4 (PCL)
   ...

3. To list attributes in a section for header and trailer pipelines for the que queue and the dev device, enter:
   lsvirpt -qque -ddev -s__HTP

   The output from this command is:
   Name          Description                      Value
   sh            Pipeline for Header Page          %Ide/pioburst %F[H]
                 %Idb/H.ascii | %Ide/pioformat
                  -%Idd/%Imm
                  -!%Idf/piof5202 -L! -J!
                  %IsH
   st            Pipeline for Trailer Page        %Ide/pioburst %F[H]
                 %Idb/T.ascii | %Ide/pioformat
                  -%Idd/%Imm
                  -!%Idf/piof5202-L!
                  -t%o%G_1%r%{14}%-%d
                  %IsT

4. To list all the data streams supported for the que queue and the dev device, enter:
   lsvirpt -qque -ddev -D

   The output from this command is:
   a     ASCII
   p     pass-through
   s     PostScript

5. To list names and descriptions of all attributes in a printer attribute database for the que queue and the dev device in a specific format, enter:
   lsvirpt -qque -ddev -a'*.*' -f' %3$5.5s: %6$s\n'

   The output from this command is:
   __FLG: Values That May Be Overridden With Flags
   _A:    stderr returned?
   _E:    Double spacing flag
   _F:    (not used) Font file name
   _H:    Name to Replace Host Name of Burst Page
   ...

6. To list all the sections in a printer attribute data base for the que queue and the dev device in a specific format, enter:
   lsvirpt -qque -ddev -a'__.*' -f' %3$5.5s: %6$s\n'

   The output from this command is:
   __FLG: Values That May Be Overridden With Flags On the Command Line
   __SYS: Other Values Of Interest To the Streams Administrator
   __IDS: Pipelines For Input Data Streams (2 char,1st="i",2nd=data stream name)
   __PFL: Flags Prohibited For Input Data Streams (2 char,1st="I", 2nd=data stream name)
Files

/etc/qconfig  Contains the configuration file.
/usr/sbin/lsvirprt  Contains the lsvirprt command.
/var/spool/lpd/pio/@local/custom/*  Contains virtual printer attribute files.
/var/spool/lpd/pio/@local/ddi/*  Contains the digested virtual printer attribute files.

Related Information

The chvirprt command, mkvirprt command, rmvirprt command.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

Configuring a Printer without Adding a Queue in AIX 5L Version 5.2 Guide to Printers and Printing.


Adding a Printer Using the Printer Colon File in AIX 5L Version 5.2 Guide to Printers and Printing.

Isvmode Command

Purpose

Display the current video mode of the X server.

Note: This command is usable only while the X server is running.

Syntax

Isvmode

Description

The Isvmode command displays the current output device and viewport size used by the X server.

Security

Access Control: Any User

Auditing Events: None

Exit Status

The following exit values are returned:

0  Successful completion.
Examples
To display the current video mode of the X server.

lsvmode

Something similar to the following displays:
Current video mode information
Logical screen size   [1024x768]
Viewport size        [640x480]
Vertical sync. (Hz)   [60]
Active output device  [LCD][CRT]

Files
/usr/bin/X11/lsvmode       Contains the lsvnode command.

Related Information
The chvmode command.

lsvsd Command

Purpose
lsvsd – Displays configured virtual shared disks and their characteristics.

Syntax
lsvsd [−l | −s[ vsd_name...]] | [−l]

Description
The lsvsd command displays information about virtual shared disks currently configured on the node on which the command is run. If a list of virtual shared disks follows the flags, information about those virtual shared disks is displayed. lsvsd with no arguments or flags lists the names of all the virtual shared disks currently configured on the node.

The lsvsd command displays information about both the configuration and the usage of a virtual shared disk.

You can use the System Management Interface Tool (SMIT) to run the lsvsd command. To use SMIT, enter:
smit vsd_mgmt

and select the Show All Managed Virtual Shared Disk Characteristics option.

Flags
−l       Lists the name of the virtual shared disk, the minor number, the state, the current server node number, and, at the server only, the major and minor number of the logical volume. (This flag is lowercase l, as in list.)

The state field can have one of the following values:
STP Stopped
SUS Suspended
ACT Active

An asterisk (*) in front of any of these values indicates that the virtual shared disk has been fenced from this node.

This flag is not compatible with the -s flag.

The server_list of the virtual shared disk is listed.

-s
Lists usage statistics about the virtual shared disks. It lists the number of local logical read and write operations, the number of remote logical read and write operations, the number of client logical read and write operations, the number of physical reads and writes, and the number of 512-byte blocks read and written. The number of blocks read and written is cumulative, so issue ctlvsd -V to reset this count before measuring it.

The local logical operations are requests which were made by a process executing at the local node, whereas the remote logical operations were made by a process executing on a remote node. Client operations are those local logical requests that cannot be satisfied locally, and have to be sent to a remote node. Physical operations are those server operations which must be passed to the underlying disk device.

This flag is not compatible with the -I flag.

-i
Lists the “node to IP address” map that is currently used by the virtual shared disk driver.

Parameters

vsd_name Specifies a virtual shared disk. This parameter is valid only with the -I and -s flags.

Security

You must be in the AIX bin group to run this command.

Exit Status

0 Indicates the successful completion of the command.

nonzero Indicates that an error occurred.

Restrictions

You must issue this command from a node that is online in the peer domain. To bring a peer domain online, use the startrpdomain command. To bring a particular node online in an existing peer domain, use the startrpnode command. For more information on creating and administering an RSCT peer domain, refer to the RSCT: Administration Guide.

Standard Output

For the following command:
createvsd -n 1/2:hdisk13/ -s 1024 -g testvg -v testvsd -T 16

The messages to standard output will be similar to:
createvsd: calls Getopts.
createvsd: parsing node list.
createvsd: creates task tables.
createvsd: calls checkclvm.perl on the nodes c164n11.ppd.pok.ibm.com
createvsd: calls domkvglv.perl.
OK:1:mkvg -f -y testvg -s 16 hdisk13
OK:1:mklv -a c -y lvtestvsd1n1 -e x testvg 64 hdisk13
It took about 8 seconds in mkkvglv.
createvsd: calls dovaryoffvg.perl testvg on the primary node c164n11.ppd.pok.ibm.com
OK:1:chvg -a m testvg
OK:1:varyoffvg testvg
createvsd: calls doimportvg.perl testvg on the nodes c164n12.ppd.pok.ibm.com with 000048186b991a6f
importvg : testvg
importvg : OK:2:importvg -y testvg hdisk5
importvg : OK:2:chvg -a n testvg
importvg : timestamp 2 testvg 3e036cb33403c8c8
importvg : OK:2:varyoffvg testvg
importvg : It took about 10 seconds.
It took about 12 seconds in importvg.
createvsd: calls vsdvg.
OK:1:vsdvg -g testvgn1b2 testvg
It took about 12 seconds in vsdvg.
createvsd: calls doimportvg.perl testvg on pri nodes c164n11.ppd.pok.ibm.com
OK:1:varyonvg testvg
createvsd: calls defvsd.
OK:1:defvsd lvtestvsd1n1 testvgn1b2 testvsd1n1
It took about 5 seconds in defvsd.

Examples
1. To list all virtual shared disks in the system, enter:

   lsvsd

   The system displays a message similar to the following:

   vsd00

   vsd01

   .

   .

2. To list virtual shared disks and their characteristics, enter:

   lsvsd -l

   The system displays a message similar to the following:

   minor state server lv_major lv_minor vsd_name size (MB)
   83 STP -1 0 0 vsdn80v3 20
   84 STP -1 0 0 vsdn80v4 16

3. To list statistics about virtual shared disks and precede the column output with a header, enter:

   lsvsd -s

   The system displays a message similar to the following:

   lc-rd lc-wt rm-rd rm-wt c-rd c-wt p-rd p-wt br bw vsd_name
   84 84 2858 169 0 0 348 253 164 184 vsd.vsd1
   0 0 0 0 0 0 0 0 0 0 vsd.r101
   0 0 0 0 0 0 0 0 0 0 vsd.r102

The following table spells out the names of the headers used in the displays for the -l and -s options:

Header               Meaning
minor                Virtual shared disk minor number
state                State of this virtual shared disk: active, stopped, suspended
server               Primary node for this virtual shared disk
lv major             Logical volume major number
lv minor
   Logical volume minor number

vsd_name
   Name of this virtual shared disk

lc-rd  Local logical reads
lc-wt  Local logical writes
rm-rd  Remote logical reads
rm-wt  Remote logical writes
c-rd   Client logical reads
c-wt   Client logical writes
p-rd   Physical reads
p-wt   Physical writes
br     Blocks read
bw     Blocks written

Location
   /opt/rsct/vsd/bin/lsvsd

Related Information
   Commands: cfgvsd, preparevsd, resumevsd, startvsd, stopvsd, suspendvsd, ucfgvsd

lswlmcconf Command

Purpose
   Lists Workload Manager (WLM) configurations.

Syntax
   lswlmcconf [ -r | -s | -c | -d Config ] [ -l ] [ -t TimeSpec ]

Description
   The lswlmcconf command lists by default all WLM configurations and, using its flags, it is able to do the following:
   • Tell which is the current configuration or set name.
   • Give the list of all existing regular WLM configurations.
   • Give the list of all existing WLM configuration sets.
   • Tell which configuration of a set is (or would be) applicable currently (or at some time of the week).
   • Tell the type of a configuration.

Flags
   -c          Restricts the displayed configurations to the current configuration or set.
   -d Config   Restricts the displayed configurations to the Config configuration or set.
   -l          Modifies the way dates are displayed for configuration sets (ineffective for regular configurations). Sets are displayed with their currently applicable regular configuration, in the form confset/config.
   -r          Restricts the displayed configurations to regular configurations only.
Restricts the displayed configurations to configuration sets only.

-t TimeSpec

Uses TimeSpec instead of the current time to display applicable regular configuration of sets. TimeSpec consists in the day of the week (0 for Sunday to 6 for Saturday) and the time of the day in 24 hours format separated with a comma, in a form similar to time ranges as described in the confsetcntrl command. For example, to know which configuration would apply on Mondays at noon, use -t 1,12:00.

Note: The -t flag is only effective with -l flag.

Examples

The following examples demonstrate how to display, change, and use WLM configurations using the lswlmconf command, the confsetcntrl command, the wlmcheck command, and the wlmcntl command.

1. To find the WLM configurations, type:

   lswlmconf

   The output to this command might look similar to the following:

   standard
template
fvtrules
fvttlimits
fvtrregul
fvtdfct
fvtsynt
fvttthreads

2. To show the current WLM configuration, type:

   lswlmconf -c

   The output might look similar to the following:

   fvtlimits

3. To show configuration sets, use the lswlmconf with the -s flag as follows:

   lswlmconf -s

   Since this example configuration contains no configuration sets, this command produces a message indicating that no matching configuration was found.

4. In order to create a configuration set using "standard" as the default configuration, type:

   confsetcntrl -C confset1 standard

5. Now, use the lswlmconf command to show the new configuration set, as follows:

   lswlmconf -s

   The command now produces the following output:

   confset1

6. In order to use the "fvttlimits" configuration for "confset1" on week days (Monday through Friday) by specifying a time range, type:

   confsetcntrl -d confset1 -a fvttlimits 1-5

7. You might want this configuration only in the morning. You cannot change a time range. Instead you must remove the time range and then create a new time range.

   First, remove the old time range, as follows (confsetcntrl accepts day names, as reported by locale day or locale abday commands):

   confsetcntrl -d confset1 -r fvttlimits monday-friday

   Then create the new time range, as follows:

   confsetcntrl -d confset1 -a fvttlimits 1-5,8:00-12:00

8. In order to add another time range for using the "fvtrregul" configuration on Sundays, type:
In order to display configuration set "confset1", type:
confsetcntrl -d confset1

In this example, this command produces the following output:

```
fvtlimits:
  time = "1-5,8:00-12:00"

fvtregul:
  time = "0"

standard:
  time = "-"
```

In order to create a configuration set called "confset2" using "template" as the default configuration, type:
confsetcntrl -C confset2 template

In order change "confset2" so it will use the configuration "fvtsynt" every night, type:
confsetcntrl -d confset2 -a fvtsynt 18:00-10:00

In order to display the list of regular configurations, type:
lswlmcnf -r

In this example, this produces the following output, (which demonstrates that in this example the list of regular configurations has not changed):

```
standard
template
fvtrules
fvtlimits
fvtregul
fvtdefct
fvtsynt
fvtthreads
```

However, as expected, the list of configurations sets in this example has changed, as shown by the following command:
lswlmcnf -s

This command produces the following output in this example:

```
confset1
confset2
```

In order to show which configuration would be currently active when that the date command reports the current time as "Tue Jul 16 18:55:10 EET 2002" with configuration set "confset2", type:
lswlmcnf -d confset2 -l

In this example, this command produces the following output:

```
confset2/fvtsynt
```

You can also show which configurations would be active at another time. To show which configurations would be active on Sunday at 9:00am, type:
lswlmcnf -l -t 0,9:00

This command produces the following output in this example:

```
standard
template
fvtrules
fvtlimits
```

confsetcntrl -d confset1 -a fvtsyregul 0
In order to display this information only for configuration sets, type:
```
lswlmcnf -s -l -t 0,9:00
```
This produces the following output in this example:
```
confset1/fvtregul
confset2/fvtsynt
```

13. In order to remove configuration set "confset2", type:
```
confsetcntrl -D confset2
```

```
lswlmcnf -s
```
now produces the following output in this example:
```
confset1
```

14. In order to check configuration set "confset1", use the wlmcheck command as follows:
```
wlmcheck -d confset1
```
In this example, this produces the following output:
```
WLM is not running.
Checking classes and rules for 'confset1' configuration...
```
```
confset1/system
confset1/default
confset1/shared
confset1/login
confset1/system
confset1/default
confset1/shared
standard/system
standard/default
standard/shared
```

15. In order to start using configuration set "confset1" used in this example, type:
```
wlcntrl -a -d confset1
```
The command `lswlmcnf -c` now produces the following output:
```
confset1
```
The command `lswlmcnf -cl`, which shows the active regular configuration, now produces the following output:
```
confset1/standard
```

**Files**
The configurations or sets files are subdirectories of `/etc/wlm`.

**Related Information**
The `wlcntrl` command, `confsetcntrl` command.

**lvmstat Command**

**Purpose**
Reports input/output statistics for logical partitions, logical volumes and volume groups.
Syntax

`lvmstat [-v] Name [-e] [{-F} | {-C} | {-c} Count] [-s] [ Interval [ Iterations ] ]`

Description

The `lvmstat` command generates reports that can be used to change logical volume configuration to better balance the input/output load between physical disks.

By default, the statistics collection is not enabled in the system. You must use the `-e` flag to enable this feature for the logical volume or volume group in question. Enabling the statistics collection for a volume group enables for all the logical volume in that volume group.

The first report generated by `lvmstat` provides statistics concerning the time since the system was booted. Each subsequent report covers the time since the previous report. All statistics are reported each time `lvmstat` runs. The report consists of a header row followed by a line of statistics for each logical partition or logical volume depending on the flags specified.

If the `-I` flag is specified, `Name` is the logical volume name, and the statistics are for the physical partitions of this logical volume. The mirror copies of the logical partitions are considered individually for the statistics reporting. They are listed in descending order of number of i/os (iocnt) to the partition.

The `Interval` parameter specifies the amount of time, in seconds, between each report. The first report contains statistics for the time since the volume group startup, `varyonvg`. Each subsequent report contains statistics collected during the interval since the previous report. If the `Count` parameter is specified, only the top `Count` lines of the report are generated. For a logical volume if `Count` is 10, only the 10 busiest partitions are identified. If the `Iterations` parameter is specified in conjunction with the `Interval` parameter, then only that many iterations are run. If no `Iterations` parameter is specified, `lvmstat` generates reports continuously. If `Interval` is used to run lvmstat more than once, no reports are printed if the statistics did not change since the last run. A single period . (period) is printed instead.

The `lvmstat` command is useful in determining whether a physical volume is becoming a hindrance to performance by identifying the busiest physical partitions for a logical volume.

**Note:** The `lvmstat` commands reports I/O statistics of the local node only.

Reports

The `lvmstat` command generates two types of reports, per partition statistics in a logical volume and per logical volume statistics in a volume group. The reports has the following format:

<table>
<thead>
<tr>
<th>Column</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Log_part</td>
<td>Logical partition number</td>
</tr>
<tr>
<td>mirror#Log_part</td>
<td>Mirror copy number of the logical partition</td>
</tr>
<tr>
<td>iocntLog_part</td>
<td>Number of read and write requests</td>
</tr>
<tr>
<td>Kb_readLog_part</td>
<td>The total number of kilobytes read</td>
</tr>
<tr>
<td>Kb_wrtnLog_part</td>
<td>The total number of kilobytes written</td>
</tr>
<tr>
<td>KbpsLog_part</td>
<td>The amount of data transferred in kilobytes per second</td>
</tr>
</tbody>
</table>

Flags

- `-c Count` Prints only the specified number of lines of statistics.
- `-C` Causes the counters that keep track of the iocnt, Kb_read and Kb_wrtn be cleared for the specified logical volume/volume group.
- `-d` Specifies that statistics collection should be disabled for the logical volume/volume group in question.
- `-e` Specifies that statistics collection should be enabled for the logical volume/volume group in question.
- `-F` Causes the statistics to be printed colon-separated.
Specifies the name of the stanza to list.
-s Suppresses the header from the subsequent reports when *Interval* is used.
-v Specifies that the *Name* specified is the name of the volume group.

**Security**

To use *lvmstat*, you must have root user authority.

**Examples**

1. To enable the statistics collection for volume group datavg (all the LVs in datavg are enabled), type:
   
   `lvmstat -v datavg -e`

2. To display the history of all the partitions of logical volume hd2, type:
   
   `lvmstat -l hd2`

3. To display the history of top five logical volumes of volume group uservg, type:
   
   `lvmstat -v uservg -c 5`

4. To display a continuous report at two second intervals for the logical volume ramlv, type:
   
   `lvmstat -l ramlv 2`

5. To display six reports at two second intervals for the volume group rootvg, type:
   
   `lvmstat -v rootvg 2 6`

6. To reset the counters for statistics for all the logical volumes in the volume group uservg, type:
   
   `lvmstat -v uservg -C`

7. To disable statistics collection for datalv, type:
   
   `lvmstat -l datalv -d`

**Files**

`/usr/sbin/lvmstat` Contains the *lvmstat* command.

**Related Information**

The `knlist` subroutine.

The `/dev/kmem` special file.

**m4 Command**

**Purpose**

Preprocesses files, expanding macro definitions.

**Syntax**

```
m4 [ -e ] [ -l ] [ -s ] [ -B Number ] [ -D Name [ =Value ] ] ... [ -H Number ] [ -I Directory ] [ -S Number ] [ -T Number ] [ -U Name ] ... [ File ... ]
```

**Description**

The *m4* command is a macro processor used as a preprocessor for C and other languages. You can use it to process built-in macros or user-defined macros.

Each *File* parameter is processed in order. If you do not specify a *File* parameter or if you specify the - (dash) as a file name, the *m4* command reads standard input. It writes the processed macros to standard output. Macro calls follow the form:
macroname{argument . . . }

The left parenthesis must immediately follow macroname. If the left parenthesis does not follow the name of a defined macro, the m4 command reads it as a macro call with no arguments. Macro names consist of ASCII alphabetic letters, digits, and the _ (underscore) character. Extended characters are not allowed in macro names. The first character cannot be a digit.

While collecting arguments, the m4 command ignores unquoted leading blanks, tabs, and new-line characters. Use single quotation marks to quote strings. The value of a quoted string is the string with the quotation marks stripped off.

When the m4 command recognizes a macro, it collects arguments by searching for a matching right parenthesis. If you supply fewer arguments than appear in the macro definition, the m4 command considers the trailing arguments in the definition to be null. Macro evaluation proceeds normally during the collection of the arguments. All commas or right parentheses within the value of a nested call are translated literally; they do not need an escape character or quotation marks. After collecting arguments, the m4 command pushes the value of the macro back onto the input stream and scans again.

**Built-in Macros**

The m4 command makes available the following built-in macros. You may redefine them, but you will lose the original meaning. The values of these macros are null unless otherwise stated:

- **define(Name,NewName)**  
  Replaces the macro Name with the value of NewName. The NewName string can take the form $n . . . (where n is a digit). In this case, each occurrence of n in the replacement text is replaced by the n'th argument of Name. $0 is the name of the macro. The null string replaces missing arguments. The number of arguments replaces $#. A comma-separated list of all arguments replaces $*. $@ acts like $*, but each argument is quoted with the current quotation character (see changequote).

- **undefine(Name)**  
  Removes the definition of Name.

- **defn(Name . . .)**  
  Returns the quoted definition of Name.

- **pushdef(Name, NewName)**  
  Redefines Name with NewName as in define, but saves any previous definition.

- **popdef(Name . . .)**  
  Removes the current definition of Name and returns to the previous definition, if one existed.

- **ifdef(Name,True,[False])**  
  Returns the value of True only if Name is defined and is not defined to be 0, otherwise returns False. If you do not supply False, its value is null.

- **shift(Argument . . .)**  
  Returns all but the first argument. The other arguments are quoted and pushed back with commas in between. The quoting nullifies the effect of the extra scan that is subsequently performed.

- **changequote(L,R)**  
  Changes quote symbols to L and R. The symbols can be up to 5 bytes long. changequote without arguments restores the original values (" ").

- **changecom(L,R)**  
  Changes left and right comment markers from the default # and new-line character to L and R. With no arguments, the comment mechanism is disabled. With one argument, the left marker becomes the parameter and the right marker becomes a new-line character. With two arguments, both markers are affected. Comment markers can be up to 5 bytes long.

- **divert(Number)**  
  Changes the current output stream to stream Number. There are 10 output streams, numbered 0-9. The final output is the concatenation of the streams in numerical order. Initially, stream 0 is the current stream. The m4 command discards output diverted to a stream other than 0-9.

- **undivert(Number . . .)**  
  Causes immediate output of text from the specified diverisions (or all diversions if there is no argument). Text may be undiverted into another diversion. Undiverting discards the diverted text.

- **divnum**  
  Returns the value of the current output stream.

- **dnl**  
  Reads and discards characters up to and including the next new-line character.
ifelse([String1, String2, True, False] ... ) If String1 and String2 are the same then the value is True. If they are not and if there are more than four arguments, the m4 command repeats the process with the additional arguments (4, 5, 6, and 7). Otherwise, the value is either False or null if you provide no value for False.

cr(N) Returns the value of its argument incremented by 1.

dcr(N) Returns the value of its argument decreased by 1.

eval(Expr[, N1[, N2]]) Evaluates its first argument as an arithmetic expression, using 32-bit arithmetic. The operators you can use are +, -, *, /, ^ (exponentiation), bitwise & , | , ^, and ^ relational functions. Octal and hex numbers can be specified as in C. N1 specifies the radix for the result of the expression. The default radix is 10. The optional N2 specifies the minimum number of digits in the result.

ten(S) Returns the number of bytes in String.

dlen(S) Returns the number of displayable characters in String; that is, two-byte extended characters are counted as one displayable character.

index(S1, S2) Returns the position in the String1 string where the String2 string begins (zero origin), or -1 if the second parameter does not occur.

substr(S, I, N) Returns a substring of String. The beginning of the substring is selected with I, and N indicates the length of the substring. Without N, the substring includes everything to the end of the first string.

translit(S, F, T) Transliterates the characters in String from the set given by F to the set given by T. No abbreviations are permitted. Two-byte extended characters are correctly mapped into the corresponding replacement characters.

include(F) Returns the contents of File or displays an error message if it cannot access the file.

sinclude(F) Returns the contents of File, but it gives no error message if File is inaccessible.

syscmd(C) Runs the Command. No value is returned.

gsysal Returns the return code from the last call to syscmd.

maketemp(... nnn ... ) Replaces nnn in its argument with the current process ID number.

m4exit(V) Exits from m4 immediately, returning the specified exit value (the default is 0).

m4wrap(L) Runs LastMacro after reading the end-of-file character. For example, m4wrap `cleanup ()` runs the cleanup macro at the end of m4.

errprint(M) Includes Message on the diagnostic output file.

dumpdef([N ... ]) Writes to standard output the current names and definitions for the named items or for all if no arguments are provided.

traceon(M) Turns on tracing for Macro. If none is named, tracing is turned on for all macros.

traceoff(M ... ) Turns off trace globally and for any Macro specified. Macros specifically traced by traceon can be untraced only by specific calls to traceoff.

Flags

-B N Makes the N variable the size of the push-back and parameter collection buffers (the default is 4096).

-e Operates interactively. Interrupts are ignored and the output is not buffered.

-H N Makes the N variable the size of the symbol table hash array (the default is 199). The size must be a prime number.

-I D (Uppercase i) Searches the Directory variable first, then searches the directories on the standard list for include (built-in macro) files with names that do not begin with a i (slash).

-l (Lowercase L) Enables line-numbering output for the assembler (.xline ...).

-s Enables the line-sync output for the C preprocessor (#line ...).

-S N Makes the N variable the size of the call stack (the default is 100 slots). Macros take three slots, and non-macro arguments take one.

-T N Makes the N variable the size of the token buffer (the default is 512 bytes).
The preceding flags must appear before any file names and before any -D or -U flags.

- 

- D  Name[]=Value] Defines the Name variable as the Value variable. If the Value variable is not specified, the Name variable becomes null.

- U  Name Undefines a the Name variable previously defined with the -D flag.

Exit Status
This command returns the following exit values:

0 Successful completion.
>0 An error occurred.

If the m4exit macro is used, the exit value can be specified by the input file.

Examples
To preprocess a C language program with the m4 command and compile it, enter:

```bash
m4 prog.m4 > prog.c
c
```

Files

/usr/ccs/bin/m4 Contains the m4 command.

Related Information

The m4 Macro Processor Overview in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.


The as command, cc command, cpp command.

mach Command

Purpose
Displays the processor type of the current host.

Syntax
mach

Description
The mach command displays the architecture of the system processor.

Exit Status

0 The command completed successfully.
>0 An error occurred.
Examples
1. To display the processor type of the current host use the mach command in the following way:
   mach

Files
/usr/bin/mach
Contains the System V mach command.

Related Information
The uname command.

machstat Command

Purpose
Reports the value of the first 4 bits of the power status register.

Syntax
machstat -p

Description
The machstat command returns the value of a status register. There is no standard output or error.

Flags
-p Displays the first 4 bits of the power status register.

Exit Status
The machstat command returns a value of 255 if an error occurs. Otherwise it returns the value of the register.

Security
Access Control: root only

Examples
To see the current value of the power status register, enter:
machstat -p
echo $?

Files
/etc/rc.powerfail Shuts down a system when a power failure is detected

Related Information
The rc.powerfail command.
macref Command

Purpose
Produces a cross-reference listing of macro files.

Syntax
macref [ [ -n ] ] [ -s ] [ -t ] [ - ] [ File ... ]

Description
The macref command reads the named English-language files (which are assumed to be nroff or troff command input) and produces a cross-referenced listing of the symbols in the input.

The default output is a list of the symbols found in the input, each accompanied by a list of all references to that symbol. The macref command lists the symbols alphabetically in the left column, with references following to the right. Each reference is given in the following form:

```
[ \[( NMName ) \]
 MName ]
 Type LNumber
 [ "\]"
```

Generated names are listed under the artificial symbol name ~sym.

Input Parameters

File Specifies the nroff or troff file from which the macref command produces output containing a list cross-referencing macros.

Output Parameters

NMName The name of the macro within which MName is defined.
MName The name of the macro within which the reference occurs. This field is not present if the reference occurs outside a macro.
Type The type associated, by context, with this occurrence of the symbol. The types can be the following:
- r Request
- m Macro
- d Diversion
- s String
- n Number register
- p Parameter. For instance, \$x is a parameter reference to x.

Note: Parameters are never modified, and the only valid parameter symbol names are 1, 2, \ldots, 9.
LNumber The line number on which the reference occurred.
# This reference modifies the value of the symbol.

Flags

- -n Causes one line to be printed for each reference to a symbol.
-s Causes symbol-use statistics to be printed.
-t Causes a macro table of contents to be printed.

The flags can be grouped behind one - (minus sign). Use a — (dash) to delimit the end of flags.

Note: The macref command does not accept - as standard input.

Files

/tmp/macref.tXXXXXX Contains a temporary file.
/tmp/macref.sXXXXXX Contains a temporary file.
/tmp/macref.cXXXXXX Contains a temporary file.

Related Information

The mm command, mmt command, mv command, nroff command, troff command.

The man macro package, mm macro package, mv macro package.

mail, Mail, or mailx Command

Purpose
Sends and receives mail.

Syntax

To Read Incoming Mail
mail -e

mail [-f -dHnN] [-F] [- FileName ]

mail [-dHnN] [-F] [-u UserID ]

To Send Mail
mail [-s Subject] [-c Address(es)] [-dinNv] Address

Description
The mail command invokes the mail utility, enabling you to:
• Read incoming mail.
• Send mail.

In addition, you can use the available options and subcommands to customize the way you send and receive mail.

The mail command operates on two types of mailboxes, the system mailbox and the personal mailbox.

Incoming mail is stored in the system mailbox. By default, a user’s system mailbox is a file located in the /var/spool/mail directory. The mailbox file is named after the userID. For example, if your userID is jeanne, then your system mailbox is /var/spool/mail/jeanne.

By default, when a user has read, deleted, or saved all the mail in their system mailbox, the mailbox is deleted. To prevent the mailbox from being deleted, use the set subcommand to set the keep option.
In addition to the system mailbox, there is the user’s personal mailbox. As messages are read, if they are not deleted or saved to a file, they will be marked to be moved to the personal mailbox. The personal mailbox, by default, is $HOME/mbox. For example, if your home directory is /home/lance, then /home/lance/mbox is your personal mailbox. The messages remain in your personal mailbox until you move them to a folder or delete them.

Folders provide a way to save messages in an organized fashion. You can create as many folders as you need. Name each folder with a name that pertains to the subject matter of the messages it contains.

**Note:** Results can be unpredictable when running multiple instances of the mail command on one mailbox.

### Examining the Contents of Your Mailbox

To process your mail, type `mail` at the system prompt. The Mail program displays a one-line entry for each piece of mail in your system mailbox:

```
Mail [5.2 UCB] [AIX 4.1] Type ? for help.
"/var/spool/mail/lance": 2 messages 2 new
>N 1 karen Thu Sep 17 14:36 13/359 "Dept Meeting"
N 2 lance@zeus Thu Sep 17 15:06 10/350 "Delay"
N 3 karen Thu Sep 17 14:36 13/359 "Meeting Cancel"
```

The current message is marked by a > at the beginning of a line in the header summary.

Each one-line entry displays the following fields:

- **status** Indicates the current class of a piece of mail. The status can be any of the following:
  - **N** A new message
  - **P** A message to be preserved in system mailbox.
  - **U** An unread message. An unread message is a message that was listed in the mailbox last time you invoked the Mail program, but whose contents you did not examine.
  - ***** A message that was saved or written to a file or folder.

  A message without a status indicates that the message has been read but has not been deleted or saved.

- **number** Identifies the numerical order of the message.
- **sender** Identifies the address of the person who sent the mail.
- **date** Specifies the date the message was received.
- **size** Defines the number of lines and characters contained in the letter (this includes the header).
- **subject** Identifies the subject of the message.

Finally, following the list of mail, the Mail program displays the mailbox prompt, which by default is `?`, to indicate that it is waiting for input.

### Flags

- **-c Address(es)** Specifies the list of users to which a copy of the message is sent. You can specify one or more addresses. When specifying more than one address, the list of addresses must be in (" ") quotes.

- **-e** Tests for the presence of mail in the system mailbox. The mail utility will write nothing and exit with a successful return code if there is mail to read.

- **-f FileName** Reads messages from the named file. If a file operand is not specified, then reads messages from mbox. When you quit from reading the messages, undeleted messages are written back to this file.

- **-F** Records the message in a file named after the recipient. The name is the portion of the address found first on the To: line in the mail header. Overrides the record variable if set.
---

**-H**  
Wants a header summary only.

**-i**  
Causes tty interrupt signals to be ignored.

**-n**  
Inhibits reading the /usr/share/lib/Mail.rc file.

**-N**  
Suppresses the initial printing of headers.

**-s **  
Specifies a subject for a message to be created.

**-u User ID**  
Specifies an abbreviated equivalent of doing mail -f /var/spool/mail/User ID. Starts the Mail program for a specified user's mailbox. You must have access permission to the specified mailbox.

**-v**  
Puts the Mail program into verbose mode. Displays the details of delivery on the user's terminal.

**Environmental Variables**

The following environment variables affect the execution of mail:

**DEAD**  
Pathname of the file in which to save partial messages in case of interrupts or delivery errors.

**EDITOR**  
Pathname of the editor to use when the edit or -e command is used.

**HOME**  
Pathname of the user's home directory.

**LISTER**  
String representing the command for writing the contents of the folder directory to standard output when the folders command is given. Any string acceptable as a command_string operand to the sh -c command is valid. If this variable is null or not set, the output command will be Is. The default value is unset.

**MAILBOX**  
Specifies the location of the system mailbox for the mail command. The MAILBOX value is where the mail command searches for mail messages. The system default value if the MAILBOX environment variable is not specified is the /var/spool/mail directory.

**MAILRC**  
Pathname of your personal startup file. The default is $HOME/.mailrc.

**MBOX**  
Pathname of your personal mailbox where messages are saved from the system mailbox that have been read. The exit command overrides this function, as will saving the message explicitly in another file. The default is $HOME/mbox.

**PAGER**  
String representing an output filtering or pagination command for writing the output to the terminal. Any string acceptable as a command_string operand to the sh -c command is valid. When standard output is a terminal device, the message output will be piped through the command if the mail internal variable crt is set to a value less than the number of lines in the message. If the PAGER variable is null or not set, the paginot is the pg shell command.

**SHELL**  
Pathname of a preferred command interpreter.

**VISUAL**  
Pathname of a utility to invoke when the visual command or -v command-escape is used. If this variable is not set, the full screen editor will be vi.

**Internal Variables in Mail**

**allnet**  
Treats all network names, whose login name components match, identically. Causes the msglist message specifications to behave similarly. The default is noallnet.

**append**  
Adds the message saved in your mailbox to the end, rather than the beginning, of the $HOME/mbox file. The default is noappend.

**ask, asksub**  
Prompts for the subject of each message if not specified on the command line with the -s option. If you do not wish to create a subject field, press Enter at the prompt. It is not possible to set both ask and noasksub, or noask and ask sub. The default is asksub.

**askbcc**  
Prompts for the addresses of people to add to the blind copy list. If you do not wish to send blind copies, press Enter at the prompt.

**askcc**  
Prompts for the addresses of people who should receive copies of the message. If you do not wish to send copies, press Enter at the prompt.

**autoprint**  
Sets the delete subcommand to delete the current message and display the next message.

---

Alphabetical Listing of Commands  421
crt

Specifies the minimum number of lines that a message must contain before any output filtering or pagination is used when the message is displayed.

deb ug

Displays debugging information. Messages are not sent while in debug mode. This is the same as specifying the -d flag on the command line.

dot

Interprets a period entered on a line by itself as the end of a message you are sending.

e scape=c

Sets the command escape character to be the character c. By default the command escape character is ~ (tilde).

Replyall, flipr

Reverses the meanings of the Respond and respond or Reply and reply commands. The default is noflip.

folder=directory

The directory name in which to store mail folders. After the directory is defined, you can use the + (plus sign) notation to refer to it when using the FileName parameter with mailbox subcommands.

header

Enables writing of the header summary when entering mail in receive mode. The default is header.

hold

Holds messages that you have read but have not deleted or saved in the system mailbox instead of in your personal mailbox. The default is nohold.

ignore

Ignores interrupts while entering messages. Echoes interrupts as @ (at) characters.

ignoreeof

Sets the mail command to refuse the Ctrl+D key sequence as the end of a message. Input can be terminated only by entering a period . (period) on a line by itself or by the ~. command escape. The default is noignoreeof.

indentprefix=string

A string that will be prefixed to each line that is inserted into the message by the ~m command escape. This variable defaults to one tab character.

keep

When a system mailbox, secondary mailbox, or mbox is empty, truncate it to zero length instead of removing it. The default is nokeep.

keepsave

Keep messages that have been saved with the (s)ave or (w)rite subcommands in the system mailbox instead of deleting them. The default is nokeepsave.

metoo

Includes the sender in the alias expansion if sender’s name is part of the alias. By default, expanding the alias removes the sender.

onehop

When responding to a message that was originally sent to several recipients, the other recipient addresses are usually forced to be relative to the originating author’s machine for the response. This flag disables alteration of the recipient’s addresses, improving efficiency in a network where all machines can send directly to all other machines (that is, one hop away). The default is noonehop.

outfolder

Causes the files used to record outgoing messages to be located in the directory specified by the folder variable unless the pathname is absolute. The default is nooutfolder. See the record and folder variables.

page

Insert a form-feed after each message sent through the pipe created by the pipe command. The default is nopage.

prompt=string

Set the command-mode prompt to string. If string is null or if noprompt is set, no prompting will occur. The default is to prompt with the “?” string.

quiet

Refrain from writing the opening message and version when entering mail. The default is noquiet.

record=file

Defines a file in which to record all outgoing mail. The default is norecord.

save

Enables saving of messages in the dead.letter file on interrupt or delivery error. The default is save.

screen=number

Sets the number of lines in a screenful of headers for the headers and z commands.

sendmail=shell_command

Alternative command for delivering messages.

sendwait

Wait for the background mailer to finish before returning. The default is nosendwait.
showto

When the sender of the message was the user who is invoking mail, write the information from the To: line instead of the From: line in the header summary. The default is nohowto.

sign=string

Inserts string into the text of a message when the ~a command escape is given. The default is nosign. The character sequences /t and /n are recognized in the string as tab and newline characters, respectively.

Sign=string

Inserts string into the text of a message when the ~A command escape is given. The default is noSign.

toplines=number

Number of lines displayed by the top subcommand.

verbose

Displays the actual delivery of messages on the terminal. This is the same as specifying the -v flag on the command line.

Setting Environment Variables

The Bourne shell (bsh command) uses and checks the following variables. These variables can be set in $HOME/.profile.

MAIL

Specifies the location and name of the user’s system mailbox that is checked by the Bourne shell to determine whether or not you have mail. If the system mailbox is not empty, the Bourne shell sends a message that you have new mail. The Bourne shell checks the system mailbox periodically based on the value of the MAILCHECK environment variable.

MAILCHECK

Specifies the interval at which the Bourne shell checks the system mailbox for mail.

MAILMSG

Specifies the message sent to your console shell by the system when you have mail. The default message is similar to the following:

YOU HAVE NEW MAIL

Examples

1. To start the Mail program and list the messages in your mailbox, type the following at the command line prompt:

   mail

   The mail command lists every messages in your system mailbox. The mail system then displays the mailbox prompt (?) to indicate waiting for input. When you see this prompt, enter any mailbox subcommand. To see a list of subcommands, type:

   ?

   This entry lists the Mail subcommands.

2. To send the message letter to the recipient user1@host1 and copies to user2@host2 and user3@host3, type:

   mail -c "user2@host2 user3@host3" user1@host1<letter

3. To look at the contents of your personal mailbox, type:

   mail -f

   This command displays a list of the messages in your personal mailbox, $HOME/mbox.

4. To look at the contents of a specific mail folder, type:

   mail -f +dept

   This command displays a listing of the messages in the dept folder.

5. To send a message to a user on your local system, type:

   mail ron
When you finish typing the message to user ron, press Enter and press either . (period) or Ctrl+D to exit the editor and send the message. To determine if a user is on your local system, check for the user's name in your /etc/passwd file.

If your message is delivered successfully, you receive no notification. If your message could not be delivered, an error message is sent to you.

6. To mail a file to another user on your local system, type:
   
   mail karen < letter

   This command sends the contents of the file letter to user karen on your local system. After the command sends the file, the Mail program displays the command line prompt.

7. To send a message to a user on a remote system, type:
   
   mail dale@zeus

   You now can create a message to dale. In this example, you are sending a message to user dale on remote system zeus. To send a message to a user on another system connected to your system through a network, you must know that person's login ID and the name of the other system.

**Mailbox Subcommands for the mail, Mail, and mailx Command**

From the mail prompt, ? (question mark), you can enter subcommands to manipulate mail in your mailbox. Subcommands that work on more than one message at a time use the MessageList parameter. Subcommands that work with files or folders use the FileName parameter. These parameters are discussed in "Summary of Mailbox Subcommands in the Mail Program" in AIX 5L Version 5.2 System User's Guide: Communications and Networks.

The following list describes the Mailbox subcommands and their functions:

- **=** Echoes the number of the current message.
- **#** Comment character for writing comments in mail script files.
- **-** Displays the previous message.
- **?** Displays a brief summary of mailbox subcommands. Identical to the help subcommand.
- !Command**
  - **alias**
    - (a) With no arguments, displays all currently defined aliases and their corresponding addresses. With one argument, displays one alias. With more than one argument, creates a new alias or changes an old alias. Identical to the group subcommand.
- **alternates AlternatesList**
  - (alt) The alternates subcommand is useful if you have accounts on several machines. Use the subcommand to inform the Mail program that the addresses listed in AlternatesList all refer to you. When you use the reply subcommand to reply to messages, the Mail program does not send a copy of the message to any of the addresses given in AlternatesList. If you enter the alternates subcommand with no argument, the Mail program displays the current set of alternate names.
- **chdir Directory**
  - (cd) Changes your working directory to the indicated Directory. If no directory is given, it changes to your login directory.
- **copy [MessageList] File**
  - (c, co) Appends each message in MessageList to the end of File. Displays the file name in quotes, followed by the line count and character count, on the user's terminal. Does not delete any messages when you quit.
- **Copy [MessageList]**
  - (C) Saves the specified message in a file whose name is derived from the author of the message to be saved, without marking the messages as saved. Otherwise, it is equivalent to the save subcommand.
delete [MessageList] (d) Marks the messages in MessageList to be deleted when you quit the Mail program. Entering the d subcommand without a message list deletes the current message. Deleted messages are not saved in your $HOME/mbox file nor are they available for most other commands. However, you can use the undelete subcommand to restore messages you have deleted while in the same mailbox session. If you delete a message and either change to another mailbox or quit the mailbox with the quit subcommand, the deleted message cannot be recalled.

discard [FieldList] (di) Identical to the ignore subcommand.

Note: The retain subcommand overrides the discard subcommand.

dp Deletes the current message and displays the next message. If there is no next message, the Mail program displays EOF. Identical to the dt subcommand.

dt Deletes the current message and displays the next message. If there is no next message, the Mail program displays EOF. Identical to the dp subcommand.

echo String Displays the character string String on the command line.

edit [MessageList] (e) Starts the alternate editor using the MessageList as input files. To define an alternate editor, use the set EDITOR= command or place an entry in your $HOME/mailrc file. Any message specified by the MessageList parameter retains the changes made during the editor session.

exit (ex or x) Leaves the mailbox and returns to the operating system without changing the original contents of the mailbox. The mailbox returns to the condition that it was when the Mail program was started. Messages marked to be deleted are not deleted. Identical to the xit subcommand.

file [Name] (fi) Identical to the folder subcommand.

folder [Name] (fo) Switches to a new mail file or folder. With no arguments, the subcommand displays the name of the current mailbox. If an argument is included, it stores the current mailbox with changes (such as messages deleted) and reads in the new mailbox specified by the Name parameter. Identical to the file subcommand.

Some special conventions are recognized for the Name:

# Refers to the previous file.

% Refers to the system mailbox (/var/spool/mail/UserID).

& Refers to your personal mailbox ($HOME/mbox).

+Name Refers to a file in your folder directory.

folders Lists the names of the folders in your folder directory.

followup [message] (fo) Responds to a message, recording the response in a file whose name is derived from the author of the message. Overrides the record variable, if set.

Followup [MessageList] (F) Responds to the first message in the msglist, sending the message to the author of each message in the msglist. The subject line is taken from the first message and the response is recorded in a file whose name is derived from the author of the first message.

from [MessageList] (f) Displays the headings of messages in MessageList.

group (g) Identical to the alias subcommand.
headers [Message]

(h) Lists the headings in the current group of messages (each group of messages contains 20 messages by default; change this with the set screen= statement). If the mailbox contains more messages than can be displayed on the screen at one time, information about only the first group of messages will be displayed. To see information about the rest of the messages, use the h subcommand with a message number in the next range of messages, or use the z subcommand to change the current message group.

help

Displays a brief summary of mailbox subcommands. Identical to the ? subcommand.

hold [MessageList]

(ho) Marks each message in MessageList to be saved in your system mailbox (/var/spool/mail/UserID) instead of in your $HOME/mbox file. Does not override the delete subcommand. Identical to the preserve subcommand.

if Condition

else
endif

Construction for conditional execution of the mail subcommands. Subcommands following if are executed if Condition is true. Subcommands following else are executed if Condition is not true. The else is not required. The endif ends the construction and is required. The Condition can be receive (receiving mail) or send (sending mail).

ignore [FieldList]

Adds the header fields in FieldList to the list of fields to be ignored. Ignored fields are not displayed when you look at a message with either the type or print subcommand. Use this subcommand to suppress machine-generated header fields. Use either the Type or Print subcommand to print a message in its entirety, including ignored fields. The ignore subcommand with no arguments lists all header fields that are not included when you use a type or print subcommand to display a message. Identical to the discard subcommand.

list

(l) Displays a list of all mailbox subcommands with no explanation of what they do.

mail AddressList

(m) Starts the mail editor. Enables you to create and send a message to people specified in AddressList. The newly created message is independent from any receive messages.

mbox [MessageList]

Indicates that the messages in MessageList are to be sent to your personal mailbox (SHOME/mbox) when you quit the Mail program. This operation is the default action for messages that you have read if you are looking at your system mailbox (/var/spool/mail/UserID) and the hold option is not set.

more [MessageList]

(mo) Displays the messages in MessageList using the defined pager program to control display to the screen. Identical to the page subcommand.

More [MessageList]

(Mo) Similar to the more subcommand, but also displays ignored header fields.

new [MessageList]

Marks each message in MessageList as not having been read. Identical to the New, unread, and Unread subcommands.

New [MessageList]

Marks each message in MessageList as not having been read. Identical to the new, unread, and Unread subcommands.

next [Message]

(n) Makes the next message in the mailbox the current message and displays that message. With an argument list, it displays the next matching message.

page [MessageList]

(pa) Displays the messages in MessageList using the defined pager program to control display to the screen. Identical to the more subcommand.

Page [MessageList]

(Pa) Similar to the page subcommand but also displays ignored header fields.

pipe [[msglist command]]

l [[msglist] command]

(pi) Pipe the messages through the given command by invoking the command interpreter specified by SHELL with two arguments: -c and command. The command must be given as a single argument. This can be accomplished by quoting. If no arguments are given, the current message will be piped through the command specified by the value of the cmd variable. If the page variable is set, a form-feed character will be inserted after each message.

preserve

(pre) Identical to the hold subcommand.
print [MessageList]

(P) Displays the text of a specific message. Identical to the type subcommand.

Print [MessageList]

(P) Displays the text of a specific message along with the ignored header fields. Identical to the Type subcommand.

quit

(q) Leaves the mailbox and returns to the operating system. All messages read, but not deleted or saved are stored in your personal mailbox ($HOME/mbox). All messages you have marked to be deleted are removed from the mailbox and cannot be recovered. All messages marked with the hold or preserve option and messages you have not viewed are saved in the system mailbox (/var/spool/mail/UserID). If the quit subcommand is given while editing a mailbox file with the -f flag, the edit file is saved with changes. If the edit file cannot be saved, the Mail program does not exit. Use the exit subcommand to exit without saving the changes.

reply [Message]

(r) Allows you to reply to the sender of a message and to all others who receive copies of the message. Identical to the respond subcommand.

Reply [Message]

(R) Allows you to reply to only the sender of a message. Identical to the Respond subcommand.

respond [Message]

Allows you to reply to the sender of a message and to all others who receive copies of a message. Identical to the reply subcommand.

Respond [Message]

Allows you to reply to only the sender of a message. Identical to the Reply subcommand.

retain [FieldList]

Adds the header fields in FieldList to the list of fields to be retained. Retained fields are displayed when you look at a message with the type subcommand or print subcommand. Use this subcommand to define which header fields you want displayed. Use the Type or Print subcommand to print a message in its entirety, including fields that are not retained. If the retain subcommand is executed with no arguments, it lists the current set of retained fields.

Note: The retain subcommand overrides the discard subcommand.

save [ File ]

(s) Saves the current message including header information to a file or folder. If the file already exists, the message is appended to the file. If File is omitted, the message will be saved to the user's mbox.

save [MessageList] File

(s) Saves a MessageList including heading information to a file or folder. If the file already exists, the MessageList is appended to the file. Displays the file name and the size of the file when the operation is complete. If you save a message to a file, that message is not returned to the system mailbox (/var/spool/mail/UserID) nor saved in your personal mailbox ($HOME/mbox) when you quit the Mail program.
Save [MessageList]

Set [OptionList | Option= Value...]

Shell
Size [MessageList]
Source File
Top [MessageList]
Touch [MessageList]
Type [MessageList]
Type [MessageList]
Unalias
Uname [MessageList]
Unread [MessageList]
Unread [MessageList]
Unset OptionList

(S) Saves the specified messages in a file whose name is derived from the author of the first message. The name of the file is taken to be the author’s name with all network addressing stripped off.

(se) With no arguments, displays the options that are currently enabled. Otherwise, sets an option as specified. The argument following the set command can be either:

- An OptionList giving the name of a binary option (an option that is either set or unset)
- An Option=Value entry used to assign a value to an option

The options are listed in the .mailrc file format.

Note: The form unset name is equivalent to noname.

(sh) Starts an interactive version of the shell.

Disables the specified alias.

(so) Reads and executes the mail subcommands from File.

Displays the top few lines of the messages specified by MessageList. The number of lines displayed is determined by the valued option topolines and defaults to five.

Within your system mailbox (var/spool/mail/UserId), this subcommand marks the messages in MessageList to be moved to your personal mailbox (HOME/mbox) when you quit the Mail program. The messages are moved even though you have not read them. The messages are displayed in your personal mailbox as unread messages. The last message in MessageList becomes the current message.

(t) Displays the text of a specific message. Identical to the print subcommand.

(T) Displays the text of a specific message along with the ignored header fields. Identical to the Print subcommand.

Deletes the specified alias names.

(u) Removes the messages in MessageList from the list of messages to be deleted when you quit the Mail program. Entering the u subcommand without a message list recalls the last deleted message.

(U) Marks each message in MessageList as not having been read. Identical to the new, New, and Unread subcommands.

Unread [MessageList]

Marks each message in MessageList as not having been read. Identical to the new, New, and unread subcommands.

Disables the values of the options specified in OptionList. This action is the inverse of the set subcommand.

Note: The form unset name is equivalent to noname.
version

(ve) Displays the version banner for the Mail program.

visual [MessageList]

(v) Starts the visual editor using the MessageList as the input field. (This editor can be defined with the set VISUAL= statement.) Any changes made during the editor session are saved back to the messages in the MessageList.

write [MessageList] File

(w) Saves a message without heading information to a file or folder. Displays the file name and the size of the file when the operation is complete. Does not include message headers in the file.

xit

(x) Identical to the exit subcommand.

z [+ l -]

Changes the current message group (group of 20 messages) and displays the headings of the messages in that group. If a + or no argument is given, then headings in the next group are shown. If a - argument is given, the headings in the previous group are shown.

Mail Editor Subcommands for the mail, Mail Command

By default, the Mail program treats lines beginning with the ~ (tilde) character as subcommands. The following list describes the subcommands used while in the mail editor. The editor recognizes subcommands only if you enter them at the beginning of a new line.

~?

Displays a summary of the mail subcommands.

~!Command

The command interpreter specified by SHELL will be invoked with two arguments: -c and command. The standard output of command will be inserted into the message.

~a

Inserts the value of the sign variable into the text of the message, followed by a newline character. Identical to -i sign.

~A

Inserts the value of the Sign variable into the text of the message, followed by a newline character. Identical to -I Sign.

~b AddressList

Adds names in AddressList to the list of addresses to receive blind copies of the message. The -b subcommand can only be used to add to, not change or delete, the contents of the Bcc: List.

~c AddressList

Adds names in AddressList to the list of people to receive copies of the message. The -c subcommand can only be used to add to, not change or delete, the contents of the Cc: List.

~d

Appends the contents of the dead.letter file to the end of the message.

~e

Starts the alternate editor using the message text as the input file. (This editor can be defined with the set EDITOR= statement in the Bourne shell.) When you exit that editor, you return to the mail editor, where you may add text, or send the message by exiting the Mail program.

~f [MessageList]

Includes a MessageList in the current message to forward the message to another user. This subcommand reads each message in the MessageList and appends it to the current end of the message, but does not indent the appended message. This subcommand is also used to append messages for reference whose margins are too wide to embed with the m subcommand. This subcommand works only if you entered the mail editor from the mailbox prompt using either the mail, reply or Reply subcommand.

~F [MessageList]

Equivalent of the ~f, except that all headers will be included in the message, regardless of previous discard, ignore, and retain commands.

~h

Enables you to add or change information in all of the heading fields. The system displays each of the four heading fields, one at a time. You can view the contents of each field and delete or add information to that field. Press the Enter key to save any changes to the field and to display the next field and its contents.
~i string

Inserts the value of the named variable, followed by a newline character, into the text of the message. If the string is unset or null, the message will not be changed.

~m [MessageList]

Includes a MessageList in the current message for reference purposes. This subcommand reads each message in the MessageList and appends it to the current end of the message. The included message is indented one tab character from the usual left margin of the message. This subcommand works only if you entered the mail editor from the mailbox prompt using either the mail, reply, or reply subcommand.

~M [MessageList]

Equivalent of the ~m, except that all headers will be included in the message, regardless of previous discard, ignore, and retain commands.

~p

Displays the entire message, including header information.

~q

Quits the editor without sending the message. Saves the message in the dead.letter file in your home directory, unless the nosave option is set. The previous contents of the dead.letter file are replaced with the partially completed message.

Note: You can also quit the editor by using the Interrupt (Ctrl+C) key sequence twice.

~r File

Reads the contents of a file into the current message.

~s String

Changes the subject field to the phrase specified in String. You cannot append to the subject field with this subcommand.

~t AddressList

Adds the addresses in AddressList to the To: field of the message. The -t subcommand can only be used to add to, not change or delete, the contents of the To: List.

~v

Starts the visual editor using the message text as the input file. This editor can be defined with the set VISUAL statement in the Bourne shell.) When you exit that editor, you return to the mail editor where you may add text to the message, or send the message by exiting the Mail program.

~w File

Writes the message to the named file.

~x

Exits as with ~q, except the message will not be saved in the dead.letter file.

~: Subcommand

Executes the subcommand specified by Subcommand and returns to the mail editor.

~! Command

Pipes the message through the command Command as a filter. If Command gives no output or terminates unusually, it retains the original text of the message. Otherwise, the output of Command replaces the current message. The fmt command is often used as Command to format the message.

~< file

Reads the contents of a file into the current message.

~<!Command

Allows you to run a shell command. The shell runs with the -c flag and the Command specified. The standard output of Command is inserted into the message.

~~

Allows you to use the ~ (tilde) character in a message without it being interpreted as a command prefix. The ``` key sequence results in only one ~ character being sent in the message.

Files

$HOME/.mailrc

Contains the mail subcommands to customize the Mail program for a specific user.

$HOME/mbox

Contains your personal mailbox.

/usr/share/lib/Mail.rc

Contains the file with mail subcommands to change the Mail program for all users on the system.

/var/spool/mail/*

Contains system mailboxes for all users.

/usr/bin/mail

Contains the mail command.

/usr/bin/Mail

Contains the Mail command.

/usr/bin/mailx

Contains the mailx command.
Related Information
The `bellmail` command, `fmt` command, `pg` command, `sendmail` command.

The `.mailrc` file format.

Mail Overview, Creating and Sending Mail, Receiving and Handling Mail in AIX 5L Version 5.2 System User's Guide: Communications and Networks.

Reading Mail, Replying to Mail, Displaying the Contents of Your Mailbox, Editing a Message, Folders in AIX 5L Version 5.2 System User's Guide: Communications and Networks.

Customizing the Mail Program, Starting the Mail Editor, Displaying a Message While in the Mail Editor, Changing or Adding to the Heading Fields of a Message in AIX 5L Version 5.2 System User's Guide: Communications and Networks.

Checking for Misspelling in the Mail Editor, Reformating a Message in the Mail Editor, Changing Text Editors Used for Entering Messages in AIX 5L Version 5.2 System User's Guide: Communications and Networks.

Mailbox Subcommands for the mail Command.

**mailq Command**

**Purpose**
Prints the contents of the mail queue.

**Syntax**
```
/usr/sbin/mailq [-v]
```

**Description**
The `mailq` and `MAILQ` commands print a list of messages that are in the mail queue. The first line printed for each message shows:
- The internal identifier used on this host for the message with a possible status character
- The size of the message in bytes
- The date and time the message is accepted into the queue
- the envelope sender of the message

The second line shows the error message that caused the message to be retained in the queue, it is not displayed if the message is being displayed for the first time. The status characters are either:
- * Indicates the job is being processed
- X Indicates that the load is too high to process the job
- - Indicates that the job is too young to process

The following lines show message recipients, one per line.

The `mailq` command is the same as the `sendmail -bp` command.

Specify the `-v` flag to display message priority.
Flags

-v  Prints verbose information. This adds the priority of the message and a single character indicator (+ or blank) indicating whether a warning message has been sent on the first line of the message. Additionally, extra lines may be intermixed with the recipients indicating the controlling user information; this shows who owns any program that are executed on behalf of this message and the name of the alias this command expanded from, if any.

Exit Status

The command returns the following exit values:

0     Exits successfully.
>0    An error occurred.

Examples

The mailq command prints two types of lists:

- The mailq command lists the mail queue as shown in the following example:

    Mail Queue (1 request)
    ---QID----- --Size-- -----Q-Time----- ------Sender/Recipient-----
    AA02508     3     Thu Dec 17 10:01     root
    (User unknown)     bad_user

- The mailq -v command lists the mail queue as follows:

    Mail Queue (1 request)
    ---QID----- --Size-- -Priority- ---Q-Time--- --Sender/Recipient--
    AA02508     3     1005     Dec 17 10:01     root
    (User unknown)     bad_user

The fields have the following meanings:

QID    Contains the message queue ID of the message.
Size   Contains the number of bytes in the body of the message (heading information not included).
Priority Contains the priority of the message, based primarily on the size of the message.
Q-Time Contains the time the message entered the queue.
Sender/Recipient Contains the user ID of the sender and the recipient of the message. A message on the line between the sender and the recipient indicates the status of the message.

Files

/usr/sbin/mailq     Contains the mailq command.
/var/spool/mqueue    directory     Contains the log file and temporary files associated with the messages in the mail queue.

Related Information

The sendmail command.
**mailstats Command**

**Purpose**
Displays statistics about mail traffic.

**Syntax**
```
mailstats [ -o ] [ -p ] [ -C CfFile ] [ -f StatFile ]
```

**Description**
The `mailstats` command displays the current mail statistics. The time at which the statistics started displays reads the information in the format specified by `ctime`. The statistics for each mailer are displayed on a single line, with the following fields:

- `M` Contains the mailer number.
- `msgsfr` Contains the number of messages received by the local machine from the indicated mailer.
- `bytes_from` Contains the number of Kbytes in the messages received by the local machine from the indicated mailer.
- `msgsto` Contains the number of messages sent from the local machine using the indicated mailer.
- `bytes_to` Contains the number of bytes in the messages sent from the local machine using the indicated mailer.
- `msgsrej` Contains the number of messages rejected.
- `msgsdis` Contains the number of messages discarded.
- `Mailer` Contains the name of mailer.

After the statistics are displayed, a line totaling the value of all of the mailers displays, preceded with a `T`. This information is separated from the statistics by a line containing only `=` (equal characters). Another line preceded with a `C` lists the number of connections.

**Flags**

- `-C CfFile` Specifies use of the `cfFile` instead of the default `sendmail.cf` file.
- `-o` Specifies that the name of the mailer does not display in the output.
- `-p` Outputs information in program readable mode and clears the statistics.

**Exit Status**

- `0` Successful completion.
- `>0` An error occurred.

**Files**

- `/etc/mail/statistics` Default `sendmail` statistics file.
- `/etc/mail/sendmail.cf` Default `sendmail` configuration file.

**Related Information**
The `mailq` command and `sendmail` command.

Managing the Mail Queue Files and Directories in *AIX 5L Version 5.2 System Management Guide: Communications and Networks.*
Note: When the make command encounters a line beginning with the word include followed by another word that is the name of a makefile (for example, include depend), the make command attempts to open that file and process its contents as if the contents were displayed where the include line occurs. This behavior occurs only if the first noncomment line of the first makefile read by the make command is not the .POSIX target; otherwise, a syntax error occurs.
Comments: Comments begin with a # character, anywhere but in a shell command line, and continue to the end of the line.

Environment: The make command uses the MAKEFLAGS environment variable, if it exists.

Target Rules
Target rules have the following format:

```
target[target...] : [prerequisite...] [ ;command ]
<Tab>command
```

Multiple targets and prerequisites are separated by spaces. Any text that follows the ; (semicolon) and all of the subsequent lines that begin with a Tab character are considered commands to be used to update the target. A new target entry is started when a new line does not begin with a Tab or # character.

Note: The list of prerequisites can be empty.

Special Targets
Special targets cannot be included with other targets; that is, they must be the only target specified. These targets control the operation of the make command. These targets are:

- `.DEFAULT` - This is used as the rule for any target (that was used only as a prerequisite) that the make command cannot figure out any other way to create. Only the shell script is used. The < (left angle bracket) variable of a target that inherits .DEFAULT's commands is set to the target's own name.
- `.IGNORE` - Prerequisites of this target are targets themselves; this causes errors from commands associated with them to be ignored. If no prerequisites are specified, this is the equivalent of specifying the -i flag.
- `.POSIX` - Causes the make command to use a different default rules file. The file, /usr/ccs/lib/posix.mk, provides the default rules as specified in the POSIX standard.
- `.PRECIOUS` - Prerequisites of this target are targets themselves. .PRECIOUS prevents the target from being removed. If no prerequisites are specified, the .PRECIOUS attribute is applied to every target in the file. Usually, when make is interrupted (for example, with SIGHUP, SIGTERM, SIGINT, or SIGQUIT), it removes any partially made targets. If make was invoked with the -n, -p, or -q flags, the target is considered to have the .PRECIOUS attribute.
- `.SILENT` - Prerequisites of the target are targets themselves. This causes commands associated with the target to not be written to standard output before they are executed. If no prerequisites are specified, the .SILENT attribute is applied to every command in the file.
- `.SUFFIXES` - Use this name to add more suffixes to the list of file suffixes that make recognizes. Prerequisites of the target are appended to the list of known suffixes. If no suffixes are specified, any previously specified suffixes are deleted. These suffixes are used by the inference rules. To change the order of suffixes, you need to specify an empty .SUFFIXES entry and then a new list of .SUFFIXES entries. A makefile must not associate commands with .SUFFIXES.

Inference Rules
The make command has a default set of inference rules, which you can supplement or overwrite with additional inference rules definitions in the makefile. The default rules are stored in the external file, /usr/ccs/lib/aix.mk. You can substitute your own rules file by setting the MAKERULES variable to your own file name from the command line. The following line shows how to change the rules file from the command line:

```
make MAKERULES=/pathname/filename
```

Inference rules consist of target suffixes and commands. From the suffixes, the make command determines the prerequisites, and from both the suffixes and their prerequisites, the make command determines how to make a target up-to-date. Inference rules have the following format:

```
rule:
<Tab>command
...
```
where rule has one of the following forms:

• **.s1** A single-suffix inference rule that describes how to build a target that is appended with one of the single suffixes.
• **.s1.s2** A double-suffix inference rule that describes how to build a target that is appended with .s2 with a prerequisite that is appended with .s1.

The .s1 and .s2 suffixes are defined as prerequisites of the special target, **.SUFFIXES**. The suffixes .s1 and .s2 must be known suffixes at the time the inference rule is displayed in the makefile. The inference rules use the suffixes in the order in which they are specified in **.SUFFIXES**. A new inference rule is started when a new line does not begin with a <Tab> or # character.

If rule is empty, for example:

```bash
rule: ;
```

execution has no effect, and the **make** command recognizes that the suffix exists, but takes no actions when targets are out-of-date.

A ~ (tilde) in the preceding rules refers to an SCCS file. Therefore, the rule, **.c~.o**, would transform an SCCS C language prerequisite file into an object file (**.o**). Because the **s.** of the SCCS file is a prefix, it is incompatible with the **make** command’s suffix view. The ~ (tilde) is a way of changing any file reference into an SCCS file reference.

**Libraries**

A target or prerequisite can also be a member of an archive library and is treated as such if there are parentheses in the name. For example, **library(name)** indicates that **name** is a member of the archive library **library**. To update a member of a library from a particular file, you can use the format **.s1.a**, where a file with the **.s1** suffix is used to update a member of the archive library. The **.a** refers to an archive library.

**Using Macros**

In makefiles, macro definitions are defined in the format:

```bash
variable=value
```

Macros can be displayed throughout the makefile, as follows:

- If a macro is displayed in a target line, it is evaluated when the target line is read.
- If a macro is displayed in a command line, it is evaluated when the command is executed.
- If a macro is displayed in a macro definition line, it is evaluated when the new macro is displayed in a rule or command.

If a macro has no definition, it defaults to **NULL**. A new macro definition overwrites an existing macro of the same name. Macros assignments can come from the following, in the listed order:

1. Default inference rules
2. Contents of the environment
3. Makefiles
4. Command lines.

**Note:** The **-e** flag causes environment variables to override those defined in the makefile.

The **SHELL** macro is special. It is set by the **make** command to the path name of the **shell** command interpreter (**/usr/bin/sh**). However, if it is redefined in the makefile or on the command line, this default setting is overridden.

**Note:** The **SHELL** macro does not affect, and is not affected by, the **SHELL** environment variable.
Shell Commands
Each target can have associated with it a series of shell commands, usually used to create the target. Each of the commands in this script must be preceded by a Tab. While any target can be displayed on a dependency line, only one of these dependencies can be followed by a creation script, unless the :: operator is used.

If the first, or first two characters, of the command line are one or all of @ (at sign), - (hyphen), and + (plus sign), the command is treated specially, as follows:

@ Causes the command not to be echoed before it is executed.
- Causes any nonzero exit status of the command line to be ignored.
+ Causes a command line to be executed, even though the options -n, -q, or -t are specified.

A command that has no metacharacters is directly executed by the make command. For example, the make command consigns the first command in the following example to the shell because it contains the > (greater than sign) shell metacharacter. The second command in the following example does not contain any shell metacharacters, so the make command executes it directly:

```
target: dependency
  cat dependency > target
  chmod a+x target
```

Bypassing the shell saves time, but it can cause problems. For example, attempting to execute a C shell script from within a makefile by setting the SHELL macro to /bin/csh will not work unless the command line also contains at least one shell metacharacter.

```
SHELL=/bin/csh

target: dependency
  my_csh_script
```

This makefile fails because the make command attempts to run my_csh_script instead of consigning it to the C shell.

Variable Assignments
Variables in the make command are much like variables in the shell and consist of all uppercase letters. The = operator assigns values to variables. Any previous variable is then overridden.

Any white space before the assigned value is removed; if the value is being appended, a single space is inserted between the previous contents of the variable and the appended value.

Variables are expended by surrounding the variable name with either {} (braces) or () (parentheses) and preceding it with a $ (dollar sign). If the variable name contains only a single letter, the surrounding braces or parentheses are not required. This shorter form is not recommended.

Variable substitution occurs at two distinct times, depending on where the variable is being used. Variables in dependency lines are expanded as the line is read. Variables in shell commands are expanded when the shell command is executed.

The four classes of variables (in order of increasing precedence) are:

- **Environment** Variables defined as part of the make command's environment.
- **Global** Variables defined in the makefile or in included makefiles.
- **Command line** Variables defined as part of the command line.
Local Variables defined specific to a certain target. The local variables are as follows:

- **$<** Represents either the full name of a prerequisite that made a target out-of-date (inference rule), or the full name of a target (.DEFAULT rule).
- **$*** Represents the file name section of a prerequisite that made a target out-of-date (in an inference rule) without a suffix.
- **$@** Represents the full target name of the current target or the archive file name part of the library target.
- **$%** Represents a library member in a target rule if the target is a member of the archive library.

You can also use these local variables appended with D or F:

- **D** Indicates that the local variable applies to the directory part of the name. This is the path name prefix without a trailing / (slash). For current directories, D is a . (period).
- **F** Indicates that the local variable applies to the file name part of the name.

In addition, the make command sets or knows about the following variables:

- **$** A single $ (dollar sign); that is, $$ expands to a single dollar sign.
- **LANG** Determines the locale to use for the locale categories when both LC_ALL and the corresponding environment variable (beginning with LC_) do not specify a locale.
- **LC_ALL** Determines the locale to be used to override any values for locale categories specified by the setting of LANG or any other LC_ environment variable.
- **LC_CTYPE** Determines the locale for the interpretation of sequences of bytes of text data as characters; for example, single- versus multibyte characters in arguments.
- **LC_MESSAGES** Determines the language in which messages should be written.

The environment variable, MAKEFLAGS, can contain anything that can be specified on make's command line. Anything specified on make's command line is appended to the MAKEFLAGS variable, which is then entered into the environment for all programs that make executes. Note that the operation of the -f and -p flags in the MAKEFLAGS variable is undefined. Command line flags take precedence over the -f and -p flags in this variable.

- **VPATH** Allows you to specify a list of directories to search for prerequisites. The list of directories works like the PATH variable in the SHELL. The VPATH variable can specify multiple directories separated by colons. For example:

```
VPATH=src:/usr/local/src
```

This tells the make command to search for the following directories in the order given:

- The current directory (this happens even without VPATH)
- src (a subdirectory in the current directory )
- /usr/local/src.

Flags

- **D Variable** Sets the value of Variable to 1.
-dOption Displays detailed information about the files and times that make examines (debug mode). The -d flag without any options or with the A option displays all the debug information available. Individually selectable debug options follow:

A Displays all possible debug information.
a Displays debug information about archive searching and caching.
d Displays debug information about directory searching.
g1 Displays debug information about input graph before making anything.
g2 Displays debug information about input graph after making everything, or before exiting on an error.
m Displays debug information about making targets, including modification dates.
s Displays debug information about suffix searching.
v Displays debug information about variable assignments.
-e Specifies that environmental variables override macro assignments within makefiles.
-f MakeFile Specifies a makefile to read instead of the default makefile. If MakeFile is - (hyphen), standard input is read. Multiple makefiles can be specified and are read in the order specified.
-i Ignores nonzero exit of shell commands in the makefile. Equivalent to specifying - (hyphen) before each command line in the makefile.
-k Continues processing after errors are encountered, but only on those targets that do not depend on the target whose creation caused the error.
-n Displays commands, but does not run them. However, lines beginning with a + (plus sign) are executed.
-p Displays the complete set of macro definitions and target descriptions before performing any commands.
-q Returns a zero status code if the target file is up-to-date; returns a one status code if the target file is not up-to-date. However, a command line with the + (plus sign) prefix will be executed.
-r Does not use the default rules.
-S Terminates the make command if an error occurs. This is the default and the opposite of -k flag.
-s Does not display commands on the screen as they are performed.
-t Creates a target or updates its modification time to make it seem up-to-date. Executes command lines beginning with a + (plus sign).

Target Specifies a target name of the form Target or sets the value of variables.

Exit Status
When the -q flag is specified, this command returns the following exit values:

0 Successful completion.
1 The target was not up-to-date.
>1 An error occurred.

Otherwise, this command returns the following exit values:

0 Successful completion.
>1 An error occurred.

Examples
1. To make the first target found in the makefile, type:

   make

2. To display, but not run, the commands that the make command would use to make a file:
make -n search.o

Doing this will verify that a new description file is correct before using it.

3. To create a makefile that says that pgm depends on two files, a.o and b.o, and that they, in turn, depend on their corresponding prerequisite files (a.c and b.c) and a common file, incl.h, type:

```
pgm: a.o b.o
  c89 a.o b.o -o pgm
a.o: incl.h a.c
   c89 -c a.c
b.o: incl.h b.c
   c89 -c b.c
```

4. To make optimized .o files from .c files, type:

```
.c.o:
   c89 -c -o $*.c
or:
   .c.o:
   c89 -c -o $<
```

5. To view the contents of the built-in rules, type:

```
make -p -f /dev/null 2>/dev/null
```

### Files

- **makefile**: Contains a list of dependencies.
- **Makefile**: Contains a list of dependencies.
- **s.makefile**: Contains a list of dependencies. It is an SCCS file.
- **s.Makefile**: Contains a list of dependencies. It is an SCCS file.
- **/usr/ccs/lib/posix.mk**: Contains default POSIX rules for the `make` command.
- **/usr/ccs/lib/aix.mk**: Contains default rules for the `make` command.

### Related Information

The `sh` command.

The `make Command Overview` in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.


### makedbm Command

#### Purpose

Makes a Network Information Services (NIS) database map.

#### Syntax

**To Create an NIS Map**

```
/usr/sbin/makedbm [-b] [ -i NISInputFile ] [ -o NISOutputFile ] [ -d NISDomainName ] [ -m NISMasterName ] [ InputFile OutputFile ]
```

**To Create a Non-dbmap Formatted NIS Map**

```
/usr/sbin/makedbm [ -u dbmFileName ]
```
Description

The **makedbm** command makes an NIS map. It does this by converting the file named in the *InputFile* parameter into two output files: *OutputFile.pag* and *OutputFile.dir*. Each line in each input file is converted into a single Data Base Manager (DBM) record.

The **makedbm** command is most often invoked from the */var/yp/Makefile* file to generate NIS maps. All characters leading up to the first space or tab in each line of the */var/yp/Makefile* file form the key. The rest of the line contains value data. If a line ends with a \ (backslash), data for that record is continued on the next line. NIS clients must interpret the # (pound sign) symbol since the **makedbm** command does not treat it as a comment character. If the *InputFile* parameter is a - (minus sign), the **makedbm** command reads standard input instead.

This command generates a special entry in the output map by using the YP_LAST_MODIFIED key, which is the date that the file specified by the *InputFile* parameter was created (or the current time, if the *InputFile* parameter is a - (minus sign)).

Flags

- **-b** Propagates a map to all servers using the named name server.
- **-i** Creates a special entry with the YP_INPUT_FILE key.
- **-o** Creates a special entry with the YP_OUTPUT_FILE key.
- **-d** Creates a special entry with the YP_DOMAIN_NAME key.
- **-m** Creates a special entry with the YP_MASTER_NAME key.
- **-u** Undoes a DBM file. That is, prints out a DBM file one entry per line, with a single space separating keys from values.

Files

*/var/yp/Makefile* Contains rules for making NIS maps.

Related Information

The **ypinit** command, **yppush** command.


How to Create a Non-Standard NIS Maps in AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide.

List of NDBM and DBM Programming References in AIX 5L Version 5.2 Communications Programming Concepts.

NIS Reference in AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide.

makedepend Command

Purpose

Create dependencies in makefiles.
Syntax


Description

The makedepend command reads each SourceFile in sequence and parses it like a C-preprocessor. It processes all #include, #define, undef, ifdef, ifndef, endif, #if, and #else directives to determine which #include directives need to be used in a compilation. Any #include directives can reference files having other #include directives, and parsing occurs in these files as well.

Every file that a SourceFile includes, directly or indirectly, is what makedepend calls a "dependency." These dependencies are then written to a makefile in such a way that the make command can determine which object files must be recompiled when a dependency has changed.

By default, makedepend places its output in the file named makefile if it exists, otherwise Makefile. An alternate makefile may be specified with the -f flag. makedepend first searches the available makefile for the line:

# DO NOT DELETE THIS LINE - make depend depends on it.

or one provided with the -s flag, as a delimiter for the dependency output. If it finds the line, it deletes everything following the line to the end of the makefile and puts the output after the line. If makedepend does not find the line, it appends the delimited string to the end of the makefile and places the output immediately after the string.

For each SourceFile appearing on the command line, makedepend puts lines in the makefile in the following form.

SourceFile.o: df ile ...

Where SourceFile.o is the name from the command line with its suffix replaced with .o, and dfile is a dependency discovered in an #include directive while parsing the SourceFile or one of the files it included.

The algorithm used in this command assumes that all files compiled by a single makefile will be compiled with roughly the same -I and -D flags, and that most files in a single directory will include largely the same files.

Given these assumptions, makedepend expects to be called once for each makefile, with all source files that are maintained by the make file appearing on the command line. It parses each source and include file only once, maintaining an internal symbol table for each. As a result, the first file on the command line takes an amount of time proportional to the amount of time that a normal C preprocessor takes. On subsequent files, if it encounters an include file that it has already parsed, it does not parse again.

For example, imagine you are compiling two files, file1.c and file2.c, each includes the header file header.h. The header.h file includes the files def1.h and def2.h. When you run the command:

makedepend file1.c file2.c

then makedepend will first parse file1.c and consequently, header.h and then def1.h and def2.h. It then decides that the dependencies for this first file are:

file1.o: header.h def1.h def2.h

But when the program parses the second file, file2.c and discovers that it, too, includes header.h, it does not parse the file, but simply adds header.h, def1.h and def2.h to the list of dependencies for file2.o.

Note: If you do not have the source for cpp (the Berkeley C preprocessor), then makedepend will compile in such a way that all #if directives will evaluate to False, regardless of their actual value.
This may cause the wrong #include directives to be evaluated. In these cases, it is recommended
that you write a new parser for #if expressions. The need for a new parser should be clear from the
following example:

Imagine you are parsing two files file1.c and file2.c, each includes the file def.h. The list of files that
def.h includes might be very different when def.h is included by file1.c than when it is included by
file2.c. But once makedepend arrives at a list of dependencies for a file, it is cast in concrete.

Flags

Note: The makedepend command ignores flags it does not understand. Flag usage is similar to that of the cc command.

-DName=Def or -DName
 Places a definition for the Name variable in the makedepend command’s symbol table. Without the =Def specifier, the symbol is defined as 1.
-IIncludeDir
 Prepends the IncludeDir variable to the list of directories searched by the makedepend command when it encounters an #include directive. By
default, the makedepend command searches only the /usr/include directory.
-YIncludeDir
 Replaces all of the standard include directories with a single specified include directory, you can omit IncludeDir to prevent searching the standard
include directories.
-a
 Appends the dependencies to the end of the file instead of replacing them.
-fMakeFile
 Enables you to specify an alternate makefile in which to place command output.
-oObjSuffix
 Specifies an object suffix. For example, some systems may have object files whose suffix is something other than.o. This flag allows you to specify
another suffix, such as “.b” with -o.b or “.obj” with -o.obj and so forth.
-pObjPrefix
 Prepends the object file prefix to the name of the object file. This flag is used to designate a different directory for the object file. The default is the empty
string.
-sString
 Specifies the starting string delimiter. This flag permits you to specify a different string for makedepend to search for in the makefile.
-wWidth
 Changes the maximum line width of output lines. The default maximum is 78 characters.
-v
 Causes makedepend to display a list of files included by each input file on standard output.
-m
 Causes makedepend to display a warning if any input file includes another file more than once. In previous version of makedepend this was the
default behavior. This flag is provided for backward compatibility and to aid in debugging problems related to multiple inclusion.

—Options—

Ignores any unrecognized argument contained within a beginning and ending double hyphen. When makedepend encounters a double hyphen (—) in the argument list, any unrecognized argument following it is silently ignored; a second double hyphen terminates this treatment. The double hyphens enable makedepend to safely ignore esoteric compiler arguments that might normally be found in a CFLAGS make command macro (see the Examples section). All flags that makedepend recognizes and that appear between the pair of double hyphens are processed normally.

Examples

Normally, makedepend will be used in a makefile target so that typing makedepend updates the
dependencies for the makefile.

SRCS=file1.c file2.c ...
CFLAGS=-O -DHACK -I../foobar -xyz
depend:
   makedepend -- $(CFLAGS) -- $(SRCS)
Related Information
The cc command, make command.

makedev Command

Purpose
Creates binary description files suitable for reading by the troff command and its postprocessors.

Syntax
makedev DESC | FontFile ...

Description
The makedev command creates binary files suitable for reading by the troff command and its postprocessors. When the DESC file is specified, the makedev command creates a DESC.out file and a set of font description files using the information contained in the DESC file. When a font file is specified, the makedev command creates the corresponding font description file.

Options
DESC Causes a DESC.out file to be created.

Parameters
FontFile Causes a FontFile.out file to be created.

Examples
The following command:
makedev B
creates a B.out file, which contains the font tables for the Times-Bold fonts.

Related Information
The troff command.
The troff Font File Format

makekey Command

Purpose
Generates an encryption key.

Syntax
makekey [ ASCIICharacters ... ]

Description
The makekey command generates an encryption key for use with programs that perform encryption. Its input and output are usually pipes.
The `makekey` command reads 10 characters from standard input and writes 13 characters to standard output. The first 8 of the 10 input characters can be any sequence of ASCII characters, as specified by the `ASCIICharacters` parameter. The last two input characters, called the salt, are chosen from the sets 0 through 9, a through z, A through Z, . (period), and / (slash). The salt characters are repeated as the first two characters of the output. The remaining 11 output characters are chosen from the same set as the salt and constitute the output key that you use as the encryption key parameter of programs that perform encryption.

**Examples**

Entering the following example at the command line generates an encryption key:

```
makekey
1234567890
```

The `makekey` command generates an encryption key based on the input string 1234567890 and displays the following encryption key below, with the `_${` (shell prompt) appearing immediately after the generated key and on the same line.

```
90y744T/NXw1U$_
```

**Related Information**

The `crypt`, `encrypt`, or `setkey` subroutine.

---

**makemap Command**

**Purpose**

Creates database maps for `sendmail`.

**Syntax**

```
makemap [-C][-N][c] CacheSize [-d][-e][-f][-l][-o][-r][-s][-u][-v] Maptype Mapname
```

**Description**

The `makemap` command creates the database maps used by the keyed map lookups in the `sendmail` command. It reads input from the standard input and outputs them to the indicated `Mapname`.

**Parameters**

- `Maptype` Depending upon how it is compiled, this command handles up to three different database formats:
  - `dbm` DBM format maps. This requires the `ndbm` library.
  - `btree` B-tree format maps. This requires the new Berkeley DB library.
  - `hash` Hash format maps. This requires the new Berkeley DB library.

  **Note:** In all cases, this command reads lines from the standard input, consisting of two words separated by white space. The first is the database key, the second is the value. The value may contain "%n" strings to indicate parameter substitution. Literal percents should be doubled ("%%"). Blank lines and lines beginning with a "#" are ignored.

- `Mapname` Name of the map.

**Flags**

- `-c CacheSize` Specifies to use the hash and B-tree cache size.
-C Indicates to use the specified sendmail configuration file for looking up the TrustedUser option.
-d Allows duplicate keys in the map. This is only allowed on B-Tree format maps. If two identical keys are read, they are both inserted into the map.
-e Allows empty values, right hand side.
-f Disables the function of folding all upper case letters in the key to lowercase. This is intended to mesh with the -f flag in the K line in the sendmail.cf file. The value is never case folded.
-i Lists supported map types.
-N Includes the Null byte that terminates strings in the map. This must match the -N flag in the sendmail.cf "K" line.
-o Specifies to append to an existing file. This allows you to augment an existing file.
-r Allows replacement of existing keys. Normally, the makemap command complains if you repeat a key and does not perform the insert.
-s Ignores safety checks on maps being created. This includes checking for hard or symbolic links in world writeable directories.
-u Dumps, unmaps, the contents of the database to standard output.
-v Specifies that the command verbosely print its status.

Related Information
The sendmail command.

man Command

Purpose
Displays manual entries online.

Syntax
man \([ [ [C] [F] [L] [Section]\] ] [\[-k [f]\] [\[-M Path\] [r] Title ...\]

Description
The man command provides reference information on topics, such as commands, subroutines, and files. The man command provides one-line descriptions of commands specified by name. The man command also provides information on all commands whose descriptions contain a set of user-specified keywords.

The man command formats a specified set of manual pages. If you specify a section for the Section parameter, the man command searches in that section of the manual pages for the title specified by the Title parameter. The value of the Section parameter can be either an Arabic number from 1 through 8 or a letter.

The Section letters are:

C Specifies commands (including system management commands).
F Specifies file-type manual pages.
L Specifies library functions.
n Specifies new.
l Specifies local.
o Specifies old.
p Specifies public.

Note: The n, l, o, and p section specifiers are not valid for reading the hypertext information bases, which contain the operating system documentation.
The Section numbers are:

1. Indicates user commands and daemons.
2. Indicates system calls and kernel services.
3. Indicates subroutines.
4. Indicates special files, device drivers, and hardware.
5. Indicates configuration files.
6. Indicates games.
7. Indicates miscellaneous commands.
8. Indicates administrative commands and daemons.

Note: The operating system documentation in the hypertext information databases is grouped into three sections only: command manual pages (in section 1, equivalent to section C), subroutine manual pages (in section 3, equivalent to section L), and file manual pages (in section 4, equivalent to section F). When searching for hypertext information, specifying section 1, 6, 7, or 8 will default to the command manual pages, section 2 or 3 will default to the subroutine manual pages, and section 4 or 5 will default to the file manual pages.

If the Section parameter is omitted, the man command searches all sections of the manual.

The search path the man command uses is a list of directories separated by a : (colon) in which manual subdirectories can be found. The MANPATH environment variable value is used for the default path. The MANPATH environment variable is not valid when reading the hypertext information bases.

The man command displays the manual pages as follows:

1. The man command searches the nroff directories (man?) under the /usr/share/man directory.
2. The man command searches the formatted version directories (cat?) under the /usr/share/man directory. If the formatted version is available, and if it has a more recent modify time than the nroff command source, the man command displays the formatted version. Otherwise, the manual page is formatted with the nroff command and displayed. If the user has permission, the formatted manual page is deposited in the proper place, so that later invocations of the man command do not format the page again.

Note: There is no nroff source for the supplied manual pages. However, you can put nroff source for manual pages into the man directories and the man command can locate and process the nroff source.

3. If the man command does not find a manual page in the /usr/share/man/man or /usr/share/man/cat directory, the man command reads from the hypertext information bases. The hypertext information bases reside in the /usr/share/man/info directory structure and contain the operating system documentation. When reading from the hypertext databases, the man command does not put any manual pages in the /usr/share/man/cat directory structure. The man command strips formatting information from the manual page, wraps lines so the lines fit on the display, and displays the manual page using the command described by the PAGER environment variable.

When accessing the HTML databases, man looks for the operating system library before it proceeds to other LPP libraries. Within these libraries, it processes information in the following order:

cmds Commands Reference
libs Subroutines, System Calls
files Files Reference

If the standard output is a tty, the man command pipes its output using the more command with the -s and -v flags. The -s flag eliminates multiple blank lines and stops after each page on the screen. The -v
flag suppresses the display of nonprinting characters to the screen. To continue scrolling, press the space
bar. To scroll an additional 11 lines when the output stops, press the Ctrl-D key sequence.

The PAGER environment variable can be set to whatever pager is desired. The default value is the more
command. To change the default pager, enter:

PAGER=Somepager
export PAGER

For example, if there are customized manual pages which are formatted with reverse or fractional line
feeds, the PAGER environment variable may be set to /usr/bin/pg so that the line feeds are not printed as
control characters. This procedure is not necessary for the manual pages.

When the man command uses a hypertext database, it can retrieve several articles. For example, man
open displays several articles. The use of SIGINT (Ctrl-C) exits the man command completely. On the
other hand, man open close also displays several articles but the use of SIGINT (Ctrl-C) causes man to
display the close command information instead of exiting. Using SIGINT (Ctrl-C) again exits the man
command completely.

When specifying one of the Network Computing System library routines that contains a $ (dollar sign) in its
name, enter a \ (backslash) preceding the $.

Flags

-c Displays the manual information using the cat command.
-f Displays entries in the keyword database related only to the command name given as the final
parameter. You can enter more than one command name, each separated by a space. Use this flag to
search for command articles only. To use the -f flag, a root user must have previously entered catman -x
create the /usr/share/man/whatis file.
-k Displays each line in the keyword database that contains a string of characters matching the title given
as the final parameter. You can enter more than one title, each separated by a space. To use the -k flag,
a root user must have previously entered catman -x to create the /usr/share/man/whatis file.
-MPath Changes the standard location where the man command searches for manual information. The search
path the man command uses is a list of directories separated by a : (colon) in which manual
subdirectories can be found. The MANPATH environment variable value is used for the default path.
Note: The -M flag is not valid when the man command reads from the hypertext databases.
-r Searches remotely for the manual information. If for any reason (either the remote machine is not
reachable, or there is a problem reading URL or the Java is not installed on the local machine) the
remote search fails, then man performs a local search for the requested man page.
Note: The DOCUMENT_SERVER_MACHINE_NAME environment variable should be set to the name of
the documentation search server machine the user wants to use. If the AIX Base Documentation is not
supported for the host's locale, the man command searches for the documentation for an alternate
locale. If the search is successful, the documentation page is displayed after conversion to the local
host's locale. If the alternate locale is not installed on the local host, the man command fails to display
the documentation page.
-t Formats the manual information using the troff command. This flag is ignored if the manual page is
found in a hypertext information base.

Exit Status
This command returns the following exit values:

0 Successful completion.
>0 An error occurred.
Examples

1. To display information about the `grep` command, enter:
   
   ```
   man grep
   ```

2. To display information about the `rpc_Register` library routine, enter:
   
   ```
   man rpc_registered
   ```

3. To display all entries in the `/usr/share/man/whatis` keyword database that contain the "mkdir" string, enter:
   
   ```
   man -k mkdir
   ```

   The output is equivalent to the `apropos` command. You receive output from the `-k` flag only when the `/usr/share/man/whatis` keyword database already exists.

4. To display all entries from the keyword database related to the `nroff` and `troff` commands, enter:
   
   ```
   man -f nroff troff
   ```

   The output is equivalent to the `whatis` command. You receive output from the `-f` flag only when the `/usr/share/man/whatis` keyword database already exists.

5. To display all `ftp` command related articles in the `/usr/share/man` or `/usr/share/man/local` path, enter:
   
   ```
   man -M/usr/share/man:/usr/share/man/local ftp
   ```

Files

- `/usr/bin/man` Contains the `man` command.
- `/usr/share/man` Standard manual directory structure.
- `/usr/share/man/whatis` Contains the keyword database.

Related Information

The `apropos` command, `catman` command, `more` command, `whatis` command, `whereis` command.

managefonts Command

Purpose

Provides the user with a simple menu-based interface to update or change the set of installed font families on the system.

**Note:** You must have root user authority to run the `managefonts` script. The `managefonts` script is contained in the `/usr/lib/ps/ditroff.fonts/managefonts` file.

Syntax

```
managefonts [ Option ]
```  

Description

The `managefonts` command provides the user with a simple menu-based interface to update or change the set of installed font families on the system. If no command line arguments are provided, the menu-based interface is used. Command-line arguments can be used to provide the equivalent of the menu selections.

A set of font families is installed on the system at the time the TranScript Tools option of the Text Formatter Services Package is installed on the system. This default setup includes the standard 13 fonts
comprising the Times, Courier, and Helvetica font families. You can use the program called up by the managefonts command to erase the current configuration and replace it with a new one. There are several predefined packages of font families that can be installed this way:

<table>
<thead>
<tr>
<th>Package</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Times Family Only</td>
<td>This is the most minimal configuration that allows the TranScript Tools option to run.</td>
</tr>
<tr>
<td>Standard13 Package</td>
<td>This package builds the Times, Courier, and Helvetica font families. This was the package installed on your system with TranScript.</td>
</tr>
<tr>
<td>Standard35 Package</td>
<td>This font family package includes the Standard13 package font families in addition to the following: Avant Garde, Bookman, New Century Schoolbook, and Palatino font families.</td>
</tr>
<tr>
<td>All Font Families</td>
<td>This package installs all the font families available for installation.</td>
</tr>
</tbody>
</table>

You can also use the managefonts command to add new font families one at a time. A menu of available fonts is displayed and users can select which font family they want to be built. The program prevents building of font families that are already installed.

The managefonts command includes help screens to assist the user in installing font families.

**Notes:**

1. Font families cannot be deleted directly. To delete font families, it is first necessary to install a package containing the minimal subset of families desired. After the package is installed, it is possible to add font families, one at a time, from the Individual Fonts Menu. For instance, if your current configuration is Times, Courier, and Helvetica, and you want only Times and Courier, you can use the managefonts program to install the Times Only Package.

2. There is no command-line syntax equivalent to the menu items in the managefonts program.

The command line arguments are acted upon in the order they are given, reading left to right. The following are the valid values for the option parameter and their meanings:

- init0: Initialize for the installation of a font package.
- clean: Remove all temporary files and previously installed fonts.
- cleanall: Remove all the temporary files, the previously installed fonts, and the TranScript troff font files installed.
- default: Install the Standard 13 fonts.
- standard35: Install the Standard 35 fonts.
- all: Install all possible fonts.
- CourierFamily: Install the Courier Family.
- HelveticaFamily: Install the Helvetica Family.
- HelvNarrowFamily: Install the Helvetica Narrow Family.
- AvantGardeFamily: Install the Avant Garde Family.
- BookmanFamily: Install the Bookman Family.
- GaramondFamily: Install the Garamond Family.
- LubalinFamily: Install the Lubalin Family.
- NewCenturyFamily: Install the New Century Family.
- OptimaFamily: Install the Optima Family.
- PalatinoFamily: Install the Palatino Family.
- SouvenirFamily: Install the Souvenir Family.
- ZapfFamily: Install the Zapf Family.
- BaseFamily: Install the Base Family, such as Times Roman.
Examples

1. To install the standard 13 fonts:
   
   ```bash
   managefonts cleanall standard13
   ```

2. To install the standard 35 fonts:
   
   ```bash
   managefonts cleanall standard35
   ```

3. To install all the fonts:
   
   ```bash
   managefonts cleanall all
   ```

4. To install the Courier Family (the Times Roman or Base Family must have been previously installed):
   
   ```bash
   managefonts init0 CourierFamily clean
   ```

Related Information

The `enscript` command, `ps630` command, `psrev` command, `ps4014` command, `psdit` command, `psplot` command.

mant Command

Purpose

Typesets manual pages.

Syntax

```bash
mant [-M Media] [-a] [-c] [-e] [-t] [-z] [-T Name] [ troffFlags ] [ File ... ]
```

Description

The `mant` command uses the manual page macros (`man` macro package) to typeset manual pages. The `File` parameter specifies the files to be processed by the `mant` command. Files must be displayed after all flags. If no file name is specified, the `mant` command prints a list of its flags. If a `-` (minus sign) is specified for the `File` parameter, standard input is read.

The `mant` command has flags to specify preprocessing by the `tbl` command, `cw` command, or `eqn` command. Flags from the `troff` command can be specified with the `troffFlags` parameter.

If the input contains a `troff` command comment line consisting solely of the string `"\" x (single quotation mark, backslash, double quotation mark, x), where x is any combination of the three letters c, e, and t, and where there is exactly one character space between the double quotation mark and x, then the input is processed through the appropriate combination of the `cw` command, `eqn` command, and `tbl` command, respectively, regardless of the command-line options.

>Note: Use the `-o List` flag of the `troff` command to specify ranges of pages to be output. Calling the `mant` command with one or more of the `-c` flag, `-e` flag, `-t` flag, and `-` (minus) flags together with the `-o List` flag of the `troff` command, give a broken pipe message if the last page of the document is not specified by the `List` variable. This broken pipe message is not an indication of any problem and can be ignored.

The `mant` command, unlike the `troff` command, automatically pipes its output to a specific postprocessor, according to the following flags, environment variable, or default setting unless specifically requested not to do so:

- `-z` Indicates that no postprocessors are used.
- `-T Name` Prepares the output for the printing device specified by the `Name` variable.
- `TYPESETTER` Specifies a particular printing device for the system environment.
- `default` Sends to `ibm3816`. 

452  Commands Reference, Volume 3
Flags, other than the ones in the following list, are passed to the \texttt{troff} command or to the macro package, as appropriate. All flags must be displayed before the specified file names.

**Flags**

All flags must appear before the specified file names.

- \texttt{-a}  
  Calls the \texttt{-a} flag of the \texttt{troff} command.

- \texttt{-c}  
  Preprocesses the input files with the \texttt{cw} command.

- \texttt{-e}  
  Preprocesses the input files with the \texttt{eqn} command.

- \texttt{-M Media}  
  Specifies a paper size in order to determine the amount of imageable area on the paper. Valid values for the \texttt{Media} variable are:

  - \texttt{A4}  
    Specifies a paper size of 8.3 X 11.7 inches (210 X 297 mm).

  - \texttt{A5}  
    Specifies a paper size of 5.83 X 8.27 inches (148 X 210 mm).

  - \texttt{B5}  
    Specifies a paper size of 6.9 X 9.8 inches (176 X 250 mm).

  - \texttt{EXEC}  
    Specifies a paper size of 7.25 X 10.5 inches (184.2 X 266.7 mm).

  - \texttt{LEGAL}  
    Specifies a paper size of 8.5 X 14 inches (215.9 X 355.6 mm).

  - \texttt{LETTER}  
    Specifies a paper size of 8.5 X 11 inches (215.9 X 279.4 mm). This is the default value.

- \texttt{-t}  
  Preprocesses the input files with the \texttt{tbl} command.

- \texttt{-z}  
  Prepares the output without the postprocessor.

- \texttt{-TName}  
  Prepares the output for the specified printing device. Possible \texttt{Name} variables are:

  - \texttt{ibm3812}  
    3812 Pageprinter II.

  - \texttt{ibm3816}  
    3816 Pageprinter.

  - \texttt{hplj}  
    Hewlett-Packard LaserJet II.

  - \texttt{ibm5587G}  
    5587-G01 Kanji Printer multi-byte language support.

  - \texttt{psc}  
    PostScript printer.

  - \texttt{X100}  
    AIXwindows display.

- -  
  Forces input to be read from standard input.

**Related Information**

The \texttt{cw} command, \texttt{eqn} command, \texttt{nroff} command, \texttt{tbl} command, \texttt{tc} command, \texttt{troff} command.

**mark Command**

**Purpose**

Creates, modifies, and displays message sequences.

**Syntax**

\begin{verbatim}
mark [ +Folder ] [ -list ] [ -sequence Name [ Messages... ] ] [ -add | -delete ] [ -zero | -nozero ] [ -public | -nopublic ]
\end{verbatim}
Description

The **mark** command creates, deletes, adds, and lists messages in a sequence. The **mark** command by default lists all of the sequences and their messages for the current folder. If you use the **-add** or **-delete** flag, you must also use the **-sequence** flag. When all messages are deleted from a sequence, the **mark** command removes the sequence name from the folder.

To create a new sequence, enter the **-sequence** flag with the name of the sequence you want to create. The **mark** command creates the sequence starting with the current message. By default, the **mark** command places the sequence in the current folder. If you specify a folder, that folder becomes the current folder.

Flags

- **-add**
  Adds messages to a sequence. The **-add** flag is the default. If you do not specify a message, the **mark** command uses the current message.

  **Note:** You can only use this flag with the **-sequence** flag.

- **-delete**
  Deletes messages from a sequence. If you do not specify a message, the current message is deleted by default.

  **Note:** You can only use this flag with the **-sequence** flag.

- **Folder**
  Specifies the folder to examine.

- **-help**
  Lists the command syntax, available switches (toggles), and version information.

  **Note:** For MH, the name of this flag must be fully spelled out.

- **-list**
  Displays the messages in a sequence. By default, the **-list** flag displays all the sequence names and messages defined for the current folder. To see a specific sequence, use the **-sequence** flag with the **-list** flag.

- **-nopublic**
  Restricts a sequence to your usage. The **-nopublic** flag does not restrict the messages in the sequence, only the sequence itself. This option is the default if the folder is write-protected from other users.

- **-nozero**
  Modifies the sequence by adding or deleting only the specified messages. This flag is the default.

- **-public**
  Makes a sequence available to other users. The **-public** flag does not make protected messages available, only the sequence itself. This flag is the default if the folder is not write-protected from other users.

- **-sequence Name**
  Specifies a sequence for the **-list** [**-add**] and **-delete** flags.

- **-zero**
  Clears a sequence of all messages except the current message. When the **-delete** flag is also specified, the **-zero** flag places all of the messages from the folder into the sequence before deleting any messages.

Messages

Specifies messages in a sequence. You can specify more than one message at a time. Messages are identified with following references:

- **Number**
  Number of the message

- **all**
  All the messages in a folder

- **cur or .** (period)
  Current message (the default)

- **first**
  First message in a folder

- **last**
  Last message in a folder

- **next**
  Message following the current message

- **prev**
  Message preceding the current message

If the **-list** flag is used, the default for the **Messages** parameter is **all**. Otherwise, the default is the current message.
Profile Entries
The following entry is found in the UserMHDirectory/context file:

Current-Folder: Specifies the default current folder.

The following entry is found in the $HOME/.mh_profile file:

Path: Specifies the MH directory.

Examples
1. To see the list of all sequences defined for the current folder, enter:
   
   mark
   
   The system displays a message similar to the following:
   
   cur: 94
test: 1-3 7 9

   In this example, message 94 is the current message number in the current folder. The message sequence called test includes message numbers 1, 2, 3, 7, and 9.

2. To see the list of all the sequences defined for the meetings folder, enter:
   
   mark +meetings
   
   The system displays a message similar to the following:
   
   cur: 5
dates: 12 15 19

3. To create a new message sequence called schedule in the current folder, enter:
   
   mark -sequence schedule
   
   The system displays the shell prompt to indicate that the schedule sequence was created. By default, the system adds the current message to the new sequence.

4. To delete message 10 from the schedule sequence, enter:
   
   mark -sequence schedule 10 -delete

Files

$HOME/.mh_profile Specifies the MH user profile.

/usr/bin/mark Contains the mark command.

Related Information

The pick command.

The mh_alias file format, mh_profile file format.

Mail Overview in AIX 5L Version 5.2 System User’s Guide: Communications and Networks.
**mesg Command**

**Purpose**
Permits or refuses write messages.

**Syntax**

```
mesg [n | y]
```

**Description**

The `mesg` command controls whether other users on the system can send messages to you with either the `write` command or the `talk` command. Called without arguments, the `mesg` command displays the current workstation message-permission setting.

The shell startup process permits messages by default. You can override this default action by including the line `mesg n` in your `$HOME/.profile` file. A user with root user authority can send write messages to any workstation, regardless of its message permission setting. Message permission has no effect on messages delivered through the electronic mail system (`sendmail`).

If you add `mesg y` to your `$HOME/.profile`, you will be able to receive messages from other users via the `write` command or the `talk` command.

If you add `mesg n` to your `$HOME/.profile`, you will not be able to receive messages from other users using the `write` command or the `talk` command.

**Flags**

- `n` Allows only the root user the permission to send messages to your workstation. Use this form of the command to avoid having others clutter your display with incoming messages.
- `y` Allows all workstations on the local network the permission to send messages to your workstation.

**Exit Status**

This command returns the following exit values:

- `0` Receiving messages is allowed.
- `1` Receiving messages is not allowed.
- `>1` An error occurred.

**Examples**

1. To allow only the root user the permission to send messages to your workstation, enter:

   ```
   mesg n
   ```

2. To allow everyone the permission to send messages to your workstation, enter:

   ```
   mesg y
   ```

3. To display what your current message-permission setting is, enter:

   ```
   mesg
   ```

   Information similar to the following is displayed:

   ```
   is y
   ```
In the previous example, the current message-permission setting is [X] (allowing all users on the local network the permission to send messages to your workstation). If you change the message-permission setting to [n] (allowing only the root user the permission to send messages to your workstation), information similar to the following is displayed:

```
is  n
```

Files
/dev/tty* Supports the controlling terminal interface.
$HOME/.profile Controls startup processes and daemons.

Related Information
The `sendmail`, `talk`, command, `write` command.

Network Overview in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

**mgmtsvr Command**

**Purpose**
mgmtsvr – Displays or sets the management server entry for the Managed node.

**Syntax**
mgmtsvr [-h] [-k] [-v | -V] [-N] [-n hostname] [host | -d]

**Description**
The `mgmtsvr` command runs on Managed nodes to display or set the management server for the node.

**Flags**

- **-d** Deletes the management server entry.
- **-h** Writes the command’s usage statement to standard output.
- **host** Specifies the host name or IP address of the CSM management server that will be managing this node. If no host is specified, and the `-d` flag is not specified, the current setting will be displayed. In this case, the exit status can be examined to distinguish between nodes and the management server.
- **-k** Exchanges RSCT public keys, host name, and node identifier between the node and the management server. Only use this flag if the host name, node identifier, or public key of either machines has changed. At the time this flag is used, you must trust your network to ensure that a rogue machine is not going to insert its public key in place of the public key of the real node.
- **-n hostname**
  Specifies the host name known to the CSM management server for this node. Passing a node name to the `updatenode` command is normally only needed for diagnostic purposes. The `updatenode` command uses the `-n` flag to ensure correct operation when the `hostname` value on the node resolves to a different name than the name known to the management server. The default is `/bin/hostname` Before using the `updatenode` command with the `-n` flag, you must run `chnode nodename AllowManageRequest=1`.
- **-N** Displays the value for the local node name and management server. This flag is valid only if a host name or IP address for the management server is not specified.
-v | -V
  Writes the command’s verbose messages to standard output.

**Parameters**
None.

**Security**
The command requires root access to the cluster management server.

**Exit Status**

0  This is a Managed node only and the management server value is set.
1  The combination of flags is not valid.
11 This is a Managed node only and the management server value is not set.
21 This node is its own management server.
22 This node is also a management server for a separate cluster.
23 This machine is a CSM management server and a node, but the node does not currently have a management server set for it.
31 This machine is a CSM management server only.
101 This machine is neither a CSM management server or node.

**Environment Variables**

**DSH_FANOUT**
  Sets the maximum number of concurrent remote shell commands. This can also be set by the -f flag.

**DSH_LIST**
  Specifies a file that contains definitions of the set of nodes that comprise the node list.

**DSH_LOG**
  Enables logging when set to the absolute path and file name of the log file. This variable is overridden by the --log and --noLogging flags.

**DSH_PATH**
  Sets the path that is used on the remote nodes. If DSH_PATH is not set, the default path for the remote shell is used. For example, DSH_PATH=$PATH sets the path on the remote node to the same path that is used on the source node.

**DSH_REMOTE_CMD**
  Specifies the path of the remote shell executable to use instead of the default.

**DSH_REMOTE_OPTS**
  Includes the options specified in the remote command when the command is forwarded to the remote nodes.

**DSH_REPORTS_DIRECTORY**
  Enables reporting and output file writing when set to the reports directory path. This variable is overridden by the --reports and the --noReporting flags.

**DSH_TIMEOUT**
  Specifies the time, in seconds, to wait for response from the remote shell. This can also be set with the -t flag.

**WCOLL**
  Equivalent to DSH_LIST.
Standard Error

When the dshbak filter is used and standard error messages are generated, all error messages on standard error appear before all standard output messages. This is true with and without the -c flag.

Examples

To query the current CSM management server for this node, enter:

mgmtsvr

Location

/opt/csm/bin/mgmtsvr

Related Information

- The updatenode command.
- See the cluster security section in the IBM CSM for AIX 5L: Administration Guide for information on editing the RMC ACL file.

mhl Command

Purpose

Produces formatted listings of messages.

Syntax

mhl [ [ -form FormFile ] | [ -folder +Folder ] | [ -moreproc Command | -nomoreproc ] | [ -bell | -nobell ] | [-clear | -noclear ] ] [ -length Number ] [ -width Number ]

Description

The mhl command creates formatted lists of messages. The command is usually started through the showproc: profile entry or through the -showproc flag in other MH commands. When displaying messages, the mhl command uses the directions listed in the format file. If you specify more than one message, the mhl command provides a prompt before displaying each screen of messages.

If the -nomoreproc flag is specified, the mhl command prompts the user to press the Return key (the Ctrl-D key sequence is also acceptable) to see the next message. To stop the current message output and receive a prompt for the next message, press the Ctrl-D key sequence. Press the QUIT key sequence to stop the command output.

Note: To use the mhl command, you must make the folder you wish to work with the current directory.

Flags

-bell

Produces a bell at the end of each page. When the -nomoreproc flag is specified or the moreproc: profile entry is defined, but empty, the -bell flag is the default.

-clear

Clears the screen after each page when the output device is a display. The mhl command uses the STERM environment variable to determine the type of display. When the output device is not a display, the -clear flag inserts a form feed character at the end of each message. This flag affects the mhl command only if the moreproc: profile entry is defined and empty.

-folder +Folder

Identifies the folder to be used for the mhl.format file’s MessageName: entry. The default is the value of the Smhfolder environment variable.
**-form FormFile**

Specifies a file containing an alternate output format. The default format is described in the UserMHDirectory/mhl.format file. If this file does not exist, the mhl command uses the system default format described in the /etc/mh/mhl.format file.

**-help**

Lists the command syntax, available switches (toggles), and version information.

*Note:* For MH, the name of this flag must be fully spelled out.

**-length Number**

Sets the screen length for the output. The default is the value indicated by the $TERM environment variable. If that value is not appropriate, the default is 40 lines.

**-moreproc Command**

Uses the value of the Command variable instead of the value of the moreproc: entry specified in the $HOME/.mh_profile file.

**-nobell**

Suppresses the bell at the end of each page. This flag affects the mhl command only if the output device is a display, the -nomoreproc flag is used, or the moreproc: profile entry is defined and empty.

**-noclear**

Prevents clearing of the screen at the end of each page when the output device is a display. When the output device is not a display, the -clear flag does not insert a form-feed character at the end of each message. This flag is the default when the -moreproc flag is used or the moreproc: entry is defined and is empty.

**-nomoreproc**

Sets the moreproc: entry as an empty value.

**-width Number**

Sets the screen width for the output. The default is the value indicated by the $TERM environment variable. If that value is not appropriate, the default is 80 characters.

---

**Profile Entries**

The following entry is found in the UserMHDirectory/.mh_profile file:

moreproc: Specifies the interactive program for communicating with the user.

---

**Examples**

1. To list message 5 in the inbox folder, change the directory to inbox:
   
   ```
   cd /home/mickey/Mail/inbox
   ```
   
   Then enter:
   
   ```
   /usr/lib/mh/mhl 5
   ```
   
   A display similar to the following appears:
   
   ```
   --- Using template MHL.FORMAT ---
   Date:
   To:
   cc:
   From:
   Subject:
   Message Text
   ```

2. To display more than one message, enter:
   
   ```
   /usr/lib/mh/mhl 5 6 7
   ```

---

**Files**

- **$HOME/mh_profile**
  - Contains the MH user profile.

- **/etc/mh/mhl.format**
  - Defines the default MH message template.

- **UserMHDirectory/mhl.format**
  - Specifies a user's default message template. (If it exists, it overrides the default MH message template.)
Related Information

The `ap` command, `dp` command, `next` command, `prev` command, `show` command.

The `mh_alias` file format, `mh_profile` file format.

Mail Overview in AIX 5L Version 5.2 System User’s Guide: Communications and Networks.

mhmail Command

Purpose

Sends or receives mail.

Syntax

```
mhmail User ... [ -cc User ... ] [ -from User ... ] [ -subject "String" ] [ -body "String" ]
```

Description

The `mhmail` command composes, sends, and files messages. To file a message, enter the `mhmail` command without any flags. The default folder is $HOME/inbox.

If you specify one or more user addresses with the `User` parameter, the `mhmail` command accepts text from your terminal and composes a message. You can end the message text by pressing the Ctrl-D key sequence. The `mhmail` command sends a copy of the message to each specified address.

Flags

- **-body "String"**
  
  Sends a message with the specified string as the body. You must enclose the string in quotes. When you specify the `-body` flag, the `mhmail` command does not accept text from the terminal.

- **-cc User...**
  
  Sends a copy of the message to the specified users. The `mhmail` command puts the addresses in the `cc:` field.

- **-from User...**
  
  Places the specified user address in the `From:` field of the message.

- **-help**
  
  Lists the command syntax, available switches (toggles), and version information.

- **-subject "String"**
  
  Places the specified text string in the `Subject:` field of the message.

Note: For MH, the name of this flag must be fully spelled out.

Examples

1. To receive new mail and file it into the default mail folder, $USER/MAIL/inbox, enter:

   
   ```
   mhmail
   ```

   The system displays a message similar to the following:

   ```
   Incorporating new mail into inbox...
   65+ 04/08 jim@athena.a Meeting <<The meeting will
   66 04/08 jim@athena.a Schedule <<Schedule change
   ```

   In this example, two messages are filed in the inbox file. The subject of the first message is `Meeting`, and the first line starts with the words `The meeting will`. The subject of the second message is `Schedule`, and the first line starts with the words `Schedule change`.

2. To send a message regarding a schedule change to user `jamie` on system `venus`, enter:
mhmail jamie@venus -subject "Schedule Change"

The system waits for you to enter the text of the message. After completing the last line of the text, press the Enter key and then the Ctrl-D key sequence to send the message.

**Files**

/var/spool/Mail/$USER Defines the location of the mail drop.
/usr/bin/mhmail Contains the `mhmail` command.

**Related Information**
The `inc` command, `post` command.
The `mh_alias` file format, `mh_profile` file format.


---

**mhpath Command**

**Purpose**
Prints full path names of messages and folders.

**Syntax**

```
mhpath [+Folder] [ Messages [,Messages] ... ]
```

**Description**
The `mhpath` command lists the path names of folders and messages. By default, the command lists the path name of the current folder.

**Flags**

+Folder Specifies which folder path to list.
-help Lists the command syntax, available switches (toggles), and version information.

**Note:** For MH, the name of this flag must be fully spelled out.
**Messages**  
Specifies the messages for which you want to list path names. The *Messages* parameter can specify several messages, a range of messages, or a single message. Use the following references to specify messages.

**Number**  
Number of the message. When specifying multiple messages, separate each message number with a comma. When specifying a range of messages, separate the upper and lower ends of the range with a hyphen.

*Note:* You cannot use the *new* variable when specifying a range.

**Sequence**  
A group of messages specified by the user. Recognized values include:

- **all**  
  All the messages in a folder.
- **cur or . (period)**  
  Current message.
- **first**  
  First message in a folder.
- **last**  
  Last message in a folder.
- **new**  
  Path name that the system will assign to the next message that is incorporated.
- **next**  
  Message following the current message.
- **prev**  
  Message immediately before the current message.

---

**Profile Entries**  
The following entries are entered in the *UserMhDirectory*.mh_profile file:

- **Current-Folder:**  
  Sets the default current folder.
- **Path:**  
  Specifies a user’s MH directory.

**Examples**

1. To list the path name of the current folder, enter:

   ```
   mhpah
   ```

   The system responds with a message similar to the following:

   ```
   /home/tom/Mail/inbox
   ```

2. To list the path names for messages 2 through 4 in the *source* folder, enter:

   ```
   mhpah source 2-4
   ```

   The system responds with a message similar to the following:

   ```
   /home/tom/Mail/source/2
   /home/tom/Mail/source/3
   /home/tom/Mail/source/4
   ```

3. To list the path name the system will assign to the next message added to the current folder, enter:

   ```
   mhpah new
   ```

   The system responds with a message similar to the following:

   ```
   /home/tom/Mail/source/5
   ```
In this example, the next message will be message 5 in user tom's current folder, /home/tom/Mail/source.

**Files**

$HOME/.mh_profile          Defines the user's MH profile.
/usr/bin/mhpath             Contains the mhpath command.

**Related Information**

The [folder](#) command.

The [mh_alias](#) file format, [mh_profile](#) file format.


**migratelp Command**

**Purpose**

Moves allocated logical partition from one physical partition to another physical partition on a different physical volume.

**Syntax**

migratelp  *LVname*/LPartnumber[ /Copynumber ] *DestPV*/[PPartNumber]

**Description**

The `migratelp` moves the specified logical partition `LPartnumber` of the logical volume `LVname` to the `DestPV` physical volume. If the destination physical partition `PPartNumber` is specified it will be used, otherwise a destination partition is selected using the intra region policy of the logical volume. By default the first mirror copy of the logical partition in question is migrated. A value of 1, 2 or 3 can be specified for `Copynumber` to migrate a particular mirror copy.

**Note:** You must consider the partition usage, reported by `lvmstat`, on the other active concurrent nodes in case of a concurrent volume group.

The `migratelp` command fails to migrate partitions of striped logical volumes.

**Security**

To use `migratelp`, you must have root user authority.

**Examples**

1. To move the first logical partitions of logical volume lv00 to hdisk1, type:

   migratelp lv00/1 hdisk1

2. To move second mirror copy of the third logical partitions of logical volume hd2 to hdisk5, type:

   migratelp hd2/3/2 hdisk5

3. To move third mirror copy of the 25th logical partitions of logical volume testlv to 100th partition of hdisk7, type:

   migratelp testlv/25/3 hdisk7/100
Files

/usr/sbin Directory where the `migratelp` resides.

Related Information

The `lslv` command, and `vmstat` command.


migratepv Command

Purpose

Moves allocated physical partitions from one physical volume to one or more other physical volumes.

Syntax

`migratepv [ -l ] [ LogicalVolume ] SourcePhysicalVolume DestinationPhysicalVolume...`

Description

The `migratepv` command moves allocated physical partitions and the data they contain from the `SourcePhysicalVolume` to one or more other physical volumes. To limit the transfer to specific physical volumes, use the names of one or more physical volumes in the `DestinationPhysicalVolume` parameter; otherwise, all the physical volumes in the volume group are available for the transfer. All physical volumes must be within the same volume group. The specified source physical volume cannot be included in the list of `DestinationPhysicalVolume` parameters.

Notes:

1. To use this command, you must either have root user authority or be a member of the `system` group.
2. The `migratepv` command is not allowed on a snapshot volume group or a volume group that has a snapshot volume group.

The allocation of the new physical partitions follows the policies defined for the logical volumes that contain the physical partitions being moved.

The `migratepv` command (only when the source and target physical volumes are specified) fails when a boot logical volume is found on the source physical volume. When you migrate a physical volume, the boot logical volume must remain intact. Two contiguous physical partitions and the new boot image must be built on the new boot logical volume.

If you specify a logical volume that contains the boot image, the `migratepv -l` command attempts to find enough contiguous partitions on one of the target physical volumes. If the migration is successful, the `migratepv` command prints a message that recommends the user run the `bosboot` command to indicate a change in the boot device. The attempted migration fails if the `migratepv -l` command is unable to find enough contiguous space to satisfy the request.

Note: All Logical Volume Manager migrate functions work by creating a mirror of the logical volumes involved, then resynchronizing the logical volumes. The original logical volume is then removed. If the `migratepv` command is used to move a logical volume containing the primary dump device, the system will not have an accessible primary dump device during the execution of the command. Therefore, a dump taken during this execution may fail. To avoid this, reassign the primary dump device using the `sysdumpdev` command or ensure there is a secondary dump device defined before using `migratepv`. 
You can use the Volumes application in Web-based System Manager (wsm) to change volume characteristics. You could also use the System Management Interface Tool (SMIT) `smit migratetev` fast path to run this command.

**Note:** For concurrent mode volume groups `migratetev` can only be used while active in enhanced concurrent mode or active in concurrent mode on SSA disks.

### Flags

- `-i` Reads the `DestinationPhysicalVolume` parameter from standard input.
- `-l LogicalVolume` Moves only the physical partitions allocated to the specified logical volume and located on the specified source physical volume.

### Examples

1. To move physical partitions from `hdisk1` to `hdisk6` and `hdisk7`, enter:

   ```
   migratetev hdisk1 hdisk6 hdisk7
   ```

   Physical partitions are moved from one physical volume to two others within the same volume group.

2. To move physical partitions in logical volume `lv02` from `hdisk1` to `hdisk6`, enter:

   ```
   migratetev -l lv02 hdisk1 hdisk6
   ```

   Only those physical partitions contained in `lv02` are moved from one physical volume to another.

### Files

- `/usr/sbin` Directory where the `migratetev` command resides.
- `/tmp` Directory where the temporary files are stored while the command is running.

### Related Information

The `cplv` command, `lsv` command.


For information on installing the Web-based System Manager, see [Chapter 2: Installing Web-based System Manager](#) in *AIX 5L Version 5.2 Web-based System Manager Administration Guide.*

[System Management Interface Tool (SMIT)](#) in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.*


---

### mirrord Daemon

#### Purpose

Controls and monitors the mirror module for remote maintenance.
Syntax

mirrord

Description

The mirrord daemon controls and monitors the mirror module. The mirror daemon and mirror module work together to provide console mirroring, which is the two-way echoing of commands between the local operator’s system console and a remote service expert’s console. The local console, or BUMP console, is connected to line S1, and the remote console is connected to line S2 using a modem. The mirrord daemon is used to perform remote service support when the operating system is running.

Note: To use remote service, the Software Error Logging and Dump Service Aids Package must be installed, remote service support must be valid, and the remote authorization flag must be set.

Normally, the mirrord daemon is started during the boot phase, just after the console configuration, but it can also be started from the command line. If the remote service support flag is not set (no remote service agreement) or if the remote authorization flag is not set, the daemon does nothing and exits.

If both flags are set, the daemon checks the key mode switch. If your machine has a key mode switch and it is in the Normal or Secure position, the daemon sleeps until the key is placed in the Service position. When the key is placed in the Service position, the daemon wakes up and checks the remote authorization flag and that the tty lines S1 and S2 (special files /dev/ttyS1 and /dev/ttyS2) are managed by streams. If one of these checks fails, mirrord cannot perform console mirroring and returns a message explaining why not. If the checks pass, mirrord creates a lock file /etc/locks/mirror, kills processes belonging to line S2, pushes the mirror module, initializes line S2, and starts echoing in the mirror module. If your machine has the key mode switch, regardless of this switch, applications belonging to line S1 are never affected.

If your machine has a mode switch and it is already in the Service position and line S2 is connected when the daemon is started, the daemon simply pushes the mirror module (if necessary) and activates the echo mode. The daemon does not kill processes belonging to /dev/ttyS2 in this case, since a remote service session may be underway. The modem used for the line S2 connection is configured according to the file /usr/share/modems/mir_modem, and its tty is configured according to the file /usr/lib/mir_tty. Normally, these files are installed by service personnel and do not need to be modified.

If the mirrord command is executed when the daemon is already installed, an error message is returned.

The portmir command (available in AIX 4.2.1 and later) can be used on most systems to mirror the console.

Note: The mirrord command works only on multiprocessor systems with Micro Channel I/O. For IBM systems, this includes the IBM 7012 Model G Series, the IBM 7013 Model J Series, and the IBM 7015 Model R Series.

Signals

The daemon can be stopped using the SIGTERM or SIGKILL signals.

Examples

To start the daemon from the command line, simply enter:

mirrord

Files

/usr/lib/drivers/mirror The mirror streams module.
Related Information

The `portmir` command.

mirrorvg Command

Purpose
Mirrors all the logical volumes that exist on a given volume group. This command only applies to AIX 4.2.1 or later.

Syntax
`mirrorvg [ -S | -s ] [ -Q ] [ -C Copies ] [ -m ] VolumeGroup [ PhysicalVolume ... ]`

Description
The `mirrorvg` command takes all the logical volumes on a given volume group and mirrors those logical volumes. This same functionality may also be accomplished manually if you execute the `mklvcopy` command for each individual logical volume in a volume group. As with `mklvcopy`, the target physical drives to be mirrored with data must already be members of the volume group. To add disks to a volume group, run the `extendvg` command.

By default, `mirrorvg` attempts to mirror the logical volumes onto any of the disks in a volume group. If you wish to control which drives are used for mirroring, you must include the list of disks in the input parameters, `PhysicalVolume`. Mirror strictness is enforced. Additionally, `mirrorvg` mirrors the logical volumes, using the default settings of the logical volume being mirrored. If you wish to violate mirror strictness or affect the policy by which the mirror is created, you must execute the mirroring of all logical volumes manually with the `mklvcopy` command.

When `mirrorvg` is executed, the default behavior of the command requires that the synchronization of the mirrors must complete before the command returns to the user. If you wish to avoid the delay, use the `-S` or `-s` option. Additionally, the default value of 2 copies is always used. To specify a value other than 2, use the `-c` option.

Notes:
1. To use this command, you must either have root user authority or be a member of the `system` group.
2. The `mirrorvg` command is not allowed on a snapshot volume group.

Attention: The `mirrorvg` command may take a significant amount of time before completing because of complex error checking, the amount of logical volumes to mirror in a volume group, and the time is takes to synchronize the new mirrored logical volumes.

You can use the Volumes application in Web-based System Manager (wsm) to change volume characteristics. You could also use the System Management Interface Tool (SMIT) `smit mirrorvg` fast path to run this command.
Flags

-c Copies

Specifies the minimum number of copies that each logical volume must have after the `mirrorvg` command has finished executing. It may be possible, through the independent use of `mklvcopy`, that some logical volumes may have more than the minimum number specified after the `mirrorvg` command has executed. Minimum value is 2 and 3 is the maximum value. A value of 1 is ignored.

-m exact map

Allows mirroring of logical volumes in the exact physical partition order that the original copy is ordered. This option requires you to specify a `PhysicalVolume(s)` where the exact map copy should be placed. If the space is insufficient for an exact mapping, then the command will fail. You should add new drives or pick a different set of drives that will satisfy an exact logical volume mapping of the entire volume group. The designated disks must be equal to or exceed the size of the drives which are to be exactly mirrored, regardless of if the entire disk is used. Also, if any logical volume to be mirrored is already mirrored, this command will fail.

-Q Quorum Keep

By default in `mirrorvg`, when a volume group’s contents becomes mirrored, volume group quorum is disabled. If the user wishes to keep the volume group quorum requirement after mirroring is complete, this option should be used in the command. For later quorum changes, refer to the `chvg` command.

-S Background Sync

Returns the `mirrorvg` command immediately and starts a background `syncvg` of the volume group. With this option, it is not obvious when the mirrors have completely finished their synchronization. However, as portions of the mirrors become synchronized, they are immediately used by the operating system in mirror usage.

-s Disable Sync

Returns the `mirrorvg` command immediately without performing any type of mirror synchronization. If this option is used, the mirror may exist for a logical volume but is not used by the operating system until it has been synchronized with the `syncvg` command.

The following is a description of `rootvg`:

**rootvg mirroring**

When the `rootvg` mirroring has completed, you must perform three additional tasks: `bosboot`, `bootlist`, and `reboot`.

The `bosboot` command is required to customize the bootrec of the newly mirrored drive. The `bootlist` command needs to be performed to instruct the system which disk and order you prefer the mirrored boot process to start.

Finally, the default of this command is for Quorum to be turned off. For this to take effect on a `rootvg` volume group, the system must be rebooted.

**non-rootvg mirroring**

When this volume group has been mirrored, the default command causes Quorum to deactivate. The user must close all open logical volumes, execute `varyoffvg` and then `varyonvg` on the volume group for the system to understand that quorum is or is not needed for the volume group. If you do not `revaryon` the volume group, mirror will still work correctly. However, any quorum changes will not have taken effect.

**rootvg and non-rootvg mirroring**

The system dump devices, primary and secondary, should not be mirrored. In some systems, the paging device and the dump device are the same device. However, most users want the paging device mirrored. When `mirrorvg` detects that a dump device and the paging device are the same, the logical volume will be mirrored automatically.

If `mirrorvg` detects that the dump and paging device are different logical volumes, the paging device is automatically mirrored, but the dump logical volume is not. The dump device can be queried and modified with the `sysdumpdev` command.
Examples

1. To triply mirror a volume group, enter:

   ```
mirrorvg -c 3 workvg
   ```

   The logical partitions in the logical volumes held on `workvg` now have three copies.

2. To get default mirroring of `rootvg`, enter:

   ```
mirrorvg rootvg
   ```

   `rootvg` now has two copies.

3. To replace a bad disk drive in a mirrored volume group, enter

   ```
unmirrorvg workvg hdisk7
reducevg workvg hdisk7
rmdev -l hdisk7 -d
replace the disk drive, let the drive be renamed hdisk7
extendvg workvg hdisk7
mirrorvg workvg
   ```

   **Note:** By default in this example, `mirrorvg` will try to create 2 copies for logical volumes in `workvg`. It will try to create the new mirrors onto the replaced disk drive. However, if the original system had been triply mirrored, there may be no new mirrors created onto `hdisk7`, as other copies may already exist for the logical volumes.

4. To sync the newly created mirrors in the background, enter:

   ```
mirrorvg -S -c 3 workvg
   ```

5. To create an exact mapped volume group, enter:

   ```
mirrorvg -m datavg hdisk2 hdisk3
   ```

Files

`/usr/sbin` Directory where the `mirrorvg` command resides.

Related Information

The `mklvcopy` command, `unmirrorvg` command, `syncvg` command, `extendvg` command, `reducevg` command, `sysdumpdev` command.

For information on installing the Web-based System Manager, see [Chapter 2: Installing Web-based System Manager](#) in *AIX 5L Version 5.2 Web-based System Manager Administration Guide*.

The `Logical Volume Storage Overview` in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices*.

**mk_niscachemgr Command**

**Purpose**

Uncomments the entry in the `/etc/rc.nfs` file for the `nis_cachemgr` daemon and invokes the daemon by using the `startsrc` command.

**Syntax**

```
/usr/sbin/mk_niscachemgr [-I] [-B] [-N] 
```
Description

The `mk_niscachemgr` command uncomments the entry in the `/etc/rc.nfs` file for the `nis_cachemgr` daemon. The `mk_niscachemgr` command starts the daemon by using the `startsrc` command.

Note: The `mk_nisd`, `mk_cachemgr`, `mk_nispasswdd`, `rm_nisd`, `rm_cachemgr`, and `rm_nispasswdd` commands do two things:

- Alter the entries of daemon startup calls in `/etc/rc.nfs`.
- Alter the default behavior of the daemon `src` entities.

For example, if the `rpc.nisd` daemon is supposed to start with the `-Y` flag, this will not be explicitly set in the `/etc/rc.nfs` entry for starting the `rpc.nisd` daemon. Instead, a `chssys` is executed to place the default options which are added (if any) to the daemons during startup. To verify that these options exist, use the `lssrc -S -s subsystem` command to show the default options.

Flags

- `-I` Uncomments the entry in the `/etc/rc.nfs` file to start the `nis_cachemgr` daemon on the next system restart.
- `-B` Uncomments the entry in the `/etc/rc.nfs` file to start the `nis_cachemgr` daemon and uses the `startsrc` command to start the `nis_cachemgr` daemon. This flag is the default.
- `-N` Uses the `startsrc` command to start the `nis_cachemgr` daemon. This flag does not change the `/etc/rc.nfs` file.

Examples

To modify the `/etc/rc.nfs` file to invoke the `nis_cachemgr` daemon on the next system restart, enter:

```
mk_niscachemgr -I
```

Files

```
/etc/rc.nfs
```

Contains the startup script for the NFS and NIS daemons.

Related Information

The `smit` command, `startsrc` command, and the `nis_cachemgr` daemon.

[Network Information Services+ (NIS+) Overview for System Management in AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide](#)

[Network File System (NFS) Overview for System Management Guide: Communications and Networks](#)

[How to Start and Stop the NIS+ Daemons](#)

How to Export a File System Using Secure NFS

For information on installing the Web-based System Manager, see [Chapter 2: Installing Web-based System Manager Administration Guide](#).
mk_nisd Command

Purpose
Uncomments the entry in the /etc/rc.nfs file for the rpc.nisd daemon and invokes the daemon by using the startsrc command.

Syntax

Description
The mk_nisd command uncomments the entry in the /etc/rc.nfs file for the rpc.nisd daemon. The mk_nisd command starts the daemon by using the startsrc command.

Note: The mk_nisd, mk_cachemgr, mk_nispasswdd, rm_nisd, rm_cachemgr, and rm_nispasswdd commands do two things:
• Alter the entries of daemon startup calls in /etc/rc.nfs.
• Alter the default behavior of the daemon src entities.

For example, if the rpc.nisd daemon is supposed to start with the -Y flag, this will not be explicitly set in the /etc/rc.nfs entry for starting the rpc.nisd daemon. Instead, a chssys is executed to place the default options which are added (if any) to the daemons during startup. To verify that these options exist, use the lssrc -S -s subsystem command to show the default options.

Flags
- I Uncomments the entry in the /etc/rc.nfs file to start the rpc.nisd daemon on the next system-restart.
- B Uncomments the entry in the /etc/rc.nfs file to start the rpc.nisd daemon and uses the startsrc command to start the rpc.nisd daemon. This flag is the default.
- N Uses the startsrc command to start the rpc.nisd daemon. This flag does not change the /etc/rc.nfs file.
- s Starts the rpc.nisd with no DES authentication. If this flag is not used, the default rpc.nisd behavior is to always start with DES authentication. The -s option is used to make the rpc.nisd compatible with NIS(YP) clients.
- y Causes the rpc.nisd daemon to emulate a NIS(YP) service. This is not the default setting of rpc.nisd or mk_nisd.
- b Causes the rpc.nisd daemon to emulate the NIS(YP) DNS resolver service. This is not the default setting of rpc.nisd or mk_nisd.

Note: The settings that result from using the -a, -y, and -b flags remain the default behavior of rpc.nisd after a system reboot if the -I or -B flags were used. The only way to restore settings is by executing rm_nisd and then executing mk_nisd once again.

Examples
1. To modify the /etc/rc.nfs file to invoke the rpc.nisd daemon on the next system-restart, enter:
mk_nisd -I
2. To start the rpc.nisd daemon without DES authentication and to modify the /etc/rc.nfs file to invoke the rpc.nisd daemon without DES authentication upon reboot:
mk_nisd -B -s

Files
/etc/rc.nfs Contains the startup script for the NFS and NIS daemons.
Related Information
The `smit` command and the `startsrc` command.

The `rpc.nisd` daemon.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide

---

**mk_nispasswdd Command**

### Purpose

Uncomments the entry in the `/etc/rc.nfs` file for the `rpc.nispasswdd` daemon and invokes the daemon by using the `startsrc` command.

### Syntax

```
/usr/sbin/mk_nispasswdd [ -I ] [ -B ] [ -N ]
```

### Description

The `mk_nispasswdd` command uncomments the entry in the `/etc/rc.nfs` file for the `rpc.nispasswdd` daemon. The `mk_nispasswdd` command starts the daemon using the `startsrc` command.

**Note:** The `mk_nisd`, `mk_cachemgr`, `mk_nispasswdd`, `rm_nisd`, `rm_cachemgr`, and `rm_nispasswdd` commands do two things:

- Alter the entries of daemon startup calls in `/etc/rc.nfs`.
- Alter the default behavior of the daemon `src` entities.

For example, if the `rpc.nisd` daemon is supposed to start with the `-Y` flag, this will not be explicitly set in the `/etc/rc.nfs` entry for starting the `rpc.nisd` daemon. Instead, a `chssys` is executed to place the default options which are added (if any) to the daemons during startup. To verify that these options exist, use the `lssrc -S -s subsystem` command to show the default options.

### Flags

- `-I` Uncomments the entry in the `/etc/rc.nfs` file to start the `rpc.nispasswdd` daemon on the next system restart.
- `-B` Uncomments the entry in the `/etc/rc.nfs` file to start the `rpc.nispasswdd` daemon and uses the `startsrc` command to start the `rpc.nispasswdd` daemon. The `-B` flag is the default.
- `-N` Uses the `startsrc` command to start the `rpc.nispasswdd` daemon. The `-N` flag does not change the `/etc/rc.nfs` file.
Examples
1. To modify the /etc/rc.nfs file to invoke the rpc.nispasswdd daemon on the next system restart, enter:
   mk_nispasswdd -I

Files
/etc/rc.nfs Contains the startup script for the NFS and NIS daemons.

Related Information
The smit command and startsrc command.

The rpc.nispasswdd daemon.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

NIS+ Reference

mkboot Command

Purpose
Creates the boot image, the boot record, and the service record. This command is not a user-level command and is NOT supported in AIX 4.2 or later.

Syntax
mkboot [d Device [b] [D] [c] [h] [f] [l] [k Kernel [e Expander] [l] [s] [r] [p Offset] [w] [f] FileSystem

Description
The mkboot command combines a kernel and file system into a boot image. The resulting image is written to standard out. It is copied to a boot device with the appropriate boot record information. The boot image can be made compressed or uncompressed and with or without a boot record at the beginning of the image. An image created for a tape is compressed with the boot record at the start of the image file. A disk boot image may be created without compression and has no boot record. The boot record is written to the first sector of the disk. The record contains information about the size and location of the image after it is written to the boot logical volume on that disk.

If the boot logical volume is mirrored, the mkboot command not only writes the boot image to each copy of the boot logical volume but also writes a boot record to each physical disk comprising the mirror. As long as the mkboot command is able to update at least one of the copies of a mirrored boot logical
volume, no error is returned. To enable booting from each copy of a mirrored boot logical volume, each of
the physical disks must be specified using the `bootlist` command. For more information regarding mirrored
logical volumes, see [Logical Volume Storage Overview](https://www.aix.ibm.com/5l/platform/scala/docs.htm) in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices*.

The `mkboot` command is usually called by the `bosboot` command. However, you can run the `mkboot`
command a second time to put expand code at the beginning of a compressed boot image.

### Flags

- **-b**
  Zeros out save-base fields. This flag is optional.
- **-d Device**
  Specifies the device required for the IPL record. This flag is required.
- **-c**
  Zeros out the boot record on the device. This flag is optional.
- **-D**
  Loads the low-level debugger at boot time.
- **-e Expander**
  Specifies kernel expansion code to create a compressed boot image file.
  Either the `-e` flag or the `-k` flag must be specified.
- **-f FileSystem**
  Specifies the boot file system. This flag is required.
- **-h**
  Prevents the `mkboot` command from Updating the boot header. This flag is optional.
- **-i**
  Writes the normal portion of the boot record.
- **-I (uppercase i)**
  Invokes the low-level debugger at boot time.
- **-k Kernel**
  Specifies the kernel in the boot image. Either the `-k` flag or the `-e` flag
  must be specified.
- **-l (lowercase L) LVDev**
  Specifies the logical volume device that contains the loadable boot code.
- **-L**
  Enables lock instrumentation for MP systems. This flag has no effect on
  systems that are not using the MP kernel.
- **-p Offset**
  Specifies the address to use as `boot_pr_start` field in the boot record.
  This flag is used in creating the CD-ROM boot image. This flag is
  optional.
- **-r**
  Creates an image that is read-only (ROS) emulation code.
- **-s**
  Writes the service portion of the boot record.
- **-w**
  Outputs first two blocks of boot logical volume before the boot image.
  This flag is applicable to disk boot image only.

### Security

Access Control: Only the root user can read and execute this command.

### Examples

1. To create an uncompressed boot image, using the kernel `/usr/lib/boot/unix` and the `/tmp/bootfs` file
   system for the device `/dev/hdisk0`, enter:
   ```
   mkboot -d /dev/hdisk0 -k /usr/lib/boot/unix -f /tmp/bootfs \
   -b -i -s > /tmp/image
   ```
2. To clear the boot record but leave the PVID for disk `hdisk0`, enter:
   ```
   mkboot -d /dev/hdisk0 -c
   ```
3. Although the `mkboot` command combines a kernel and a random access memory (RAM) file system
to create one boot image, you can run the `mkboot` command a second time to put expand code at
the beginning of a compressed boot image. For example, enter:
   ```
   mkboot -b -d /dev/rmt0 -k unix -f ramfs | compress > /tmp/image
   mkboot -b -i -s -d /dev/rmt0 -k bootexpand -f /tmp/image \
   > bootfile
   ```
   for a bootable tape where:
   ```
   unix
   ```
   Specifies the kernel.
ramfs
Specifies the RAM disk file system.

compress
Specifies the compression or compact routine.

bootexpand
Specifies the expansion or kernel uncompact routine.

Files

/files/usr/include/sys/bootrecord.h
Specifies the structure of the boot record.

Related Information

The bosboot command, and lockstat command.


mkC2admin Command

Purpose
Configure a system to operate in C2 Security Mode.

Syntax

mkC2admin {{ -m } | { -a address } hostname}

Description

The mkC2admin command initializes the security directories for use in a C2 System configuration. The distributed database directories are created and symbolic links initialized. When a system is being configured as the Administrative Host (using the -m flag), an additional file system is created to hold the master copies of the administrative database files. Those files are stored in the directory /etc/data.master which has a logical volume name of hd10sec.

The administrative database files are divided into three categories. Those files that must be shared, those files that optionally may be shared, and those files that may not be shared. Optionally sharable files are described in the file /etc/security/files.config. That file consists of multiple lines of the format:

[y|n] filename

and is editable by the administrator. To select an optionally sharable filename, the administrator sets the first field to the value y. To make an optionally sharable file be unshared, the field is set to the value n. All hosts in the C2 System must have an identical /etc/security/files.config file.

The system hostname must be defined in the /etc/hosts file at the time this command is run. If not, the IP address of the new C2 System Administrative Host may be provided with the -a option, and an entry will be added to /etc/hosts.

Flags

-a address
Use address as the IP address of hostname.

-m
Configure the host as the administrative master.

Parameters

hostname
Specifies the hostname.
Exit Status
0  The system has been properly configured to operate in the C2 mode.
1  The system was not installed with the C2 option.
2  The system could not be successfully configured to operate in C2 mode.
3  The system was previously configured to operate in C2 mode without having first been unconfigured.

Files
/usr/sbin/mkC2admin Contains the mkC2admin command.

Related Information
The chC2admin command, isC2host command, lsC2admin command, rmC2admin command.

mkcatdefs Command

Purpose
Preprocesses a message source file.

Syntax
mkcatdefs SymbolName SourceFile ... [-h]

Description
The mkcatdefs command preprocesses a message source file for input to the gencat command.

The SourceFile message file contains symbolic identifiers. The mkcatdefs command produces the SymbolName_msg.h file, containing statements that equate symbolic identifiers with the set numbers and message ID numbers assigned by the mkcatdefs command.

The mkcatdefs command creates two outputs. The first is a header file called SymbolName_msg.h. You must include this SymbolName_msg.h file in your application program to associate the symbolic names to the set and message numbers assigned by the mkcatdefs command.

The mkcatdefs command sends message source data, with numbers instead of symbolic identifiers, to standard output. This output is suitable as input to the gencat command. You can use the mkcatdefs command output as input to the gencat command in the following ways:

- Use the mkcatdefs command with a > (redirection symbol) to write the new message source to a file. Use this file as input to the gencat command.
- Pipe the mkcatdefs command output file directly to the gencat command.
- Use the runcat command rather than the mkcatdefs command. The runcat command automatically sends the message source file through the mkcatdefs command and then pipes the file to the gencat command.

After running the mkcatdefs command, you can use symbolic names in an application to refer to messages.
Flags

-h  Suppresses the generation of a SymbolName_msg.h file. This flag must be the last argument to the mkcatdefs command.

Examples

To process the symb.msg message source file and redirect the output to the symb.src file, enter:

```
mkcatdefs symb symb.msg > symb.src
```

The generated symb_msg.h file looks similar to the following:

```c
#ifdef _H_SYMB_MSG
#define _H_SYMB_MSG
#include <limits.h>
#include <nl_types.h>
#define MF_SYMB "symb.cat"
/* The following was generated from symb.src. */
/* definitions for set MSFAC */
#define SYM_FORM 1
#define SYM_LEN 2
#define MSG_H 6
#endif
```

The mkcatdefs command also creates the symb.src message catalog source file for the gencat command with numbers assigned to the symbolic identifiers:

```c
$quote " Use double quotation marks to delimit message text
$delset 1
$set 1
1 "Symbolic identifiers can only contain alphanumeric \n characters or the _ (underscore character)\n"
2 "Symbolic Identifiers cannot be more than 65 \n characters long"
5 "You can mix symbolic identifiers and numbers\n"
$quote
6 remember to include the "msg_h" file in your program
```

The assigned message numbers are noncontiguous because the source file contained a specific number. The mkcatdefs program always assigns the previous number plus 1 to a symbolic identifier.

Note: The mkcatdefs command inserts a $delset command before a $set command in the output message source file. This means you cannot add, delete, or replace single messages in an existing catalog when piping to the gencat command. You must enter all messages in the set.

Files

```
/usr/bin/mkcatdefs
```

Contains the mkcatdefs command.

Related Information

The dspcat command, dspmsg command, gencat command, runcat command.

The catclose subroutine, catgets subroutine, catopen subroutine.

mkCCadmin Command

**Purpose**
Configure a system to operate in Common Criteria enabled Security Mode.

**Syntax**
```plaintext
mkCCadmin { [-m] | [-a address] hostname }
```

**Description**
The `mkCCadmin` command initializes the security directories for use in a Common Criteria enabled System configuration. The distributed database directories are created and symbolic links initialized. When a system is being configured as the Administrative Host (using the `-m` flag), an additional file system is created to hold the master copies of the administrative database files. Those files are stored in the directory `/etc/data.master` which has a logical volume name of `hd10sec`.

The administrative database files are divided into three categories. Those files that must be shared, those files that optionally may be shared, and those files that may not be shared. Optionally sharable files are described in the file `/etc/security/files.config`. That file consists of multiple lines of the format:

```
[y|n] filename
```

and is editable by the administrator. To select an optionally sharable filename, the administrator sets the first field to the value `y`. To make an optionally sharable file be unshared, the field is set to the value `n`. All hosts in the Common Criteria enabled System must have an identical `/etc/security/files.config` file.

The system `hostname` must be defined in the `/etc/hosts` file at the time this command is run. If not, the IP address of the new Common Criteria enabled System Administrative Host may be provided with the `-a` option, and an entry will be added to `/etc/hosts`.

**Flags**
- `-a address` Use address as the IP address of `hostname`.
- `-m` Configure the host as the administrative master.

**Parameters**

*hostname* Specifies the hostname.

**Exit Status**

<table>
<thead>
<tr>
<th>Status Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>The system has been properly configured to operate in the Common Criteria enabled mode.</td>
</tr>
<tr>
<td>1</td>
<td>The system was not installed with the Common Criteria enabled option.</td>
</tr>
<tr>
<td>2</td>
<td>The system could not be successfully configured to operate in Common Criteria enabled mode.</td>
</tr>
<tr>
<td>3</td>
<td>The system was previously configured to operate in Common Criteria enabled mode without having first been unconfigured.</td>
</tr>
</tbody>
</table>

**Files**

* `/usr/sbin/mkCCadmin`* Contains the mkCCadmin command.
Related Information
The `chCCadmin` command, `isCChost` command, `IsCCadmin` command, `rmCCadmin` command.

---

**mkcd Command**

**Purpose**

Creates a multi-volume CD (or CDs) from a `mksysb` or `savevg` backup image.

**Syntax**

```bash
```

**Description**

The `mkcd` command creates a system backup image (`mksysb`) to CD-Recordable (CD-R) or DVD-Recordable (DVD-R, DVD-RAM) from the system `rootvg` or from a previously created `mksysb` image. It also creates a volume group backup image (`savevg`) to CD-R from a user-specified volume group or from a previously created `savevg` image.

For DVD media, system backups made with the `mkcd` command have a limitation in that they expect the media to be 4.7 GB or larger per side. The `mkcd` command will not process the next volume until it writes over 4 GB on the current volume, thus the use of smaller media would result in corruption when going beyond the media's capacity.

With the `mkcd` command, you can create bootable and non-bootable CDs in Rock Ridge (ISO9660) or UDF (Universal Disk Format) format.

See the `-L` flag for details about creating DVD-sized images. What applies to CDs also applies to DVDs, except where noted.

**Note:** The functionality required to create Rock Ridge format CD images and to write the CD image to the CD-R, DVD-R or DVD-RAM device is not part of the `mkcd` command. You must supply additional code to `mkcd` to do these tasks. The code will be called via shell scripts and then linked to `/usr/sbin/mkrr_fs` (for creating the Rock Ridge format image) and `/usr/sbin/burn_cd` (for writing to the CD-R device). Both links are called from the `mkcd` command.

Some sample shell scripts are included for different vendor-specific routines. You can find these scripts in `/usr/samples/oem_cdwriters`.

If you do not supply any file systems or directories as command parameters, `mkcd` creates the necessary file systems and removes them when the command finishes executing. File systems you supply are checked for adequate space and write access.

**Note:** If `mkcd` creates file systems in the backup volume group, they are excluded from the backup.

If you need to create multi-volume CDs because the volume group image does not fit on one CD, `mkcd` gives instructions for CD replacement and removal until all the volumes have been created.

**Flags**

`-d cd_device` Indicates the CD-R, DVD-R or DVD-RAM device (`/dev/cd1`, for instance). This flag is required unless you use the `-S` flag.
-r directory

Indicates existing directory structure to burn onto a CD or DVD. This makes a CD image that is a copy of the given directory structure.

-m mksysb_image

Specifies a previously created mksysb image. If you do not give the -m flag mkcd calls mksysb. (See the -M flag for more information about where the mksysb image is placed.)

-s savevg_image

Indicates a previously created savevg image. See the Notes below.

-v savevg_volume_group

Denotes the volume group to be backed up using the savevg command. See the Notes below. (See the -M flag for more information about where the savevg image is placed.)

-C cd_fs_dir

Specifies the file system used to create the CD file system structure, which must have at least 645MB of available disk space (up to 4.38 GB for DVD sized images). The CD image will only consume as much room as necessary to contain all the data on the CD.

If you do not specify the -C flag and the /mkcd/cd_fs directory exists, mkcd uses that directory. If you do not give the -C flag and the /mkcd/cd_fs directory does not exist, mkcd creates the file system /mkcd/cd_fs and removes it when the command finishes executing. The command creates the file system in the volume group indicated with the -V flag, or rootvg if that flag is not used. Each time you invoke the mkcd command, a unique subdirectory (using the process id) is created under the /mkcd/cd_fs directory, or in the directory specified with the -C flag.

Note: If performing DVD sized backups, the filesystems need to be large file enabled. This also requires setting the file ulimit size to unlimited.

-M mksysb_target

States the directory or file system where the mksysb or savevg image is stored if a previously created backup is not given with the -m or -s flags. If the -M flag is not used and a mksysb or savevg image is not provided, mkcd verifies that /mkcd/mksysb_image exists. If the directory does not exist, then mkcd creates a separate file system, /mkcd/mksysb_image, where the mksysb or savevg images are temporarily stored. The command creates the file system in the volume group given with the -V flag, or in rootvg if that flag is not used.

Note: If performing DVD sized backups, the filesystems need to be large file enabled. This also requires setting the file ulimit size to unlimited.

-I cd_image_dir

Specifies the directory or file system where the final CD images are stored before writing to the CD-R, DVD-R or DVD-RAM device. If this flag is not used, mkcd uses the /mkcd/cd_images directory if it already exists. If not, the command creates the /mkcd/cd_images file system in the volume group given with the -V flag, or in rootvg if that flag is not used.

If mkcd creates the file system, it is removed upon command completion, unless either the -R or -S flag is used. If the -R or -S flag is used, consideration must be made for adequate file system, directory, or disk space, especially when creating multi-volume CDs. The CD image will only consume as much room as necessary to contain all the data on the CD.

Note: If performing DVD sized backups, the filesystems need to be large file enabled. This also requires setting the file ulimit size to unlimited.

-V cdfs_volume_group

Indicates the volume group used when creating the file systems needed for the mkcd command. If the -V flag is not given and a file system is needed but not there (because it was not supplied with other flags), then rootvg is the default volume group for creating the file systems. If mkcd creates the file systems in the backup volume group, those file systems are not included as part of the backup image. mkcd-created file systems are removed upon the command's completion.
-p pkg_source_dir

Names the directory or device that contains device and kernel package images. The device can only be a CD device (for example, /dev/cd0). If you use the same CD-R, DVD-R or DVD-RAM device that you gave with the -d flag, the product CD media must be inserted into the CD-R drive first. mkcd then prompts you to insert the writeable CD before the actual CD creation.

-B

Prevents mkcd from adding boot images (non-bootable CD) to the CD. Use this flag if creating a mksysb CD that you will not boot. Before installing the non-bootable mksysb CD you must boot a same level (V.R.M.) product CD. The mkcd command defaults to creating a bootable CD for the machine type of the source system. See the [Notes] below.

-R

Prevents mkcd from removing the final CD images. mkcd defaults by removing everything that it creates when it finishes executing. The -R flag allows multiple CD image sets to be stored, or for CD creation (burn) to occur on another system. If multiple volumes are needed, the final images are uniquely named using the process ID and volume suffixes.

-S

Stops mkcd before writing to the CD-R, DVD-R or DVD-DRAM without removing the final CD images. The -S flag allows multiple CD sets to be created, or for CDs to be created on another system. The images remain in the directory marked by the -I flag, or in the /mkcd/cd_images directory if the -I flag is not used. If multiple volumes are required, the final images are uniquely named using the process ID and volume suffixes.

-u bosinst.data

Specifies the user-supplied bosinst.data file. This data file takes precedence over the bosinst.data file in the mksysb image. If you do not give the -u flag, then mkcd restores bosinst.data from the given mksysb image, or generates a new bosinst.data file during the creation of mksysb.

-i image.data

Specifies the user-supplied image.data file. This data file takes precedence over the image.data file in the mksysb image. If you do not give the -i flag, then mkcd restores the image.data from the given mksysb image, or generates a new image.data file during the creation of mksysb.

[Note: The -i flag cannot be used to specify a user-supplied vgname.data file for use with a savevg image.]

-e

Excludes the files and/or directories from the backup image listed in the /etc/exclude.volume_group file. You cannot use this flag with the -m or -s flags.

-P

Creates physical partition mapping during the mksysb or savevg creation. You cannot use this flag with the -m or -s flags.

-I package_list

Specifies the file containing a list of additional packages you want copied to the /usr/lpp/inst.images directory of the CD file system. The images are copied from the location named with the -p flag. If you use the -I flag you must also use the -p flag.

-L

Creates final CD images that are DVD sized (up to 4.38 GB).

-b bundle_file

Gives the full pathname of the file containing a list of files to be installed after the mksysb is restored. This file is copied to /usr/sys/inst.data/user_bundles/bundle_file in the CD file system and also copied to RAM in case the CD is unmounted. The file would be listed as BUNDLES=/usr/sys/inst.data/user_bundles/bundle_file in the bosinst.data file.

-z custom_file

States the full pathname of the file to be copied to the root directory of the CD file system. This file could be a customization script specified in the bosinst.data file, such as CUSTOMIZATION_FILE=filename.

For example: If the file my_script is in /tmp on the machine where mkcd is running, then enter -z/tmp/my_script and specify CUSTOMIZATION_FILE=my_script. The code copies the script to the root directory of the RAM file system before it executes.
-D  Turns on the debug output information feature. The default is no debug output.

-U  Creates a UDF (Universal Disk Format) file system on DVD-RAM media. It does not require the amount of free space needed to create Rock Ridge format backups. It does not need the /mkcd/cd_fs and /mkcd/cd_images file systems. Therefore, the only temporary disk space it needs is to create the backup image that will be copied to the media. This means that the -I and -C flags do not apply to the -U flag. Because the backup is copied to the media, images cannot be created and burned later or on another system. So, the -R flag and -S flag do not apply when using the -U flag. You need to specify a device to write to with the -d flag. The -U flag does not use the /usr/sbin/mkrr_fs or /usr/sbin/burn_cd file systems.

-Y  Accepts licenses.

-n  Backs up user volume group information and administration data files. This backs up files such as /tmp/vgdata/vgname/vgname.data and map files, if any exist. This does not backup user data files. This backup can be used to create a user volume group without restoring user data files. This cannot be done to rootvg.

Notes:
1. If you are creating a non-bootable CD (using the -B flag), you cannot use the -p or -I flags.
2. If you are creating a non-bootable CD with a savevg image (using the -s or -v flags), you cannot use the -p, -I, -U, -z, or -b flags.

Examples
1. To generate a bootable system backup to the CD-R device named /dev/cd1, enter:
   mkcd -d /dev/cd1
2. To generate a system backup to the DVD-R or DVD-RAM device named /dev/cd1, enter:
   mkcd -d /dev/cd1 -L
3. To generate a non-bootable volume group backup of the volume group myvg to /dev/cd1, enter:
   mkcd -d /dev/cd1 -v myvg

   Note: All savevg backup images are non-bootable.
4. To generate a non-bootable system backup, but stop mkcd before the CD is created and save the final images to the /mydata/my_cd file system, and create the other mkcd filesystems in myvg, enter:
   mkcd -B -I /mydata/my_cd -V myvg -S
5. To create a CD or DVD that duplicates an existing directory structure
   /mycd/a
   /mycd/b/d
   /mycd/c/f/g
   use the following command:
   mkcd -r /mycd -d /dev/cd1
   After mounting with mount -o ro /dev/cd1 /mnt, cd to /mnt; a find . -print command displays:
   ./a
   ./b
   ./b/d
   ./c
   ./c/f
   ./c/f/g

Files

/usr/bin/mkcd  Contains the mkcd command.
Related Information

The `mksysb` command and `savevg` command.

The `/image.data` file and the `/bosinst.data` file.

A procedure to verify the backup can be found in the article "Verifying a System Backup" in the AIX 5L Version 5.2 Installation Guide and Reference.

For more information about CD-R drives (including DVD-R and DVD-RAM drives) and CD-R creation software, refer to the following README file: `/usr/lpp/bos.sysmgt/README.oem_cdwriters`

### mkcfsmnt Command

**Purpose**

Mounts a CacheFS directory.

**Syntax**

```
```

**Description**

The `mkcfsmnt` command constructs an entry that will be appended to the `/etc/filesystems` file, thus making a file system available for use as a cache file system. If the mount is to be permanent, this entry will remain. If the mount is temporary, the flags will be used directly for the `mount` command. CacheFS file systems are used to cache accesses to backing file systems. Backing file systems are generally NFS mounts.

**Flags**

- `-d PathName` Specifies the mount point for the cache directory.
- `-t` Selects file systems to be cached.
  - `-nfs` Specifies that the CacheFS file system is backed by an NFS mount.
  - `-cdrom` Specifies that the CacheFS file system is backed by a CDROM file system. (Currently not supported.)
- `-h RemoteHost` Specifies the NFS server that is exporting the directory.
- `-p RemoteDirectory` Specifies the directory that is mounted on the path name specified. This is commonly a remote file system that will be mounted via NFS or a local device name in the case of CDROM (Currently not supported.)
- `-c CacheDirectory` Specifies the location of the CacheFS file system. This must have been previously created by execution of the `cfsadmin` command.
- `-d RemoteDirectory` Specifies the directory that is mounted on the path name specified.
- `-o MountOptions` Specifies a comma-separated string of mount options that are dependent on the backing file system type. For instance, if it is NFS, the options would be those typically specified by the `-o` Options string to mount. See the `mount` command documentation for the acceptable values.
- `-b BackingFileSystem` Specifies a backing file system if it is already mounted. If this is not specified, then the command will do the mount itself on a temporary mount point. If this is not specified, then RemoteHost and RemoteDirectory must be specified.
- `-I` Causes an entry to be added to the `/etc/filesystems` file. The directory is not mounted.
-B Adds an entry to the `/etc/filesystems` file and attempts to mount the file system. This flag is the default.
-N Mounts the directory with the options specified, but does not modify the `/etc/filesystems` file.

Examples
To specify a CacheFS mount, type:
```
/usr/sbin/mkcfsmnt -t nfs -d /usr/share/man -p /usr/share/man -h host1 -c /cache/cachel -o ro, intr -N
```
In this example, the `mkcfsmnt` command caches the remote directory `/usr/share/man` that resides on host1 on the local `/usr/share/man` directory. The cache is kept in `/cache/cachel`, which was created with the `cfsadmin` command. CacheFS takes care of doing the NFS backing mount, because the `-b` flag has not been specified.
```
/usr/sbin/mkcfsmnt -t nfs -d /usr/share/man -p /usr/share/man -h host1 -c /cache/cachel -b /backs/man -o ro, intr -N
```
In this example, the `mkcfsmnt` command caches the remote directory `/usr/share/man` residing on host1 on the local `/usr/share/man` directory. The cache is kept in `/cache/cachel`, which was created with the `cfsadmin` command. The backing file system has already been mounted on `/backs/man`.

Files
```
/etc/filesystems
```
Lists the remote file systems to be mounted during the system restart.

---

**mkclass Command**

**Purpose**
Create a Workload Management class.

**Syntax**
```
mkclass [ -a Attribute=Value ... ] [ -c | -m | -b [ -C | -B | -P | -T | -L | -A KeyWord=Value ] ] [ -d ] Config_Dir ] [[-S SuperClass ] Name
```

**Description**
The `mkclass` command creates a superclass or a subclass identified by the `Name` parameter. The class must not already exist. The `Name` parameter can contain only upper and lower case letters, numbers and underscores. The name is in the format `supersName or subName` (with the `-S supersName flag) or supersName.subName. SuperName and subName are each limited to 16 characters in length. The names `Default`, `System`, and `Shared` are reserved. They refer to predefined classes. Any `Attribute=Value` or `KeyWord=Value` argument initialise the specified attribute or resource limit. See `chclass` for more information. To set the process total limits (the limits that apply to each process of the class), use one or more of the options `-C` (totalCPU), `-B` (totalDiskIO) or `-A` (totalConnectTime), with the keyword value of hardmax. To set the class total limits (the limits that apply to the whole class), use one or more of the options `-P` (totalProcesses), `-T` (totalThreads) or `-L` (totalLogins), with the keyword value of hardmax. To reset any total limit, use `"-"` for Value. Process, class, or both total limits may be disabled when starting or updating the WLM (see `wlmcntrl` command).

Normally, `mkclass` adds the class and its attributes in the relevant WLM property files, and the modifications is applied to the in-core class definitions (active classes) only after an update of WLM using the `wlmcntrl` command.
If an empty string is passed as the configuration name (Config_dir) with the -d flag, the class is created only in the WLM in-core data structures, and no property file is updated, making the new class temporary (the change is lost if WLM is stopped and restarted or the system is rebooted).

**Note:** This command cannot apply to a set of time-based configurations (do not specify a set with the -d flag). If the current configuration is a set, the -d flag must be given to indicate which regular configuration the command should apply to.

**Flags**

- **A hardmax=** Value
  Sets the maximum amount of time a login session in the class can stay active. Value is specified as an integer, possibly appending the unit (s for seconds, m for minutes, h for hours, d for days, and w for weeks, default is seconds). As a user approaches this connection time limit, WLM will send a warning message to the session terminal. When the limit is reached, the user will be notified and the session leader will be sent the SIGTERM signal, and after a short grace period, the session will be terminated (SIGKILL).

- **B hardmax=** Value
  Sets the total amount of disk I/Os allowed for each process in the class. Value is specified as an integer, possibly appending the unit (KB for kilobytes, MB for megabytes, TB for terabytes, PB for petabytes, and EB for exabytes, default is kilobytes). After a process has used this amount of disk I/Os, the process will be sent the SIGTERM signal, and after a grace period, it will be killed (SIGKILL).

- **C hardmax=** Value
  Sets the total amount of CPU time allowed for each process in the class. Value is specified as an integer, possibly appending the unit (s for seconds, m for minutes, h for hours, d for days, and w for weeks, default is seconds). After a process has used this amount of time, the process will be sent the SIGTERM signal, and after a grace period, it will be killed (SIGKILL).

- **d Config_Dir**
  Use /etc/wlm/Config_Dir as an alternate directory for the properties files. When this flag is not used, mkclass uses the configuration files in the directory pointed to by /etc/wlm/current. If an empty string is passed as the configuration name (-d "") the new class is created only in the WLM in-core data structures and no configuration file is modified.

- **L hardmax=** Value
  Sets the total number of login sessions simultaneously available in the class. If a user tries to log onto the system and the login shell would end up in a class that has reached the total logins limit, the login operation will fail.

- **P hardmax=** Value
  Sets the maximum number of processes allowed in the class. If an operation would result in a new process entering the class when the class has this many processes in it, the operation will fail.

- **S Superclass**
  Specifies the name of the superclass when creating a subclass. There are two ways of creating the subclass Sub of superclass Super:

  1. Specify the full name of the subclass as Super.Sub for Name and not use -S
  2. Specify the -S flag to give the superclass name and use the short name for the subclass:

     ```
     mkclass options -S Super Sub
     ```

- **T hardmax=** Value
  Sets the maximum number of threads allowed in the class. If an operation would result in a new thread entering the class when the class has this many processes in it, the operation will fail. The total thread limit must be at least as large as the total process limit for a class. If a class has a total thread limit but no total process limit specified, the total process limit will be set to the total thread limit.

**Security**

Access control: Only the root user can create a superclass. Only root or authorized users whose user ID or group ID matches the user name or group name specified in the attributes admuser and admingroup of a superclass can create a subclass of this superclass.
Files

classes  Contains the names and definitions of the classes.
limits   Contains the resource limits enforced each class.
shares  Contains the resource shares attributed to each class.

Related Information

The `wlmcntrl` command, `lsclass` command, `chclass` command, and `rmclass` command.

---

**mkclient Command**

**Purpose**

Uncomments the entry in the `/etc/rc.nfs` file for the `ypbind` daemon and starts the `ypbind` daemon to configure a client.

**Syntax**

```
/usr/sbin/mkclient [-I | -B | -N] [-S server]
```

**Description**

The `mkclient` command uncomments the entry to the `/etc/rc.nfs` file to start the `ypbind` daemon to configure a client. The `mkclient` command starts the `ypbind` daemon by using the appropriate System Resource Controller (SRC) command.

You can use the File Systems application in Web-based System Manager (WBM) to change file system characteristics.

You could also use the System Management Interface Tool (SMIT) `smit mkclient` fast path to run this command.

**Flags**

- **-I**  Uncomments the entry for starting the `ypbind` daemon to the `/etc/rc.nfs` file. This entry causes the `ypbind` daemon to start during the next system restart.
- **-B**  Uncomments the entry to the `/etc/rc.nfs` file and starts the `ypbind` daemon. This flag is the default.
- **-N**  Causes the `startsrc` command to start the `ypbind` daemon. This flag does not affect the `/etc/rc.nfs` file.
- **-S**  Specifies which NIS `server` to use instead of broadcasting for one. This option must be used when no NIS server exists on the networks directly connected to the client machine.

**Examples**

To modify the `/etc/rc.nfs` file so that the `ypbind` daemon is started on the next system restart, enter:

```
mkclient -I
```

**Files**

```
/var/yp/directoryname directory  Contains the NIS maps for the NIS domain.
/etc/rc.nfs                   Contains the startup script for the NFS and NIS daemons.
```
Related Information

The `mkmaster` command, `rmyp` command, `smit` command, `startsrc` command.

The `ypbind` daemon, `yppasswd` daemon, `ypserv` daemon, `ypupdated` daemon.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


NIS Reference

---

**mkcomg Command**

**Purpose**

Creates a new communication group definition for a peer domain.

**Syntax**

```
mkcomg [-s sensitivity] [-p period] [-t priority] [-x (br) [-e NIM_path] [-m NIM_parameters] [-i n:network_interface1[:node1] [,network_interface2[:node2]...]] | -S n:”network_interface_selection_string” [-h] [-TV] communication_group
```

**Description**

The `mkcomg` command creates a new communication group definition for an online peer domain with the name specified by the `communication_group` parameter. The communication group is used to define heartbeat rings for use by topology services and to define the tunables for each heartbeat ring. The communication group determines which devices are used for heartbeating in the peer domain. There can be more than one communication group in a peer domain.

The `mkcomg` command must be run on a node that is currently online in the peer domain where the communication group is to be defined. More than half of the nodes must be online to create a new communication group for the domain.

The `-e` and `-m` flags are used to set the network interface module (NIM) path and parameters. The NIM path is the path to the NIM that supports the adapter types used in the communication group. The NIM parameters are passed to NIM when it is started. If `-m` is not specified, the parameters predefined by topology services are used.

The communication group can be assigned to one or more interface resources. Use the `-i` flag to assign the communication group to a specific interface resource name. The interface resource can be limited to one on a particular node. An interface resource can also be specified using the `-S` flag and a selection
string. This is used when specifying the interface resource name is not sufficient. The -i and -S flags cannot be used together. The **chcomg** command can also be used to assign a communication group to an interface resource.

**Flags**

- **s sensitivity**
  Specifies the heartbeat sensitivity. This is the number of missed heartbeats that constitute a failure. The sensitivity value is an integer greater than or equal to 2. The default value is 4.

- **p period**
  Specifies the number of seconds between heartbeats. The period is an integer greater than or equal to 1. The default value is 1.

- **t priority**
  Specifies the priority. This value indicates the importance of this communication group with respect to others. It is used to order the heartbeat rings. The lower the number means the higher the priority. The highest priority is 1. The default value is 1 for IP networks and 255 for RS232 networks.

- **x br**
  Excludes controls for heartbeat mechanisms. This flag indicates that one or more controls for heartbeat mechanisms should not be used even if the underlying media support it. The following features can be excluded:

  - **b**
    Specifies that the broadcast feature should not be used even if the underlying media support it. If `-x b` is not specified, the broadcast feature will be used if the underlying media support it.

  - **r**
    Specifies that the source routing feature should not be used even if the underlying media support it. If `-x r` is not specified, the source routing feature will be used if the underlying media support it.

  To exclude more than one control, specify the feature characters consecutively: `-x br`.

- **e NIM_path**
  Specifies the network interface module (NIM) path name. This character string specifies the path name to the NIM that supports the adapter types in the communication group.

- **m NIM_parameters**
  Specifies the NIM start parameters. This character string is passed to the NIM when starting it.

- **i n:network_interface1[:node1] [network_interface2[:node2]]...**
  Assigns this communication group to the network interface resource defined by the network interface resource name and optionally the node name where it can be found.

  If `-i` is specified, `-S` cannot be specified.

- **S n:“network_interface_selection_string”**
  Assigns this communication group to the interface specified by the network interface selection string.

  If `-S` is specified, `-i` cannot be specified.

- **h**
  Writes the command’s usage statement to standard output.

- **T**
  Writes the command’s trace messages to standard error. For your software service organization’s use only.

- **V**
  Writes the command’s verbose messages to standard output.
Parameters

communication_group
Specifies the name of the new communication group that is to be created for the online peer domain. The name can contain any printable character.

Security
The user of the mkcomg command needs write permission for the IBM.CommunicationGroup resource class. Write permission for the IBM.NetworkInterface resource class is required to set the communication group for a network interface resource. By default, root on any node in the peer domain has read and write access to these resource classes through the configuration resource manager.

Exit Status
0  The command ran successfully.
1  An error occurred with RMC.
2  An error occurred with a command-line interface script.
3  An incorrect flag was entered on the command line.
4  An incorrect parameter was entered on the command line.
5  An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

Restrictions
This command must be run on a node that is defined and online to the peer domain where the communication group is to be defined.

Standard Input
When the -f "-" flag is specified, this command reads one or more node names from standard input.

Standard Output
When the -h flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
1. To define the communication group ComGrp1 for the peer domain ApplDomain and nodeA is defined and online to ApplDomain, run this command on nodeA:
   mkcomg ComGrp1
2. To define the communication group ComGrp1 for the peer domain ApplDomain, using a sensitivity of 1 and period of 3, and nodeA is defined and online to ApplDomain, run this command on nodeA:
   mkcomg -s 1 -p 3 ComGrp1
3. To define the communication group ComGrp1 for the peer domain ApplDomain, not using broadcast, using a priority of 3, and nodeA is defined and online to ApplDomain, run this command on nodeA:
   mkcomg -x b -t 3 ComGrp1
4. To define the communication group ComGrp1 for the peer domain ApplDomain, not using broadcast, not using source routing, and nodeA is defined and online to ApplDomain, run the following command on nodeA:
   mkcomg -x br ComGrp1
5. To define the communication group ComGrp1 for the peer domain ApplDomain, using a NIM path of /usr/sbin/rsct/bin/hats_nim, NIM parameters -l 5 to set the logging level, and nodeA is defined and online to ApplDomain, run this command on nodeA:
   mkcomg -e /usr/sbin/rsct/bin/hats_nim -m "-l 5" ComGrp1
6. To define the communication group ComGrp1 for the peer domain ApplDomain, assign ComGrp1 to the network interface resource named eth0 on nodeB, and nodeA is defined and online to ApplDomain, run this command on nodeA:
   mkcomg -i i:n:eth0:nodeB ComGrp1
7. To define the communication group ComGrp1 for the peer domain ApplDomain, assign ComGrp1 to the network interface resource that uses the subnet 9.123.45.678, and nodeA is defined and online to ApplDomain, run this command on nodeA:
   mkcomg -S n:"Subnet == 9.123.45.678" ComGrp1

Location

/usr/sbin/rsct/bin/mkcomg Contains the mkcomg command

Related Information

Commands: Iscomg, lsrdomain, lsrpnode, mkrpdomain, preprpnode, rmcomg, startrpdomain

Files: rmccli, for general information about RMC-related commands

---

**mkcondition Command**

**Purpose**

Creates a new condition definition which can be monitored.

**Syntax**

```
mkcondition -r resource_class -e event_expression [-E rearm_expression] [-d event_description] [-D rearm_description] [-m l | m | p ] [-n node_name1[,node_name2...]] [-s "selection_string"] [-p node_name] [-S c | w | i] [-h] [-TV] condition
```

```
```

**Description**

The mkcondition command creates a new condition with the name specified by the condition parameter. The condition is used to monitor a resource for the occurrence of the condition (or event). One or more
responses to an event can be defined by using the *mkresponse* command. The conditions can then be linked to the responses by using the *mkcondresp* command, or by using the *startcondresp* command to link the responses and start monitoring.

When in a cluster environment, use the `-p` flag to specify the node in the domain that is to contain the condition definition. If the `-p` flag is not specified, the condition is defined on the local node. If the node where the condition will be defined is:

- in a cluster of nodes, the condition can monitor resources on more than one node. Use the `-n` flag to specify the nodes on which the condition will be monitored.
- the management server in a management domain, a management scope (-m) of local (l) or management domain (m) can be specified to indicate how the condition applies. The selection string will be evaluated using the entire management domain when management scope is set to the management domain and the node is the management server.
- a managed node in a management domain, only a management scope (-m) of local (l) can be used.
- in a peer domain, a management scope (-m) of peer domain (p) or local (l) can be used to indicate how the condition and the selection string apply.
- in both a management domain and a peer domain, a management scope (-m) of management domain (m), peer domain (p), or local (l) can be used to indicate how the condition and its selection string apply.

To lock a condition so it cannot be modified or removed, use the *chcondition* command (with its -L flag).

**Flags**

- `c existing_condition[:node_name]`
  Copies an existing condition. The existing condition is defined on node_name. If node_name is not specified, the local node is used. node_name is a node within the scope determined by the CT_MANAGEMENT_SCOPE environment variable. If any other flags are specified, update the new condition as indicated by the flags. Links with responses are not copied.

- `r resource_class`
  Specifies the resource class to be monitored by this condition. You can display the resource class names using the *lsrsrcdef* command.

- `e "event_expression"`
  Specifies an event expression, which determines when an event occurs. An event expression consists of a dynamic attribute or a persistent attribute of resource_class, a mathematical comparison symbol (> or <, for example), and a constant. When this expression evaluates to TRUE, an event is generated.

- `E "rearm_expression"
  Specifies a rearm expression. After event_expression has evaluated to TRUE and an event is generated, the rearm expression determines when monitoring for the event_expression will begin again. Typically, the rearm expression prevents multiple events from being generated for the same event evaluation. The rearm expression consists of a dynamic attribute of resource_class, a mathematical comparison symbol (> or <, for example), and a constant.

- `d "event_description"
  Describes the event expression.

- `D "rearm_description"
  Describes the rearm expression.

- `n node_name1[,node_name2...]`
  Specifies the host name for a node (or a list of host names separated by commas for multiple nodes) where this condition will be monitored. You must specify the -m flag with a value of m or p if you want to use the -n flag. This way, you can monitor conditions on specific nodes instead of the entire domain.
-s "selection_string"
  Specifies a selection string that is applied to all of the resource_class attributes to determine which resources should be monitored by the event_expression. The default is to monitor all resources within the resource_class. The resources used to evaluate the selection string is determined by the management scope (the -m flag). The selection string must be enclosed within double or single quotation marks.

-S c | w | i
  Specifies the severity of the event:
  c  Critical
  w  Warning
  i  Informational (the default)

-m l | m | p
  Specifies the management scope to which the condition applies. The management scope determines how the condition is registered and how the selection string is evaluated. The scope can be different from the current configuration, but monitoring cannot be started until an appropriate scope is selected. The valid values are:
  l  Specifies local scope. This is the default. The condition applies only to the local node (the node where the condition is defined; see the -p flag). Only the local node is used in evaluating the selection string.
  m  Specifies management domain scope. The condition applies to the management domain in which the node where the condition is defined belongs (see the -p flag). All nodes in the management domain are used in evaluating the selection string. The node where the condition is defined must be the management server in order to use management domain scope.
  p  Specifies peer domain scope. The condition applies to the peer domain in which the node where the condition is defined belongs (see the -p flag). All nodes in the peer domain are used in evaluating the selection string.

-p node_name
  Specifies the name of the node where the condition is defined. This is used in a cluster environment and the node name is the name by which the node is known in the domain. The default node_name is the local node on which the command runs. node_name is a node within the scope determined by the CT_MANAGEMENT_SCOPE environment variable.

-h
  Writes the command’s usage statement to standard output.

-T
  Writes the command’s trace messages to standard error. For your software service organization’s use only.

-V
  Writes the command’s verbose messages to standard output.

Parameters

condition  The condition name is a character string that identifies the condition. If the name contains spaces, it must be enclosed in quotation marks. A name cannot consist of all spaces, be null, or contain embedded double quotation marks.

Security

The user needs write permission for the IBM.Condition resource class to run mkcondition. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status

0  The command ran successfully.
1  An error occurred with RMC.
An error occurred with a command-line interface script.

An incorrect flag was entered on the command line.

An incorrect parameter was entered on the command line.

An error occurred that was based on incorrect command-line input.

Environment Variables

**CT_CONTACT**
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

**CT_MANAGEMENT_SCOPE**
Determines the management scope that is used for the session with the RMC daemon in processing the resources of the event-response resource manager (ERRM). The management scope determines the set of possible target nodes where the resources can be processed. The valid values are:

0  Specifies *local* scope.
1  Specifies *local* scope.
2  Specifies *peer domain* scope.
3  Specifies *management domain* scope.

If this environment variable is not set, *local* scope is used.

Standard Output
When the -h flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
These examples apply to standalone systems:

1. To define a condition with the name “FileSystem space used” to check for percentage of space used greater than 90% and to rearm when the percentage is back down below 85%, enter:

   ```bash
   mkcondition -r IBM.FileSystem "  
   -e "PercentTotUsed > 90" -E "PercentTotUsed < 85" "FileSystem space used"
   ```

2. To define a condition with the name “tmp space used” to check for percentage of space used greater than 90% for /tmp and to rearm when the percentage is back down below 85%, including comments, enter:

   ```bash
   mkcondition -r IBM.FileSystem "  
   -e "PercentTotUsed > 90" -E "PercentTotUsed < 85" "  
   -d "Generate event when tmp > 90% full" "  
   -D "Restart monitoring tmp again after back down < 85% full" "  
   -s 'Name=="/tmp"' "tmp space used"
   ```

3. To define a condition with the name “Space used” as a copy of “FileSystem space used”, enter:

   ```bash
   mkcondition -c "FileSystem space used" "Space used"
   ```
4. To define a condition with the name "var space used" as a copy of "tmp space used", but change the selection to /var, enter:

   mkcondition -c "tmp space used" -s 'Name="/var"' \
   "var space used"

These examples apply to management domains:

1. To define a condition with the name "FileSystem space used" to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor all nodes in the domain, run this command on the management server:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -m d "FileSystem space used"

2. To define a condition with the name "FileSystem space used" to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor nodes nodeA and nodeB in the domain, run this command on the management server:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -n nodeA,nodeB -m d \
   "FileSystem space used"

3. To define a condition with the name "nodeB FileSystem space used" on nodeB to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor the condition with local scope, run this command on the management server:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -m l -p nodeB \
   "nodeB FileSystem space used"

4. To define a condition with the name "local FileSystem space used" to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor the local node, run this command on a managed node:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -m l "local FileSystem space used"

These examples apply to peer domains:

1. To define a condition on nodeA with the name "FileSystem space used" to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor all nodes in the domain, run this command:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -m p -p nodeA "FileSystem space used"

2. To define a condition on nodeC with the name "FileSystem space used" to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor nodes nodeA and nodeB in the domain, run this command:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -n nodeA,nodeB -m p -p nodeC \
   "FileSystem space used"

3. To define a condition with the name "local FileSystem space used" on nodeB to check for percentage of space used greater than 90%, to rearm when the percentage is back down below 85%, and to monitor the local node only, run this command:

   mkcondition -r IBM.FileSystem -e "PercentTotUsed > 90" \
   -E "PercentTotUsed < 85" -m l "local FileSystem space used"

**Location**

/usr/sbin/rsct/bin/mkcondition

Contains the **mkcondition** command
Related Information
Commands: chcondition, lscondition, mkcondresp, mkresponse, rmcondition, startcondresp

Files: rmccli

mkcondresp Command

Purpose
Creates a link between a condition and one or more responses.

Syntax
mkcondresp [-h] [-TV] condition[:node_name] response1 [response2...]

Description
The mkcondresp command creates a link between a condition and one or more responses. A link
between a condition and a response is called a condition/response association. Note that, while this
command creates one or more condition/response associations, it does not start monitoring. In a cluster
environment, the condition and the response must be defined on the same node. You can start monitoring
for this condition and its linked responses later by using the startcondresp command.

To lock a condition/response association, use the -L flag of the rmcondresp, startcondresp, or
stopcondresp command.

Flags

- h Writes the command’s usage statement to standard output.
- T Writes the command’s trace messages to standard error. For your software service organization’s
use only.
- V Writes the command’s verbose messages to standard output.

Parameters

condition Specifies the name of the condition to be linked to the response. The condition is always
specified first.

node_name Specifies the node in the domain where the condition is defined. If node_name is not
specified, the local node is used. node_name is a node within the scope determined by
the CT_MANAGEMENT_SCOPE environment variable.

response1 [response2...]
Specifies one or more response names. All responses are linked to condition.

Security
The user needs write permission for the IBM.Association resource class to run mkcondresp.
Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status

0 The command ran successfully.
1 An error occurred with RMC.
2 An error occurred with a command-line interface script.
3 An incorrect flag was entered on the command line.
An incorrect parameter was entered on the command line.

An error occurred that was based on incorrect command-line input.

Environment Variables

**CT_CONTACT**
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

**CT_MANAGEMENT_SCOPE**
Determines the management scope that is used for the session with the RMC daemon in processing the resources of the event-response resource manager (ERRM). The management scope determines the set of possible target nodes where the resources can be processed. The valid values are:

- 0  Specifies local scope.
- 1  Specifies local scope.
- 2  Specifies peer domain scope.
- 3  Specifies management domain scope.

If this environment variable is not set, local scope is used.

Standard Output
When the -h flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
These examples apply to standalone systems:

1. To link the condition "FileSystem space used" to the response "Broadcast event on-shift", run this command:
   ```
   mkcondresp "FileSystem space used" "Broadcast event on-shift"
   ```

2. To link the condition "FileSystem space used" to the responses "Broadcast event on-shift" and "E-mail root anytime", run this command:
   ```
   mkcondresp "FileSystem space used" "Broadcast event on-shift" "E-mail root anytime"
   ```

These examples apply to management domains:

1. To link the condition "FileSystem space used" on the management server to the response "Broadcast event on-shift" (also on the management server), run this command on the management server:
   ```
   mkcondresp "FileSystem space used" "Broadcast event on-shift"
   ```

2. To link the condition "FileSystem space used" on the management server to the response "Broadcast event on-shift", run this command on one of the nodes in the domain:
   ```
   mkcondresp "FileSystem space used":nodeA "Broadcast event on-shift"
   ```

This example applies to peer domains:
1. To link the condition "FileSystem space used" on node `nodeA` to the response "Broadcast event on-shift" (also on `nodeA`), run this command on one of the nodes in the domain:

   `mkcondresp "FileSystem space used":nodeA "Broadcast event on-shift"

**Location**

`/usr/sbin/rsct/bin/mkcondresp`

Contains the `mkcondresp` command

**Related Information**

Commands: `lscondresp`, `mkcondition mkresponse`, `rmcondresp`, `startcondresp`, `stopcondresp`

Files: `rmccli`

---

**mkdev Command**

**Purpose**

Adds a device to the system.

**Syntax**

```
mkdev { [-c Class] [-s Subclass] [-t Type] [-l Name] [-a Attribute=Value] ... [ -d ] [-S] [-R] [ -f File ] [ -h ] [ -p ParentName ] [ -q ] [ -w ConnectionLocation ]
```

```
mkdev [-Name] [-h] [-q] [-S]
```

**Description**

**Attention:** To protect the Configuration Database, the `mkdev` command is not interruptible. Stopping this command before execution is complete could result in a corrupted database.

The `mkdev` command either defines and makes available a device with the given device class (`-c Class` flag), type (`-t Type` flag), subclass (`-s Subclass` flag), connection location (`-w ConnectionLocation` flag), and the device logical name of the parent (`-p ParentName` flag), or makes available the previously defined device specified by the given device logical name (`-l Name` flag). You can use any combination of the `-c`, `-s`, and `-t` flags needed to uniquely identify the predefined device.

If you specify the `-d` flag, the `mkdev` command only defines the device. If you specify the `-S` flag, the `mkdev` command brings the device to the Stopped state, if this state is supported, and does not make the device available. If you do not specify either the `-d` flag or the `-S` flag, the `mkdev` command makes the device available.

If you specify the `-R` flag, the `mkdev` command configures any parents of the specified device that are not already configured. Parents must be previously defined. The `-R` flag is not compatible with the `-d` and `-S` flags.

By using the `-l` flag with the `-c`, `-s`, and `-t` flags, you can specify the name you want the device to be known by. If you do not use the `-l` flag, a name will be automatically generated and assigned. Not all devices support user-supplied names.

**Note:** Queue device names must begin with an alphabetic character.

When using the `mkdev` command, you can supply the flags either on the command line or from the specified `File` parameter.
You can use the Devices application in Web-based System Manager (wsm) to change device characteristics.

You can use the Web-based System Manager Devices application (wsm devices fast path) to run this command. You could also use the System Management Interface Tool (SMIT) smit mkdev fast path to run this command.

Flags

-a Attribute=Value Specifies the device attribute value pairs to be used instead of the defaults. The Attribute=Value variable can be used to specify one attribute value pair or multiple attribute value pairs for one -a flag. If you use an -a flag with multiple attribute value pairs, the list of pairs must be enclosed in quotation marks with a blank space between the pairs. For example, entering -a 'Attribute1=Value1 Attribute2=Value2' lists more than one attribute value pair. This flag cannot be used with the [-c] flag unless the [-s] and [-f] flags are also used.

-c Class Specifies the device class.

-d Defines the device in the Customized Devices object class. If you specify the -d flag, the mkdev command does not make the device available. This flag cannot be used with the [-s] flag.

-f File Reads the needed flags from the File parameter.

-h Displays the command usage message.

-l Name Specifies the already defined device, indicated by the Name variable, in the Customized Devices object class when not used with the [-c] and [-h] flags. The [-a], [-p], and [-w] flags cannot be used in this case. Queue device names must begin with an alphabetic character.

-p ParentName Specifies the device name, indicated by the ParentName variable, that you want assigned to the device when used with the [-c] and [-h] flags. Not all devices support this feature. This flag cannot be used with the [-f] flag unless the [-c], [-s], and [-f] flags are also used.

-q Suppresses the command output messages from standard output and standard error.

-R Configures any parents of the device that are not already configured. This flag is not compatible with the [-d] and [-S] flags.

-S Prevents the device from being set to the Available state. This flag is only meaningful for those devices that support the Stopped state. This flag cannot be used with the [-d] flag.

-s Subclass Specifies the subclass, indicated by the Subclass variable, of the device.

-t Type Specifies the device type from the Predefined Devices object class.

-w ConnectionLocation Specifies the connection location, indicated by the ConnectionLocation variable, on the parent. This flag cannot be used with the [-f] flag unless the [-c], [-s], and [-f] flags are also used.

Security

Privilege Control: Only the root user and members of the system group should have execute (x) access to this command.

Auditing Events:

<table>
<thead>
<tr>
<th>Event</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEV_Create</td>
<td>Method name, parameters</td>
</tr>
<tr>
<td>DEV_Configure</td>
<td>Errors</td>
</tr>
<tr>
<td>DEV_Start</td>
<td>Device name</td>
</tr>
<tr>
<td>DEV_Change</td>
<td>Parameters</td>
</tr>
</tbody>
</table>
Examples
1. To define (but not configure) a 4.0 GB 4mm Tape Drive tape drive connected to the SCSI adapter scsi0 and using SCSI ID 5 and LUN of 0, type:
   ```
   mkdev -d -c tape -t4mm2gb -s scsi -p scsi0 -w 5,0
   ```
   The system displays a message similar to the following:
   ```
   rmt4 defined
   ```
2. To make the already defined tape device rmt0 available to use, type:
   ```
   mkdev -l rmt0
   ```
   The system displays a message similar to the following:
   ```
   rmt0 available
   ```
3. To define and configure an RS-232 tty device connected to port 0 on the IBM 8-Port EIA-232/RS-422A (PCI) Adapter with the speed attribute set to 19200, and other attributes set from the foo file, type:
   ```
   mkdev -t tty -s rs232 -p sa3 -w 0 -a speed=19200 -f foo
   ```
   The system displays a message similar to the following:
   ```
   tty0 available
   ```

Files
/usr/sbin/mkdev Contains the mkdev command.

Related Information
The chdev command, lsattr command, lsconn command, lsdev command, lsparent command, rmdev command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mkdir Command

Purpose
Creates one or more new directories.

Syntax
```bash
mkdir [-m Mode ] [-p] Directory ...
```

Description
The mkdir command creates one or more new directories specified by the Directory parameter. Each new directory contains the standard entries . (dot) and .. (dot-dot). You can specify the permissions for the new directories with the -m Mode flag. You can use the umask subroutine to set the default mode for the mkdir command.
The owner-ID and group-ID of the new directories are set to the process’s effective user-ID and group-ID, respectively. The setgid bit setting is inherited from the parent directory. To change the setgid bit, you can either specify the -m Mode flag or issue the chmod command after the creation of the directory.

**Note:** To make a new directory you must have write permission in the parent directory.

**Flags**

- **-m Mode**
  
  Sets the permission bits for the newly-created directories to the value specified by the Mode variable. The Mode variable takes the same values as the Mode parameter for the chmod command, either in symbolic or numeric form.

  When you specify the -m flag using symbolic format, the op characters + (plus) and - (minus) are interpreted relative to the assumed permission setting a=rwx. The + adds permissions to the default mode, and the - deletes permissions from the default mode. Refer to the chmod command for a complete description of permission bits and formats.

- **-p**
  
  Creates missing intermediate path name directories. If the -p flag is not specified, the parent directory of each newly created directory must already exist.

  Intermediate directories are created through the automatic invocation of the following mkdir commands:

  ```
  mkdir -p -m $(umask -S),u+wx $(dirname Directory) &&
  mkdir [-m Mode] Directory
  ```

  where the [-m Mode] represents any option supplied with your original invocation of the mkdir command.

  The mkdir command ignores any Directory parameter that names an existing directory. No error is issued.

**Exit Status**

This command returns the following exit values:

<table>
<thead>
<tr>
<th>Exit Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>All the specified directories were created successfully, or the -p option was specified and all the specified directories now exist.</td>
</tr>
<tr>
<td>&gt;0</td>
<td>An error occurred.</td>
</tr>
</tbody>
</table>

**Examples**

1. To create a new directory called Test in the current working directory, enter:

   ```
   mkdir Test
   ```

   The Test directory is created with default permissions.

2. To create a new directory called Test with rwxr-xr-x permissions in the previously created /home/demo/sub1 directory, enter:

   ```
   mkdir -m 755 /home/demo/sub1/Test
   ```

3. To create a new directory called Test with default permissions in the /home/demo/sub2 directory, enter:

   ```
   mkdir -p /home/demo/sub2/Test
   ```

   The -p flag creates the /home, /home/demo, and /home/demo/sub2 directories if they do not already exist.

**Files**

/usr/bin/mkdir Contains the mkdir command.
Related Information
The chmod command, rm command.
The mkdir subroutine, umask subroutine.

mkdirhier Command

Purpose
Creates a hierarchy of directories or a single directory.

Syntax
mkdirhier Directory ...

Description
The mkdirhier command creates the specified directories. Unlike the mkdir command, if any of the parent directories of the specified directory do not exist, the mkdirhier command creates those directories as well as the specified directory.

Example
To create a directory named foo2 or to create a hierarchy of directories named foo, foo1, and foo2, enter:

    mkdirhier ~/foo/foo1/foo2

If foo and foo1 already exist then the command creates foo2. However, if none of them exist then the command creates all three new directories.

Related Information
The mkdir command.

mkfifo Command

Purpose
Makes first-in-first-out (FIFO) special files.

Syntax
mkfifo [-m Mode ] File ...

Description
The mkfifo command creates FIFO special files specified by the File parameter, in the order specified. If the -m Mode flag is not specified, the file mode of the FIFO file is the bitwise inclusive OR of the S_IRUSR, S_IWUSR, S_IRGRP, S_IWRGP, S_IROTH, and S_IWOTH permissions as modified by the file mode creation (see the umask command).

The mkfifo command functions similarly to the mkfifo subroutine.
Flags

-m Mode  Sets the file permission bits of the newly created FIFO file to the specified mode values. The Mode variable is the same as the mode operand defined for the chmod command. The characters + (plus sign) and - (minus sign), if used, are interpreted relative to the initial value a=rw (that is, having permissions of rw-rw-rw-).

Exit Status
This command returns the following exit values:

0  All the specified FIFO special files were created successfully.
>0  An error occurred.

Examples
1. To create a FIFO special file with permissions prw-r--r--, enter:

   mkfifo -m 644 /tmp/myfifo

   This command creates the /tmp/myfifo file with read/write permissions for the owner and read permission for the group and for others.

2. To create a FIFO special file using the - (minus sign) operand to set permissions of prw-r----, enter:

   mkfifo -m g-w,o-rw /tmp/fifo2

   This command creates the /tmp/fifo2 file, removing write permission for the group and all permissions for others.

   Note: If more than one file is created using the - (minus sign) operand, separate each mode specifier with a comma and no spaces.

Files

/usr/bin/mkfifo Contains the mkfifo command.

Related Information
The chmod command.

The mkfifo subroutine, umask subroutine.

mkfilt Command

Purpose
Activates or deactivates the filter rules.

Syntax

mkfilt [-v] [d] [-u] [zP] [D] [-g] [start | stop] [d]

Description
Use the mkfilt command to activate or deactivate the filter rules. This command can also be used to control the filter logging function.
Flags

-\v
IP version of the rules you want to activate. The value of 4 specifies IP version 4 and the value of 6 specifies IP version 6. The default (when this flag is not used) is to activate both IP version 4 and IP version 6. All the filter rules defined in the filter rule table for the IP version(s) will be activated or deactivated.

-\d
Deactivates the active filter rules. This flag cannot be used with the -\u flag.

-\u
Activates the filter rules in the filter rule table. This flag cannot be used with the -\d flag.

-\z
Sets the action of the default filter rule to Permit (P) or Deny (D). The default filter rule is the last rule in the filter rule table that will apply to traffic that does not apply to any other filter rules in the table.
Setting the action of this rule to Permit will allow all traffic that does not apply to any other filter rules. Setting this action to Deny will not allow traffic that does not apply to any other filter rules.

-\g
This flag is used to either start (start) or stop (stop) the log functionality of the filter rule module.

-\i
Initialization flag. This flag only applies when the -\u flag is also used. If the -\i flag is used, all the filter rules with an "active" status will be activated. If not used, all the filter rules in the filter rule table will be activated.

mkfont Command

Purpose
Adds a font path name to the Object Data Manager (ODM) that is loaded by the low function terminal (LFT) at boot time.

Syntax
mkfont [\texttt{FontPathName}]

Description
The \texttt{mkfont} command adds a fully qualified font file path name to the ODM. At boot time, the LFT loads the new font and any other fonts found in the ODM. The list of font information acquired by the LFT is passed to the default display device driver. The display driver selects from this list the font that best fits the display. If a default font was selected using the \texttt{chfont} command, the device driver uses that font.

\textbf{Note:} This command can be run only from an LFT.

You can use the Devices application in Web-based System Manager (wsm) to change device characteristics. You could also use the System Management Interface Tool (SMIT) \texttt{smit mkfont} fast path to run this command.

The user must have root authority to execute this command.

Parameter

\textit{FontPathName} \hspace{1cm} The fully qualified pathname of a font file.

Examples
To add the font file \texttt{/usr/lpp/fonts/Rom10.snf}, enter the following command:
\texttt{mkfont /usr/lpp/fonts/Rom10.snf}

Files

\texttt{/bin/mkfont} \hspace{1cm} Contains the \texttt{mkfont} command.
\texttt{/usr/lpp/fonts} \hspace{1cm} Contains the font directory.
Related Information
The `chfont` command, `lsfont` command.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

**mkfontdir Command**

**Purpose**
Creates a `fonts.dir` file from a directory of font files.

**Syntax**
`mkfontdir [ DirectoryName ... ]`

**Description**
The `mkfontdir` command creates a `fonts.dir` file from a directory of font files. For each directory argument, the `mkfontdir` command reads all of the bitmapped font files in the directory, searching for properties named FONT or the name of the file stripped of its suffix. These are used as font names, which are written to the `fonts.dir` file in the directory along with the name of the font file. The `fonts.dir` file is then used by the X server and the Font server to determine which fonts are available.

The kinds of font files read by the `mkfontdir` command depend upon the configuration parameters and typically include the following formats:

<table>
<thead>
<tr>
<th>Format</th>
<th>Suffix</th>
</tr>
</thead>
<tbody>
<tr>
<td>Portable Compile Format</td>
<td>.pcf</td>
</tr>
<tr>
<td>Compressed PCF</td>
<td>.pcf.Z</td>
</tr>
<tr>
<td>Server Natural Format</td>
<td>.snf</td>
</tr>
<tr>
<td>Compressed SNF</td>
<td>.snf.Z</td>
</tr>
<tr>
<td>Bitmap Distribution Format</td>
<td>.bdf</td>
</tr>
<tr>
<td>Compressed BDF</td>
<td>.bdf.Z</td>
</tr>
</tbody>
</table>

If a font exists in multiple formats, the most efficient format is used (PCF format before SNF then BDF formats).

Scalable fonts are not automatically recognized by `mkfontdir`. You can construct a `fonts.scale` file (the format is identical to that in the `fonts.dir` file) containing entries for scalable fonts. Then, when you run `mkfontdir` on a directory, it copies entries from the `fonts.scale` file in that directory into the `fonts.dir` file it constructs in that directory.

You can create the `fonts.alias` file, which can be put in any directory of the font path, to map new names to existing fonts. This file should be edited by hand. The format is two columns separated by white space, with the first column containing aliases and the second column containing font-name patterns.

When a font alias is used by an X client, the X server searches for the name it references by looking through each font directory in turn. Therefore, the aliases and the font files do not need to be in the same directory.

To embed white space in aliases or font-name patterns, enclose them in double-quotation marks. To embed double-quotation marks, or any other characters, precede each character with a \\ (backslash).
If the character string FILE_NAMES_ALIAS stands alone on a line, each file name in the directory when stripped of its suffix (such as .pcf or .pcf.Z) is used as an alias for that font.

The X server and the Font Server look for fonts.dir and fonts.alias files in each directory in the font path each time the font path is set.

**Examples**

To create a fonts.dir file from a directory of font files, enter:

```
mkfontdir DirectoryName
```

If no directory name is specified, the `mkfontdir` command reads the current directory.

**Files**

```
/usr/lib/X11/fonts
```

Is the directory containing font files, fonts.dir and fonts.alias files.

---

**mkfs Command**

**Purpose**

Makes a file system.

**Syntax**

```
```

**Description**

The `mkfs` command makes a new file system on a specified device. The `mkfs` command initializes the volume label, file system label, and startup block.

The `Device` parameter specifies a block device name, raw device name, or file system name. If the parameter specifies a file system name, the `mkfs` command uses this name to obtain the following parameters from the applicable stanza in the `/etc/filesystems` file, unless these parameters are entered with the `mkfs` command:

- `dev`  Device name
- `vol`  Volume ID
- `size` File system size
- `boot` Program to be installed in the startup block
- `vfs`  Definition of the virtual file system
- `options` File-system implementation-specific options of the form `Keyword, Keyword=Value`

**Notes:**

1. The file system is created with the setgid (set group ID) bit enabled. The setgid bit determines the default group permissions. All directories created under the new file system have the same default group permissions.

2. The `mkfs` command does not alter anything in a mounted file system, including the file system label. The file system label changes when you change the mount point, unless the file system is mounted.

3. For information about creating a filesystem on a striped logical volume, refer to “File Systems on Striped Logical Volumes” on page 521 in the `mklv` documentation.
Flags

- **Boot**
  Names the program to be installed in block 0 of the new file system.

- **i-Nodes**
  Specifies the initial number of i-nodes on the file system. This flag is ignored when creating a journaled file system.

- **Label**
  Specifies the file system label for the new file system.

- **Options**
  Specifies a comma-separated list of virtual file system implementation-specific options.

The following options are specific to the Journaled File System (JFS):

- **-o ag= [8 | 16 | 32 | 64]**
  Specifies the allocation group size in megabytes. An allocation group is a grouping of i-nodes and disk blocks similar to BSD cylinder groups. The default ag value is 8.

- **-o bf= [true | false]**
  Specifies a large file enabled file system. See "Understanding Large File Enabled File Systems" for more information. If you do not need a large file enabled file system, set this option to false; this is the default. Specifying bf=true requires a fragment size of 4096 and compress=no.

- **-o frag= [512 | 1024 | 2048 | 4096]**
  Specifies the JFS fragment size in bytes. A file system fragment is the smallest unit of disk storage that can be allocated to a file. The default fragment size is 4096 bytes.

- **-o compress= [no | LZ]**
  Specifies data compression. If you do not want data to be compressed, set this option to no. Selecting compression requires a fragment size of 2048 or less.

- **-o nbpi= [512 | 1024 | 2048 | 4096 | 8192 | 16384 | 32768 | 65536 | 131072]**
  Specifies the number of bytes per i-node (nbpi). The nbpi is the ratio of file system size in bytes to the total number of i-nodes. The default nbpi value is 4096 bytes. The values 32768, 65536, and 131072 only apply to AIX 4.2 or later.

Notes:

1. File systems created with an ag value greater than 8 is not recognized on an AIX 4.1 system.
2. The ag, bf, compress, frag, and nbpi attributes are set at file system creation and cannot be changed after the file system is successfully created. The size attribute defines the minimum file system size, and you cannot decrease it after the file system is created.
3. The root filesystem (/) cannot be compressed.
4. Some nbpi values and allocation group sizes are mutually exclusive. See "Understanding JFS Size Limitations" for information.

The following options are specific to the Enhanced Journaled File System (JFS2):

- **-o agblksize= [512 | 1024 | 2048 | 4096]**
  Specifies the JFS2 block size in bytes. A file system block is the smallest unit of disk storage that can be allocated to a file. The default block size is 4096 bytes.

- **-o name=mountpoint**
  Specifies the mount point for the file system.

- **-o log=LVName**
  Specifies the log logical volume name. The specified logical volume is the logging device for the new JFS2.

- **-o log=INLINE**
  Specifies to place the log in the logical volume with the JFS2 file system. The INLINE log will default to .4% of the logical volume size if logsize is not specified.

- **-o logsize=Value**
  Specifies the size for an INLINE log in MBytes. Ignored if INLINE log not being used. Cannot be greater than 10% of the size of the filesystem.

Note: The agblksize attribute is set at file system creation and cannot be changed after the file system is successfully created. The size attribute defines the minimum file system size, and you cannot decrease it after the file system is created.

- **-p Prototype**
  Specifies the name of the prototype file. Options specified on the command line override attributes in the prototype file.

- **-s Size**
  Specifies the size of the file system. Size can be specified in units of 512-byte blocks, Megabytes (suffix M should be used) or Gigabytes (suffix G should be used). See "Understanding JFS Size Limitations" for more information.
Notes:
1. The volume group in which the file system resides defines a maximum logical volume size and also limits the file system size.
2. The \texttt{-s} Size flag specifies the minimum file size and cannot be decreased after the file system has been successfully created.

\texttt{-v VolumeLabel} \hspace{1cm} Specifies the volume label for the new file system.
\texttt{-V VfsName} \hspace{1cm} Specifies the virtual file system (VFS) type. The VFS must have an entry in the \texttt{/etc/vfs} file.

Examples
1. To specify the volume and file system name for a new file system, type:
   \texttt{mkfs -lworks -vvol001 /dev/hd3}
   This command creates an empty file system on the \texttt{/dev/hd3} device, giving it the volume serial number \texttt{vol001} and file system name \texttt{works}. The new file system occupies the entire device. The file system has a default fragment size (4096 bytes) and a default nbpi ratio (4096).

2. To create a file system with nondefault attributes, type:
   \texttt{mkfs -s 8192 -o nbpi=2048,frag=512 /dev/lv01}
   This command creates an empty 4MB file system on the \texttt{/dev/lv01} device with 512-byte fragments and 1 i-node for each 2048 bytes.

3. To create a large file enabled file system, type:
   \texttt{mkfs -V jfs -o nbpi=131072,bf=true,ag=64 /dev/lv01}
   This creates a large file enabled JFS file system with an allocation group size of 64 megabytes and 1 i-node for every 131072 bytes of disk. The size of the file system will be the size of the logical volume \texttt{lv01}.

4. To create a file system with nondefault attributes, type:
   \texttt{mkfs -s 4M -o nbpi=2048, frag=512 /dev/lv01}
   This command creates an empty 4MB file system on the \texttt{/dev/lv01} device with 512-byte fragments and one i-node for each 2048 bytes.

Files
\texttt{/etc/vfs} \hspace{1cm} Contains descriptions of virtual file system types.
\texttt{/etc/filesystems} \hspace{1cm} Lists the known file systems and defines their characteristics.

Related Information
The \texttt{fsck} command, \texttt{mkproto} command, \texttt{proto} command.
The \texttt{ioctl} subroutine.
The \texttt{dir} file, \texttt{filesystems} file, \texttt{filsys.h} file.

Understanding JFS Size Limitations in \textit{AIX 5L Version 5.2 System Management Concepts: Operating System and Devices}.
**mkgroup Command**

**Purpose**
Creates a new group.

**Syntax**
mkgroup [ -R load_module ] [ -a ] [ -A ] [ Attribute=Value ... ] Group

**Description**
The `mkgroup` command creates a new group. The Group parameter must be a unique string of 8-byte or less and cannot be the ALL or default keywords. By default, the `mkgroup` command creates a standard group. To create an administrative group, specify the -a flag. You must be the root user or a user with GroupAdmin authorization to create an administrative group.

To create a group with an alternate Identification and Authentication (I&A) mechanism, the -R flag can be used to specify the I&A load module used to create the group. Load modules are defined in the `/usr/lib/security/methods.cfg` file.

You can use the Users application in Web-based System Manager (wsm) to change user characteristics. You could also use the System Management Interface Tool (SMIT) smit mkgroups fast path to run this command.

**Restrictions on Creating Group Names**
To prevent login inconsistencies, you should avoid composing group names entirely of uppercase alphabetic characters. While the `mkgroup` command supports multi-byte group names, it is recommended that you restrict group names to characters with the POSIX portable filename character set.

To ensure that your user database remains uncorrupted, you must be careful when naming groups. Group names must not begin with a - (dash), + (plus sign), @ (at sign), or ~ (tilde). You cannot use the keywords ALL or default in a group name. Additionally, do not use any of the following characters within a group-name string:

- : Colon
- " Double quote
- # Pound sign
- , Comma
- = Equal sign
- \ Back slash
- / Slash
- ? Question mark
- ' Single quote
- ` Back quote

Finally, the Name parameter cannot contain any space, tab, or new-line characters.

**Flags**
- `-a` Creates an administrative group. Only the root user can use this flag.
- `-A` Sets the group administrator to the person who invoked the `mkgroup` command.
- `-R load_module` Specifies the loadable I&A module used to create the user.
- `Attribute=Value` Initializes a group with a specific attribute. See the `chgroup` command for more information about the group attributes.
Security

Access Control: This command should grant execute (x) access only to the root user and members of the security group. This command should be installed as a program in the trusted computing base (TCB). The command should be owned by the root user with the `setuid` (SUID) bit set.

Files Accessed:

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>rw</td>
<td>/etc/passwd</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/user</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/limits</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/environ</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/group</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/group</td>
</tr>
<tr>
<td>r</td>
<td>/usr/lib/security/mkuser.default</td>
</tr>
<tr>
<td>x</td>
<td>/usr/lib/security/mkuser.sys</td>
</tr>
</tbody>
</table>

Auditing Events:

<table>
<thead>
<tr>
<th>Event</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>USER_Create</td>
<td>user</td>
</tr>
</tbody>
</table>

Limitations

Creating a group may not be supported by all loadable I&A modules. If the loadable I&A module does not support creating a group, an error is reported.

Examples

1. To create a new group account called finance, type:

   `mkgroup finance`

2. To create a new administrative group account called payroll, type:

   `mkgroup -a payroll`

   Only the root user can issue this command.

3. To create a new group account called managers and set yourself as the administrator, type:

   `mkgroup -A managers`

4. To create a new group account called managers and set the list of administrators to steve and mike, type:

   `mkgroup adms=steve,mike managers`

   The users steve and mike must already exist on the system.

5. To create a new group that is a LDAP I&A loadable module user, type:

   `mkgroup -R LDAP monsters`

Files

- `/usr/bin/mkgroup`: Contains the `mkgroup` command.
- `/etc/group`: Contains the basic attributes of groups.
- `/etc/security/group`: Contains the extended attributes of groups.
- `/etc/passwd`: Contains basic user information.
- `/etc/security/passwd`: Contains password information.
Related Information

The `chgroup` command, `chgrpmem` command, `chuser` command, `isgroup` command, `lsuser` command, `mkuser` command, `passwd` command, `pwdadm` command, `rmgroup` command, `rmuser` command, `setgroups` command, `setsenv` command.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the AIX 5L Version 5.2 Security Guide.

mkhosts Command

Purpose
Generates the host table file.

Syntax
```
/usr/sbin/mkhosts [ -v ] HostFile
```

Description
The `mkhosts` command can be used to generate a hashed host database, using the filename specified by the `HostFile` parameter. It is not used if name resolution is performed by the `named` daemon. The host file is usually the `/etc/hosts` file, and in any case must be in the same format as the `/etc/hosts` file.

The `mkhosts` command generates database files named `hostfile.pag` and `hostfile.dir`. Updates to these files are built in a set of temporary files named `hostfile.new.pag` and `hostfile.new.dir`. The temporary files are copied into the database files only if the `hostfile.new.pag` and `hostfile.new.dir` files are built without errors.

The host file is used by one version of the `gethostbyaddr` and `gethostbyname` library routines for name resolution.

Note: The version of the `gethostbyaddr` and `gethostbyname` library routines on this operating system do not support the `hostfile.pag` and `hostfile.dir` files.

After creating the host file, you can edit it to include the desired host entries.

Flags

-v Lists each host as it is added to the host file specified by the `HostFile` parameter.

Examples

Use the following command to generate the `/etc/hosts.pag` and `/etc/hosts.dir` files:
```
mkhosts /etc/hosts
```

This command creates two host files called `/etc/hosts.pag` and `/etc/hosts.dir`.

Files

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>hostfile.pag</code></td>
<td>One of two files containing the real database for name resolution.</td>
</tr>
<tr>
<td><code>hostfile.dir</code></td>
<td>One of two files containing the real database for name resolution.</td>
</tr>
</tbody>
</table>
Related Information
The `gettable` command, `ntable` command.
The `named` daemon.
The `gethostbyname` subroutine, `gethostbyaddr` subroutine.
The `hosts` file format.

Naming in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

mkitab Command

Purpose
Makes records in the `/etc/inettab` file.

Syntax
`mkitab [-I] Identifier { [ Identifier ] : [ RunLevel ] : [ Action ] : [ Command ] }

Description
The `mkitab` command adds a record to the `/etc/inettab` file. The `Identifier:RunLevel:Action:Command` parameter string specifies the new entry to the `/etc/inettab` file. You can insert a record after a specific record using the `-i` `Identifier` flag. The command finds the field specified by the `Identifier` parameter and inserts the new record after the one identified by the `-i` `Identifier` flag.

Parameters
The `Identifier:RunLevel:Action:Command` parameter string specifies the record in the `/etc/inettab` file, as follows:

- `Identifier`: A 14-character parameter that uniquely identifies an object. The `Identifier` must be unique. If the `Identifier` is not unique, the command is unsuccessful. The `Identifier` cannot be changed; if you try to change it, the command is unsuccessful.

- `RunLevel`: A 20-character parameter defining the run levels in which the `Identifier` can be processed. Each process started by the `init` command can be assigned one or more run levels in which it can be started.

- `Action`: A 20-character parameter that informs the `init` command how to process the `Command` parameter that you specify. The `init` command recognizes the following actions:
  - `respawn`: If the process identified in this record does not exist, start the process. If the process currently exists, do nothing and continue scanning the `/etc/inettab` file.
  - `wait`: When the `init` command enters the run level specified for this record, start the process and wait for it to stop. While the `init` command is in the same run level, all subsequent reads of the `/etc/inettab` file ignore this object.
  - `once`: When the `init` command enters the run level specified for this record, start the process, do not wait for it to stop and when it does stop do not restart the process. If the system enters a new run level while the process is running, the process is not restarted.
**boot**  
Read this record only when the system boots and reads the `/etc/inittab` file. The `init` command starts the process. Do not wait for the process to stop and when it does stop, do not restart the process. The run level for this process should be the default, or it must match the run level specified by the `init` command at startup time.

**bootwait**  
Read this record only when the system boots and reads the `/etc/inittab` file. The `init` command starts the process. Wait for it to stop, and when it does stop, do not restart the process.

**powerfail**  
Start the process identified in this record only when the `init` command receives a SIGPWR power fail signal.

**powerwait**  
Start the process identified in this record only when the `init` command receives a SIGPWR power fail signal, and wait until it stops before continuing to process the `/etc/inittab` file.

**off**  
If the process identified in this record is currently running, send the warning signal SIGTERM and wait 20 seconds before sending the SIGKILL kill signal. If the process is nonexistent, ignore this line.

**hold**  
When the process identified in this record is terminated, do not start a new one. The `hold` action can only be activated by the `phold` command.

**ondemand**  
Functionally identical to respawn. If the process identified in this record does not exist, start the process. If the process currently exists, do nothing and continue scanning the `/etc/inittab` file. Specify this action to perform the respawn action when using a, b, or c run levels.

**initdefault**  
A line with this action is processed only when the `init` command is originally invoked. The `init` command uses this line to determine which run level to originally enter. The command does this by taking the highest run level specified in the `RunLevel` parameter and using that as the command's initial state. If the `RunLevel` parameter is empty, its value is interpreted as 0123456789, and the `init` command enters a run level of 9. If the `init` command does not find an initdefault line in the `inittab` file, it requests an initial run level from the operator at initial program load (IPL) time.

**sysinit**  
Start the process identified in this record before the `init` command tries to access the console. For example, you might use this to initialize devices.

*Command*  
A 1024-character field specifying the shell command.

**Attention:** To avoid possible corruption of system files, the `stdin`, `stdout`, and `stderr` files must be specified in the `Command` parameter with redirection, or they must be explicitly opened by the program being run by the command line.

### Flags

- `-i` *Identifier*  
  Specifies which record in the `/etc/inittab` file the new record follows.

### Examples

1. To add a new record to the `/etc/inittab` file, telling the `init` command to handle a login on tty2, type:
   ```bash
   mkitab "ttys002:2:respawn:/usr/sbin/getty /dev/tty2"
   ```

2. To add a new record to the `/etc/inittab` file, telling the `init` command to execute the `/etc/rc.tcpip` file after the `/usr/sbin/srcmstr` file is started, type:
   ```bash
   mkitab -i srcmstr "rctcpip:2:wait:/etc/rc.tcpip > /dev/console"
   ```
3. To add a new record to the `/etc/inittab` file, telling the `init` command to execute the `/etc/rc` file and send its output to the boot log, type:

   ```bash
   mkitag ((rc:2:wait:/etc/rc 2>&1 | alog -tboot > /dev/console))
   ```

**Files**

`/etc/inittab` Contains the `mkitab` command.

**Related Information**

The `chitab` command, `lsitab` command, `rmitab` command, `init` command.

---

### mkinstallp Command

**Purpose**

Creates software packages in `installp` format.

**Syntax**

```bash
mkinstallp [ -d BaseDirectory ] [ -T TemplateFile ]
```

**Description**

The `mkinstallp` command allows users to create their own software packages for AIX. Packages created with the `mkinstallp` command are in `installp` format and can be installed or removed with the `installp` command.

Files to be packaged by the `mkinstallp` command must be in a directory structure such that the location of the file relative to the root build directory is the same as the destination of the file after installation. For example, if `/usr/bin/somecommand` is to be installed through a `mkinstallp` package, the `somecommand` parameter must be in the `buildroot/usr/bin` directory when the `mkinstallp` command is run.

After the contents of a package are located in the correct directory structure, the `mkinstallp` command prompts for basic package data. This data includes the package name, requisites, descriptions of files to be packaged, and more. The `mkinstallp` command will then generate a template file based on responses given by the user. To prevent command-line prompting, template files can be created and edited directly by the user and passed to the `mkinstallp` command with the `-T` flag.

**Flags**

- `-d BaseDirectory` Specifies the root build directory containing the files to be packaged. If not specified, the current working directory is used.

- `-T TemplateFile` Specifies the full path name of the template file to be passed to the `mkinstallp` command. If not specified, the `mkinstallp` command prompts for package information and creates a new template file based on user responses.

**Examples**

This example demonstrates how to package the file `/usr/bin/foo` using the `/tmp/packages` directory as the root build directory.

First, create the directory structure by typing the following at the command line:

```bash
mkdir -p /tmp/packages/usr/bin
```

Then, type the following to create the file `/usr/bin/foo`:
touch /tmp/packages/usr/bin/foo

Then, type the following to create the package using the `mkinstallp` command:
```
mkinstallp -d /tmp/packages
```

For more examples, see the `/usr/lpp/bos/README.MINSTALLP` file.

**Files**

```
/usr/sbin/mkinstallp
```

Contains the `mkinstallp` command.

**Related Information**

The `/usr/lpp/bos/README.MINSTALLP` file.

---

**mkkeyserv Command**

**Purpose**

Uncomments the entry in the `/etc/rc.nfs` file for the `keyserv` daemon and invokes the daemon by using the `startsrc` command.

**Syntax**

```
/usr/sbin/mkkeyserv [-I | -B | -N]
```

**Description**

The `mkkeyserv` command uncomments the entry in the `/etc/rc.nfs` file for the `keyserv` daemon. The `mkkeyserv` command starts the daemon by using the `startsrc` command.

You can use the File Systems application in Web-based System Manager (wsm) to change file system characteristics. You could also use the System Management Interface Tool (SMIT) `smit mkkeyserv` fast path to run this command.

**Flags**

- **-I**
  Uncomments the entry in the `/etc/rc.nfs` file to start the `keyserv` daemon on the next system restart.

- **-B**
  Uncomments the entry in the `/etc/rc.nfs` file to start the `keyserv` daemon and uses the `startsrc` command to start the `keyserv` daemon. This flag is the default.

- **-N**
  Uses the `startsrc` command to start the `keyserv` daemon. This flag does not change the `/etc/rc.nfs` file.

**Examples**

To modify the `/etc/rc.nfs` file to invoke the `keyserv` daemon on the next system restart, enter:
```
mkkeyserv -I
```

**Files**

```
/etc/rc.nfs
```

Contains the startup script for the NFS and NIS daemons.
Related Information

The `smit` command, `startsrc` command.

The `keyserv` daemon.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mkkrb5clnt Command

Purpose

Configures a Kerberos client.

Syntax

```
```

Description

This command configures the Kerberos client. The first part of the command reads realm name, KDC, VDB path, and domain name from the input and generates a `krb5.conf` file.

```
/etc/krb5/krb5.conf: Values for realm name, Kerberos admin server, and domain name are set as specified on the command line. Also updates the paths for default_keytab_name, kdc, and kadmin log files.
```

If DCE is not configured, this command creates a link to `/etc/krb5/krb5.conf` from `/etc/krb5.conf`.

The command also allows you to configure root as admin user, configure integrated Kerberos authentication, and configure Kerberos as default authentication scheme.

For integrated login, the `-i` flag requires the name of the database being used. For LDAP, use the load module name that specifies LDAP. For local files, use the keyword files.

Standard Output Consists of information messages when the `-h` flag is used.

Standard Error Consists of error messages when the command cannot complete successfully.

Flags

- `-a Admin` Specifies the principal name of the Kerberos server admin.
- `-A` Specifies root to be added as a Kerberos administrative user.
-c KDC Specifies the KDC server.
-d Domain Specifies the complete domain name for the Kerberos client.
-h Specifies that the command is only to display the valid command syntax.
-I Database Configures integrated Kerberos authentication.
-K Specifies Kerberos to be configured as the default authentication scheme.
-L ldapserver|ldapserver:port For servers, specifies the LDAP directory used to store the Network Authentication Service principal and policy information.
For clients, specifies the LDAP directory server to use for Administration server and KDC discovery using LDAP. If the -I flag is used, then the KDC and server flags are optional. If the -I option is not used, the KDC and server flags must be specified. The port number can optionally be specified.
Note: Only the client configuration is updated.
-R Realm Specifies the full realm name for which the Kerberos client is to be configured.
-S Server Specifies the fully qualified host name for Kerberos admin server.
-T Specifies the flag to acquire server admin TGT based admin ticket.
-U Undo the setup from the previous configuration command.

Exit Status
Failure of this command to execute successfully may result in incomplete client configuration.

0 Indicates the successful completion of the command.
1 Indicates that an error occurred.

Security
Only the root user is authorized to use this command.

Examples
1. To display the command syntax, type:
   mkkrb5clnt -h
2. To configure testbox.austin.ibm.com as a client to sundial.austin.ibm.com where KDC is also running on sundial.austin.ibm.com, type:
   mkkrb5clnt -c sundial.austin.ibm.com -r UD3A.AUSTIN.IBM.COM
   -s sundial.austin.ibm.com -d austin.ibm.com
3. To configure testbox.austin.ibm.com as the client, make root as the server admin, configure integrated login, configure Kerberos as default authentication scheme, type:
   mkkrb5clnt -c sundial.austin.ibm.com -r UD3A.AUSTIN.IBM.COM
   -s sundial.austin.ibm.com -d austin.ibm.com
   -A -i files -K

Files
/usr/krb5/sbin Contains the mkkrb5clnt command.

mkkrb5srv Command

Purpose
Configures a Kerberos server.
Syntax

```
mkkrb5srv [h] [R Realm [s Server] [d Domain [a AdminName] [l ldapserver | ldapserver:port] [p
ldap_DN] [p ldap_DN_pw] [f (keyring | keyring:entry_dn)]] [k keyring_pw] [b bind_type] [m
masterkey_location] [U]
```

Description

The `mkkrb5srv` command configures the Kerberos server. This command creates the `kadm5.acl` file, the `kdc.conf` file, and the Kerberos database. It also adds the administrator to the database and updates the `/etc/inittab` file with Kerberos daemons. This command does the initial configuration once the variables are set. They can be modified by editing the following files:

```
/etc/krb5/krb5.conf:
```
Values for realm name, Kerberos admin server, and domain name are set as specified on the command line. Also updates the paths for `default_keytab_name`, `kdc`, and `kadim` log files.

```
/var/krb5/krb5kdc/kdc.conf:
```
This command sets the value for `kdc_ports`. Paths for database name, `admin_keytab`, `acl_file`, `dict_file`, `key_stash_file`. Values for `kadim_port`, `max_life`, `max_renewable_life`, `master_key_type`, and `supported_enctypes`.

```
/var/krb5/krb5kdc/kadm5.acl:
```
Sets up the acs for admin, root, and host principals.

If DCE is not configured, this command creates a link to `/etc/krb5/krb5.conf` from `/etc/krb5.conf`.

**Standard Output**: Consists of information messages when the `-h` flag is used.

**Standard Error**: Consists of error messages when the command cannot complete successfully.

Flags

- `a AdminName`
  Specifies the Kerberos Principal name for the administrator.

- `b bind_type`
  Specifies the LDAP bind type. Supported values are the following:
  - simple
  - cram-md5
  - external

  These bind types can be specified in either upper case or lower case.

- `d Domain`
  Specifies the domain name for the Kerberos realm.

- `f (keyring | keyring:entry_dn)`
  Specifies the LDAP keyring database file name if you are using SSL communication.

- `h`
  Specifies that the command is only to display the valid command syntax.

- `k keyring_pw`
  Specifies the password for the LDAP keyring database file. If not specified, SSL uses the password that is encrypted in the appropriate password stash file.

- `l ldapserver | ldapserver:port`
  For servers, specifies the LDAP directory used to store the Network Authentication Service principal and policy information.

  For clients, specifies the LDAP directory server to use for Administration server and KDC discovery using LDAP. If the `-l` flag is used, then the KDC and server flags are optional. If the `-l` option is not used, the KDC and server flags must be specified. The port number can optionally be specified.

  For clients and servers, the port number can optionally be specified. If the port number is not specified, the client connects to the default LDAP server port 389 or 636 for SSL connections.

  **Note**: Only the client configuration is updated.

- `m masterkey_location`
  Specifies the fully qualified file name for storing the master key in the local file system when using LDAP to store data.

  **Note**: This flag is only for use with the LDAP directory.
-p ldap_DN_pw
  Specifies the password for the entry being used for the ldap_DN_pw.
-r Realm
  Specifies the realm for which the Kerberos server is to be configured.
-s Server
  Specifies the fully qualified name of Kerberos Admin Server.
-u ldap_DN
  Specifies the LDAP entry to be used as the ldap_DN.
  Note: With external bind, the -u and -p flags are not required, and the values come form the certificate.
-U
  Undo the setup from the previous configuration command.

Exit Status
Failure of this command to execute successfully results in incomplete server configuration.
0
  Indicates the successful completion of the command.
1
  Indicates that an error occurred.

Security
Only the root user is authorized to use this command.

Examples
1. To display the command syntax, type:
   mkkrb5srv -h
2. To configure sundial as a Kerberos server, type:
   mkkrb5srv -r UD3A.AUSTIN.IBM.COM -s sundial.austin.ibm.com -d austin.ibm.com

Files
/usr/sbin/mkkrb5srv
  Contains the mkkrb5srv command.

mklost+found Command

Purpose
Creates a lost and found directory for the fsck command.

Syntax
mklost+found

Description
The mklost+found command creates a lost and found directory in the current directory. A number of empty files are created within the lost and found directory and then removed so that there are empty slots for the fsck command. The fsck command reconnects any orphaned files and directories by placing them in the lost and found directory with an assigned i-node number. The mklost+found command is not normally needed, since the fsck command automatically creates the lost and found directory when a new file system is created.

Examples
To make a lost+found directory for the fsck command, enter:
mklost+found
Files

/show/sbin/mklost+found Contains the mklost+found command.

Related Information

The fsck command, mksfs command.


mklv Command

Purpose

Creates a logical volume.

Syntax

[-G Groupid] [-f Mode] VolumeGroup Number [ PhysicalVolume ... ]

Description

The mklv command creates a new logical volume within the VolumeGroup. For example, all file systems must be on separate logical volumes. The mklv command allocates the number of logical partitions to the new logical volume. If you specify one or more physical volumes with the PhysicalVolume parameter, only those physical volumes are available for allocating physical partitions; otherwise, all the physical volumes within the volume group are available.

The default settings provide the most commonly used characteristics, but use flags to tailor the logical volume to the requirements of your system. After a logical volume is created, its characteristics can be changed with the chlv command.

The default allocation policy is to use a minimum number of physical volumes per logical volume copy, to place the physical partitions belonging to a copy as contiguously as possible, and then to place the physical partitions in the desired region specified by the -a flag. Also, by default, each copy of a logical partition is placed on a separate physical volume.

The -m flag specifies exact physical partitions to be used when creating the logical volume.

If the volume group in which the logical volume is being created is in big vg format, U, G, and P flags can be used to set the ownership, group, and permissions respectively, of the special device files. Only root user will be able to set these values. If the volume group is exported, these values can be restored upon import if R flag is specified with importvg command.

You can specify logical volumes sizes in 512 Blocks/KB/MB/GB when using the mklv command. (See "Examples" on page 524).

Physical partitions are numbered starting at the outermost edge with number one.

Notes:

1. Changes made to the logical volume are not reflected in the file systems. To change file system characteristics use the chfs command.
2. Each logical volume has a control block. This logical volume control block is the first few hundred bytes within the logical volume. Care has to be taken when reading and writing directly to the logical volume to allow for the control block. Logical volume data begins on the second 512-byte block.

3. To use this command, you must either have root user authority or be a member of the system group.

4. When creating a striped logical volume using the -S flag, you must specify two or more physical volumes or use the -u flag.

5. When creating a striped logical volume, the number of partitions must be an even multiple of the striping width.

6. To create a striped logical volume with more than one copy, all active nodes should be at least AIX 4.3.3 or later when the volume group is in the concurrent mode.

7. The mklv command is not allowed on a snapshot volume group.

You can use the Volumes application in Web-based System Manager to change volume characteristics. You could also use the System Management Interface Tool (SMIT) smit mklv fast path to run this command.

**File Systems on Striped Logical Volumes**

If you want to create a filesystem on a striped logical volume, you should create the striped logical volume before you run the crfs command or mkfs command to create the filesystem. In order to maximize the use of disk space within the striping width, you should choose hard disks of the same size when creating the striped logical volume. The striping width is the number of hard disks that form the striped logical volume.

**Flags**

- **-a Position**
  
  Sets the intra-physical volume allocation policy (the position of the logical partitions on the physical volume). The Position variable can be one of the following:
  
  m  Allocates logical partitions in the outer middle section of each physical volume. This is the default position.
  
  c  Allocates logical partitions in the center section of each physical volume.
  
  e  Allocates logical partitions in the outer edge section of each physical volume.
  
  ie  Allocates logical partitions in the inner edge section of each physical volume.
  
  im  Allocates logical partitions in the inner middle section of each physical volume.
  
- **-b BadBlocks**
  
  Sets the bad-block relocation policy. The Relocation variable can be one of the following:
  
  y  Causes bad-block relocation to occur. This is the default.
  
  n  Prevents bad-block relocation from occurring.
  
- **-c Copies**
  
  Sets the number of physical partitions allocated for each logical partition. The Copies variable can be set to a value from 1 to 3; the default is 1.
-d Schedule

Sets the scheduling policy when more than one logical partition is written. The Schedule variable can be one of the following:

p Establishes a parallel scheduling policy. This is the default for scheduling policy.

ps Parallel write with sequential read policy. All mirrors are written in parallel but always read from the first mirror if the first mirror is available.

pr Parallel write round robbin read. This policy is similar to the parallel policy except an attempt is made to spread the reads to the logical volume more evenly across all mirrors.

s Establishes a sequential scheduling policy.

-e Range

Sets the inter-physical volume allocation policy (the number of physical volumes to extend across, using the volumes that provide the best allocation). The Range value is limited by the UpperBound variable, (set with the -u flag) and can be one of the following:

x Allocates across the maximum number of physical volumes.

m Allocates logical partitions across the minimum number of physical volumes. This is the default range.

-G Groupid

Specifies group ID for the logical volume special file.

-i

Reads the PhysicalVolume parameter from standard input. Use the [-i] flag only when PhysicalVolume is entered through standard input.

-L

Sets the logical volume label. The default label is None. The maximum size of the label file is 127 characters.

Note: If the logical volume is going to be used as a journaled file system (JFS), then the JFS will use this field to store the mount point of the file system on that logical volume for future reference.

-m MapFile

Specifies the exact physical partitions to allocate. Partitions are used in the order given in the MapFile parameter. Used partitions in the MapFile parameter are not legal, because the new logical volume cannot occupy the same physical space as a previously allocated logical volume. All physical partitions belonging to a copy are allocated before allocating for the next copy of the logical volume. The MapFile parameter format is: PVname:PPnum1[-PPnum2]. In this example, PVname is a physical volume name (for example, hdisk0) as specified by the system. It is one record per physical partition or a range of consecutive physical partitions. PPnum is the physical partition number.

PVname Name of the physical volume as specified by the system.

PPnum Physical partition number.

-o Y / N

Turns on/off serialization of overlapping IOs. If serialization is turned on then overlapping IOs are not allowed on a block range and only a single IO in a block range is processed at any one time. Most applications like file systems and databases do serialization so serialization should be turned off. The default for new logical volumes is off.

-P Modes

Specifies permissions (file modes) for the logical volume special file.
-r Relocate

Sets the reorganization relocation flag. For striped logical volumes, the Relocate parameter must be set to n (the default for striped logical volumes). The Relocate parameter can be one of the following:

y Allows the logical volume to be relocated during reorganization. This is the default for relocation.

n Prevents the logical volume from being relocated during reorganization.

-s Strict

Determines the strict allocation policy. Copies of a logical partition can be allocated to share or not to share the same physical volume. The Strict parameter is represented by one of the following:

y Sets a strict allocation policy, so copies for a logical partition cannot share the same physical volume. This is the default for allocation policy.

n Does not set a strict allocation policy, so copies for a logical partition can share the same physical volume.

s Sets a super strict allocation policy, so that the partitions allocated for one mirror cannot share a physical volume with the partitions from another mirror.

-S StripeSize

Specifies the number of bytes per striped. Must be a power of two, between 4K and 128K, for example 4K, 8K, 16K, 32K, 64K, or 128K.

Note: The -d, -e, and -s flags are not valid when creating a striped logical volume using the -S flag.

-t Type

Sets the logical volume type. The standard types are jfs (journal file systems), jfslog (journal file system logs), jfs2 (enhanced journal file system), jfs2log (enhanced journal file system logs), and paging (paging spaces), but a user can define other logical volume types with this flag. You cannot create a striped logical volume of type boot. The default is jfs. If a log is manually created for a filesystem, the user must run the logform command to clean out the new jfslog before the log can be used. For example, to format the logical volume logdev, type:

logform /dev/logdev

where /dev/logdev is the absolute path to the logical volume.

-U Userid

Specifies user ID for logical volume special file.

-u UpperBound

Sets the maximum number of physical volumes for new allocation. The value of the Upperbound variable should be between one and the total number of physical volumes. When using striped logical volumes or super strictness the upper bound indicates the maximum number of physical volumes allowed for each mirror copy.

Note: When creating a super strict logical volume you must specify physical volumes or use the -u flag.

-v Verify

Sets the write-verify state for the logical volume. Causes (y) all writes to the logical volume to either be verified with a follow-up read, or prevents (n) the verification of all writes to the logical volume. The Verify parameter is represented by one of the following:

n Prevents the verification of all write operations to the logical volume. This is the default for the -v flag.

y Causes the verification of all write operations to the logical volume.
-w MirrorWriteConsistency

**y or a** Turns on active mirror write consistency that ensures data consistency among mirrored copies of a logical volume during typical I/O processing.

**p** Turns on passive mirror write consistency that ensures data consistency among mirrored copies during volume group synchronization after a system interruption.

**Note:** This functionality is only available on Big Volume Groups.

**n** No mirror write consistency. See the flag of the syncvg command.

-x Maximum

Sets the maximum number of logical partitions that can be allocated to the logical volume. The default value is 512. The number represented by the Number parameter must be equal to or less than the number represented by the Maximum variable. The maximum number of logical partitions per logical volume is 32,512.

-y NewLogicalVolume

Specifies the logical volume name to use instead of using a system-generated name. Logical volume names must be unique system wide name, and can range from 1 to 15 characters. If the volume group is varied on in concurrent mode, the new name should be unique across all the concurrent nodes the volume group is varied on. The name cannot begin with a prefix already defined in the PdDv class in the Device Configuration Database for other devices.

-Y Prefix

Specifies the Prefix to use instead of the prefix in a system-generated name for the new logical volume. The prefix must be less than or equal to 13 characters. The name cannot begin with a prefix already defined in the PdDv class in the Device Configuration Database for other devices, nor a name already used by another device.

**Examples**

1. To make a logical volume in volume group vg02 with one logical partition and a total of two copies of the data, type:
   
   mklv -c 2 vg02 1

2. To make a logical volume in volume group vg03 with nine logical partitions and a total of three copies spread across a maximum of two physical volumes, and whose allocation policy is not strict, type:
   
   mklv -c 3 -u 2 -s n vg03 9

3. To make a logical volume in vg04 with five logical partitions allocated across the center sections of the physical volumes when possible, with no bad-block relocation, and whose type is paging, type:
   
   mklv a d -t paging b n vg04 5

4. To make a logical volume in vg03 with 15 logical partitions chosen from physical volumes hdisk5, hdisk6, and hdisk9, type:
   
   mklv vg03 15 hdisk5 hdisk6 hdisk9

5. To make a striped logical volume in vg05 with a stripe size of 64K across 3 physical volumes and 12 logical partitions, type:
   
   mklv -u 3 -s 64K vg05 12

6. To make a striped logical volume in vg05 with a stripe size of 8K across hdisk1, hdisk2, and hdisk3 and 12 logical partitions, type:
   
   mklv -s 8K vg05 12 hdisk1 hdisk2 hdisk3

7. To request a logical volume with a minimum size of 10MB, type:
   
   mklv VGNAME 10M #
The `mklv` command will determine the number of partitions needed to create a logical volume of at least that size.

You can use uppercase and lowercase letters as follows:

- `B/b` 512 byte blocks
- `K/k` KB
- `M/m` MB
- `G/g` GB

**Files**

- `/usr/sbin` Directory where the `mklv` command resides.
- `/tmp` Directory where the temporary files are stored while the command is running.
- `/dev` Directory where the character and block device entries for the logical volume are created.

**Related Information**

The `chfs` command, `chlv` command, `chpv` command, `extendlv` command, `mklvcopy` command, `rmllvcopy` command, `syncvg` command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


**mklvcopy Command**

**Purpose**

Provides copies of data within the logical volume.

**Syntax**

```
mklvcopy [ -a Position ] [ -e Range ] [ -k ] [ -m MapFile ] [ -s Strict ] [ -u UpperBound ]
LogicalVolume Copies [PhysicalVolume...]
```

**Description**

The `mklvcopy` command increases the number of copies in each logical partition in `LogicalVolume`. This is accomplished by increasing the total number of physical partitions for each logical partition to the number represented by `Copies`. The `LogicalVolume` parameter can be a logical volume name or logical volume ID. You can request that the physical partitions for the new copies be allocated on specific physical volumes (within the volume group) with the `PhysicalVolume` parameter; otherwise, all the physical volumes within the volume group are available for allocation.

The logical volume modified with this command uses the `Copies` parameter as its new `copy` characteristic. The data in the new copies are not synchronized until one of the following occurs: the `-k` option is used, the volume group is activated by the `varyonvg` command, or the volume group or logical volume is synchronized explicitly by the `syncvg` command. Individual logical partitions are always updated as they are written to.
The default allocation policy is to use minimum numbering of physical volumes per logical volume copy, to place the physical partitions belong to a copy as contiguously as possible, and then to place the physical partitions in the desired region specified by the -a flag. Also, by default, each copy of a logical partition is placed on a separate physical volume.

Notes:
1. To use this command, you must either have root user authority or be a member of the system group.
2. To create a copy of a striped logical volume, all active nodes using the volume group must be at least AIX 4.3.3 or later. Older versions will not be able to use the volume group after a mirror copy has been added to the striped logical volume.
3. The mklvcopy command is not allowed on a snapshot volume group.

You can use the Volumes application in Web-based System Manager (wsm) to change volume characteristics. You could also use the System Management Interface Tool (SMIT) smit mklvcopy fast path to run this command.

Flags
- Note: The -e, -m, -s, and -u flags are not valid with a striped logical volume.

-a Position
Sets the intra-physical volume allocation policy (the position of the logical partitions on the physical volume). The Position variable can be one of the following:
- m Allocates logical partitions in the outer middle section of each physical volume. This is the default position.
- c Allocates logical partitions in the center section of each physical volume.
- e Allocates logical partitions in the outer edge section of each physical volume.
- ie Allocates logical partitions in the inner edge section of each physical volume.
- im Allocates logical partitions in the inner middle section of each physical volume.

-e Range
Sets the inter-physical volume allocation policy (the number of physical volumes to extend across, using the volumes that provide the best allocation). The Range value is limited by the Upperbound variable (set with the -u flag), and can be one of the following:
- x Allocates across the maximum number of physical volumes.
- m Allocates logical partitions across the minimum number of physical volumes. This is the default for the -e flag.

-k
Synchronizes data in the new partitions.

-m MapFile
Specifies the exact physical partitions to allocate. Partitions are used in the order given by the file designated by the MapFile parameter. Used partitions in the file are skipped. All physical partitions belonging to a copy are allocated before allocating for the next copy. The MapFile format is:

PVname: PPnum1[-PPnum2]
where PVname is a physical volume name (for example, hdisk0). It is one record per physical partition or a range of consecutive physical partitions.

PVname Name of the physical volume as specified by the system.
PPnum Physical partition number.
-s Strict

Determines the strict allocation policy. Copies of a logical partition can be allocated to share or not to share the same physical volume. The Strict variable is represented by one of the following:

y Sets a strict allocation policy, so copies for a logical partition cannot share the same physical volume. flag.

n Does not set a strict allocation policy, so copies for a logical partition can share the same physical volume.

s Sets a super strict allocation policy, so that the partitions allocated for one mirror cannot share a physical volume with the partitions from another mirror.

Note: When changing a nonsuper strict logical volume to a super strict logical volume, you must specify physical volumes or use the -u flag.

-u UpperBound

Sets the maximum number of physical volumes for new allocation. The value of the Upperbound variable should be between one and the total number of physical volumes. The default is the total number of physical volumes in the volume group. When using striped logical volumes or super strictness the upper bound indicates the maximum number of physical volumes allowed for each mirror copy.

Example

To add physical partitions to the logical partitions in the logical volume lv01, so that a total of three copies exists for each logical partition, enter:

mklvcopy lv01 3

The logical partitions in the logical volume represented by directory lv01 have three copies.

Files

/usr/sbin/mklvcopy

Contains the mklvcopy command.

Related Information

The chlv command, lslv command, mklv command, syncvg command, varyonvg command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mkmaster Command

Purpose

Executes the ypinit command and starts the NIS daemons to configure a master server.

Syntax

/usr/sbin/mkmaster [s HostName [,HostName ...]] [O | C] [E | B] [P | P] [U | U] [C | C] [I | B | N]
Description

The `mkmaster` command invokes the `ypinit` command to build the NIS maps for the current domain, if the domain name of the system is currently set. After the `ypinit` command completes successfully, the `mkmaster` command uncomments the entries in the `/etc/rc.nfs` file for the `ypserv` command, `yppasswd` command, `ypupdated` command, and `ypbind` command.

You can use the File Systems application in Web-based System Manager (wsm) to change file system characteristics. You could also use the System Management Interface Tool (SMIT) `smit mkmaster` fast path to run this command.

Flags

- `-s HostName [,HostName ...]` Specifies the slave host names for this master server. These slave hosts must be configured after the master server has been configured. The `mkmaster` command automatically adds the current host to this list.
- `-O` Overwrites existing maps for this domain.
- `-o` Prevents the overwriting of existing maps for this domain. This flag is the default.
- `-E` Prevents further action if errors are encountered while building new maps. This is true for both the `ypinit` command and the `mkmaster` command. This flag is the default.
- `-e` Does not exit from the `ypinit` command and the `mkmaster` command if errors are encountered.
- `-P` Starts the `yppasswd` daemon along with the `ypserv` daemon.
- `-p` Suppresses the start of the `yppasswd` daemon. This flag is the default.
- `-U` Starts the `ypupdated` daemon along with the `ypserv` daemon.
- `-u` Suppresses the start of the `ypupdated` daemon. This flag is the default.
- `-C` Starts the `ypbind` daemon along with the `ypserv` daemon. This flag is the default.
- `-c` Suppresses the start of the `ypbind` daemon.
- `-I` Directs the `mkmaster` command to change the `/etc/rc.nfs` file to start the appropriate daemons on the next system restart. The execution of the `ypinit` command occurs when this command is invoked.
- `-B` Executes the `ypinit` command, uncomments the entries in the `/etc/rc.nfs` file, and starts the daemons. This flag is the system default.
- `-N` Executes the `ypinit` command and starts the appropriate daemons without changing the `/etc/rc.nfs` file.

Example

To execute the `ypinit` command, overwrite any existing maps for the current domain, and make host1 and host3 slave servers, enter:

```
mkmaster -s host1,host3 -O -p -u -B
```

This command will not start the `yppasswd` daemon or the `ypupdated` daemon.

Files

- `/var/yp/domainname` directory Contains the NIS maps for the NIS domain.
- `/etc/rc.nfs` Contains the startup script for the NFS and NIS daemons.

Related Information

The `chmaster` command, `rmyp` command, `smit` command, `ypinit` command.

The `ypbind` daemon, `yppasswd` daemon, `ypserv` daemon, `ypupdated` daemon.
**System Management Interface Tool (SMIT): Overview** in *AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.*

**Network File System (NFS) Overview for System Management** in *AIX 5L Version 5.2 System Management Guide: Communications and Networks.*

For information on installing the Web-based System Manager, see *Chapter 2: Installing Web-based System Manager* in *AIX 5L Version 5.2 Web-based System Manager Administration Guide.*

**Network Information Services (NIS) Overview for System Management** in *AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide.*

**NIS Reference**

---

**mknamsv Command**

**Purpose**
Configures TCP/IP-based name service on a host for a client.

**Syntax**

```
mknamsv { -a "Attribute=Value ..." | -A FileName }
```

**Description**
The `mknamsv` high-level command configures a TCP/IP instance to use a name server. It calls the `namerslv` low-level command to configure the `resolv.conf` file appropriately.

You can use the Devices application in Web-based System Manager (wsm) to change device characteristics. You could also use the System Management Interface Tool (SMIT) `smit mknamerslv` fast path to run this command.

**Flags**

- `-A FileName`
  Specifies the name of the file containing `named` daemon initialization information.
- `-a "Attribute=Value ..."`
  Specifies a list of attributes with corresponding values to be used for updating the `named` server initialization files in the database. Attributes available are:
  
  **Domain**
  Domain name
  
  **NameServer**
  Internet address of name server in dotted decimal format

**Examples**

1. To configure the name server initialization files, enter the command in the following format:

   ```
   mknamsv -a "domain=austin.century.com nameserver=192.9.200.1"
   ```

   In this example the domain name and name server address are updated. The previous domain and name server are overwritten.

2. To configure name server initialization files according to information in another file, enter the command in the following format:

   ```
   mknamsv -A namsv.file
   ```
In this example, the file that contains the configuration information is `namsv.file`.

**Files**

`/etc/resolv.conf` Contains DOMAIN name server information for local resolver routines.

**Related Information**

The `namerslv` command.

Naming in *AIX 5L Version 5.2 System Management Guide: Communications and Networks*.

Configuring Name Servers in *AIX 5L Version 5.2 System Management Guide: Communications and Networks*.

For information on installing the Web-based System Manager, see *Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide*.

The SMIT Interface for TCP/IP in *AIX 5L Version 5.2 System Management Guide: Communications and Networks*.


---

**mknetid Command**

**Purpose**

Generates data for the `netid.byname` map for use by the Network Information Services (NIS).

**Syntax**

To Create an NIS Map:

```
```

**Description**

The `mknetid` command is used to produce the data for the `netid.byname` NIS map. It will parse the files specified on the command line and build the corresponding netid keys and values. Users will get the following entries:

```
unix.<uid>@<domainname> <uid>:<gid1>,<gid2>,...
```

Hosts will get the following entries:

```
unix.<hostname>@<domainname> 0:<hostname>
```

The domainname that is used is the same that is configured on the system at the time `mknetid` is run. The generated data is sent to `stdout`. Each line contains one entry, with the key and the data separated by a space.

**Flags**

- `q` Quiet mode - do not report any warnings about the data.
- `p PasswordFile` Specifies which passwd file to be used for reading the list of users.
- `g GroupFile` Specifies which groups file to be used for reading the list of group memberships.
- `h HostsFile` Specifies which hosts file to be used for reading the list of hostnames.
-m NetidFile Specifies a file from which to read any additional netid entries to be included.

Files

/var/yp/Makefile mkenetid is most commonly used when rebuilding the NIS databases using /var/yp/Makefile.
/etc/passwd Where PasswordFile resides.
/etc/groups Where GroupFile resides.
/etc/hosts Where HostsFile resides.
/etc/netid Where NetidFile resides.

Related Information

The ypinit command.

AIX 5L Version 5.2 Network Information Services (NIS and NIS+) Guide

mknfs Command

Purpose

Configures the system to run NFS.

Syntax

/usr/sbin/mknfs [-I] [-N] [-B]

Description

The mknfs command configures the system to run the Network File System (NFS) daemons. The mknfs command adds an entry to the inittab file so that the /etc/rc.nfs file is executed on system restart.

Flags

-B Adds an entry to the inittab file to execute the /etc/rc.nfs file on system restart. The mknfs command also executes the /etc/rc.nfs file immediately to start the NFS daemons. This flag is the default.
-I Adds an entry to the inittab file to execute the /etc/rc.nfs file on system restart.
-N Starts the /etc/rc.nfs file to start the NFS daemons immediately. When started this way, the daemons run until the next system restart.

Files

inittab Controls the initialization process of the system.
/etc/rc.nfs Contains the startup script for the NFS and NIS daemons.

Related Information

The chnfs command, mknfsexp command, mknfsmnt command, rmnfs command.

NFS Installation and Configuration in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

**mknfsexp Command**

**Purpose**
Exports a directory to NFS clients.

**Syntax**
```
/usr/sbin/mknfsexp -d Directory [ -t Exports_File ] [ -t [ { rw | ro | rm } HostName [ , HostName ... ] ] ]
[ -a UID ] [ -r HostName [ , HostName ... ] ] [ -c HostName [ , HostName ... ] ] [ -s | -n ] [ -i | -B | -N ] [ -P | -p ]
```

**Description**
The `mknfsexp` command takes the flags and parameters specified and constructs a line that is syntactically correct for the `/etc/exports` file. If this command is invoked with the `-B` flag, an entry will be added to the `/etc/exports` file and the `exportfs` command is invoked to export the directory specified. Alternatively, the `-I` flag adds an entry to the `exports` file and does not export the directory, or the `-N` flag does not add an entry to the `exports` file but does export the directory.

**Flags**
- **-d Directory**
  Specifies the directory that is to be exported or changed.
- **-f Exports_File**
  Specifies the full path name of the exports file to use if other than the `/etc/exports` file.
- **-t Type**
  Specifies whether the directory is read-write, read-only, or read-mostly. The possible values for the `Type` variable are:
  - `rw` Exports the read-write directory. This is the system default.
  - `ro` Exports the read-only directory.
  - `rm` Exports the read-mostly directory. If chosen, the `-h` flag must be used to specify the hosts that have read-write permission.
- **-h HostName [ , HostName ] ...**
  Specifies which hosts have read-write access to the directory. This option is valid only when the exported file is to be read-mostly.
- **-a UID**
  Uses the UID variable as the effective user ID only if a request comes from an unknown user. The default value of this option is `-2`.  
  **Note:** Root users (UID 0) are always considered unknown by the NFS server, unless they are included in the root option. Setting the value of UID to `-1` disables anonymous access.
- **-r HostName [ , HostName ] ...**
  Gives root users on the specified hosts access to the directory. The default is for no hosts to be granted root access.
- **-c HostName [ , HostName ] ...**
  Gives mount access to each of the clients listed. A client can either be a host or a netgroup. The default is to allow all hosts access.
- **-s**
  Requires clients to use a more secure protocol when accessing the directory.
- **-n**
  Does not require the client to use the more secure protocol. This flag is the default.
- **-B**
  Adds an entry to the `/etc/exports` file and the `exportfs` command is executed to export the directory. This flag is the default.
- **-I**
  Adds an entry to the `/etc/exports` file so that the next time the `exportfs` command is run during system restart, the directory will be exported.
- **-N**
  Does not add an entry to the `/etc/exports` file but the `exportfs` command is run with the correct parameters so that the directory is exported.
- **-P**
  Specifies that the exported directory is to be a public directory. This flag only applies to AIX 4.2.1 or later.
- **-p**
  Specifies that the exported directory is not a public directory. This flag only applies to AIX 4.2.1 or later.
**Examples**

1. To export a directory with read-only permission, enter:
   
   ```
   mknfsexp -d /usr -t ro
   ```
   
   In this example, the `mknfsexp` command exports the `/usr` directory with read-only permission.

2. To export a directory with read-mostly permission and a secure protocol to specific hosts, enter:
   
   ```
   mknfsexp -d /home/guest -t rm -h bighost,littlehost -s
   ```
   
   In this example the `mknfsexp` command exports the `/home/guest` directory with read-mostly permission, using more secure protocol.

3. To export a directory with read-write permission to a specific netgroup and specific hosts, and to make the export effective on the next system restart, enter:
   
   ```
   mknfsexp -d /user -t rw -c host1,host3,grp3 -I
   ```
   
   In the above example, the `mknfsexp` command exports the `/user` directory and gives read and write permission to `host1`, `host2`, and `grp3`. The `-I` flag makes this change effective on the next system restart.

4. To export a directory with read-only permission to an exports file other than `/etc/exports`, enter:
   
   ```
   mknfsexp -d /usr -t ro -f /etc/exports.other
   ```
   
   In the above example, the `mknfsexp` command exports the `/usr` directory with read-only permission to the `/etc/exports.other` file.

**Files**

| `/etc/exports` | Lists the directories that the server can export. |

**Related Information**

The `chnfsexp` command, `exports` command, `rmnfsexp` command.


[List of NFS Commands](#)

**mknfsmnt Command**

**Purpose**

Mounts a directory from an NFS server.

**Syntax**

```
/usr/sbin/mknfsmnt [-f PathName | d RemoteDirectory | h RemoteHost | t { rw | ro } ] [-m
MountTypeName [-w {bg | bg, ro}] [-x] [S-H] [v-v] [Z-Z] [-e | E] [a-A] [H] [J] [-G | g | G] [S-n] [-B N] [-m TimeOut] [-p PortNumber] [-u AcRegMin] [-U AcRegMax] [-v AcDirMin] [-V
AcDirMax] [-T AcTimeOut] [-p NumBiods] [-K { any | 2 | 3 }] [-k { any | tcp | udp }]
```

**Description**

The `mknfsmnt` command constructs an entry that will be appended to the `/etc/filesystems` file, thus making a file system available for mounting. If the mount is to be permanent, this entry will remain. If the
mount is temporary, the flags will be used directly for the mount command. If the mount is soft, the system returns an error if the server does not respond. If the mount is hard, the client continues trying until the server responds. The hard mount is the default.

**Flags**

- **-A** The /etc/filesystems entry for this file system will specify that it should be automatically mounted at system restart.

- **-a** The /etc/filesystems entry for this file system will specify that it should not be automatically mounted at system restart. This is the default.

- **-B** Adds an entry to the /etc/filesystems file and attempts to mount the file system. This flag is the default.

- **-b ReadBufferSize** Indicates the size of the read buffer in bytes specified by the ReadBufferSize variable.

- **-c WriteBufferSize** Indicates the size of the write buffer in bytes specified by the WriteBufferSize variable.

- **-d RemoteDirectory** Specifies the directory that is mounted on the path name specified.

- **-E** Allows keyboard interrupts on hard mounts.

- **-e** Prevents keyboard interrupts on hard mounts. This is the default.

- **-f PathName** Specifies the mount point for the remote directory.

- **-G** Directs any file or directory created on the file system to inherit the group ID of the parent directory.

- **-g** Does not direct new files or directories created on the file system to inherit the group ID of the parent directory. This is the default.

- **-H** Creates a hard mount, which causes the client to continue retrying until the server responds. This is the default.

- **-h RemoteHost** Specifies the NFS server that is exporting the directory.

- **-l** Causes an entry to be added to the /etc/filesystems file. The directory is not mounted.

- **-J** Indicates that acl is used on this mount.

- **-j** Indicates that acl is not used on this mount. This is the default.

- **-K** Specifies the NFS version used for this NFS mount. This flag only applies to AIX 4.2.1 or later. Options are:

  - any Uses the mount command to determine the correct match, first attempting the highest NFS version available.


  - 3 Specifies NFS Version 3.

- **-k** Specifies the transport protocol used for the mount. This flag only applies to AIX 4.2.1 or later. Options are:

  - any Uses the mount command to select the protocol to use. TCP protocol is the preferred protocol.

  - tcp Specifies the TCP protocol.

  - udp Specifies the UDP protocol.

- **-m MountTypeName** Specifies the type of file system to mount. File system types are specified in the /etc/filesystems file with the type variables. When the mount -t MountTypeName command is issued, all of the currently unmounted file systems with a type equal to the MountTypeName are mounted.

- **-N** Mounts the directory with the options specified but does not modify the /etc/filesystems file.

- **-n TimeOut** Instructs the mount not to use a more secure protocol. This flag is the default.

- **-o TimeOut** Indicates the length of the NFS timeout in tenths of a second as specified by the TimeOut variable.

- **-P PortNumber** Indicates the Internet Protocol port number for the server.
-p NumBiods  
Specifies the number of biod daemons that are allowed to work on a particular file system. The biod daemons handle client requests and the default number of daemons is 6 (six).

-Q  
Requests that no posix pathconf information be exchanged and made available on an NFS Version 2 mount. Requires a mount Version 2 rpc.mountd at the NFS server.

-q  
Specifies that no posix pathconf information is exchanged if mounted as an NFS Version 2 mount. This is the default.

-r TimesToRetry  
Indicates the number of times to retry a mount. The default is 1000.

-R NumRetrans  
Specifies, for a soft mount, the number of times that a request is to be transmitted if it is not acknowledged by the server. If the request goes unacknowledged after NumRetrans transmissions, the client gives up on the request. If this flag is not specified, the default value of 3 is used.

-S  
Creates a soft mount, which means the system returns an error if the server does not respond.

-s  
Instructs the mount to use a more secure protocol.

-T Actimeo  
Sets minimum and maximum times allowed for regular files and directories to the number of seconds specified by the Actimeo variable. If this flag is specified, the other cached attribute times are overridden.

-t Type  
Specifies that the directory is either read-write or read-only.

  rw  Mounts the directory read-write. This type is the default for the system.

  ro  Mounts the directory read-only.

-U AcRegMax  
Holds cached attributes for no more than the number of seconds specified by the AcRegMax variable after file modification.

-u AcRegMin  
Holds cached attributes for at least the number of seconds specified by the AcRegMin variable after file modification.

-V AcDirMax  
Holds cached attributes for no more than the number of seconds specified by the AcDirMax variable after directory update.

-v AcDirMin  
Holds cached attributes for at least the number of seconds specified by the AcDirMin variable after directory update.

-w Location  
Indicates where the mount should be attempted. The Location variable can have one of the following values:

  fg  Attempts the mount in the foreground. This is the default value.

  bg  Attempts the mount in the background. If background is specified and the attempt to mount the directory fails, the mount will be retried in the background.

-x  
Specifies that the server does not support long device numbers.

Use this flag when mounting from an NFS server that does not correctly handle device numbers that are 32 bits long.

-X  
Specifies that the server does support long device numbers. This is the default.

-y  
Indicates that the execution of suid and sgid programs is not allowed in this file system.

-Y  
Indicates that the execution of suid and sgid programs are allowed in this file system. This is the default.

-z  
Indicates that device access through this mount is not allowed; that is, the device cannot be opened on this mount point.

-Z  
Indicates that device access through this mount is allowed. This is the default.

Example
To add the mount of a remote directory, enter:

mknfsmnt -f /usr/share/man -d /usr/share/man -h host1

In this example, the mknfsmnt command mounts the remote directory /usr/share/man on the /usr/share/man directory that resides on host1.
**Files**

`/etc/filesystems` Lists the remote file systems to be mounted during the system restart.

**Related Information**

The `chnfsmnt` command, `mount` command, `rmnfsmnt` command.

How to Mount a NFS File System Explicitly in *AIX 5L Version 5.2 System Management Guide: Communications and Networks.*

Network File System (NFS) Overview for System Management in *AIX 5L Version 5.2 System Management Guide: Communications and Networks.*

List of NFS Commands

**mknod Command**

**Purpose**

Creates a special file.

**Syntax**

*Only executed by root or system group member*

```bash
mknod Name {b|c} Major Minor
```

*Creates FIFOs (named pipelines)*

```bash
mknod Name {p}
```

**Description**

The `mknod` command makes a directory entry and corresponding i-node for a special file. The first parameter is the name of the `Name` entry device. Select a name that is descriptive of the device. The `mknod` command has two forms that have different flags.

The first form of the `mknod` command can only be executed by root or a member of the system group. In the first form, the `b` or `c` flag is used. The `b` flag indicates the special file is a block-oriented device (disk, diskette, or tape). The `c` flag indicates the special file is a character-oriented device (other devices).

The last two parameters of the first form are numbers specifying the `Major` device, which helps the operating system find the device driver code, and the `Minor` device, that is the unit drive or line number, which may be either decimal or octal. The major and minor numbers for a device are assigned by the device’s configure method and are kept in the `CuDvDr` class in ODM. It is important that major and minor numbers be defined in this object class to insure consistency of device definitions through the system.

In the second form of the `mknod` command, the `p` flag is used to create FIFOs (named pipelines).

**Flags**

- `b` Indicates the special file is a block-oriented device (disk, diskette, or tape).
- `c` Indicates the special file is a character-oriented device (other devices).
- `p` Creates FIFOs (named pipelines).
Examples
To create the special file for a new diskette drive, enter:

mknod /dev/fd2 b 1 2

This creates the /dev/fd2 special file that is a special block file with the major device number 1 and the minor device number 2.

Files
/usr/sbin/mknod Contains the mknod command.

Related Information
The mknod subroutine.

mknotify Command

Purpose
Adds a notify method definition to the Notify object class.

Syntax
mknotify -n NotifyName -m NotifyMethod

Description
The mknotify command adds a notify method definition to the Notify object class. When a notify method is defined for both a subsystem name and a group name, the subsystem name takes precedence. For example, if the subsystem notify method is executed by the System Resources Controller (SRC), the group notify method is not performed.

The SRC places the name of the unsuccessful subsystem as the first argument to the method and the name of the unsuccessful subsystem group as the second argument.

Flags

-m NotifyMethod Specifies an absolute path to an executable program that starts when the subsystem stops abnormally.

-n NotifyName Specifies the subsystem or group name to which the notify method belongs. The NotifyName variable must exist as either a valid subsystem name or a valid group name in the Subsystem object class. The mknotify command is unsuccessful if the NotifyName variable already exists in the Notify object class.

Examples
1. To add a notify method for the srctest subsystem, enter:

    mknotify -n srctest -m /usr/lpp/srctest/failure

This adds a subsystem notify method for the srctest subsystem, with a notify method designated in the /usr/lpp/srctest/failure file.
2. To add a notify method for the tcpip group, enter:

```
mmnotify -n tcpip -m /usr/lpp/tcpip/tcpfailure
```

This adds a group notify method for the tcpip group, with a notify method designated in the /usr/lpp/tcpip/tcpfailure file.

**Files**

/etc/objrepos/SRCsubsys Specifies the SRC Subsystem Configuration object class.
/etc/objrepos/SRCnotify Specifies the SRC Notify Method object class.

**Related Information**

The rmnotify command, lssrc command, mkssys command, rmssys command.

System Resource Controller Overview in the AIX 5L Version 5.2 System Management Concepts: Operating System and Devices

System Resource Controller (SRC) Overview for Programmers in the in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Defining Your Subsystem to the SRC in the in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

**mkpasswd Command**

**Purpose**
Organizes the basic user database for efficient searches.

**Syntax**

```
mkpasswd [-v] [-f] [-d] [-c] indexname
```

**Description**

The mkpasswd generates indexes over certain security files. These indexes are used by the getpwnam, getpwuid, getuserattr, and putuserattr library subroutines.

This approach significantly enhances performance for large user base systems. The following indexes, defined in /usr/include/usersec.h, are created:

```
/etc/passwd.nm.idx: Index over /etc/passwd file using username as key.
/etc/passwd.id.idx: Index over /etc/passwd file using userid number as key.
/etc/security/passwd.idx: Index over /etc/security/passwd file.
/etc/security/lastlog.idx: Index over /etc/security/lastlog file.
```

**Notes:**
1. Modifying the security files over which indexes are built by an editor disables the use of indexing mechanism.
2. Indexed read of a data file is automatically done if a corresponding index exists over the file and is not older than it (except for lastlog index).
3. In order for indexed mechanism to be used at login, the mkpasswd command must have generated indexes.
4. The indexing mechanism replaces the previous hashing mechanism which used dbm files.

**Flags**

- **-v** Reports progress if index built.
- **-f** Forces building of all indexes.
- **-d** Deletes all indexes.
- **-c** Checks all indexes and rebuilds the ones that look suspicious.
- **indexname** Forces building of a particular index.

**Security**

Access Control: Only the root user and members of the security group should have execute (x) access to this command. The command should be setuid to the root user so the command has access to the user database. Members of the security group should have access to all the files listed in the **Files** section. This command should have the trusted computing base attribute.

**Files Accessed:**

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>/etc/passwd</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/passwd.nm.idx nnnn.tmp and /etc/passwd.id.idx nnnn.tmp</td>
</tr>
<tr>
<td></td>
<td>where nnnn is the process id.</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/passwd</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/passwd.idx</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/passwd.idx nnnn.tmp</td>
</tr>
<tr>
<td></td>
<td>where nnnn is the process id</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/lastlog</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/lastlog.idx</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/lastlog.idx nnnn.tmp</td>
</tr>
<tr>
<td></td>
<td>where nnnn is the process id</td>
</tr>
</tbody>
</table>

**Examples**

1. To create and enable indexed read of security files, enter:
   
   mkpasswd -f

2. To create and enable indexed read of only the /etc/security/passwd file, enter:
   
   mkpasswd /etc/security/passwd.idx

3. To check and rebuild outdated or bad indexes, enter:
   
   mkpasswd -c

**Files**

- **/usr/sbin/mkpasswd** Contains the mkpasswd command.
- **/etc/passwd** Contains basic user attributes.
- **/etc/security/passwd** Contains user password attributes.
- **/etc/security/lastlog** Contains lastlog related attributes.
Related Information
The `passwd` command, `pwdadm` command `mkuser` command `chuser` command `rmusers` command.
The `getpwnam` subroutine, `getpwuid` subroutine, `getuserattr` subroutine, `putuserattr` subroutine.
For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the AIX 5L Version 5.2 Security Guide

mkpath Command

Purpose
Adds to the system another path to an MPIO capable device.

Syntax
mkpath [-I Name] [-P Parent] [-W Connection] [-d]

mkpath --h

Description
The `mkpath` command defines, and possibly configures, one or more paths to the target device (-I Name). The paths are identified by a combination of the -I Name, -P Parent, and -W Connection flags. Both the target device and parent must be previously defined in the system to define a path. They both must be "AVAILABLE" to configure a path.

If the -d flag is specified, the `mkpath` command only defines the new path definition to the system. If the -d flag is not specified, the `mkpath` command attempts to define the path, if it does not already exist, before it attempts to configure the path. Configuring a path requires the path to already be defined and both the device and the parent device to already be configured.

The `mkpath` command displays a status message upon completion. It is possible for some paths to configure and others to fail.

Note that any device that cannot be manually defined using the `mkdev` command will not be able to have paths manually defined to using the `mkpath` command. These limitations are due to the way that path information is stored for these devices. Fiber channel devices fall into this category.

The `mkpath` command provides status messages about the results of operation. Messages in one of the following formats will be generated:

`path [ available | defined ]`
This message is displayed when `mkpath` is run on a single path. If the path is successfully configured the message "path available" is displayed. If the path is not successfully configured and there is no explicit error code returned by the method, the message "path defined" is displayed.

`paths available`
This message is displayed if multiple paths were identified and all paths were successfully configured.

`somes paths available`
This message is displayed if multiple paths were identified, but only some of them were successfully configured.

`no paths processed`
This message is generated if no paths were found matching the selection criteria.
Flags

- **d**
  Defines a new path to a device by adding a path definition to the system. The new path will not automatically be configured when the -d flag is specified. Note that only one path may be defined at a time.

- **h**
  Displays the command usage message.

- **l Name**
  Specifies the logical device name of the target device to which the path(s) are being added. The path(s) to be added are qualified by the -p and -w flags.

- **p Parent**
  Indicates the logical device name of the parent device associated with the path(s) to be added. This flag is required if the -d flag is specified.

- **w Connection**
  Indicates the connection information associated with the path to be added. This flag is required if the -d flag is specified.

Security

*Privilege Control:* Only the root user and members of the system group have execute access to this command.

*Auditing Events:*

<table>
<thead>
<tr>
<th>Event</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>DEV_Change</td>
<td>mkpath,Define,&lt;define method arguments&gt;</td>
</tr>
<tr>
<td>DEV_Change</td>
<td>mkpath,Configure,&lt;configure method arguments&gt;</td>
</tr>
</tbody>
</table>

Examples

1. To define and configure an already defined path between scsi0 and the hdisk1 device at SCSI ID 5 and LUN 0 (i.e., connection 5,0), enter:
   
   ```
   mkpath -l hdisk1 -p scsi0 -w 5,0
   ```
   
   The system displays a message similar to the following:
   
   path available

2. To configure an already defined path from 'fscsi0' to fiber channel disk 'hdisk1', the command would be:
   
   ```
   mkpath -l hdisk1 -p fscsi0
   ```
   
   The message would look similar to:
   
   path available

3. To only add to the Customized Paths object class a path definition between scsi0 and the hdisk1 disk device at SCSI ID 5 and LUN 0, enter:
   
   ```
   mkpath -d -l hdisk1 -p scsi0 -w 5,0
   ```
   
   The system displays a message similar to the following:
   
   path defined

Files

`/usr/sbin/mkpath` Contains the `mkpath` command.

Related Information

The `chpath` command, `lspath` command, `rmpath` command.
**mkproto Command**

**Purpose**
Constructs a prototype file system.

**Syntax**

```
mkproto Special Prototype
```

**Description**
The `mkproto` command is used to construct a prototype for a new file system. It exists solely for Berkeley Software Distribution (BSD) compatibility.

The `Special` parameter can be a block device name, raw device name, or file system name. The `Prototype` parameter is the name of the prototype file that specifies the structure and contents of the file system to be created. The `mkproto` command calls the `mkfs` command with the `Prototype` and `Special` parameters.

**Prototype Files**
The `mkproto` and `mkfs` commands require an extended prototype file to create a Journaled File System (JFS). A prototype file is a formatted listing of the contents and structure of a file system. A prototype file describes the file system by a series of tokens separated by spaces and new lines. The main body of a prototype file defines the objects of the file system.

A JFS prototype file consists of the main body, which can be created by the `proto` command, preceded by five special tokens. These five tokens are defined as follows:

1st token  Name of a file to be copied onto block 0 as the bootstrap program or the special token `<noboot>`.
2nd token  Size of the file system. For a JFS, the size is expressed in units of 512-byte blocks. If the 2nd token is 0, the `mkfs` command creates the file system to fill the entire logical volume.
3rd token  Number of i-nodes on the file system. This token is not used by a JFS but must be provided to preserve the position.
4th token  Size of the file system fragment in bytes. If the 4th token is 0 (zero), the `mkfs` command uses the default fragment size. For JFS, the token must be either 0 (default value used), 512, 1024, 2048, or 4096. The default fragment size is 4096 for a JFS. An invalid fragment size causes the `mkfs` command to fail.
5th token  Number of bytes per i-node (nbpi). If this token is 0, the `mkfs` command uses the default nbpi. For a JFS, this token must be either 0 (default value used), 512, 1024, 2048, 4096, 8192, or 16384. The default number of bytes per i-node is 4096 for a JFS. An invalid nbpi causes the `mkfs` command to fail.

The remaining tokens define the contents and structure of the file system. These tokens are grouped into sets, with each set defining one object of the file system. The syntax of each set is as follows:

```
{ ( [ Name ] [ -d | b | c | l | L | p ] [ -u ] [ -g ] [ -t ] ) Mode Owner Group [ ( [ Major Minor ] [ SourceFile ] DirectoryListing ] ) } { S }
```

where:

**Name**  Specifies the name of the object as it is to appear in the new file system. The `Name` token is required for every object except for the root directory definition.
Represents a string of 4 positional characters, where:

{-ldbcllllp}

Defines the object type. Valid types are:

- Regular file
  d Directory
  b Block special file
  c Character special file
  l Symbolic link
  L Hard link
  p Named pipe

{-lu}

Toggles the set UID bit of the object, as follows:

u Set UID on execution
- Do not set UID on execution

{-lg}

Toggles the set group ID (GID) bit of the object, as follows:

g Set GID on execution
- Do not set GID on execution

{-lt}

Toggles the sticky bit of the object, as follows:

t Sticky bit on
- Sticky bit off

This 4-character token is required for every object.

Mode

Represents a string of 3 octal characters defining the read, write, and execute permissions of the object. The Mode token is required of every object. See the `chmod` command for more information about permissions.

Owner

Specifies the UID of the owner of the object. The owner token is required of every object.

Group

Specifies the GID of the owner of the object. The group token is required of every object.

Major Minor

Specifies the major and minor device numbers of the object if its type is a block or character special file. If the object is not a block or character special file, these tokens are omitted.

SourceFile

Applies only to regular file, hard link, and symbolic link objects. For regular files, this token is the path name to the file from which the object file is to be initialized. For both symbolic and hard links, this token is the source of the link. The source of the link is relative to the new file system for hard links.

DirectoryListing

Defines the contents of the object if it is a directory. The contents of the directory are defined using the token syntax described here. For example, a directory listing can include one or more regular files, one or more block files, and one or more directory listings. The `mkfs` command creates the directory entries . (dot) and .. (dot dot). Each directory listing is terminated with the special $ token.

$ Ends the current directory listing or indicates the end of the prototype file.
Example Prototype Specification

The following prototype specification describes a JFS that does not have a boot program in block 0 and occupies the entire device. The 3rd token is ignored. The 4th and 5th tokens define the fragment size as 1024 bytes and the number of bytes per i-node as 2048. The main body of this prototype defines the file system contents.

```
<noboot> 0 0 1024 2048
d--- 755 0 0
dir1 d--- 755 0 2
  block_dev b--- 644 0 0 880 881
  char_dev c--- 644 0 0 990 991
  named_pipe p--- 644 0 0
regfile3 ---- 644 0 0 /tmp/proto.examp/dir1/regfile3
regfile4 ---- 644 0 0 /tmp/proto.examp/dir1/regfile4
$ dir2 d--- 755 205 300
  regfile6 ---- 644 0 0 /tmp/proto.examp/dir2/regfile6
  symlinkInFs l--- 644 0 0 /tmp/proto.examp/dir2/regfile6
  symlinkNoExist l--- 644 0 0 /home/foobar
  symlinkInFs l--- 644 0 0 /dir2/regfile6
regfile5 ---- 644 0 0 /tmp/proto.examp/dir2/regfile5
hardlink L--- 644 0 0 /dir2/regfile5
$ dir3 d--- 755 0 0
  setgid --g- 755 0 0 /tmp/proto.examp/dir3/setgid
  setuid --u- 755 0 0 /tmp/proto.examp/dir3/setuid
 sticky ---- t 755 0 0 /tmp/proto.examp/dir3/sticky
$ dir4 d--- 755 0 0
  dir5 d--- 755 0 0
  dir6 d--- 755 0 0
$ dir7 d--- 755 0 0
$ regfile7 ---- 644 0 0 /tmp/proto.examp/dir4/regfile7
$ regfile1 ---- 555 205 1 /tmp/proto.examp/regfile1
regfile2 ---- 744 0 0 /tmp/proto.examp/regfile2
$  
```

Three entries for the dir2 object deserve further examination:

```
symlinkOutoffS l--- 644 0 0
/tmp/proto.examp/dir2/regfile6

This entry defines a symbolic link to a file outside the file system to be created. The command ls -l lists something similar to symlinkOutoffS -> /tmp/proto.examp/dir2/regfile6.

symlinkNoExist l--- 644 0 0 /home/foobar

This entry defines a symbolic link to a file outside the file system to be created to a file that does not exist. The command ls -l lists something similar to symlinkNoExist -> /home/foobar.

symlinkInFs l--- 644 0 0 /dir2/regfile6

This entry defines a symbolic link to a file within the file system to be created. The command ls -l lists something similar to symlinkInFs -> /dir/regfile6.
```

Examples

To make a prototype JFS using the prototype file described in the "Example Prototype File Specification":

1. Generate the main body of the prototype file using the proto command or a text editor. For the purposes of this example, call the file /tmp/ProtoFile.

2. Add the first 5 tokens as required for a JFS. In the example prototype file, the tokens are:
3. Create a logical volume to hold the file system, as follows:

```
  mklv -y protolv -t jfs SomeVGname 5
```

This command creates a logical volume named protolv in the SomeVGname volume group. The size of the logical volume is 5 logical partitions.

4. Add an appropriate stanza to the `/etc/filesystem` file. A minimal example stanza is:

```
/protofs:
  dev = /dev/protolv
  vfs = jfs
  log = /dev/loglv00
  mount = false
```

5. Run the following `mkproto` command:

```
  mkproto /dev/protolv /tmp/ProtoFile
```

This command creates a JFS on the protolv logical volume. The size of the JFS is 5 logical partitions, its fragment size is 1024 bytes, and its nbpi ratio is 2048. The structure and contents of the file system are as specified in the prototype file `/tmp/ProtoFile`.

### Files

- `/usr/sbin/mkproto` Contains the `mkproto` command.

### Related Information

- The `mkfs` command, `fsck` command, `fsdb` command, `proto` command.
- The `filsys.h` file, `dir` file.


### mkprtldap Command

#### Purpose

Configures IBM Directory (LDAP) for Directory enabled System V print. It also configures client machines to use the Directory for System V print information.

#### Syntax

To configure the IBM Directory to store System V Print information:

```
```

To configure clients to use the IBM Directory for System V Print information:

```
```

To get usage information for the `mkprtldap` command:

```
  mkprtldap [-U]
```
Description

The mkprtldap command configures the IBM Directory (LDAP) server, and one or more clients to use the Directory (LDAP) for System V Print information. This command must be run on the system being setup as the server and on all the client systems. Once the Directory (LDAP) server is configured for System V print, the directory enabled System V Print commands (dslpadmin, dslpaccess, dslpsearch, dslpenable, dslpdisable, dslpaccept, ip, lpstat, cancel and dslpreject) must be run to add, remove and manage System V print information (printers and print queues) on the Directory (LDAP) server. The mkprtldap command configures client machines to use the Directory (LDAP) server for System V print information.

The mkprtldap command requires the IBM Directory server software to be installed on the machine being configured as the server. The command also requires the IBM Directory client software to be installed on all client machines that will use the Directory (LDAP) server for System V print information.

Note: The client (-c flag) and the server (-s server) options cannot be run at the same time. When setting up a system as the server, the mkprtldap command should be run twice on that system. Once to set up the server, and again to set up the client.

During the server side configuration, using the -s flag, the mkprtldap command:

- Requires the IBM Directory Administrator’s DN and password if the Directory has been configured. If the Directory Administrator’s DN and password have not been set, mkprtldap will set them with the values passed to the command.
- Creates a db2 instance with ldapdb2 as the default instance name.
- Creates a db2 database with ldapdb2 as the default database name if one does not exist. If an existing database is found, mkprtldap adds AIX System V print information to the existing database.
- Creates the AIX Information tree DN (cn=aixdata container object) on the Directory if one is not present. The print subtree will be created under the AIX Information subtree. If an existing AIX Information subtree exists on the Directory, the print subtree will be created under it. All System V print information will be stored under the print subtree. The directory enabled System V print commands have to be run to add printers and print queues under the print subtree created.
- The default suffix and AIX Information tree for the mkprtldap command is a top level container object cn=aixdata. The Print subtree (ou=print) will be created under the AIX Information tree.
- The print subtree is ACL protected with the value of the ACLBindPasswd parameter passed to the command. The same value must be used when configuring clients to use the Directory for System V print information. Select a password value that is difficult for people or password cracking programs to guess.
- If the -d option is used and a valid existing node on the Directory is passed to the command, the AIX Information subtree is created under the given node. The print subtree is then created under the AIX Information subtree.
- Starts the IBM Directory server after all the above is done
- Adds the IBM Directory server process (slapd) to the /etc/inittab file to have the server start after a reboot.

During the client configuration, the mkprtldap command:

- Saves the IBM Directory (LDAP) server host name in the /etc/ldapsvc/server.print file.
- Saves the AIX Print Bind DN in the /etc/ldapsvc/server.print file.
- Saves the ACL Bind Password for the AIX Print Bind DN in the /etc/ldapsvc/system.print file. The value of the ACL Bind password must be the same as the one specified during the configuration of the Directory server.
- Undo a previous client configuration if the -U flag is specified. This option will replace the /etc/ldapsvc/system.print and /etc/ldapsvc/server.print files with the previous saved copies of the files (/etc/ldapsvc/server.print.save and /etc/ldapsvc/system.print.save).
**Flags**

**Server**

- `-a AdminDN` Specifies the Directory (LDAP) Administrator's DN.
- `-d nodeDN` This advanced option requires a valid existing node DN on the Directory under which the AIX Information tree and Print Subtree will be created.
- `-f` The force flag is required by the `mkprtldap` command to force the creation of the Print subtree (and AIX Information subtree if needed) when one or more AIX Information trees exist on the Directory.
- `-p adminpasswd` Specifies the Directory (LDAP) Administrator's password.
- `-s` Indicates the command is being run to configure the Directory for System V print.
- `-w ACLBindPasswd` Specifies the password to ACL protect the Print Subtree on the Directory. Select a password value that is difficult for people or password cracking programs to guess.

**Client**

- `-c` Indicates the command is being run to configure clients to use the Directory for System V Print information.
- `-d PrintBindDN` Specifies the Print Bind DN. The default Print Bind DN is `ou=print,cn=aixdata`. The Print Bind DN to use during Client configuration is displayed at the end of the server setup of the `mkprtldap` command.
- `-h DirectoryServerHostname` Hostname of the IBM Directory server setup to store System V Print information.
- `-U` Undo a previous configuration of a client.
- `-w ACLBindPasswd` The ACL Bind Password for the print subtree. The ACL Bind password is specified during the server setup of the `mkprtldap` command. The value of the ACL Bind Password must match the one used during the setup of the Directory server.

**Usage**

`?` Displays usage information for the `mkprtldap` command.

**Security**

This command can be run by the `root` user only.

**Examples**

1. To configure a new installation of IBM Directory for System V print with the Administrator DN `cn=root` and password root, type:
   ```
   mkprtldap -s -a cn=root -p root -w aclpasswd
   ```
   where the ACL Bind password is the password used to ACL protect the print subtree. The ACL Bind password is specified during the configuration of System V Print on the Directory. This configuration will also set the Directory Administrator's DN and password to `cn=root` and `root`. Running the command will setup a suffix and top level object `cn=aixdata`. The Print subtree (`ou=print`) will be created under this AIX Information tree (`cn=aixdata object`). Select a ACL Bind password value that is difficult for people or password cracking programs to guess.

2. To configure System V print on a machine with a configured IBM Directory server -
   ```
   mkprtldap -s -a cn=admin -p passwd -w pass123wd
   ```
   The Administrator DN and password are required to configure System V print on the Directory. Assume the existing Administrator's DN and password are `cn=admin` and `passwd`. 
3. The `mkprtldap` command provides the option to configure the IBM Directory to store the print information under a pre-existing node (e.g. `o=ibm,c=us`) on the Directory [Advanced Option]. This is only recommended when it is necessary to store the print information under the existing node on the Directory for specific reasons. The recommend option is to store the print subtree in the default location on the Directory by not specifying the `-d` option. The Administrator DN and password are required to configure System V print on the Directory. Assume the existing Administrator’s DN and password are `cn=admin` and `passwd`.

```
> mkprtldap -a cn=admin -p passwd -w acl123passwd -d o=ibm, c=us
```

Running the command will create an AIX Information tree (`cn=aixdata`) under the `o=ibm, c=us` object. The print subtree will be created under this new object (`cn=aixdata`, `o=ibm, c=us`).

4. To configure System V print on a machine with a configured IBM Directory server and an existing AIX Information tree. There might be situations where the Directory contains an existing AIX information tree with other subsystem specific information (e.g Security or NIS information). It might be required to store the print information in a separate location on the Directory under a different AIX Information tree. The command, by default, will not create a new AIX Information tree if one exists on the Directory. To force the command to create a new AIX Information tree to store the print information, use the `-f` flag with the command. Consider the case where the Security and NIS subsystem information is stored under the AIX Information tree at `cn=aixdata,o=ibm, c=us`. To create a new AIX Information tree for print information different from the existing one, run the command with the `-f` flag and specify the default location or another node. The Administrator DN and password are required to configure System V print on the Directory. Assume the existing Administrator’s DN and password are `cn=admin` and `passwd`.

```
> mkprtldap -a cn=admin -p passwd -w passwd123 -f
```

Running the command will create a new AIX Information tree (`cn=aixdata`) with the suffix (`cn=aixdata`) and the print information will be stored under this new AIX Information tree (`ou=print, cn=aixdata`). There will be two AIX Information trees on the Directory in this example `cn=aixdata,o=ibm, c=us` and `cn=aixdata`. The print information will be under the `cn=aixdata` object (suffix `- cn=aixdata`). For `mkprtldap`, it is recommend to use the default location to add the print information to the Directory.

5. To configure a client to use an IBM Directory setup for System V Print on host `server.ibm.com`, type:

```
> mkprtldap -c -h server.ibm.com -w passwd
```

Please ensure that the ACL Bind Password (`passwd`) is the same as the one specified during the setup of the Directory Server. Running the command without specifying a Print Bind DN value with the `-d` option will cause the command to use the default Print Bind DN `ou=print,cn=aixdata`. The Print Bind DN must match the one displayed at the end of running the `mkprtldap` command to configure the server.

6. To change the information in the client side configuration files, run the `mkprtldap` command with the new information.

```
> mkprtldap -c -h server.ibm.co.uk -w aclpasswd -d ou=print,cn=aixdata,c=uk
```

Executing this command on a client that has already been configured will change the information in the `/etc/ldapsvc/server.print` and `/etc/ldapsvc/system.print` files to contain the new configuration information. The original contents of the `/etc/ldapsvc/server.print` and `/etc/ldapsvc/system.print` will stored in the `/etc/ldapsvc/server.print.save` and `/etc/ldapsvc/system.print.save` files.

### Files

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>rw</td>
<td>/etc/slapd32.conf</td>
<td>(Server configuration) - Contains the IBM Directory (LDAP) configuration information.</td>
</tr>
</tbody>
</table>
### Mode  File  Description
| rw    | /etc/ldapsvc/server.print | (Client configuration) - Contains information about the Directory Server configured to store System V Print information. (Machine name, Location of Print subtree on the Directory and LDAP port) |
| rw    | /etc/ldapsvc/system.print | (Client configuration) - Contains the ACL Bind Password for the Print subtree on the Directory. |

## Related Information


The `dslpadmin` command, `dslpaccess` command, `dslpsearch` command, `dslpenable` command, `dslpdisable` command, `dslpaccept` command, `lp` command, `lpstat` command, `cancel` command, `dslpreject` command, and `mksecldap` command.  

The `/etc/ldapsvc/server.print` and `/etc/ldapsvc/system.print` files.  

The `/etc/slapd32.conf` file.

## mkprtsv Command

### Purpose

Configures TCP/IP-based print service on a host.

### Syntax

**To Configure and Start Print Service for a Client Machine**

```
mkprtsv -c [-S] [-q QueueName] [-v DeviceName] -b "Attribute=Value ..." [-a "Attribute=Value ..."] [-A FileName]
```

**To Configure and Start Print Service for a Server Machine**

```
mkprtsv -s [-S] [-q QueueName] [-v DeviceName] -b "Attribute=Value ..." [-a "Attribute=Value ..."] [-A FileName] [-H HostName ...] [-H FileName]
```

### Description

The `mkprtsv` high-level command configures a TCP/IP-based print service on a host. The print service configuration can be done for a host functioning as a client or for a host functioning as a server.

Use the command to configure and start the print service.

To configure print service for a client, the `mkprtsv` command calls the spooler `mkque` and `mkquedev` commands to change the `/etc/lpd/qconfig` file (or its object class equivalent) appropriately and set up a spooler queue on the client machine.

To configure print service for a server, the `mkprtsv` command does the following:

1. Calls the `ruser` command to set up remote users to print on the server.
2. Calls the `mkque` and `mkquedev` commands to change the server’s `/etc/lpd/qconfig` file appropriately and set up the necessary device queues on the server machine.
3. Calls the `startsrc` command to activate the `lpd` and `qdaemon` server daemons. The `qdaemon` server daemon starts the `piobe` printer backend.
Flags

- A FileName
- a "Attribute=Value..."

Specifies name of file containing entries related to the qconfig file.

Specifies a list of attributes and their corresponding values to be used for updating the spooler’s qconfig file or object class. The -a flag is optional. Valid attribute types are listed below:

acctfile (true/false)
  Identifies the file used to save print command accounting information. The default value of false suppresses accounting. If the named file does not exist, no accounting is done.

argname
  Specifies the logical printer name.

device
  Identifies the symbolic name that refers to the device stanza.

discipline
  Defines the queue-serving algorithm. The default value of fcfs means first come, first served. A sjn value means shortest job next.

pserver
  Specifies the remote print server.

up (true/false)
  Defines the state of the queue. The default value of true indicates that it is running. A false value indicates that it is not.
-b "Attribute = Value..." Specifies a list of attributes and their corresponding values to be used for updating the spooler's qconfig file or object class. At least one attribute must be defined for the -b option. The backend attribute is required. Valid attribute types are listed below:

access (true/false) Specifies the type of access the backend has to the file specified by the file attribute. The access attribute has a value of write if the backend has write access to the file, or a value of both if the backend has both read and write access. This field is ignored if the file field has a value of false.

align (true/false) Specifies whether the backend sends a form-feed control before starting the job if the printer has been idle. The default value is false.

backend Specifies the full path name of the backend, optionally followed by flags and parameters to be passed to it. The backend attribute is required.

feed Specifies the number of separator pages to print when the device becomes idle, or takes a never value, which indicates that the backend is not to print separator pages.

file Identifies the special file where the output of the backend is to be redirected. The default value of false indicates no redirection. In this case, the backend opens the output file.

header (never/always/group) Specifies whether a header page prints before each job or group of jobs. The default value of never indicates no header page. To produce a header page before each job, specify an always value. To produce a header before each group of jobs for the same user, specify a group value.

trailer (never/always/group) Specifies whether a trailer page prints after each job or group of jobs. The default value of never indicates no trailer page. To produce a trailer page after each job, specify an always value. To produce a trailer after each group of jobs for the same user, specify a group value.

host Specifies the host name from which to print.

s_statfilter Translates short queue-status information to a format recognized by this operating system.

l_statfilter Translates long queue-status information to a format recognized by this operating system.

-c Performs print service configuration for a client machine. Use the -q flag with the -c option.

-H FileName Specifies the name of a file containing a list of host names.

-h "HostName..." Specifies a list of host names to be included in the list of remote users who can use the print server. The queuing system does not support multibyte host names.

-q QueueName Specifies the name of a queue in the qconfig file.

-S Starts print service after it is configured. If the -S flag is omitted, print service is configured but not started.

-s Performs print service configuration for a server machine. Use the -h -H and -q flags with the -s flag.

-v DeviceName Specifies the name of the device stanza in the qconfig file.

Examples
1. To configure and enable print service for a client, enter the command in the following format:
mkprtsv -c -S -a "argname=rp1 backend=piobe \ pservver=print802"

In this example, rp1 is the logical printer name, piobe is the printer backend, and print802 is the remote print server.

2. To configure a print server using initialization information and allow remote printing, enter the command in the following format:

    mkprtsv -s -H hnames -A qinfo

In this example, attribute information stored in the qinfo file initializes the spooler, and the list of host names stored in the hnames file is the list of remote hosts that have access rights to the print server.

Files

/etc/lpd/qconfig Contains configuration information for the printer queuing system.

Related Information

The mkque, mkquedev, qadm, ruser, startsrc commands.

The lpd and qdaemon daemons.

TCP/IP Daemons in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

---

**mkps Command**

**Purpose**

Adds an additional paging space.

**Syntax**

**To Add a Logical Volume for Additional Paging Space**

    mkps [-a] [-n] [-lv] [-s LogicalPartitions VolumeGroup] [ PhysicalVolume ]

**To Add Additional Paging Space On an NFS Server**

    mkps [-a] [-n] nfs ServerHostName ServerFileName

**Description**

The mkps command adds an additional paging space. Before the paging space can be used it must be activated, using the swapon command. The VolumeGroup parameter specifies the volume group within which the logical volume for the paging space is to be made. The PhysicalVolume parameter specifies the physical volume of the VolumeGroup on which the logical volume is to be made.

In the second form of the mkps command, the ServerHostName parameter specifies the NFS server where the ServerFileName resides. The ServerFileName specifies the file which will be used for the NFS paging of the system. The ServerFileName file must exist and be exported correctly to the client that will use the file for paging.

You can use the File Systems application in Web-based System Manager (wsm) to change file system characteristics. You could also use the System Management Interface
Flags

-a Specifies that the paging space is configured at subsequent restarts.
-n Activates the paging space immediately.
-s LogicalPartitions Specifies the size of the paging space and the logical volume to be made in logical partitions.
-t Specifies the type of paging space to be created. One of the following variables is required:
   lv Specifies that a paging space of type logical volume should be created on the system.
   nfs Specifies that a paging space of type NFS should be created on the system.

Examples

1. To create a paging space in volume group myvg that has four logical partitions and is activated immediately and at all subsequent system restarts, enter:

   mkps -a -n -s 4 myvg

2. To create an NFS paging space on the NFS server swapserve where the host swapserve has the /export/swap/swapclient file exported, enter:

   mkps -t nfs swapserve /export/swap/swapclient

Files

/etc/swapspaces Specifies the paging space devices that are activated by the swapon -a command.

Related Information

The mkqos command, lscps command, lsvps command, mklv command, swapon command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mkqos Command

Purpose

Configures the system to support QoS.

Syntax

/usr/sbin/mkqos [-I] [-N] [-B]
Description
The **mkqos** command configures the system to support Quality of Service (QoS).

Flags

- **-B**
  Adds an entry to the *inittab* file to execute the `/etc/rc.qos` file now and on the next system restart. This flag is the default.

- **-I**
  Adds an entry to the *inittab* file to execute the `/etc/rc.qos` file on the next system restart.

- **-N**
  Executes the `/etc/rc.qos` file to start the QoS daemons. When invoked in this way, the QoS daemons run until the next system restart.

Files

- **inittab**
  Controls the initialization process of the system.

- **/etc/rc.qos**
  Contains the startup script for the QoS daemons.

Related Information

The [rmqos](#) command.

[QoS Installation] and [TCP/IP Quality of Service (QoS)] in the *AIX 5L Version 5.2 System Management Guide: Communications and Networks.*

**mkque Command**

**Purpose**

Adds a printer queue to the system.

**Syntax**

```
mkque [-D] [-q] Name [-a 'Attribute = Value' ...]
```

**Description**

The **mkque** command adds a printer queue to the system by adding the stanza described on the command line to the end of the `/etc/qconfig` file.

You can use the Printer Queues application in Web-based System Manager (wsm) to change printer characteristics. You could also use the System Management Interface Tool (SMIT) **smit mkque** fast path to run this command.

To use the SMIT fast path to go directly to the **Add a Local Queue** dialog, enter:

```
smit mkque
```

To use the SMIT fast path to go directly to the **Add a Remote Queue** dialog, enter:

```
smit mkrque
```

**Note:** Do not edit the `/etc/qconfig` file while there are active jobs in any queue. Editing includes both manual editing and use of the **chque**, **mkque**, **rmque**, **mkquedev**, **rmquedev**, or **chquedev** commands. It is recommended that all changes to the `/etc/qconfig` file be made using these commands. However, if manual editing is desired, first issue the **enq -G** command to bring the
queuing system and the qdaemon daemon to a halt after all jobs are processed. Then edit the /etc/qconfig file and restart the qdaemon daemon with the new configuration.

Flags

-a 'Attribute = Value' Specifies a line to be added to the queue stanza in the /etc/qconfig file. This flag must be the last flag when entering the mkque command on the command line. For a list of all valid attributes, see the /etc/qconfig file.

Note: It is recommended that you do not use the 'device =' attribute. This attribute is handled automatically by the mkquedev command. Also note that the queuing system does not support multibyte host names.

-D Specifies that the queue defined by the Name variable queue is added to the top of the /etc/qconfig file and is therefore the default queue. If you do not specify this flag, the Name variable is added to the bottom of the /etc/qconfig file and is not the default queue.

-q Name Specifies the name of the queue to be added.

Note: The queue name must not exceed 20 characters.

Examples

To add the print queue lp0 specifying a host name of leo and a remote print queue named lp013, enter:

mkque -q lp0 -a 'host = leo' -a 'rq = lp013'

Files

/usr/bin/mkque Contains the mkque command.
/etc/qconfig Configuration file.

Related Information

The chque command, lsque command, mkquedev command, rmque command.

The qconfig file.

Printer Overview for System Management in the AIX 5L Version 5.2 Guide to Printers and Printing.

Spooler Overview for System Management in the AIX 5L Version 5.2 Guide to Printers and Printing.

Printer Specific Information in the AIX 5L Version 5.2 Guide to Printers and Printing.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mkquedev Command

Purpose

Adds a printer queue device to the system.
Syntax

\texttt{mkquedev [-d Name] [-q Name] -a 'Attribute = Value' ...}

Description

The \texttt{mkquedev} command adds a printer queue device to the system by adding the stanza described on the command line to the \texttt{/etc/qconfig} file.

You can use the Printer Queues application in Web-based System Manager (wsm) to change printer characteristics. You could also use the System Management Interface Tool (SMIT) \texttt{smit mkquedev} fast path to run this command.

\textbf{Note:} Do not edit the \texttt{/etc/qconfig} file while there are active jobs in any queue. Editing includes both manual editing and use of the \texttt{chque}, \texttt{mkque}, \texttt{rmque}, \texttt{mkquedev}, \texttt{rmquedev}, or \texttt{chquedev} commands. It is recommended that all changes to the \texttt{/etc/qconfig} file be made using these commands. However, if manual editing is desired, first issue the \texttt{enq -G} command to bring the queuing system and the qdaemon to a halt after all jobs are processed. Then edit the \texttt{/etc/qconfig} file and restart the qdaemon with the new configuration.

Flags

\textbf{-a 'Attribute = Value'}

Specifies the 'Attribute = Value' attribute to be added to the device stanza in the \texttt{/etc/qconfig} file. This flag must be the last flag when entering the \texttt{mkquedev} command on the command line. For a list of valid attributes, see the \texttt{/etc/qconfig} file.

\textbf{Note:} The 'backend = ' attribute must be included when entering this command on the command line.

\textbf{-d Name}

Specifies with the \texttt{Name} variable the name of the queue device to add.

\textbf{Note:} The queue device name must not exceed 20 characters.

\textbf{-q Name}

Specifies with the \texttt{Name} variable the name of the queue (this name must already exist) to which the queue device is added. The \texttt{mkquedev} command automatically adds the 'device = ' attribute to the specified queue stanza.

Examples

To add the postscript print queue device to the \texttt{lp0} queue, specify the backend program to be the \texttt{piobe} command (backend = \texttt{/usr/lib/lpd/piobe}) and direct the backend program not to align the paper (align = FALSE), enter:

\begin{verbatim}
mkquedev -q lp0 -d postscript -a 'backend = /usr/lib/lpd/piobe' \ -a 'align = FALSE'
\end{verbatim}

Files

\texttt{/usr/bin/mkquedev} Contains the \texttt{mkquedev} command.
\texttt{/etc/qconfig} Contains the configuration file.

Related Information

The \texttt{chquedev} command, \texttt{lsquedev} command, \texttt{mkque} command, \texttt{rmquedev} command.

The \texttt{/etc/qconfig} file.
mkramdisk Command

Purpose
Creates a RAM disk using a portion of RAM that is accessed through normal reads and writes.

Syntax
mkramdisk [-u] [size M | G]

Description
The mkramdisk command is shipped as part of bos.rte.filesystems, which allows the user to create a RAM disk. Upon successful execution of the mkramdisk command, a new RAM disk is created, a new entry added to /dev, the name of the new RAM disk is written to standard output, and the command exits with a value of 0. If the creation of the RAM disk fails, the command prints an internalized error message, and the command will exit with a nonzero value.

The size can be specified in terms of MB or GB. By default, it is in 512 byte blocks. A suffix of M will be used to specify size in megabytes and G to specify size in gigabytes.

The names of the RAM disks are in the form of /dev/rramdiskx where x is the logical RAM disk number (0 through 63).

The mkramdisk command also creates block special device entries (for example, /dev/ramdisk5) although use of the block device interface is discouraged because it adds overhead. The device special files in /dev are owned by root with a mode of 600. However, the mode, owner, and group ID can be changed using normal system commands.

Up to 64 RAM disks can be created.

Note: The size of a RAM disk cannot be changed after it is created.

The mkramdisk command is responsible for generating a major number, loading the ram disk kernel extension, configuring the kernel extension, creating a ram disk, and creating the device special files in /dev. Once the device special files are created, they can be used just like any other device special files through normal open, read, write, and close system calls.

RAM disks can be removed by using the rmramdisk command. RAM disks are also removed when the machine is rebooted.

By default, RAM disk pages are pinned. Use the -u flag to create RAM disk pages that are not pinned.
Flags
-u Specifies that the ram disk that is created will not be pinned. By default, the ram disk will be pinned.

Parameters
size Indicates the amount of RAM (in 512 byte increments) to use for the new RAM disk. For example, typing:
    mkramdisk 1
creates a RAM disk that uses 512 bytes of RAM. To create a RAM disk that uses approximately 20 MB of RAM, type:
    mkramdisk 40000

Exit Status
The following exit values are returned:
0 Successful completion.
>0 An error occurred.

Examples
1. To create a new ram disk using a default 512-byte block size, and the size is 500 MBs (1048576 * 512), enter:
    mkramdisk 1048576
    /dev/rramdisk0
    The /dev/rramdisk0 ramdisk is created.
2. To create a new ramdisk with a size of 500 Megabytes, enter:
    mkramdisk 500M
    /dev/rramdisk0
    The /dev/rramdisk0 ramdisk is created. Note that the ramdisk has the same size as example 1 above.
3. To create a new ram disk with a 2-Gigabyte size, enter:
    mkramdisk 2G
    /dev/rramdisk0
4. To set up a RAM disk that is approximately 20 MB in size and create a JFS file system on that RAM disk, enter the following:
    mkramdisk 40000
    ls -l /dev | grep ram
    mkfs -V jfs /dev/ramdiskx
    mkdir /ramdisk0
    mount -V jfs -o nointegrity /dev/ramdiskx /ramdiskx

    where x is the logical RAM disk number.
    For using RAM file system on JFS2, where nointegrity is not supported by the mount command, the example is as follows:
    mkramdisk 40000
    ls -l /dev | grep ram
    /sbin/helpers/jfs2/mkfs -V jfs2 -o log=INLINE /dev/ramdiskx
    mkdir /ramdiskx
    mount -V jfs2 -o log=/dev/ramdiskx /dev/ramdiskx /ramdiskx

    where x is the logical RAM disk number.
Note: If using file system on a RAM disk, the RAM disk must be pinned.

Files

| /usr/sbin/mkramdisk | Contains the mkramdisk command. |

Related Information

The rmramdisk command.

mkresponse Command

Purpose

Creates a new response definition.

Syntax

To create a response with no actions:

```
mkresponse [-p node_name] [-h] [-TV] response
```

To create a response with one action:

```
mkresponse -n action [-d days_of_week[,days_of_week...]]
[-t time_of_day[,time_of_day...]] -s action_script [-r return_code]
[-e a | r | b] [-o] [-E env_var=value[,env_var=value...]]
[-u] [-p node_name] [-h] [-TV] response
```

To copy a response:

```
mkresponse -c existing_response[:node_name] [-p node_name] [-h] [-TV] response
```

Description

The mkresponse command creates a new response definition with the name specified by the response parameter. One action can also be specified when the response is defined. Actions define commands to be run when the response is used with a condition and the condition occurs. The action defines days of the week when the action can be used, the time of day for those days of the week, the script or command to be run, what type of event causes the command to be run, the expected return code of the script or command, and whether to keep standard output. The days and times are paired so that different times can be specified for different days. A response with no actions only logs the events.

When in a cluster environment, use the -p flag to specify the node in the domain that is to contain the response definition. If the -p flag is not specified, the response is defined on the local node.

The chresponse command can be used to add actions to a response or to remove actions from a response. Monitoring can be started by using the startcondresp command. The startcondresp command links a response to a condition if they are not already linked.

To lock a response so it cannot be modified or removed, use the chresponse command (with its -L flag).

Flags

- **c existing_response[:node_name]**
  
  Copies an existing response. Links with conditions are not copied. The existing response is
defined on the node known as *node_name* in a cluster. If *node_name* is not specified, the local
node is used. *node_name* is a node within the scope determined by the
CT_MANAGEMENT_SCOPE environment variable. If any other flags are specified, update the
new response as indicated by the flags.

--n action
Specifies the name of the action being defined. Only one action can be defined when the
response is created. Use the chresponse command to add more actions to the response.

--d days_of_week
Specifies the days of the week when the action being defined can be run. *days_of_week* and
time_of_day together define the interval when the action can be run.

Enter the numbers of the days separated by a plus sign (+) or as a range of days separated by a
hyphen (-). More than one *days_of_week* parameter can be specified, but the parameters must be
separated by a comma (,). The number of *days_of_week* parameters specified must match the
number of *time_of_day* parameters specified. The default is all days. If no value is specified but a
comma is entered, the default value is used. The values for each day follow:
1 Sunday
2 Monday
3 Tuesday
4 Wednesday
5 Thursday
6 Friday
7 Saturday

--t time_of_day
Specifies the time range when *action* can be run, consisting of the start time followed by the end
time, separated by a hyphen. *days_of_week* and *time_of_day* together define the interval when the
action can be run.

The time is in 24-hour format (HHMM) where the first two digits represent the hour and the last
two digits represent the minutes. The start time must be less than the end time because the time
is specified by day of the week. More than one *time_of_day* parameter can be specified, but the
parameters must be separated by a comma (,). The number of *days_of_week* parameters
specified must match the number of *time_of_day* parameters specified. The default value is
0000-2400. If no value is specified but a comma is entered, the default value is used.

--s action_script
Specifies the fully-qualified path for the script or command to run for the action being defined. See
the man pages for logevent, notifyevent, and wallevent for descriptions of the predefined
response scripts provided with the application.

--r return_code
Specifies the expected return code for *action_script*. If the expected return code is specified, the
actual return code of *action_script* is compared to the expected return code. A message is written
to the audit log indicating whether they match. If the --r flag is not specified, the actual return code
is written to the audit log, and no comparison is performed.

--e a | r | b
Specifies the type of event that causes the action being defined to run:
a Event. This is the default.
r Rearm event.
b Both event and rearm event.

--o Directs all standard output from *action_script* to the audit log. The default is not to keep standard
output. Standard error is always directed to the audit log.
-E  env_var=value[,env_var=value...]
   Specifies any environment variables to be set before running the action. If multiple env_var=value
   variables are specified, they must be separated by commas.

-u  Specifies that the action is to be run when a monitored resource becomes undefined.

-p  node_name
   Specifies the name of the node where the response is defined. This is used in a cluster
   environment and the node name is the name by which the node is known in the domain. The
   default node_name is the local node on which the command runs. node_name is a node within the
   scope determined by the CT_MANAGEMENT_SCOPE environment variable.

-h  Writes the command’s usage statement to standard output.

-T  Writes the command’s trace messages to standard error. For your software service organization’s
   use only.

-V  Writes the command’s verbose messages to standard output.

Parameters
response  The response name is a character string that identifies the response. If the name contains
spaces, it must be enclosed in quotation marks. A name cannot consist of all spaces, be
null, or contain embedded double quotation marks.

Security
The user needs write permission for the IBM.EventResponse resource class to run mkresponse.
Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status
0     The command ran successfully.
1     An error occurred with RMC.
2     An error occurred with a command-line interface script.
3     An incorrect flag was entered on the command line.
4     An incorrect parameter was entered on the command line.
5     An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT
   Determines the system where the session with the resource monitoring and control (RMC)
   daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts
   the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the
   RMC daemon on the local system where the command is being run. The target of the RMC
   daemon session and the management scope determine the resource classes or resources that are
   processed.

CT_MANAGEMENT_SCOPE
   Determines the management scope that is used for the session with the RMC daemon in
   processing the resources of the event-response resource manager (ERRM). The management
   scope determines the set of possible target nodes where the resources can be processed. The
   valid values are:

   0     Specifies local scope.
   1     Specifies local scope.
2 Specifies peer domain scope.
3 Specifies management domain scope.

If this environment variable is not set, local scope is used.

Standard Output
When the -h flag is specified, this command’s usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
These examples apply to standalone systems:
1. To define a response with the name "Log event in audit log", run this command:
   mkresponse "Log event in audit log"
2. To define a response with the name "E-mail root anytime" that has an action named "E-mail root", to be used any time Saturday and Sunday and uses the command /usr/sbin/rsct/bin/notifyevent root for both events and rearm events, run this command:
   mkresponse -n "E-mail root" -d 1+7 \ 
   -s "/usr/sbin/rsct/bin/notifyevent root" -e b \ 
   "E-mail root anytime"
3. To define a response with the name "E-mail root anytime" that has an action named "E-mail root", to be used anytime Saturday and Sunday but only 8 am to 5 pm Monday through Friday and that uses the command /usr/sbin/rsct/bin/notifyevent root for events, run this command:
   mkresponse -n "E-mail root" \ 
   -d 1+7,2-6 -t 0800-2400,0800-1700 \ 
   -s "/usr/sbin/rsct/bin/notifyevent root" -e a \ 
   "E-mail root anytime"
4. To define a response with the name "E-mail root anytime" that has an action named "E-mail root" to be used any time Saturday and Sunday, that uses the command /usr/sbin/rsct/bin/notifyevent root for both events and rearm events, and that sets the environment variable LANG to en_US, run this command:
   mkresponse -n "E-mail root" -d 1+7 \ 
   -s "/usr/sbin/rsct/bin/notifyevent root" -e b \ 
   -E LANG="en_US" "E-mail root anytime"
5. To define a response with the name "E-mail root first shift" that has an action named "E-mail root" to be used Monday through Friday from 8 am to 6 pm, that uses the command /usr/sbin/rsct/bin/notifyevent root for rearm events, and that saves standard output in the audit log, expecting return code 5, run this command:
   mkresponse -n "E-mail root" -d 2-6 -t 0800-1800 \ 
   -s "/usr/sbin/rsct/bin/notifyevent root" -e r -o \ 
   -r 5 "E-mail root first shift"
6. To define a response with the name "Critical notifications" as a copy of "Warning notifications", enter:
   mkresponse -c "Warning notifications" "Critical notifications"

These examples apply to management domains:
1. To define a response on the management server with the name "E-mail root anytime" that has an action named "E-mail root", to be used any time Saturday and Sunday and uses the command /usr/sbin/rsct/bin/notifyevent root for both events and rearm events, run this command on the management server:
2. To define a response on the managed node nodeB with the name "E-mail root anytime" that has an action named "E-mail root", to be used any time Saturday and Sunday and uses the command /usr/sbin/rsct/bin/notifyevent root for both events and rearm events, run this command on the management server:

```
mkresponse -n "E-mail root" -d 1+7 \ 
-s "/usr/sbin/rsct/bin/notifyevent root" -e b \ 
"E-mail root anytime"
```

3. To define a response on the managed node nodeB with the name "nodeB Warning notifications" as a copy of "nodeA Warning notifications" on the managed node nodeA, run this command on the management server:

```
mkresponse -c "nodeA Warning notifications":nodeA \ 
-p nodeB "nodeB Warning notifications"
```

These examples apply to peer domains:

1. To define a response on the current node with the name "E-mail root anytime" that has an action named "E-mail root", to be used any time Saturday and Sunday and uses the command /usr/sbin/rsct/bin/notifyevent root for both events and rearm events, run this command from any node in the domain:

```
mkresponse -n "E-mail root" -d 1+7 \ 
-s "/usr/sbin/rsct/bin/notifyevent root" -e b \ 
"E-mail root anytime"
```

2. To define a response on the node nodeB in the domain with the name "E-mail root anytime" that has an action named "E-mail root", to be used any time Saturday and Sunday, that uses the command /usr/sbin/rsct/bin/notifyevent root for both events and rearm events, and that sets two environment variables (PAGE ALL and TIMER SET), run this command from any node in the domain:

```
mkresponse -n "E-mail root" -d 1+7 \ 
-s "/usr/sbin/rsct/bin/notifyevent root" -e b \ 
-p nodeB -E 'ENV1="PAGE ALL", ENV2="TIMER SET"' \ 
"E-mail root anytime"
```

3. To define a response on the node nodeB in the domain with the name "nodeB Warning notifications" as a copy of "nodeA Warning notifications" on the node nodeA in the domain, run this command from any node in the domain:

```
mkresponse -c "nodeA Warning notifications":nodeA \ 
-p nodeB "nodeB Warning notifications"
```

Location
/usr/sbin/rsct/bin/mkresponse
Contains the mkresponse command

Related Information
Commands: chresponse, lsresponse, mkcondition, mkcondresp, rmresponse, startcondresp

Files: rmccli

mkrole Command

Purpose
Creates new roles. This command applies only to AIX 4.2.1 and later.
Syntax

mkrole [ Attribute=Value ... ] Name

Description

The mkrole command creates a new role. The Name parameter must be a unique role name. You cannot use the ALL or default keywords as the role name.

You can use the Users application in Web-based System Manager to change user characteristics. You could also use the System Management Interface Tool (SMIT) to run this command.

Restrictions on Creating Role Names

To prevent inconsistencies, restrict role names to characters with the POSIX portable filename character set. You cannot use the keywords ALL or default as a role name. Additionally, do not use any of the following characters within a role-name string:

:  Colon
"  Double quotation mark
#  Comma
=  Equal sign
\  Backslash
/  Slash
?  Question mark
'  Single quotation mark
`  Back quotation mark

Finally, the Name parameter cannot contain any space, tab, or new-line characters.

Restrictions on Creating Roles

To ensure the integrity of the role information, only users with the RoleAdmin authorization can create a role.

Parameters

Attribute=Value  Initializes a role attribute. Refer to the chrole command for the valid attributes and values.

Security

Files Accessed:

Mode  File
rw   /etc/security/roles
r    /etc/security/user.roles

Auditing Events:

Event Information
ROLE_Create  role
Examples
To create the ManageObjects role and set the groups attribute to objects, type:

`mkrole groups=objects ManageObjects`

Files

/etc/security/roles
Contains the attributes of roles.

/etc/security/user.roles
Contains the role attribute of users.

Related Information

The `chrole` command, `chuser` command, `lsrole` command, `lsuser` command, `mkuser` command, `rmrole` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the AIX 5L Version 5.2 Security Guide.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

mkrpdomain Command

Purpose
Creates a new peer domain definition.

Syntax

```
mkrpdomain [-t TS_port] [-g GS_port] [-c] [-h] [-TV] peer_domain node_name1 [node_name2 ...]
```

```
mkrpdomain [-t TS_port] [-g GS_port] [-c] -f file_name | "" [-h] [-TV] peer_domain
```

Description

The `mkrpdomain` command creates a new peer domain definition with the name specified by the `peer_domain` parameter. The nodes specified by `node_name` are defined to the new peer domain. A peer domain can be used to provide high-availability services when configuring application and system resources.

The `preprpnode` command must have been run on each of the nodes to be defined to the peer domain. The `preprpnode` command prepares the security environment for the peer domain operations. See the `preprpnode` command for more information about peer domain definition requirements. Only those nodes that have the appropriate security setup will be successfully defined to the peer domain.

If the UDP port numbers for group services and topology services are not available on all of the nodes to be defined to the peer domain, the `mkrpdomain` command will fail. The command will also fail if the peer domain name is already being used or if any node cannot be successfully defined to the peer domain. Use the `-c` flag to enable `mkrpdomain` to continue when there is an error on one of the nodes.

The `mkrpdomain` command does not bring the peer domain online automatically. To bring the peer domain online, run the `startrpdomain` command. Nodes can be added to and removed from the peer domain using the `addrpnode` and `rmrpnode` commands.
A node can be defined in more than one peer domain but it can be online in only one peer domain at a time.

**Flags**

- `-t TS_port`
  Specifies the topology services port number. This UDP port is used for daemon-to-daemon communication. Any unused port in the range 1024 to 65535 can be assigned. The command will fail if the specified port is unavailable. The default is 12347.

- `-g GS_port`
  Specifies the group services port number. This UDP port is for daemon-to-daemon communication. Any unused port in the range 1024 to 65535 can be assigned. The command will fail if the specified port is unavailable. The default is 12348.

- `-c`
  Continues to run the `mkrpdomain` command on the remaining nodes.
  By default, if the `mkrpdomain` command fails on any node, it will fail on all nodes. The `-c` flag overrides this behavior, so that the `mkrpdomain` command will run on the other nodes, even if it fails on one node.

- `-f file_name` | `"-"`
  Reads a list of node names from `file_name`. Each line of the file is scanned for one node name. The pound sign (`#`) indicates that the remainder of the line (or the entire line if the `#` is in column 1) is a comment. Use `-f "-"` to specify `STDIN` as the input file.

- `-h`
  Writes the command’s usage statement to standard output.

- `-T`
  Writes the command’s trace messages to standard error. For your software service organization’s use only.

- `-V`
  Writes the command’s verbose messages to standard output.

**Parameters**

`peer_domain`

Specifies the name of the new peer domain to be created. The name can contain any printable character, except it cannot contain the slash character (`/`). In addition, the name cannot be `IW`.

`node_name1 [node_name2 ... ]`

Specifies the node (or nodes) to include in this peer domain definition. The node name is the IP address or the long or short version of the DNS hostname. The node name must resolve to an IP address.

**Security**

The user of the `mkrpdomain` command needs write permission to the `IBM.PeerDomain` resource class on each node that is to be defined to the peer domain. This is set up by running the `preprpnode` command on each node that is to be defined to the domain, specifying the name of the node on which the user will run `mkrpdomain`.

**Exit Status**

0  The command ran successfully.
1  An error occurred with RMC.
2  An error occurred with a command-line interface script.
3  An incorrect flag was entered on the command line.
4  An incorrect parameter was entered on the command line.
An error occurred that was based on incorrect command-line input.

Environment Variables

CT_CONTACT
Determines the system where the session with the resource monitoring and control (RMC) daemon occurs. When CT_CONTACT is set to a host name or IP address, the command contacts the RMC daemon on the specified host. If CT_CONTACT is not set, the command contacts the RMC daemon on the local system where the command is being run. The target of the RMC daemon session and the management scope determine the resource classes or resources that are processed.

Restrictions
Any node to be defined to the peer domain must be reachable from the node on which this command runs.

Standard Input
When the -f "-" flag is specified, this command reads one or more node names from standard input.

Standard Output
When the -h flag is specified, this command's usage statement is written to standard output. All verbose messages are written to standard output.

Standard Error
All trace messages are written to standard error.

Examples
1. To define a peer domain called ApplDomain that consists of a node called nodeA, run this command on nodeA:
   ```bash
   mkrpdomain ApplDomain nodeA
   ```
2. To define a peer domain called ApplDomain that consists of three nodes called nodeA, nodeB, and nodeC, run this command on nodeA, nodeB, or nodeC:
   ```bash
   mkrpdomain ApplDomain nodeA nodeB nodeC
   ```
3. To define a peer domain called ApplDomain that consists of two nodes called nodeA and nodeB, with a topology services port number of 1200 and a group services port number of 2400, run this command on nodeA or nodeB:
   ```bash
   mkrpdomain -t 1200 -g 2400 ApplDomain nodeA nodeB
   ```

Location

/usr/sbin/rsct/bin/mkrpdomain
Contains the mkrpdomain command

Files
The /etc/services file is modified.

Related Information
Commands: addrpnode, lsrdomain, lsrpnode, preprpnode, rmrpdomain, rmrpnode, startrpdomain, stoprpdomain

Files: rmccli, for general information about RMC-related commands
mkrset Command

Purpose
Makes an rset containing the specified CPUs and memory regions and places it in the system registry.

Syntax
```
mkrset [-c CPUlist [ -m MEMlist ]] rsetname
```

Description
The `mkrset` command creates and places into the system registry an rset with the specified set of CPUs and/or memory regions. The rset name must not exist in the registry. The owner and group IDs of the rset will be set to the owner and group IDs of the command issuer. The rset will have read/write owner permissions and read permission for group and other.

Flags
- `-c` List of CPUs to be in the rset. This can be one or more CPUs or CPU ranges.
- `-m` List of memory regions to be in the rset. This can be one or more memory regions or ranges.

Parameters
- `rsetname` The name of the rset to be placed in the system registry. The name consists of a `namespace` and an `rsname` separated by a “/” (slash). Both the namespace and rsname may contain up to 255 characters. See the `rs_registername()` service for additional information about character set limits of rset names.

Security
The user must have root authority or CAP_NUMA_ATTACH capability.

Examples
1. To make an rset containing CPUs 0-7 named `test/cpus0to7`, type:
   ```
mkrset -c 0-7 test/cpus0to7
   ```
2. To make an rset containing CPUs 1, 3, 5, 6, 7, 10 named `test/lotsofcpus`, type:
   ```
mkrset -c 1 3 5-7 10 test/lotsofcpus
   ```

Files
- `/usr/bin/mkrset` Contains the `mkrset` command.

Related Information
The `attachrset|detachrset|execrset|lsrset` and `rmrset` commands.

mkrsrc Command

Purpose
Defines a new resource.
Syntax
To define a new resource, using data that is...
• entered on the command line:
  
mkrsrc [-v] [-h] [-TV] resource_class attr=value...
• predefined in an input file:
  
mkrsrc -f resource_data_input_file [-v] [-h] [-TV] resource_class

To see examples of the mkrsrc command for a resource class:

mkrsrc -e [-h] [-TV] resource_class

Description
The mkrsrc command requests that the RMC subsystem define a new resource instance for the class specified by the resource_class parameter. At least one persistent attribute name and its value must be specified either as a parameter or by a resource definition file using the -f flag.

Before you run mkrsrc, you should run the lsrsrsrcdef command to determine which attributes are designated as reqd_for_define (required) or option_for_define (optional). Only attributes that are designated as reqd_for_define or option_for_define can be defined using the mkrsrc command. The lsrsrsrcdef command also identifies the datatype for each attribute. The value specified for each attribute must match this datatype.

To verify that all the attribute names specified on the command line or in the resource_data_input_file are defined as persistent attributes and are designated as reqd_for_define or option_for_define, use the -v flag. When the mkrsrc command is run with the -v flag, the resource is not defined. Instead, the resource attributes are merely verified to be persistent and designated as reqd_for_define or option_for_define. Once you have run mkrsrc -v to verify that the attributes specified on the command line or resource_data_input_file are valid, you can issue the mkrsrc command without the -v flag to define the new resource.

Flags
- e Displays examples of mkrsrc command-line input for:
  1. required attributes only
  2. required and optional attributes
- f resource_data_input_file
  Specifies the name of the file that contains resource attribute information.
- v Verifies that all of the attribute names specified on the command line or in the input file are defined as persistent attributes and are designated as reqd_for_define or option_for_define. The mkrsrc command does not define any resources when you use this flag.
- h Writes the command’s usage statement to standard output.
- T Writes the command’s trace messages to standard error. For your software service organization’s use only.
- V Writes the command’s verbose messages to standard output.

Parameters
resource_class
  Specifies the resource class name of the resource to be defined.

attr=value...
  Specifies the attributes of the resource being defined. When defining a new resource instance,
there are specific required attributes for each resource that must be defined. These attributes can be specified as parameters on the command line or defined in an input file by using the -f flag.

- **attr** The name of a persistent attribute for this resource. This attribute must be designated as reqd_for_define or option_for_define. Use the lsrsrccdef command to check the designation.

- **value** The value for this persistent attribute. The data type for this value must match the defined data type for the value of this attribute. Use the lsrsrccdef command to verify the data type for each attribute.

**Security**

The user needs write permission for the resource_class specified in mkrsrc to run mkrsrc. Permissions are specified in the access control list (ACL) file on the contacted system.

**Exit Status**

- **0** The command has run successfully.
- **1** An error occurred with RMC.
- **2** An error occurred with the command-line interface (CLI) script.
- **3** An incorrect flag was specified on the command line.
- **4** An incorrect parameter was specified on the command line.
- **5** An error occurred with RMC that was based on incorrect command-line input.

**Environment Variables**

**CT_CONTACT**

When the CT_CONTACT environment variable is set to a host name or IP address, the command contacts the Resource Monitoring and Control (RMC) daemon on the specified host. If the environment variable is not set, the command contacts the RMC daemon on the local system where the command is being run. The resource class or resources that are displayed or modified by the command are located on the system to which the connection is established.

**CT_MANAGEMENT_SCOPE**

Determines the management scope that is used for the session with the RMC daemon to monitor and control the resources and resource classes. The management scope determines the set of possible target nodes where the resources and resource classes can be monitored and controlled. The valid values are:

- **0** Specifies local scope.
- **1** Specifies local scope.
- **2** Specifies peer domain scope.
- **3** Specifies management domain scope.

If this environment variable is not set, local scope is used.

**Standard Output**

When the -h flag is specified, this command's usage statement is written to standard output.

The command output and all verbose messages are written to standard output.

**Standard Error**

All trace messages are written to standard error.
Examples

1. To create a new resource in the IBM.Host class, assuming you already know which persistent attributes are required when defining a resource of this class, enter:

   mkrsrc IBM.Host Name=c175n05

2. To create a new resource in the IBM.Processor class by first generating a template to aid in the defining of these resources, enter:

   lsrsrccdef -i IBM.Processor > /tmp/IBM.Processor.rdef

   Then, edit the file /tmp/IBM.Processor.rdef and enter values for all of the attributes, substituting the type for an appropriate value, or leaving it blank for the default value.

   Finally, enter:

   mkrsrc -f /tmp/IBM.Processor.rdef IBM.Processor

3. To create two new IBM.Host resources using the information defined in file /tmp/IBM.Host.rdef, enter:

   mkrsrc -f /tmp/IBM.Host.rdef IBM.Host

   where the file /tmp/IBM.Host.rdef looks like this:

   PersistentResourceAttributes:
   resource 1:
     Name = c175n04
   resource 2:
     Name = c175n05

4. This example creates a new resource in the IBM.Foo class. In this class, Name and NodeList are required attributes. The Binary, SD, StringArray, and SDArray attributes are optional. This example shows how to enter the more difficult data types from the command line. The data types for the optional attributes (Binary, SD, StringArray, and SDArray) are self-explanatory. Enter:

   mkrsrc IBM.Foo Name=c175n05 \
   NodeList={1} \ 
   Binary="0xaabbccddeeff00" \ 
   SD='[testing123,1,{2,4,6}]' \ 
   StringArray='{"testing 1 2 3","testing123","testing 1 2 3"}' \ 
   SDArray='{"testing 1 2 3",1,[1,3,5]},{testing,2,][2,4,6]}'

   Note: As discussed in the rmccli general information file, attribute values for certain data types (structured data, array of structured data, and arrays containing strings enclosed in double quotation marks) should be enclosed in single quotation marks.

Location

/usr/sbin/rsct/bin/mkrsrc Contains the mkrsrc command

Related Information

Commands: chrsrc, lsrsrcc, lsrsrccdef, rmrsrc

Files: Resource_Data_Input, rmccli

mkseckrb5 Command

Purpose

Migrates existing operating system users to Kerberos.

Syntax

mkseckrb5 [ -h ] [ -r ]
Description
This command gets the list of user names and creates Kerberos users. If the \texttt{-r} flag is not specified, the command prompts for a new password for each user.

\textbf{Standard Output} \hspace{1em} Consists of information messages when the \texttt{-h} flag is used.
\textbf{Standard Error} \hspace{1em} Consists of error messages when the command cannot complete successfully.

\textbf{Flags}
\begin{itemize}
  \item \texttt{-h} \hspace{1em} Specifies that the command is only to display the valid command syntax.
  \item \texttt{-r} \hspace{1em} Specifies that random passwords are to be used.
\end{itemize}

\textbf{Exit Status}
Failure of this command to execute successfully results in incomplete migration. The admin must check the Kerberos database for the users that were migrated before taking further action.

\begin{itemize}
  \item \texttt{0} \hspace{1em} Indicates the successful completion of the command.
  \item \texttt{1} \hspace{1em} Indicates that an error occurred.
\end{itemize}

\textbf{Security}
Only the root user is authorized to use this command.

\textbf{Examples}
1. To display the command syntax, type:
   \begin{verbatim}
   mkseckrb5 -h
   \end{verbatim}
2. To migrate existing users to Kerberos users, type:
   \begin{verbatim}
   mkseckrb5
   \end{verbatim}
3. To migrate user trojan to Kerberos user with random passwd, type:
   \begin{verbatim}
   mkseckrb5 -r trojan
   \end{verbatim}

\textbf{Files}
\begin{verbatim}
/usr/sbin/mkseckrb5
\end{verbatim}
Contains the \texttt{mkseckrb5} command.

\begin{verbatim}
mksecldap Command
\end{verbatim}

\textbf{Purpose}
Sets up an AIX cluster to use LDAP for security authentication and data management.

\textbf{Syntax}
The syntax to set up a server is:
\begin{verbatim}
mksecldap \(-s\) \(-a\) adminDN \(-p\) adminpasswd \(-S\) schematype \(\[-\text{d} baseDN\]\(\[-n\] port\]\(\[-k\] SSLkeypath\]\(\[-\text{w}\] SSLkeypasswd\]\(\[-u\] NONE\]\(\[-U\]\)
\end{verbatim}
The syntax to set up a client is:
Note:
For group user security suffix:

1. The client (-c flag) and the server (-s flag) options cannot be run at the same time. When setting up a server, the mksecldap command should be run twice on that machine. Once to set up the server, and again to set up the client.

2. The SecureWay Directory server configuration file is /etc/slapd.conf for SecureWay Directory 3.1 or earlier and /etc/slapd32.conf for SecureWay Directory 3.2 or later. AIX 5.2 only supports SecureWay Directory 3.2 and later.

For server setup, make sure that the ldap.server files set is installed. When installing the ldap.server files set, the ldap.client files set and the backend DB2 software are automatically installed as well. No DB2 preconfiguration is required to run this command for LDAP server setup. When you run the mksecldap command to set up the server, the command will:

1. Create a DB2 instance with ldapdb2 as the default instance name.
2. Create a DB2 database with ldapdb2 as the default database name. If a database already exists, mksecldap will bypass the above two steps. (This is the case when the LDAP server has been set up for other usage.) The mksecldap command will use the existing database to store the AIX user/group data.
3. Create the AIX tree DN (suffix). If no baseDN is supplied from the command line, the default suffix is set to cn=aixdata and the user/group data is placed to the cn=aixsecdb,cn=aixdata DN. This is the recommended case. Otherwise, the mksecldap command takes the user-supplied DN and prefixes it with cn=aixdata and makes the newly constructed DN the suffix. This behavior is shown in the following table. The value within the brackets represents the optional user supplied DN from command line.

| CMD-line DN: | [o=ibm] |
| suffix: | cn=aixdata,o=ibm |
| security DN: | cn=aixsecdbcn=aixdata,o=ibm |
| user DN: | ou=aixuser,cn=aixsecdbcn=aixdata,o=ibm |
| group DN: | ou=aixgroup,cn=aixsecdbcn=aixdata,o=ibm |

In case the LDAP server has already been setup in the local system, the mksecldap command searches for the cn=aixsecdbcn=aixdata keyword from the suffixes defined in the slapd32.conf configuration file and from the database. If it finds the keyword, it assumes that mksecldap has been run, and bypasses the base DN setup step and the user/group migration step, and exits.

If cn=aixsecdbcn=aixdata is not found in the suffixes and the database, the mksecldap command checks for the cn=aixdata keyword. cn=aixdata is a common base DN shared by various AIX LDAP components. If the mksecldap command find the keyword, it compares it with the user supplied DN. If they are the same, the users/groups will be put under the cn=aixsecdbcn=aixdata,[userDN]. If they are different, the mksecldap command prints an error message warning the existence of the cn=aixdata,... DN, and it will not migrate the users/groups under the user supplied DN. You can choose to use the existing cn=aixdata,... by running the mksecldap command again with that existing DN.

4. Migrates the data from the security database files from the local host into the LDAP database. Depending on the -S option, the mksecldap command migrates users/groups using one of the three LDAP schemas:
• AIX - AIX schema (aixaccount and aixaccessgroup objectclasses)
• RFC2307 - RFC 2307 schema (posixaccount, shadowaccount, and posixgroup objectclasses)
• RFC2307AIX - RFC 2307 schema with full AIX support (posixaccount, shadowaccount, and posixgroup objectclasses, plus the aixauxaccount and aixauxgroup object classes).
  
  **Attention:** Systems running AIX 4.3 and AIX 5.1 which are configured as LDAP clients will only work with servers of AIX schema type. They will not talk to ldap servers of RFC2307 or RFC2307AIX types.

5. Set the LDAP server administrator DN and password. This name/password pair is also used for access control of the AIX tree.

6. Set the SSL (secure socket layer) for secure data transfer between this server and the clients. This setup requires the **GSKIT** to be installed.

  **Note:** If this option is used, the SSL key must be created before running the **mksecldap** command. Otherwise the server may not be able to start.

7. Installs the /usr/ccs/lib/libsecldapaudit.a, a LDAP server plug-in. This plugin supports AIX audit of the LDAP server.

8. Start/restart the LDAP server after all the above is done.

9. Add the LDAP server process (slapd) to /etc/inittab to have the LDAP server start after reboot.

10. With the **-U** option, undo a previous setup for the server configuration file. The first time you run the **mksecldap** command, it saves two copies of the slapd32.conf server configuration file. One is saved to /etc/security/ldap/slapd32.conf.save.orig and the other to /etc/security/ldap/slapd32.conf.save. Each subsequent run of **mksecldap**, the current slapd32.conf is only saved to /etc/security/ldap/slapd32.conf.save file. The undo option restores the /etc/slapd32.conf server configuration file with the /etc/security/ldap/slapd32.conf.save copy.

  **Note:** The undo option applies to the server configuration file only. It has no effect on the database.

  **Note:** All the LDAP configuration is saved into the /etc/slapd32.conf LDAP server configuration file.

For client setup, make sure the LDAP server has been setup and is running. The **mksecldap** command does the follows during for client setup:

1. Saves the LDAP server(s)' host name.

2. Saves the user base DN and group base DN of the server. If no **-d** option is supplied from command line, the **mksecldap** command searches the LDAP server for aixaccount, aixaccessgroup, posixaccount, posixgroup, and aixauxaccount objectclasses from the LDAP server, and set up the base DN's accordingly. If the server has multiple user/group bases, you must supply the **-d** option with a RDN so that the **mksecldap** command can setup the base DN's to the ones within that RDN.

If the posixaccount objectclass is found during client setup, **mksecldap** will also try to search for base DN's for these entities: hosts, networks, services, netgroups, protocols, and rpc from the server and save any that is found.

3. Determines the schema type used by the LDAP server - AIX specific schema, RFC 2307 schema, or RFC 2307 schema with full AIX support (see objectclasses listed in step 2). It sets the objectclasses and attribute maps in the /etc/security/ldap/ ldap.cfg file accordingly. The **mksecldap** command does not recognize other schema types, so clients must be setup manually.

4. Sets SSL for secure data transfer between this host and the LDAP server. This step requires that the client SSL key and the key password are created in advance, and the server must be setup to use SSL for the client SSL to work.

5. Saves the LDAP server administrator DN and password. The DN/password pair must be the same as the pair specified during server setup.

6. Sets the cache size in terms of the number of entries used by the client side daemon. Valid values are 100-10,000 for users and 10-1,000 for groups. The default value is 1,000 for users and 100 for groups.
7. Sets the cache timeout of the client side daemon. Valid values are 60-3600 seconds. The default is 300 seconds. Set this value to 0 to disable caching.
8. Sets the number of threads used by the client side daemon. Valid values are 1-1,000. The default is 10.
9. Sets the time interval in seconds that the client daemon checks for the LDAP server status. Valid value are 60-3,600 seconds. The default is 300.
10. Optionally sets the list of users or all users to use LDAP by modifying their SYSTEM line in the /etc/security/user file. For more information on enabling ldap login in, see the following note.
11. Starts the client daemon process (secldapclntd).
12. Adds the client side daemon process to /etc/inittab to have this daemon start after a reboot.
13. With the -U option, undo a previous setup to the /etc/security/ldap/ldap.cfg file.

Note: The client configuration data is saved to the /etc/security/ldap/ldap.cfg file. Setting the SYSTEM to LDAP for the default stanza of /etc/security/user only allows LDAP users to login to the system. Setting the SYSTEM to LDAP or compat allows both LDAP users and local users to login to the system.

Flags
For Server Setup

-a AdminDN Specifies the LDAP server administrator DN.
-d baseDN Specifies the suffix or base DN of the AIX subtree. The default is cn=aixdata.
-k SSLkeypath Specifies the full path to the SSL key database of the server.
-n port Specifies the port number that the LDAP server listens to. Default is 389 for non-SSL and 636 for SSL.
-p adminpasswd Specifies the clear text password for the administrator DN.
-S schematypé Specifies the LDAP schema used to represent user/group entries in the LDAP server. Valid values are AIX, RFC2307, and RFC2307AIX.
-s Indicates that the command is being run to setup the server.
-w SSLkeypasswd Specifies the password for the SSL key.
-U Specifies to undo the previous server setup to the LDAP configuration file.
-u NONE Specifies not to migrate users and groups from local system. The only valid value is NONE. Any other values are ignored. No -S option is required with this option.

For Client Setup

-a AdminDN Specifies the LDAP server administrator DN. It must match the one used for the server setup.
-c Indicates the command is being run to setup the client.
-C Cachsize Specifies the maximum number of user entries used in the client side daemon cache. Valid values are 100-10,000 for user cache. The default value is 1,000. The group cache is 10% of that of user cache.
-d baseDN Specifies the base DN for the mksecldap command to search for the user base DN and group base DN. If not specified from the command line, the entire database is searched.
-h serverlist Specifies a comma separated list of hostnames (server and backup servers).
-k SSLkeypath Specifies the full path to the client SSL key.
-n serverport Specifies the port number that the LDAP server is listening to.
-p adminpasswd Specifies the clear text password for the administrator DN of the LDAP server. It must match the one used for the server setup.
-P NumberOfTreads Specifies the number of threads the client side daemon uses. Valid values are 1-1,000. The default is 10.
-t Cachetimeout Specifies the maximum time length that a cache entry expires. Valid values are 60-3,600 seconds. The default is 300 seconds. Set this value to 0 to disable caching.
-T heartBeatInt Specifies the time interval of heartbeat between this client and the LDAP server. Valid values are 60-3,600 seconds. Default is 300.
-u userlist Specifies the comma separated list of usernames. Specify ALL to enable all users on the client.
-w SSLkeyfilepath Specifies the password for the client SSL key.
-U Specifies to undo the previous client setup to the LDAP client configuration file.

Examples
1. To setup a LDAP server of AIX specific schema for users and groups, enter:
   
   mksecldap -s-a cn=admin -p adminpwd -S aix

   This sets up a LDAP server with LDAP server administrator DN being cn=admin, password being adminpwd. User and group data is migrated from local files to the default cn=aixdata suffix.

2. To setup a LDAP server with a baseDN other than the default and with SSL secure communication, enter:
   
   mksecldap -s-a cn=admin -p adminpwd -d o=mycompany,c=us -S rfc2307 \ -k /usr/ldap/serverkey.kdb \ -w keypwd

   This sets up a LDAP server with LDAP server administrator DN being cn=admin, password being adminpwd. User and group data is migrated from local files to the cn=aix-data,o=mycompany,c=us suffix. The LDAP server uses SSL communications by using the key stored at /usr/ldap/serverkey.kdb. The password to the key, keypwd, must also be supplied. Users and groups are migrated with the RFC 2307 schema.

3. To undo a previous server setup:
   
   mksecldap -s -U

   This undo the previous setup to the /etc/slapd32.conf server configuration file. Note, for safety reasons, this does not remove any database entries or database created my a previous setup. One has to remove the database entries/database manually if they are not needed any more.

4. To setup a client to use the server1.ibm.com and server2.ibm.com LDAP servers, enter:
   
   mksecldap -c -a cn=admin -p adminpwd -h server1.ibm.com,server2.ibm.com

   The LDAP server administrator DN and password must be supplied for this client to authenticate to the server. The mksecldap command contacts the LDAP server for schema type used, and sets up the client accordingly. Without the -d option from the command line, the entire server DIT is searched for the user base DN and the group base DN.

5. To setup the client to talk to the server3.ibm.com LDAP server using SSL, enter:
   
   mksecldap -c-a cn=admin -p adminpwd -h server3.ibm.com -d o=mycompany,c=us -k /usr/ldap/clientkey.kdb -w keypwd -u user1,user2

   This sets up a LDAP client similar to case 3, but with SSL communication. The mksecldap command searches the o=mycompany,c=us RDN for user base DN and group base DN. Account user1 and user2 are configured to authenticate through LDAP.

   Note: The -u ALL option enables all LDAP users to login to this client.

6. To undo a previous client setup, enter:
   
   mksecldap -c -U

576 Commands Reference, Volume 3
This undo the previous setup to the /etc/security/ldap/ldap.cfg file. This does not remove the SYSTEM=LDAP and registry=LDAP from the /etc/security/user file.

Files Accessed:

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>/etc/passwd</td>
</tr>
<tr>
<td>r</td>
<td>/etc/group</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/passwd</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/limits</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/user (on the server)</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/user (on the clients)</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/environ</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/user.roles</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/lastlog</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/smitacl.user</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/mac_user</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/group</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/smitacl.group</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/roles</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/login.cfg (on the server)</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/slapd32.conf (on the server)</td>
</tr>
<tr>
<td>rw</td>
<td>/etc/security/ldap/ldap.cfg (on the client)</td>
</tr>
</tbody>
</table>

Related Information

The secldapclntd daemon.

The start-secldapclntd, stop-secldapclntd, restart-secldapclntd, ls-secldapclntd, flush-secldapclntd sectoldif, and nistoldif commands.

The /etc/security/ldap/ldap.cfg file.


mksecpki Command

Purpose

mksecpki configures AIX PKI server components. The components of AIX PKI are Certificate Authority, Registration Authority, and Audit subsystems.

Syntax

mksecpki (-u username -f reference_file [-p CA_port] [-H ldap_host] [-D dn -w password] [-i certificate_issuer_dn] [ -U username])

Description

The mksecpki command configures AIX PKI server components. mksecpki must be run after configuring an LDAP server to publish certificates. The values for the options -H, -D, -w, and -i must be the same values as the ones specified during the LDAP configuration. Otherwise, the CA will not be able to publish certificates to LDAP.
The **-u** option specifies the AIX username which will host AIX PKI. The username must follow AIX username rules. Do not use **-u** and **-U** together. The invoker of the command will be asked to provide a password for the username. **mksecpki** will create a database instance with the same name.

The **-f** option specifies the file containing the reference number and passphrase. The client certificate requests will use these exact same values while communication with the CA. The reference number and passphrase are each specified on a separate line. The following is the contents of an example **iafile**:

```
11122333
tempwd1234
```

The **-p** option specifies the port that Certificate Authority accepts the certificate requests. If no port number is given, 1077 will be assumed.

The **-H** option specifies the hostname of the LDAP server where the certificates are published to. Prior to invoking the **mksecpki** command, an LDAP server must be setup to publish certificates. Otherwise, the certificates will not be published to LDAP, however, certificate will be returned to the requestor when certificate management commands are used. If the **-H** option is not given the localhost will be used as the hostname.

The **-D** option is used to specify the directory administrators distinguished name. This must be the same one that is specified during the configuration of the LDAP server.

The **-w** option specifies the password corresponding to the administrator DN. It is an error not to specify both the admin DN and password.

The **-i** option specifies the distinguish name of the Certificate Authority issuing the certificates. This must be the same value as the one given when setting an LDAP server for publishing certificates.

The **-U** option specifies the username that hosts the AIX PKI that will be unconfigured. The command will confirm the unconfiguration before starting its operation. This option removes the username from the system. The invokers of this command will be asked if they want to remove the home directory of the username. When this command runs without errors, it displays a message indicating the successful completion. The invoker of this command is recommended to wait for this message.

**Flags**

- **-u username**
  Specifies the name of the username that is going to be created that will host AIX PKI server components.

- **-f reference_file**
  Specifies the file which contains the reference number and passphrase that is used when making a certificate creation request.

- **-p CA_port**
  Specifies the Certificate Authority Communication Port.

- **-H ldap_host**
  Specifies the LDAP host where the certificates are going to be published.

- **-D adminDN**
  Specifies directory administrator distinguished name (DN).
  **Note:** The **-D** option requires that the **-w password** option also be specified.

- **-w password**
  Specifies directory administrator password.

- **-i certificate_issuer_dn**
  Specifies the distinguished name of the Certificate Authority issuing certificates.

- **-U username**
  Specifies the username which hosts the AIX PKI that will be unconfigured.

**Security**

This command should grant execute (x) access only to the root user and members of the security group.
Examples
To configure AIX PKI server side using pkitest.ibm.com as the LDAP host name for publish certificates and using o=aix,c=us as the issuer name, enter:

$ mksecpki -u pkiuser -f iafile -p 829 -H pkitest.ibm.com -D cn=admin -w password -i o=aix,c=us

where iafile contains the reference number and passphrase.

To unconfigure the server, enter:
$ mksecpki -U pkiuser

Files
/usr/lib/security/pki/ca.cfg

Related Information
The certadd, certcreate, certdelete, certget, certlink, certlist, certrevoke, certverify, keyadd, keydelete, keylist, and keypasswd commands.

mksensor Command

Purpose
Defines a sensor to the resource monitoring and control (RMC) subsystem.

Syntax
mksensor [-i seconds] [-e 0 | 1 | 2] [-n host] [-h ] [-v | -V] name sensor_command

Description
The mksensor command defines a sensor resource to the resource monitoring and control (RMC) subsystem. A sensor is an RMC resource that has attributes that can be monitored. Once monitored, the values of the sensor attributes are set by the sensor resource manager. It updates the values at defined intervals using the sensor_command specified on the mksensor command. The ERM commands can be used to set up monitoring of the sensor attributes. The response actions defined will run when a monitored sensor event occurs. This enables administrators to extend RMC monitoring capabilities without having to write a resource manager.

The sensor values can also be updated using the refsensor command.

The sensor is identified by the name parameter specified on the mksensor command. The sensor_command parameter specifies what will be run by the sensor resource manager to set values to the attributes of this sensor. The sensor consists of the following attributes that can be set by sensor_command:

- **Float32**: The type float32 attribute for this sensor resource.
- **Float64**: The type float64 attribute for this sensor resource.
- **Int32**: The type int32 attribute for this sensor resource.
- **Int64**: The type int64 attribute for this sensor resource.
- **Uint32**: The type uint32 attribute for this sensor resource.
- **Uint64**: The type uint64 attribute for this sensor resource.
- **String**: The type string attribute for this sensor resource.
Quantum The type quantum attribute for this sensor resource.

The sensor_command parameter sets attribute values by sending the values to standard output in a format that the sensor resource manager can parse. The format is attr=value. For example, if sensor_command sets the Int32 attribute to 57, it writes Int32=57 to standard output. To set more than one attribute value, the sensor_command can write multiple attr=value pairs to standard output. The attr=value pairs can be on one or more lines. If the standard output of sensor_command is not in attr=value form, it is assumed to be a string and the value is placed in the String attribute.

Use the -e flag to control how the exit values from sensor_command are interpreted. Depending on this setting, when the exit value of the sensor_command is considered to be an error, the sensor attributes are not set and information is written to the audit log.

The sensor_command runs using the user ID that creates the sensor resource. Once a sensor resource is monitored, sensor_command is run at interval values specified by the -i flag, which is expressed in seconds. The default interval is 60 seconds if none is specified. Specify a value of 0 to indicate that sensor_command is not to run at intervals. In this case, the refsensor command is typically used to update the sensor values.

The mksensor command can be run on any node. If you are in a management or peer domain, you can use the -n flag to define the sensor on a node in the domain.

Flags

-i seconds
Specifies the interval in which sensor_command is run to update the values of the sensor attributes. seconds is an integer value and must be greater than or equal to 10. sensor_command is run at the specified interval only when a sensor resource is monitored. The default interval is 60 seconds. If the interval is set to 0, sensor_command will not be automatically run. Using the refsensor command is independent of interval updates.

-e 0 | 1 | 2
Specifies how the sensor resource manager interprets the exit code of sensor_command. The values mean:

0 No exit value from sensor_command is an error.
1 Non-zero exit value from sensor_command is an error.
2 0 exit value from sensor_command is an error.

The default value is 1. The sensor attributes are not updated when the exit value is interpreted as an error. For an error, information is written to the audit log.

-n host
Specifies the node on which the sensor should be defined. By default, the sensor is defined on the local node. This flag is only appropriate in a management domain or a peer domain.

-h Writes the command’s usage statement to standard output.

-v | -V
Writes the command’s verbose messages to standard output.

Parameters

name Specifies the name of the sensor to be defined.

sensor_command Specifies name of a command or script that will be used by the sensor resource manager to set the attribute values of the sensor.
Security
The user needs write permission for the IBM.Sensor resource class in order to run mksensor. Permissions are specified in the access control list (ACL) file on the contacted system.

Exit Status
0 The command has run successfully.
1 An incorrect combination of flags and parameters has been entered.
\textit{n} Based on other errors that can be returned by the RMC subsystem.

Environment Variables

\textbf{CT\_CONTACT}
When the \textit{CT\_CONTACT} environment variable is set to a host name or IP address, the command contacts the resource monitoring and control (RMC) daemon on the specified host. If this environment variable is not set, the command contacts the RMC daemon on the local system where the command is being run. The resource class or resources that are displayed or modified by the command are located on the system to which the connection is established.

\textbf{CT\_MANAGEMENT\_SCOPE}
Determines the management scope that is used for the session with the RMC daemon to monitor and control the resources and resource classes. The management scope determines the set of possible target nodes where the resources and resource classes can be monitored and controlled.

The valid values are:
0 Specifies \textit{local} scope.
1 Specifies \textit{local} scope.
2 Specifies \textit{peer domain} scope.
3 Specifies \textit{management domain} scope.

If this environment variable is \textit{not} set, \textit{local} scope is used.

Examples
1. To create a new sensor called \textit{Sensor1} that runs the script /usr/bin/updateSensor1, which will update the sensor attributes every 30 seconds (once monitored), enter:

\begin{verbatim}
mksensor -i 30 Sensor1 "/usr/bin/updateSensor1"
\end{verbatim}

The contents of /usr/bin/updateSensor1 may be like:

\begin{verbatim}
#!/usr/bin/perl
my $int32 = some_fn_that_generates_i32_value;
my $string = some_fn_that_generates_string_value;
print "Int32=$int32 String=$string";
exit 0;
\end{verbatim}

A sample condition could be:

\begin{verbatim}
mkcondition -r IBM.Sensor -s "Name==Sensor1" -e "Int32 > 100" Sensor1Int32
\end{verbatim}

Using the response "E-mail root anytime", a start monitoring command may be:

\begin{verbatim}
startcondresp Sensor1Int32 "E-mail root anytime"
\end{verbatim}

Location
/usr/sbin/rsct/bin/mksensor Contains the \texttt{mksensor} command
Related Information
Commands: chsensor, lssensor, mkcondition, mkresponse, rmsensor, startcondresp

Files: see the rmclii man page for information about attr=value syntax.

**mkserver Command**

**Purpose**
Adds a subserver definition to the subserver object class.

**Syntax**
```
mkserver -c CodePoint -s Subsystem -t Type
```

**Description**
The `mkserver` command adds a subserver definition to the **Subserver** object class.

**Flags**
- `-c CodePoint` Specifies the `CodePoint` integer that identifies the subserver. This is the value by which the subsystem knows the subserver. The `mkserver` command is unsuccessful if this `CodePoint` value already exists for this subsystem. The limit for `CodePoint` storage is the same as a short integer (1 through 32,768).
- `-s Subsystem` Specifies the name that uniquely identifies the subsystem to which the subserver belongs. The `mkserver` command is unsuccessful if the `Subsystem` name is not known in the subsystem object class, or if the `Subsystem` name is that of a known subsystem in the subsystem object class but uses signals as its communication method.
- `-t Type` Specifies the name that uniquely identifies the subserver. The `mkserver` command is unsuccessful if the `Type` name is already known in the **Subserver Type** object class.

**Security**

Auditing Events: If the auditing subsystem has been properly configured and is enabled, the `mkserver` command will generate the following audit record (event) every time the command is executed:

<table>
<thead>
<tr>
<th>Event</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>SRC_Addserver</td>
<td>Lists in an audit log subsystems that have been added and the entire Object Data Management record.</td>
</tr>
</tbody>
</table>

See "Setting up Auditing" in *AIX 5L Version 5.2 Security Guide* for more details about how to properly select and group audit events and how to configure audit event data collection.

**Examples**
To add a subserver definition, enter:
```
mkserver -s srctest -t tester -c 1234
```

This adds a subserver definition to the **Subserver Type** object class, with an owning subsystem of srctest and a subserver code point of 1234.

**Files**
```
/etc/objrepos/SRCsubsys
```
Specifies the SRC Subsystem Configuration object class.
/etc/objrepos/SRCsubsvr

Specifies the SRC Subserver Configuration object class.

Related Information


Auditing Overview in the AIX 5L Version 5.2 Security Guide.

System Resource Controller (SRC) Overview for Programmers in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

Defining Your Subsystem to the SRC in AIX 5L Version 5.2 General Programming Concepts: Writing and Debugging Programs.

---

**mkslave Command**

**Purpose**
Invokes the [ypinit] command to retrieve maps from an NIS master server and starts the [ypserv] daemon to configure a slave server.

**Syntax**

```
```

**Description**
The `mkslave` command invokes the `ypinit` command to retrieve maps from the master server you specify on the command line. The `ypserv` daemon starts after the `ypinit` command has completed successfully. Use the `Master` parameter to specify the host name of the master server. The master server specified should already be configured and running.

You can use the Network application in Web-based System Manager (wsm) to change network characteristics. You could also use the System Management Interface Tool (SMIT) `smit mkslave` fast path to run this command.

**Flags**

- `-C`  Invokes the `ypinit` command with the `-n` flag. The `mkslave` command continues on errors. This flag is the default.
- `-c`  Stops execution when errors occur.
- `-O`  Overwrites any maps that exist in the domain.
- `-o`  Prevents the overwriting of maps that exist in the domain. This flag is the default.
- `-I`  Invokes the `ypinit` command immediately but does not start the `ypserv` daemon until the next system reboot.
- `-N`  Invokes the `ypinit` command and starts the `ypserv` daemon.
- `-B`  Invokes the `ypinit` command, starts the `ypserv` daemon and configures the `ypserv` to start at system reboot. This flag is the default.

**Examples**
To invoke the `ypinit` command so that the master server `host2` will be contacted for maps, enter:

```
mkslave -O host42
```
This command will overwrite the current maps.

**Files**
/var/yp/DomainName directory Contains the NIS maps for the NIS domain.

**Related Information**
The `chmaster` command, `mkclient` command, `rmyp` command, `smit` command, `ypinit` command.
The `ypbind` daemon, `yppasswdd` daemon, `ypserv` daemon, `ypupdated` daemon.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


NIS Reference

**mkssys Command**

**Purpose**
Adds a subsystem definition to the subsystem object class.

**Syntax**
```
```

**Description**
The `mkssys` command adds a new subsystem definition to the subsystem object class. If no flags are chosen after the `-p`, `-s`, and `-u` flags have been specified, the defaults are `-e /dev/console`, `-i`, `/dev/console`, `-o/dev/console`, `-Q`, `-Q`, `-K`, `-E 20`, and `-w 20`.

_Note:_ Any auditing performed by the System Resource Controller (SRC) when actions are taken for the subsystem is logged against the login ID of the user who created the subsystem by using the `mkssys` command. For example, if you are logged in with root user authority, the subsystem is added with root user authority as the audit account.

**Flags**

- `-a Arguments` Specifies any arguments that must be passed to the command, started as the subsystem. These `Arguments` variables are passed by the SRC to the subsystem according to the same rules used by the shell. For example, quoted strings are passed as a single argument, and blanks outside a quoted string delimit arguments. Single and double quotes can be used.
-d Specifies that inactive subsystems are displayed when the [lsrc] command 
  (status all) request is made. By default, if the -D and -d flags are not present, the -d 
  flag is used.

-D Specifies that inactive subsystems are not displayed when status-all or status-group 
  requests are made.

-e StandardError Specifies where the subsystem StandardError data is placed. If the -e flag is not 
  specified, the /dev/console file is used for standard error.

-E Nice Changes the execution priority of the subsystem. Valid values are 0 through 39 
  (ordinary Nice variables are mapped to all positive numbers). If the -E flag is not 
  present, the subsystem priority defaults to 20. Values between 0 and 19 are 
  reserved for users with root authority.

-f StopForce Specifies the signal sent to the subsystem when a forced stop of the subsystem is 
  requested. Use only when the subsystem uses signals. The mkssys command is 
  unsuccessful if the StopForce parameter is not a valid signal.

-G Group Specifies that the subsystem belongs to the Group specified, and that the 
  subsystem responds to all group actions on the Group.

-i StandardInput Specifies where the subsystem standard input is routed. This field is ignored when 
  the subsystem uses sockets communication. If the -i flag is not specified, by default 
  the /dev/console file is used for standard input.

-I MessageQueue Specifies that the subsystem uses message queues as the communication method. 
  The MessageQueue variable specifies the message queue key for creating the 
  message queue for the subsystem. Use the ftok subroutine with the subsystem 
  path name as input to generate a unique key.

-K Specifies that the subsystem uses sockets as its communication method. If a 
  communication method is not specified, sockets communication is used by default.

-m MessageMType Specifies the message type key the subsystem expects on packets sent to the 
  subsystem by the SRC. Use only when the subsystem uses message queues 
  communication.

-n StopNormal Specifies the signal sent to the subsystem when a normal stop of the subsystem is 
  requested. Use only when the subsystem uses signals communication. The 
  mkssys command is unsuccessful if the StopNormal variable is not a valid signal.

-o StandardOutput Specifies where the subsystem standard output is placed. If the -o flag is not 
  specified, by default the /dev/console file is used for standard out.

-O Specifies that the subsystem is not restarted if it stops abnormally. The default is no 
  restart.

-p Path Specifies the absolute path to the subsystem executable program.

-q Specifies that the subsystem can have multiple instances running at the same time.

-Q Specifies that multiple instances of the subsystem are not allowed to run at the 
  same time and the subsystem is not to share the same interprocess communication 
  (IPC) queue. If the -Q flag is not specified, the -Q flag is the default.

-R Specifies that the subsystem is restarted if the subsystem stops abnormally.

-s Subsystem Specifies a name that uniquely identifies the subsystem. The mkssys command is 
  unsuccessful if the subsystem name is already known in the subsystem object 
  class.

-S Specifies that the subsystem uses the signals communication method. You cannot 
  define subservers for a subsystem name when your communication method is 
  signals.

-t Synonym Specifies an alternate name for the subsystem. The mkssys command is 
  unsuccessful if the synonym name is already known in the subsystem object class.

-u UserID Specifies the user ID for the subsystem. The UserID that creates the subsystem is 
  used for security auditing of that subsystem.

-w Wait Specifies the time, in seconds, allowed to elapse between a stop cancel 
  (SIGTERM) signal and a subsequent SIGKILL signal. Also used as the time limit for 
  restart actions. If the subsystem stops abnormally more than twice in the time limit 
  specified by the Wait value, the subsystem is not automatically restarted. By 
  default, if the -w flag is not present, the wait time default is 20 seconds.
Security

Auditing Events: If the auditing subsystem has been properly configured and is enabled, the `mkssys` command will generate the following audit record (event) every time the command is executed:

<table>
<thead>
<tr>
<th>Event</th>
<th>Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>SRC_Addsys</td>
<td>Lists in an audit log the name of the subsystem being added to the Object Data Manager (ODM) database and the entire ODM record.</td>
</tr>
</tbody>
</table>

See “Setting up Auditing” in *AIX 5L Version 5.2 Security Guide* for details about selecting and grouping audit events, and configuring audit event data collection.

Examples

1. To add a subsystem that uses sockets as its communication type, type the following:

   ```bash
   mkssys -s srctest -p /usr/lpp/srctest/srctest -u 0 -K
   ```

   This adds a subsystem definition to the subsystem object class, with a communication type of sockets, a user ID of 0 (root), and a subsystem name of `srctest`.

2. To add a subsystem that uses message queues as its communication type, type the following:

   ```bash
   mkssys -s srctest -p /usr/lpp/srctest/srctest -u 0 -I 123456 > -m 789
   ```

   This adds a subsystem definition to the subsystem object class, with a communication type of message queues, a message queue key of 123456, and a subsystem message type of 789.

3. To add a subsystem that uses signals as its communication type, type:

   ```bash
   mkssys -s srctest -p /usr/lpp/srctest/srctest -u 0 -S -n 30 > -f 31
   ```

   This adds a subsystem definition to the subsystem object class, with a communication type of signals, a stop normal signal of 30, a stop force signal of 31.

4. To add a subsystem that uses sockets as its communication type and is always passed an argument, type:

   ```bash
   mkssys -s srctest -p /usr/lpp/srctest/srctest -u 0 -a "-x"
   ```

   This adds a subsystem definition to the subsystem object class with a communication type of sockets and a command argument of "-x".

Files

/etc/objrepos/SRCsubsys  
Specifies the SRC Subsystem Configuration object class.

/dev/SRC  
Specifies the AF_UNIX domain in the `socket.h` file.

/dev/.SRC-unix  
Specifies the location for temporary file sockets.

Related Information

The `auditpr` command, `chssys` command, `lssrc` command, `refresh` command, `rmssys` command, `startsrc` command, `stopsrc` command, `traceson` command, and `tracesoff` command.


Auditing Overview in *AIX 5L Version 5.2 Security Guide*.
**mkstr Command**

**Purpose**
Creates an error message file.

**Syntax**
mkstr \[ - ] MessageFile Prefix File ...

**Description**
The `mkstr` command creates a file of error messages that can be removed from a single C source file or from multiple source files. Its use can reduce the size of programs that contain many error diagnostics and reduce system overhead in running such programs, because error messages are then not constantly swapped in and out of the source files.

The `mkstr` command processes each file specified by the `File` parameter, placing a massaged version of the file in a file having the name specified by the `Prefix` parameter followed by the original name.

To process the error messages in the source to the file specified by the `MessageFile` parameter, the `mkstr` command keys on the string `"error("` in the input stream. The string, starting at the `""` (two double quotation marks), is placed in the message file and followed by a null character and a new-line character. The null character terminates the message so it can be easily used when retrieved. The new-line character makes it possible to see the contents of the error message file by using the `cat` command.

The massaged copy of the input file then contains an `lseek` pointer into the file, which can be used to retrieve the message to its appropriate source file, as shown in the following example:

```c
char efilname[] = "/usr/lib/pistrings";
int efil = -1;

error(a1, a2, a3, a4)
{
    char buf[256];
    if (efil < 0) {
        efil = open(efilname, 0);
        if (efil < 0) {
            oops:
                perror(efilname);
                exit(1);
        }
    }
    if (lseek(efil,(long)a1,0) <0||
        read(efil,buf, 256)<= 0)
        goto oops;
    printf(buf, a2, a3, a4);
}
```

**Flags**
- The optional - (minus sign) causes the error messages to be placed at the end of the `MessageFile` for recompiling part of a large `mkstr` program.
Examples

1. To put the error messages from the current directory C source files into the file pistrings and to put processed copies of the source for these files into file names prefixed by xx, enter:
   mkstr pistrings xx *.c

2. To append the error messages from an additional source file into the file pistrings, enter:
   mkstr -pistrings xx newfile.c

Files

/usr/ccs/bin/mkstr          Contains the mkstr command.

Related Information

The cat command. xstr command.

The lseek subroutine.

mksysb Command

Purpose

Creates an installable image of the root volume group either in a file or onto a bootable tape.

Syntax

mksysb [ b Number ] [ e ] [ h ] [ m ] [ p ] [ v ] [ V ] [ X ] Device | File

Description

Attention:  Running the mkszfile or mksysb commands with the LC_All environment variable variable set (especially to a non-C value) can cause unexpected system behavior such as a mixture of character sets in outputs. To resolve the problem, unset the LC_ALL variable and restart the program.

The mksysb command creates a backup of the operating system (that is, the root volume group). You can use this backup to reinstall a system to its original state after it has been corrupted. If you create the backup on tape, the tape is bootable and includes the installation programs needed to install from the backup.

The file-system image is in backup-file format. The tape format includes a boot image, a bosinstall image, and an empty table of contents followed by the system backup (root volume group) image. The root volume group image is in backup-file format, starting with the data files and then any optional map files.

One of the data files mksysb uses is the /bosinst.data file. If a /bosinst.data file doesn't exist, /var/adm/ras/bosinst.data is copied to / (root). In AIX 4.3.3 and later versions, mksysb always updates the target_disk_data stanzas in bosinst.data to match the disks currently in the root volume group of the system where the mksysb command is running.

If you are using a customized /bosinst.data file and do not want the target_disk_data stanzas updated, you must create the file /save_bosinst.data_file. The mksysb command does not update /bosinst.data if the /save_bosinst.data_file exists.

Notes:

1. The image the mksysb command creates does not include data on raw devices or in user-defined paging spaces.
2. If you are using a system with a remote-mounted /usr file system, you cannot reinstall your system from a backup image.

3. The mksysb command may not restore all device configurations for special features, such as /dev/netbios and some device drivers not shipped with the product.

4. Some rspc systems for AIX 5.1 and earlier do not support booting from tape. When you make a bootable mksysb image on an rspc system for AIX 5.1 and earlier that does not support booting from tape, the mksysb command issues a warning indicating that the tape will not be bootable. You can install a mksysb image from a system that does not support booting from tape by booting from a CD and entering maintenance mode. In maintenance mode you will be able to install the system backup from tape.

To create a backup of the operating system to CD, please refer to the mkcd command.

Flags

-b Number Specifies the number of 512-byte blocks to write in a single output operation. When the backup command writes to tape devices, the default is 100 for backups by name.

The write size is the number of blocks multiplied by the block size. The default write size for the backup command writing to tape devices is 51200 (100 * 512) for backups by name. The write size must be an even multiple of the tape’s physical block size.

-e Excludes files listed in the /etc/exclude.rootvg file from being backed up. The rules for exclusion follow the pattern matching rules of the grep command.

If you want to exclude certain files from the backup, create the /etc/exclude.rootvg file, with an ASCII editor, and enter the patterns of file names that you do not want included in your system backup image. The patterns in this file are input to the pattern matching conventions of the grep command to determine which files will be excluded from the backup. If you want to exclude files listed in the /etc/exclude.rootvg file, select the Exclude Files field and press the Tab key once to change the default value to yes.

For example, to exclude all the contents of the directory called scratch, edit the exclude file to read as follows:

```
scratch/
```

For example, to exclude the contents of the directory called /tmp, and avoid excluding any other directories that have /tmp in the pathname, edit the exclude file to read as follows:

```
^./tmp/
```

All files are backed up relative to . (current working directory). To exclude any file or directory for which it is important to have the search match the string at the beginning of the line, use ^ (caret character) as the first character in the search string, followed by . (dot character), followed by the filename or directory to be excluded.

If the filename or directory being excluded is a substring of another filename or directory, use ^.

(caret character followed by dot character) to indicate that the search should begin at the beginning of the line and/or use $ (dollar sign character) to indicate that the search should end at the end of the line.

-i Calls the mkszfile command, which generates the /image.data file. The /image.data file contains information on volume groups, logical volumes, file systems, paging space, and physical volumes. This information is included in the backup for future use by the installation process.

Note: Before running the mkszfile command, ensure that enough space is available in the /tmp file to store a boot image. This space is needed during both backup and installation. To determine the amount of space needed in the /tmp file, issue the following command:

```
bosboot -q -a -d device
```

If you use the -X flag with the mksysb command, you do not need to run the bosboot command to determine the amount of space needed in the /tmp file.
-m  Calls the `mkszfile` command, with the `-m` flag to generate map files.
   **Note:** The use of the `-m` flag causes the functions of the `-i` flag to be executed also.

-p  Disables software packing of the files as they are backed up. Some tape drives use their own packing or compression algorithms.

-v  Verbose mode. Lists files as they are backed up.

-V  Verifies a tape backup. This flag causes `mksysb` to verify the file header of each file on the backup tape and report any read errors as they occur.

-X  Specifies to automatically expand the `/tmp` file system if necessary. The `/tmp` file system may need to be extended to make room for the boot image when creating a bootable backup to tape.

### Parameters

*Device | File* Specifies the name of the device or file.

### Examples

1. **To generate a system backup and create an `/image.data` file (generated by the `mkszfile` command) to a tape device named `/dev/rmt0**, type:
   ```
mksysb -i /dev/rmt0
   ```

2. **To generate a system backup and create an `/image.data` file with map files (generated by the `mkszfile` command) to a tape device named `/dev/rmt1**, type:
   ```
mksysb -m /dev/rmt1
   ```

3. **To generate a system backup with a new `/image.data` file, but exclude the files in directory `/home/user1/tmp`, create the file `/etc/exclude.rootvg` containing the line `/home/user1/tmp/`, and type:
   ```
mksysb -i -e /dev/rmt1
   ```
   This command will backup the `/home/user1/tmp` directory but not the files it contains.

4. **To generate a system backup file named `/mksysb_images/node1` and a new `/image.data` file for that image, type:**
   ```
mksysb -i /userimage/node1
   ```
   **Note:** This file will not be bootable and can only be installed using Network Installation Management (NIM).

5. **To generate a system backup on the tape in `/dev/rmt0`, and then verify the readability of file headers, enter:**
   ```
mksysb /dev/rmt0 -V
   ```

### Files

*`/usr/bin/mksysb`* Contains the `mksysb` command.

### Related Information

The `backup` command, `bosboot` command, `mkcd` command, `mkszfile` command.

The `/image.data` file.

A procedure to verify the mksysb backup can be found in the article "Verifying a System Backup" in the `AIX 5L Version 5.2 Installation Guide and Reference`. 

590  Commands Reference, Volume 3
mkszfile Command

Purpose
Saves the system state for reinstallation on the current system or another system.

Syntax
mkszfile [-X] [-m]

Description
Attention: Running the mkszfile or mksysb commands with the LC_All environment variable set (especially to a non-C value) can cause unexpected system behavior such as a mixture of character sets in outputs. To resolve the problem, unset the LC_ALL variable and restart the program.

Attention: The mkszfile command overwrites an existing /image.data file with new information.

The mkszfile command saves the system state for reinstallation on the current system or on another system. The information saved includes the following:
- System installation information
- Logical volume information for the root volume group
- File system information.

The saved information allows the bosinstall routine to recreate the logical volume information as it existed before the backup.

The mkszfile command creates the /image.data file. The contents of this file are defined by the system in which the image was created. The user can edit the /image.data file before calling the mksysb command. The mksysb command, in turn, only backs up the file systems specified in the /image.data file, which reflects the requirements of the rootvg file system.

All the saved information is obtained using list commands. The commands are listed in the /image.data file as comments for the user’s reference when editing this file.

Files on tape cannot be changed. However, in order to override the data files on the tape, the user can create a diskette with the desired files.

The mkszfile command checks to be sure there is at least 8MB of free space available in the /tmp file system for the boot image.

Note: Before running the mkszfile command, ensure that enough space is available in the /tmp file to store a boot image. This space is needed during both backup and installation. To determine the amount of space needed in the /tmp file, issue the following command:
Flags

-m  Creates map files that specify the mapping of the logical-to-physical partitions for each logical volume in the volume group. This mapping can be used to allocate the same logical-to-physical mapping when the image is restored. The map file locations are stored in the MAPFILE field in the /image.data file for each logical volume. Sufficient space would exist in the /tmp file system for map creation because the installation routines place the maps in the /tmp file system before issuing the mklv command.

For example, for the hd7 logical volume, the location of the map file is /tmp/vgdata/rootvg/hd7.map. The MAPFILE field in the /image.data file for the hd7 logical volume is under the entry MAPFILE=/tmp/vgdata/rootvg/hd7.map.

The map files in the backup image are copied after the /bosinst.data and /image.data files.

-X  Expands /tmp if needed.

Files

/usr/bin/mkszfile  Contains the mkszfile command.

Related Information

The mksysb command.

The /image.data file.

mktcpip Command

Purpose
Sets the required values for starting TCP/IP on a host.

Syntax
mktcpip { -S Interface | -H HostName | -a Address | -i Interface | -s | -m SubnetMask | -r RingSpeed | -t CableType | -g Gateway | -n NameServerAddress | -d Domain | -c Subchannel | -D Destination }}

Description

The mktcpip command sets the required minimal values required for using TCP/IP on a host machine. These values are written to the configuration database. The basic functions of the mktcpip command include:

- Setting the host name in both the configuration database and the running machine.
- Setting the IP address of the interface in the configuration database.
- Making entries in the /etc/hosts file for the host name and IP address.
- Setting the domain name and IP address of the nameserver, if applicable.
- Setting the subnetwork mask, if applicable.
- Adding a static route to both the configuration database and the running machine, if applicable.
- Starting the specified TCP/IP daemons.

You can use the Network application in Web-based System Manager (wsm) to change network characteristics. You could also use the System Management Interface Tool (SMIT) smit mktcpip fast path to run this command.
### Flags

**-a Address**
Sets the Internet address of the host. Specify the address in dotted decimal notation. Each network interface on the host should have a unique Internet address. The following is the standard format for setting the Internet address:

127.10.31.2

**-c Subchannel**
Specifies the subchannel address for a System/370 channel adapter.

**-D Destination**
Sets the destination address for a static route. Specify the address in dotted decimal notation. The following is the standard format for setting the destination address for a static route:

192.9.52.1

**-d Domain**
Specifies the domain name of the name server the host should use for name resolution, if any. The domain name should be in the following format:

subdomain.subdomain.rootdomain

**-g Gateway**
Sets the gateway address for a static route. Specify the address in dotted decimal notation. The following is the standard format for setting the gateway address for a static route:

192.9.52.0

**-h Hostname**
Sets the name of the host. If using a domain naming system, the domain and any subdomains must be specified. The following is the standard format for setting the host name:

hostname

The following is the standard format for setting the host name in a domain naming system:

hostname.subdomain.subdomain.rootdomain

**-i Interface**
Specifies a particular network interface, for example:

tr0

**-m SubnetMask**
Specifies the mask the gateway should use in determining the appropriate subnetwork for routing. The subnet mask is a set of 4 bytes, as in the Internet address. The subnet mask consists of high bits (1s) corresponding to the bit positions of the network and subnetwork address, and low bits (0s) corresponding to the bit positions of the host address.

**-n NameserverAddress**
Specifies the Internet address of the name server the host uses for name resolution, if applicable. The address should be entered in dotted decimal notation, as follows:

127.1.0.1

**-r RingSpeed**
Specifies the ring speed for a token-ring adapter. Valid values for the **RingSpeed** variable are either 4- or 16-Mbps.

**-S Interface**
Retrieves information for System Management Interface Tool (SMIT) display.

**-s**
Starts the TCP/IP daemons.

**-t CableType**
Specifies cable size for Standard Ethernet or IEEE 802.3 Ethernet networks. Valid values for the **CableType** variable are **dix** for thick cable, **bnc** for thin cable, or **N/A** for Not Applicable. The **-t CableType** flag should be used only for Standard Ethernet (en) and IEEE 802.3 Ethernet (et) interfaces.

### Examples
To set the required values for starting TCP/IP enter:

```
mktcpip -h fred.austin.century.com -a 192.9.200.4 -i en0 \ -n 192.9.200.1 -d austin.century.com -s
```

### Related Information
The [hostname](#) command, [hostent](#) command.

The [resolv.conf](#) file format.
**mktun Command**

**Purpose**
Activates tunnel(s).

**Syntax**
mktun [-v 4|6] [-t tid_list] [-i] [-l]

**Description**
Use the mktun command to activate tunnel(s). For IBM tunnels, this command initiates the security protocol exchanges between the local and the destination host.

**Flags**
- `-i` Initiation flag. If the -i flag is not used, all the tunnels in the tunnel database (or those listed with the -t flag) will be activated. If the -i flag is used, only the tunnels whose tunnel definitions in the tunnel database with the status of "active" will be activated.
- `-l` If the -l flag is specified, manual tunnels will be activated.
- `-t` If the -t flag is specified, only the tunnel(s) that follows this flag will be activated. If the -t flag is not used, all tunnel(s) currently defined in the tunnel database will be activated. The tid_list can be a single tunnel ID or a sequence of tunnel IDs separated by ";" or "-" (1, 3, 5-7).
- `-v` The IP version of the tunnels to be activated. The value of 4 specifies IP version 4 tunnels. The value of 6 specifies IP version 6 tunnels. If the -v flag is not used, all tunnels for IP version 4 and IP version 6 will be activated.

**Related Information**
The **chtun** command, **exptun** command, **gentun** command, **imptun** command, **lstun** command, **rmtun** command.

**mkuser Command**

**Purpose**
Creates a new user account.

**Syntax**
mkuser [-R load_module ] [-a] [ Attribute=Value ... ] Name

**Description**
The **mkuser** command creates a new user account. The Name parameter must be a unique 8-byte or less string. You cannot use the **ALL** or **default** keywords in the user name. By default, the **mkuser** command creates a standard user account. To create an administrative user account, specify the -a flag.
To create a user with an alternate Identification and Authentication (I&A) mechanism, the -R flag can be used to specify the I&A load module used to create the user. Users created without the -R flag are created locally. Load modules are defined in the /usr/lib/security/method.cfg file.

The mkuser command does not create password information for a user. It initializes the password field with an * (asterisk). Later, this field is set with the passwd or pwdadm command. New accounts are disabled until the passwd or pwdadm commands are used to add authentication information to the /etc/security/passwd file.

You can use the Users application in Web-based System Manager to change user characteristics. You could also use the System Management Interface Tool (SMIT) smit mkuser fast path to run this command.

Restrictions on Creating User Names
To prevent login inconsistencies, you should avoid composing user names entirely of uppercase alphabetic characters. While the mkuser command supports multi-byte user names, it is recommended that you restrict user names to characters with the POSIX portable filename character set.

To ensure that your user database remains uncorrupted, you must be careful when naming users. User names must not begin with a - (dash), + (plus sign), @ (at sign), or ~ (tilde). You cannot use the keywords ALL or default in a user name. Additionally, do not use any of the following characters within a user-name string:

- Colon
- Double quote
- Pound sign
- Comma
- Equal sign
- Back slash
- Slash
- Question mark
- Single quote
- Back quote

Finally, the Name parameter cannot contain any space, tab, or new-line characters.

Flags

- **-a** Specifies that the user is an administrator. Only the root user can use this flag or alter the attributes of an administrative user.
- **-R load_module** Specifies the loadable I&A module used to create the user.

Parameters

**Attribute=Value** Initializes a user attribute. Refer to the chuser command for the valid attributes and values.

**Name** Specifies a unique 8-byte or less string.

Security
Access Control: This command should grant execute (x) access only to the root user and members of the security group. This command should be installed as a program in the trusted computing base (TCB). The command should be owned by the root user with the setuid (SUID) bit set.
Limitations
Creating a user may not be supported by all loadable I&A modules. If the loadable I&A module does not support creating a user, an error is reported.

Examples
1. To create the davis user account with the default values in the /usr/lib/security/mkuser.default file, type:
   mkuser davis
2. To create the davis account with davis as an administrator, type:
   mkuser -a davis
   Only the root user or users with the UserAdmin authorization can create davis as an administrative user.
3. To create the davis user account and set the su attribute to a value of false, type:
   mkuser su=false davis
4. To create the davis user account that is identified and authenticated through the LDAP load module, type:
   mkuser -R LDAP davis

Files
/usr/bin/mkuser
/usr/lib/security/mkuser.default
/etc/passwd
/etc/security/user
/etc/security/user.roles
/etc/security/limits
/etc/security/environ
/etc/group
/etc/security/group
/etc/security/.ids
Contains the mkuser command.
Contains the default values for new users.
Contains the basic attributes of users.
Contains the extended attributes of users.
Contains the administrative role attributes of users.
Contains password information.
Defines resource quotas and limits for each user.
Contains the environment attributes of users.
Contains the basic attributes of groups.
Contains the extended attributes of groups.
Contains standard and administrative user IDs and group IDs.
Related Information

The `chfn` command, `chgroup` command, `chgrpmem` command, `chsh` command, `chuser` command, `lsgrp` command, `lsuser` command, `mkgrou` command, `mkuser.sys` command, `passwd` command, `pwdadm` command, `rmgrp` command, `rmuser` command, `setgroups` command, `setsenv` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the AIX 5L Version 5.2 Security Guide.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

For more information about administrative roles, refer to Administrative Roles Overview in AIX 5L Version 5.2 Security Guide.

mkuser.sys Command

Purpose
Customizes a new user account.

Syntax
```
mkuser.sys Directory User Group
```

Description
The `mkuser.sys` command customizes the new user account specified by the `User` parameter. The `mkuser` command calls the `mkuser.sys` command after it has created and initialized the new account.

The program as shipped creates the home directory specified by the `Directory` parameter, with the owner specified by the `User` parameter, the primary group specified by the `Group` parameter, and a copy of the appropriate profile for the user’s shell. The shipped program can be replaced at installation by another program to customize local new-user creation. The installation-specific program should adhere to the error conventions of the supplied program.

Security
Access Control: This command should grant read (r), write (w), and execute (x) access for the root user and members of the security group so the `mkuser` command can execute the program.

Files Accessed:

<table>
<thead>
<tr>
<th>Mode</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>r</td>
<td>/etc/passwd</td>
</tr>
<tr>
<td>r</td>
<td>/etc/security/user</td>
</tr>
</tbody>
</table>

Files

```
/usr/lib/security/mkuser.sys
```
Contains the `mkuser.sys` command.

Related Information

The `mkuser` command.

For more information about the identification and authentication of users, discretionary access control, the trusted computing base, and auditing, refer to the AIX 5L Version 5.2 Security Guide.
mkvg Command

Purpose
Creates a volume group.

Syntax
mkvg [ -d MaximumPhysicalVolumes ] [ -B ] [ -C ] [ -G ] [ -f ] [ -i ] [ -c ] [ -x ] [ -L LTGSize ] [ -m ]
MaxPvSize ] [ -n ] [ -s Size ] [ -r factor ] [ -V MajorNumber ] [ -y VolumeGroup ] [ PhysicalVolume ... ]

Description
The mkvg command creates a new volume group, using the physical volumes represented by the
PhysicalVolume parameter. After creating the volume group, the mkvg command automatically varies on
the new volume group using the varyonvg command. The exception to this fact is when the volume group
is created with the -C flag. When the volume group is successfully created, the volume group will not be
varied on automatically. Instead, the user must manually varyon the volume group.

The mkvg command by default creates a volume group that can accommodate 255 logical volumes and
32 physical volumes (disks). These limits can be extended to 512 logical volumes and 128 physical
volumes by specifying the -B flag.

The mkvg command will attempt to determine a proper partition size if none is specified on the command
line.

MAXPVS 32 (128 if -B flag is used)

Notes:
1. The physical volume is checked to verify that it is not already in another volume group. If the system
believes the physical volume belongs to a volume group that is varied on, it exits. But if the system
detects a description area from a volume group that is not varied on, it prompts the user for
confirmation in continuing with the command. The previous contents of the physical volume are lost, so
the user must be cautious when using the override function.
2. To use this command, you must either have root user authority or be a member of the system group.
3. For disks greater than 4GB, make sure that the Physical Partition Size (-s) is set to a large enough
value so that 1016 physical partitions per PV limit is not violated. The default value, 4MB, along with
the default maximum number of physical partitions (1016), a disk up to 4GB can only be
accommodated. Or use large enough factor value (-t) such that (factor x 1016) is greater than the
number of partitions that would need to be created with given partition size and disk size. For example,
a partition size of at least 16MB would be needed to create a volume group with a 10GB disk. Or with
at factor size of 2, a smaller partition size of 8MB can be used. However, this limits the total number of
disks that can be added to the volume group. If a factor value is used, a maximum of MAXPVS/factor
disks can be included in the volume group.
4. After AIX 4.1, whenever you create a volume group, the operating system automatically does a varyon.
However if you create a volume group with the -C flag, the system will not autovaryon the volume
group at the end of the Concurrent Capable volume group creation. Instead, the mkvg command
notifies you to manually varyonvg the volume group in either non-concurrent or concurrent mode.
5. This command will fail to add a disk to the volume group if the disk indicates that it is managed by a
third party volume manager. To override and clear the disk of the third party volume manager use chpv
-C HDiskName.
6. On AIX 5.2 and later only Enhanced Concurrent Capable volume groups will be created when the -c or
-C flags are specified.
Because the VGDA space has been increased substantially, every VGDA update operation (creating a logical volume, changing a logical volume, adding a physical volume, and so on) might take considerably longer to run.

You can use the Volumes application in Web-based System Manager to change volume characteristics. You could also use the System Management Interface Tool (SMIT) `smit mkvg` fast path to run this command.

**Flags**

- **-B**
  Creates a big vg format `volume group`. This can accommodate up to 128 physical volumes and 512 logical volumes.
  **Note:** Because the VGDA space has been increased substantially, every VGDA update operation (creating a logical volume, changing a logical volume, adding a physical volume, and so on) may take considerably longer to run.

- **-c**
  Same as -C flag. On AIX 5.2 and later only Enhanced Concurrent Capable volume groups will be created.

- **-C**
  Creates an Enhanced Concurrent Capable volume group. Only use the -C flag with the HACMP ES. It is not usable on volume groups and systems not using the HACMP ES product.

  Use this flag to create an Enhanced Concurrent Capable volume group.

  **Notes:**
  1. Enhanced Concurrent volume groups use Group Services. Group Services ships with HACMP ES and must be configured prior to activating a volume group in this mode.
  2. Only Enhanced Concurrent Capable volume groups are supported when running with a 64 bit kernel. Concurrent Capable volume groups are not supported when running with a 64 bit kernel.

- **-d MaximumPhysicalVolumes**
  This flag is ignored.

- **-f**
  Forces the volume group to be created on the specified physical volume unless the physical volume is part of another volume group in the Device Configuration Database or a volume group that is active.

- **-G**
  Same as -B flag.

- **-i**
  Reads the `PhysicalVolume` parameter from standard input.

- **-L LTGSize**
  Sets the logical track group size, in number of kilobytes, of the volume group.
  The value of the `LTGSize` parameter must be 128, 256, 512, or 1024. In addition, it should be less than or equal to the maximum transfer size of all disks in the volume group. The default size is 128 kilobytes.

- **-m MaxPvSize**
  This flag is ignored.

- **-n**
  Specifies that the volume group is not automatically available during a system restart. The default value activates the volume group automatically.

- **-s Size**
  Sets the number of megabytes in each physical partition, where the `Size` variable is expressed in units of megabytes from 1 through 1024. The `Size` variable must be equal to a power of 2 (example 1, 2, 4, 8). The default value will be the lowest value to remain within the limit of 1016 physical partitions.

- **-t factor**
  Changes the limit of the number of physical partitions per physical volume, specified by `factor`. The `factor` should be between 1 and 16 for 32 disk volume groups and 1 and 64 for 128 disk volume groups. The maximum number of physical partitions per physical volume for this volume group changes to `factor` x 1016. The default will be the lowest value to remain within the physical partition limit of `factor` x 1016.

- **-V MajorNumber**
  Specifies the major number of the volume group that is created.

- **-x**
  This flag is ignored. On AIX 5.2 and later only Enhanced Concurrent Capable volume groups will be created.
-y VolumeGroup Specifies the volume group name rather than having the name generated automatically. Volume group names must be unique system wide and can range from 1 to 15 characters. The name cannot begin with a prefix already defined in the PdDv class in the Device Configuration database for other devices. The volume group name created is sent to standard output.

The volume group name can only contain the following characters: “A” through “Z,” “a” through “z,” “0” through “9,” or “_” (the underscore), “-” (the minus sign), or “.” (the period). All other characters are considered invalid.

Examples
To create a volume group that contains three physical volumes with partition size set to 1 megabyte, type:

```
mkvg -s 1 hdisk3 hdisk5 hdisk6
```

The volume group is created with an automatically generated name, which is displayed and available at system restart time.

```
mkvg -s 2 -t 2 -y newvg hdisk1
```

The volume group newvg is created with a physical partition size of 2MB and maximum number of physical partitions per physical volume of 2032. The above configuration means that the size of hdisk1 could not be larger than 4064MB (2032*2)

Files
/etc Directory where the mkvg command resides.
/tmp Directory where the temporary files are stored while the command is running.
/dev Directory where the character device entry for the volume group is created.

Related Information
The chvg command, lsvg command, varyonvg command.


For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mkvgdata Command

Purpose
Creates a file containing information about a volume group for use by the savevg and restvg commands.

Syntax
```
mkvgdata [-X] [-m] VGName
```
Description
The **mkvgdata** command creates a file containing information about a volume group for use by the **savevg** and **restvg** commands. The information includes the list of logical volumes, file systems and their sizes, and the volume group name. One of the following files is created, depending on the type of volume group:

```
/image.data
```

Created for information about the root volume group (**rootvg**). The **savevg** command uses this file to create a backup image that can be used by the **bosinstall** routine to reinstall the volume group to the current system or to a new system. The **mkvgdata** command overwrites this file if it already exists. The **/image.data** file is located in the */ directory.

```
vname.data
```

Created for information about a user volume group. The **vname** variable reflects the name of the volume group. The **savevg** command uses this file to create a backup image that can be used by the **restvg** command to reinstall the user volume group. The **mkvgdata** command overwrites this file if it already exists. The **vname.data** file is located in the */tmp/vgdata/vname directory, where **vname** is the volume group name.

The information in either of these files can be edited by the user before issuing the **savevg** command.

Flag

**-m**
Creates map files that specify the mapping of the logical-to-physical partitions for each logical volume in the volume group. This mapping can be used to allocate the same logical-to-physical mapping when the image is restored. The map file locations are stored in the MAPFILE field in the **/image.data** file for each logical volume. Sufficient space would exist in the */tmp file system for map creation because the installation routines place the maps in the */tmp file system before issuing the **mklv** command.

For example, for the **hd7** logical volume, the location of the map file is */tmp/vgdata/rootvg/hd7.map*. The MAPFILE field in the **/image.data** file for the **hd7** logical volume is under the entry MAPFILE=/tmp/vgdata/rootvg/hd7.map.*

The map files in the backup image are copied after the **image.data** or **vname.data** files.

**-X**
Expands */tmp* if needed.

**vname**
Name of volume group to backup.

Files

```
/image.data
```

Created when the volume group is **rootvg**.

```
/tmp/vgdata/vnamelvname.data
```

Created when the volume group is not **rootvg** and where **vname** is the name of the volume group.

Related Information

The **mkszfile** command, **restvg** command, **savevg** command.

**mkvirpr Command**

**Purpose**

Makes a virtual printer.

**Syntax**

```
mkvirpr [-A AttachmentType] [d] QueueDevice [n] Device [q] PrintQueue [-S DataStream [-d] PrinterType [-T]
```

```
mkvirpr -A AttachmentType
```

Alphabetical Listing of Commands 601
Description

The mkvirprt command creates a virtual printer definition and assigns it to the specified print queue and queue device. A virtual printer definition is a set of attribute values that describe a particular data stream for a particular printer. Before a print job can be queued with the enq, qprt, command, or lp command, a virtual printer definition must be created for the printer’s print queue and queue device.

Printers that support only one printer data stream, such as the 4201-3 Proprinter III, need only one virtual printer defined. Printers that support multiple printer data streams, such as the IBM 4216-31 Page Printer II, need a virtual printer defined for each data stream.

To create a virtual printer definition for a printer attached to an ASCII terminal, use the -T flag with the mkvirprt command.

After a virtual printer definition is created, its attribute values can be displayed with the lsvirprt command and changed with the chvirprt command.

The mkvirprt command becomes interactive if only the -A flag is specified with the command. Prompts are issued requesting the necessary parameter values. Prerequisite spooler queues and spooler queue devices are generated automatically, and all virtual printer definitions needed for the printer are defined with a single invocation of the mkvirprt command for the specified attachment type.

When the first prompt asks for a device name, if the device name entered is not that of a printer, or if an * (asterisk) precedes the device name, a list of printers is displayed. Otherwise, the printer type is assumed to be the same as that of the device.

Also, when a prompt asks for a print queue name, the queue name entered may optionally be followed by a colon and a queue device name. If no queue device name is provided, the queue device name is assumed to be the same as the device name.

Note: Queue and device names must begin with an alphabetic character.

You can use the Printer Queues application in Web-based System Manager (wsm) to change printer characteristics. You could also use the System Management Interface Tool (SMIT) smit mkvirprt fast path to run this command.

Flags

-A AttachmentType

Specifies the type of printer attachment. The most common values for the AttachmentType variable value are:

Attachment Type

  Represents

local  Locally connected printers
remote  Remote print queues
ascii  Printers attached to an ASCII terminal
file  Print output redirected to a regular file.

This flag is optional, and if the -A flag is not specified the default attachment type is file. If the -A flag is the only flag specified on the command line, the mkvirprt command goes into interactive mode and executes steps specified in the corresponding .config file.

d QueueDeviceName

Specifies the name of an existing queue device to which the virtual printer is assigned.
-n DeviceName Specifies the name of the printer device. Device names include lp0 for printer 0, lp1 for printer 1, and so on.

-q PrintQueueName Specifies the special file name of an existing print queue to which the virtual printer is to be assigned. Note that you do not have to specify the path name to the file, such as the /dev/lp0 file, you just need to specify lp0.

-s DataStreamType Specifies the printer data stream type. Data stream types include:

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>asc</td>
<td>Extended ASCII</td>
</tr>
<tr>
<td>ps</td>
<td>PostScript</td>
</tr>
<tr>
<td>pcl</td>
<td>Hewlett-Packard PCL</td>
</tr>
<tr>
<td>630</td>
<td>Diablo 630</td>
</tr>
<tr>
<td>855</td>
<td>Texas Instruments 855</td>
</tr>
<tr>
<td>gl</td>
<td>Hewlett-Packard GL</td>
</tr>
<tr>
<td>kji</td>
<td>Kanji</td>
</tr>
</tbody>
</table>

-t PrinterType Specifies the printer type. Printer types include 4201-3, ti2115, and so on. For more information on available printer types, see "Printer Support Provided" and "Printer Specific Information" in AIX 5L Version 5.2 Guide to Printers and Printing.

-T Specifies that the printer is attached to an ASCII terminal.

Examples
1. To make a virtual printer for the asc printer data stream for the 4029 printer attached locally, enter:
   ```
   mkvirprt -A local -d mypro -n lp0 -q proq -s asc -t 4019
   ```
2. To make a virtual printer for a printer connected to an ENA 4033 network adapter, and to be prompted for the parameter values, enter:
   ```
   mkvirprt -A ena
   ```

Files

<table>
<thead>
<tr>
<th>Path</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/usr/sbin/mkvirprt</td>
<td>Contains the mkvirprt command.</td>
</tr>
<tr>
<td>/etc/qconfig</td>
<td>Contains configuration files.</td>
</tr>
<tr>
<td>/usr/lib/lpd/pio/predef/*</td>
<td>Contains predefined printer attribute files.</td>
</tr>
<tr>
<td>/var/spool/lpd/pio/@local/custom/*</td>
<td>Contains customized virtual printer attribute files.</td>
</tr>
<tr>
<td>/usr/lib/lpd/pio/etc/*.attach</td>
<td>Contains attachment type files.</td>
</tr>
<tr>
<td>/usr/lib/lpd/pio/etc/*.config</td>
<td>Contains the configuration file for the printer.</td>
</tr>
<tr>
<td>/var/spool/lpd/pio/@local/ddi/*</td>
<td>Contains digested virtual printer attribute files.</td>
</tr>
</tbody>
</table>

Related Information

The `chvirprt` command, `lp` command, `lpr` command, `lsvirprt` command, `mkque` command, `mkquedev` command, `qprt` command, `rmvirprt` command.

The `/etc/qconfig` file.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


**mm Command**

**Purpose**
Prints documents formatted with memorandum macros.

**Syntax**

```
```

**Description**
The `mm` command formats documents that use the `nroff` command and the `mm` macro package. The `mm` command has flags that specify preprocessing by the `tbl` and `neqn` commands and postprocessing by various terminal-oriented output filters. The proper pipelines and the required flags for the `nroff` command are generated depending on the flags that are selected.

**Notes:**
1. Use the `-o List` flag of the `nroff` command to specify ranges of output pages. Remember that if the `mm` command is called with the `-e`, `-t`, or `-` (minus sign) flags together with the `-o List` flag, and if the last page of the document is not specified by the `List` variable, you may receive a broken pipe message. This message is not an indication of any problem and can be ignored.
2. The `mm` command calls the `nroff` command with the `-h` flag. With this flag, the `nroff` command assumes that the workstation has tabs set every 8 character positions.
3. If you use the `-s` flag of the `nroff` command (to stop between pages of output), use a linefeed (rather than the Enter key or a newline character) to restart the output. The `-s` flag of the `nroff` command does not work with the `-c` flag of the `mm` command or if the `mm` command automatically calls the `col` command.
4. Providing inaccurate information to the `mm` command about the kind of workstation its output is to be printed on will produce unsatisfactory results. However, if you are redirecting output to a file, use the `-T37` flag. Then, use the appropriate workstation filter when you print the file.

To obtain a list of `mm` command flags, enter the command name with no parameters. The flags can occur in any order, but they must come before the `File` parameter. Any other flags (for instance, `-rANumber`) are passed to the `nroff` command.
Flags

-M Media Specifies a paper size in order to determine the amount of imageable area on the paper. Valid values for the Media variable are:

<table>
<thead>
<tr>
<th>Media</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A4</td>
<td>Specifies a paper size of 8.27 X 11.69 inches (210 X 297 mm).</td>
</tr>
<tr>
<td>B5</td>
<td>Specifies a paper size of 6.93 X 9.84 inches (176 X 250 mm).</td>
</tr>
<tr>
<td>EXEC</td>
<td>Specifies a paper size of 7.25 X 10.5 inches (184.2 X 266.7 mm).</td>
</tr>
<tr>
<td>LEGAL</td>
<td>Specifies a paper size of 8.5 X 14 inches (215.9 X 355.6 mm).</td>
</tr>
<tr>
<td>LETTER</td>
<td>Specifies a paper size of 8.5 X 11 inches (215.9 X 279.4 mm). This is the default value.</td>
</tr>
</tbody>
</table>

Note: The Media variable is not case sensitive.

-c Calls the col command. Note that the col command is called automatically by the mm command for the following terminal names. The following devices can be specified by the -TName flag, the TERM shell variable, or by using the default:

- ppds
- Ip
- 2631
- 8510

-e Calls the neqn command; also causes the neqn command to read the /usr/share/lib/pub/eqnchar file. See the eqnchar file format.

-E Calls the -e flag of the nroff command.

-t Calls the tbl command.

-12 Uses 12-pitch font. Use this when the TERM shell variable is set to 300, 300s, 450, or 1620. (The pitch switch on the DASI 300 and 300s workstations must be manually set to 12 if this flag is used.)

-TName Uses the workstation type specified by the Name variable.

By default, the mm command uses the value of the TERM shell variable from the environment as the value of the Name variable. If the TERM shell variable is not set, the mm command uses Ip (the generic name for printers that can underline and tab). If several workstation types are specified, the last one listed applies.

- Forces input to be read from standard input.

Parameters

File Specifies the file that the mm command formats.

Examples

1. When the TERM shell variable is set in the environment to the hplj command, the following two command lines are equivalent:

   mm -t -rC3 File
tbl File | nroff -mm -Thplj -h -rC3

2. The mm command reads the standard input when you specify a - (minus sign) flag instead of a value for the File variable. This option allows you to use the mm command as a filter, as follows:

   cat File | mm -

   Note: Using other files together with a - (minus sign) flag leads to undesired results.

Environment Variables

TERM Specifies the terminal names.
Files

\texttt{/usr/share/lib/pub/eqnchar} Contains special character definitions for the \texttt{eqn} command and the \texttt{neqn} command.

Related Information

The \texttt{col} command, \texttt{env} command, \texttt{eqn} command, \texttt{greek} command, \texttt{hplj} command, \texttt{mmt} command, \texttt{neqn} command, \texttt{hplj} command, \texttt{tbl} command.

The \texttt{eqnchar} file, \texttt{profile} file.

The \texttt{nterm} file format describes terminal driving tables for the \texttt{nroff} command.

The article "\texttt{mm Macro Package for the mm, mmt, nroff, and troff Commands}" in the \texttt{troff} Command.

mmt Command

Purpose
Typesets documents.

Syntax


Description

Similar to the \texttt{mm} command, the \texttt{mmt} command typesets its input using the \texttt{troff} command, rather than formatting it with the \texttt{nroff} command. The \texttt{mmt} command uses the \texttt{mm} macro package. There are flags to specify preprocessing by the \texttt{tbl}, \texttt{pic}, \texttt{eqn}, and \texttt{grap} commands. The proper pipelines, required parameters, and flags for the \texttt{troff} command and the \texttt{mm} macro package are generated, depending on the flags selected.

There are several flags that are specific to the \texttt{mmt} command. Any other parameters or flags (for instance, \texttt{-rANumber} or \texttt{-a}) that you give the \texttt{mmt} command are passed to the \texttt{troff} command. You can put flags in any order, but they must be listed before any input files. \texttt{File} specifies the file that the \texttt{mmt} command formats. If you do not give \texttt{File} parameters or other flag variables, the \texttt{mmt} command prints a list of its flags.

The \texttt{mmt} command, unlike the \texttt{troff} command, automatically pipes its output to a postprocessor, unless specifically requested not to do so. The user should not specify a postprocessor when using the \texttt{mmt} command. The precedence is as follows:

1. The \texttt{-z} flag; no postprocessor is used.
2. The \texttt{-TName} flag.
3. The \texttt{TYPESETTER} environment variable is read.
4. The default is set to \texttt{ibm3816}.

The \texttt{mmt} command reads standard input when you specify a - (minus sign) instead of any \texttt{File} parameters.

Use the \texttt{-oList} flag of the \texttt{troff} command to specify ranges of pages to be output.
Note: If you call the mmt command with one or more of the -e, -c, -t, -p, -g, and - (minus sign) flags together with the -oList flag of the troff command, you may receive a broken pipe message if the last page of the document is not specified by the List variable. This broken pipe message is not an indication of any problem and can be ignored.

Flags

-M Media Specifies a paper size in order to determine the amount of imageable area on the paper.
Valid values for the Media variable are:
- A4 Specifies a paper size of 8.27 X 11.69 inches (210 X 297 mm).
- A5 Specifies a paper size of 5.83 X 8.27 inches (148 X 210 mm).
- B5 Specifies a paper size of 6.93 X 9.84 inches (176 X 250 mm).
- EXEC Specifies a paper size of 7.25 X 10.5 inches (184.2 X 266.7 mm).
- LEGAL Specifies a paper size of 8.5 X 14 inches (215.9 X 355.6 mm).
- LETTER Specifies a paper size of 8.5 X 11 inches (215.9 X 279.4 mm). This is the default value.

-a Displays readable troff output to the terminal.
-c Preprocesses the input files with the cw command.
-e Calls the eqn command; also causes the eqn command to read the /usr/share/lib/pub/eqnchar file (see the eqnchar file format).
-g Calls the grap command, which in turn calls the pic command.
-p Calls the pic command.
-t Calls the tbl command.
-z Starts no output filter to process or redirect the output of the troff command.

-DDestination Directs the output to a device specified by the Destination variable. Supported destination devices for English-language output is 4014, which is the Tektronix 4014 terminal by way of the tc command.

-TName Creates output for a troff device as specified by the Name variable. The output is sent through the appropriate postprocessor. The default value is ibm3816. Possible Name variables are:

- ibm3812 3812 Pageprinter II.
- ibm3816 3816 Pageprinter.
- hplj Hewlett-Packard LaserJet II.
- ibm5587G 5587-G01 Kanji Printer multi-byte language support.
- psc PostScript printer.
- X100 AIXwindows display.

- Forces input to be read from standard input.

Related Information

The cw command, eqn command, grap command, mm command, mvt command, pic command, tbl command, tc command, troff command.

The eqnchar file format contains special character definitions for the eqn and neqn commands.

The article "mm Macro Package for the mm, mmt, nroff, and troff Commands" in the troff Command.
mmtu Command

Purpose
Displaying, adding, and deleting maximum transfer unit (MTU) values used for path MTU discovery.

Syntax
mmtu \{-a Value | -d Value | -s\}

Description
Use the mmtu command to display, add, and delete maximum transfer unit (MTU) values to the list of potential path MTU values. Path MTU discovery uses the list of potential path MTU values to detect the path MTU. The list of potential path MTU values is only used when there are routers in the path that do not comply with RFC 1191. The user must have administrative authority to add or delete MTU values.

Flags
-\-a Value Adds the new MTU to the list of potential path MTU values.
-\-d Value Deletes the value from the list of potential path MTU values.
-\-s Displays the current list of potential path MTU values.

Examples
1. To add a value to the list of potential path MTU values, enter:
   mmtu -a mtu-value
2. To delete a value from the list of potential path MTU values, enter:
   mmtu -d mtu-value
3. To display the contents of the list of potential path MTU values, enter:
   mmtu -s

Files
/usr/sbin/mmtu Contains the mmtu command.

Related Information
The netstat command, no command.

mobip6ctrl Command

Purpose
Configures and manages the mobile IPv6 home agent and correspondent node functionality.

Syntax
mobip6ctrl \{-c | -R \} \{-b \} \{-S \{ 0 | 1 \} \} \{-n \{ 0 | 1 \} \} \{ LifeTime \} \{-a | -d \} HomeAddress CareOfAddress MyAddress \}

Description
The mobip6ctrl command is used to configure and manage the mobile IPv6 home agent and correspondent node. It can enable and disable NDP proxy and IP security checking, and it can be used to display or modify the mobile IPv6 binding cache.
NDP proxy must be enabled if the system is configured as a home agent. This allows the home agent to intercept packets addressed to mobile nodes that are not currently on their home network.

IP security checking enables checking to ensure that IP security is used for the Binding Update and Binding Acknowledgement messages sent for mobile IPv6. Because these two types of messages have the ability to affect the routing of packets addressed to a mobile node, they would represent a significant security vulnerability if not protected by IP security. If checking is enabled, the mobile IPv6 home agent or correspondent node will discard any Binding Update or Binding Acknowledgement packets that are not protected by IP security.

The mobile IPv6 binding cache on a home agent or correspondent node maps home addresses to the current care-of addresses for each mobile node. This allows the home agent to tunnel traffic to the mobile node at its current location, and allows a correspondent node to send packets directly to a mobile node at its current location. The `mobip6ctrl` command can be used to view the binding cache or manually edit it for debugging purposes.

Normally, this command is used from the `/etc/rc.mobip6` script when mobile IPv6 has been configured using system management.

### Flags

- `-a HomeAddress CareOfAddress MyAddress`  
  Adds this entry to the binding cache.
- `-b`  
  Displays all binding cache entries.
- `-c`  
  Compatibility option which enables the support of the mobiles implementing the draft #13 of the *Mobility support in IPv6* specification. Using this option, the home agent or correspondent node will accept the binding update messages sent using a Destination Option and using an Authentication Header (AH) to protect these packets with IPsec.
- `-d HomeAddress CareOfAddress MyAddress`  
  Delete this entry from the binding cache.
- `-l LifeTime`  
  Specifies the default life time value for binding cache entries in seconds.
- `-n 0 | 1`  
  Activates or deactivates NDP proxy capabilities. A value of 1 activates the NDP proxy capabilities, and a value of 0 disables NDP proxy capabilities. The default value is 0.
- `-R`  
  Resets all the binding cache entries.
- `-S 0 | 1`  
  Enables or disables checking to ensure that IP security is used for all Binding Update and Binding Acknowledgement packets. A value of 1 enables checking, and a value of 0 disables checking. The default value is 0.

### Exit Status

- `0`  
  The command completed successfully.
- `>0`  
  An error occurred.

### Security

You must be the root user or a member of the system group to execute this command.

### Examples

1. The following example enables NDP proxy and IP security checking for mobile IPv6:
   ```
   mobip6ctrl -S 1 -n 1
   ```
2. The following example displays all entries in the binding cache:
   ```
   mobip6ctrl -b
   ```
The output from this command looks similar to the following:

```
BINDING CACHE LIST (1 elem)

Home Address..........: 3ffe:300:20:1102::217
Care-Of Address......: 3ffe:300:20:1101::217
My Address...........: 3ffe:300:20:1102::223
Life time............: 518
Time since last usage: 50
Rate limit time......: 0
Retransmit count.....: 0
Sequence number......: 14
Registered by me.....: 1
Prefix length........: 64
```

**Related Information**
The `kmodctrl` command, `mobip6reqd` command, `ndpd-router` command, `rc.mobip6` command.

The `Mobile IPv6 section` in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

---

**mobip6reqd Daemon**

**Purpose**
Provides the Mobile IPv6 home agent daemon.

**Syntax**
To run the daemon using the System Resource Controller:

```
startsrc -s mobip6reqd
```

To run the daemon without using the System Resource Controller:

```
mobip6reqd
```

**Description**
The `mobip6reqd` daemon must be running in order for the system to function as a mobile IPv6 home agent. This daemon enables the home agent to perform NDP proxying for mobile nodes. The daemon is normally started automatically by the `/etc/rc.mobip6` script if the mobile IPv6 home agent has been enabled using system management.

**Exit Status**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>The command completed successfully.</td>
</tr>
<tr>
<td>&gt;0</td>
<td>An error occurred.</td>
</tr>
</tbody>
</table>

**Security**
You must be the root user or a member of the system group to execute this command.

**Examples**

1. The following example starts the `mobip6reqd` daemon:
   
   ```
   startsrc -s mobip6reqd
   ```

2. The following example stops the `mobip6reqd` daemon:
Related Information
The kmodctrl command, mobip6ctrl command, ndpd-router command, rc.mobip6 command.

The Mobile IPv6 section in AIX 5L Version 5.2 System Management Guide: Communications and Networks.

monacct Command

Purpose
Performs monthly or periodic accounting.

Syntax
/usr/sbin/acct/monacct [ Number ]

Description
The monacct command performs monthly or periodic accounting. The intervals are set in the crontab file. You can set the cron daemon to run the monacct command once each month or at some other specified time period. The monacct example shows how to set up this command for use with the cron daemon. See the crontab command for more information about setting up cron files.

The Number parameter indicates which month or other accounting period to process. The default value of the Number parameter is the current month. The monacct command creates summary files in the /var/adm/acct/fiscal file and restarts summary files in the /var/adm/acct/sum file, the cumulative summary to which daily reports are appended.

Note: You should not share accounting files among nodes in a distributed environment. Each node should have its own copy of the various accounting files.

Security
Access Control: This command should grant execute (x) access only to members of the administrative group.

Example
To produce automatically a monthly accounting report, add the following to the /var/spool/cron/crontabs/root file:

15 5 1 * * /usr/sbin/acct/monacct

This example shows the instructions that the cron daemon will read and act upon. The monacct command will run at 5:15 (15 5) the first day of each month (1). This command is only one of the accounting instructions normally given to the cron daemon. See "Setting Up an Accounting System" in AIX 5L Version 5.2 System Management Guide: Operating System and Devices for more information on typical cron accounting entries.

Files
/usr/sbin/acct Contains the accounting commands.
/var/adm/acct/fiscal Contains accounting data files.
/var/adm/acct/sum Cumulative daily accounting records.
/var/spool/cron/crontabs Contains the commands to be run by the cron daemon at regularly scheduled intervals.
Related Information
The acctcms command, prtaacct command, acctmerge command, crontab command.

The cron daemon.

For more information about the Accounting System, the preparation of daily and monthly reports, and the accounting files, see the Accounting Overview in AIX 5L Version 5.2 System Management Concepts: Operating System and Devices.

Setting Up an Accounting System in AIX 5L Version 5.2 System Management Guide: Operating System and Devices explains the steps you must take to establish an accounting system.

mon-cxma Command

Purpose
Monitor status of 128-port asynchronous subsystem and attached devices.

Syntax
To Display All 128-Port Adapters:
mon-cxma

To Display Syntax or Slots and Bus Information:
mon-cxma {-h | -x}

To Display Specific Slots and Bus Information:
mon-cxma { [-h | LogFile ] [-f | DeviceFile ] [-s | SlotNumber ] [-b | BusNumber ] }

Description
The mon-cxma command is a software tool which provides a means to monitor the status of serial devices and remote async nodes (RAN) attached to the IBM 128-port asynchronous adapter. It is used for subsystem problem determination and can be accessed locally and remotely via modem. The only restriction on modem access is that the modem can not be physically attached to the 128-port adapter being monitored.

When the user enters the mon-cxma command at the command line, it automatically detects and displays all available 128-port adapters in the system. The bus and slot location within the system is displayed for each adapter and the user can select adapter to monitor.

You can use the Software application in Web-based System Manager (wsm) to run this command. You could also use the System Management Interface Tool (SMIT) smit 128psync fast path to advance directly to the “128-Port Asynchronous Adapter” menu. When run from SMIT, the mon-cxma command automatically displays all available 128-port adapters in the system. Refer to “Running the Monitor from SMIT” in AIX 5L Version 5.2 Asynchronous Communications Guide for more information.

Flags
-b [ BusNumber ] Specifies the bus number of the device. Valid values for BusNumber are 0 to (n-1), where n is the number of buses the system has.

-f [ DeviceFile ] Specifies the device special file. Use this file to look at a specific device driver without having to make a selection. The default device special file is /dev/cxma0.

-h Shows syntax information.
-L [LogFile]  (Lowercase L) Specifies the file to be used as the log. Use this file to store information from the screen when the IMAGE key is pressed. The default log file is /tmp/mon-cxma.log.

-s [SlotNumber]  Specifies the slot number of the device. Valid values for SlotNumber are 0 to (n-1), where n is the number of slots the system has.

-x  Shows the POS (Programmable Select Option) register values for all the slots and buses.

Note: -x and -h ignore other options.

Security
Access Control: Root authority required to run this command.

Auditing Events: N/A

Examples
1. To run the mon-cxma command using the SMITfastpath, enter:
   smit 128psync
2. To display all 128-port adapters, enter:
   /usr/lbin/tty/mon-cxma

Files
/usr/lbin/tty/mon-cxma  Contains the mon-cxma command.
/tmp/mon-cxma.log  Contains the log file.

Related Topics
For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.

monitord Daemon

Purpose
Communicates with the License Use Management server and requests a concurrent-use license for each countable login.

Syntax
monitord [-t Minutes] [-v Version.Release]

Description
The operating system has multiple ways to access the system, and each of them has a different behavior upon exit. The monitord daemon provides a common interface to the License Use Management netlsd. monitord communicates with the License Use Management server and requests a concurrent-use license for each countable login.

Note: The License Use Management licensing mechanism is used only if the system has the floating license mode enabled.

After user logout, monitord requests netlsd to release the specific license the user was using, in order to make it available for further logins.
**monitord** is started when the **chlicense -f** command is used to enable the *floating license mode*. When the *floating license mode* is enabled, **monitord** is started upon system startup via an entry in `/etc/inittab`. The default (invoked without `-t` option) is an interval of fifteen minutes.

The entry in `/etc/inittab` looks like the following:

```
monitord:2:once:/usr/sbin/monitord >/dev/console 2>&1
```

**Flags**

- `-t Minutes` Sets the value in minutes of the heartbeat interval. A value of 0 sets an infinite interval. The default is fifteen minutes.

- `-v Version.Release` Enables the *floating license mode* for a license of the specified *Version* and *Release*.

---

**moo Command**

**Purpose**

Starts the number-guessing game.

**Syntax**

```
moo
```

**Description**

The **moo** command picks a combination of four random, non-repeating numbers. You guess four numbers at the *your guess?* prompt. Each correct number in an incorrect position in the four number combination scores “cow.” Each correct number in the correct position in the four number combination scores a “bull.” For example:

```
your guess?
1470
bulls = 0  cows = 1
your guess?
```

In this example, one of the four numbers (1, 4, 7, and 0) is correct but in the incorrect position. None of the numbers are correct and in the correct position.

To quit the game, press the Interrupt (Ctrl-C) or End Of File (Ctrl-D) key sequence.

**File**

```
/usr/games
```

Contains the system's games.

**Related Information**

The [arithmetic](CommandsReferenceV3.html#arithmetic), [back](CommandsReferenceV3.html#back), [bj](CommandsReferenceV3.html#bj), [craps](CommandsReferenceV3.html#craps), [fish](CommandsReferenceV3.html#fish), [fortune](CommandsReferenceV3.html#fortune), [hangman](CommandsReferenceV3.html#hangman), [number](CommandsReferenceV3.html#number), [quiz](CommandsReferenceV3.html#quiz), [ttt](CommandsReferenceV3.html#ttt), [turnoff](CommandsReferenceV3.html#turnoff), [turnon](CommandsReferenceV3.html#turnon), [wump](CommandsReferenceV3.html#wump) commands.

---

**more Command**

**Purpose**

Displays the contents of files one screen at a time.
Syntax
more [ -c ] [ -d ] [ -e ] [ -l ] [ -n ] [ -s ] [ -u ] [ -v ] [ -z ] [ -m Number ] [ -P Subcommand ] [ -T Tagstring ] [ -W Option ] [ -x ] [ -u ] [ -l ] [ -t ] [ Tabs ] [ File ... ]

Description
The more command reads files and displays the text one screen at a time. The command pauses after each screen and prints the word More at the bottom of the screen. If you then press a carriage return, the more command displays an additional line. If you press the space bar, the more command displays another full screen of text.

   Note: On some terminal models, the more command clears the screen, instead of scrolling.

Instead of naming files to read, you can either redirect or pipe standard output, such as a long directory listing, to the more command. The command adds a % (percent sign) to its prompt when reading from a file rather than a pipe. This provides the percentage of the file (in characters, not lines) that the more command has read.

The more command sets the terminal to NOECHO mode so the output can be continuous. With the exception of the / and ! subcommands, commands that are typed do not normally show up on the terminal. If the standard output is not a terminal, the more command will act just like the cat command, except that a header will be printed before each file in a series.

Environment Variables
Environment variables affect the way the more command works. You can set some environment characteristics in the /etc/environment file and system profile files, such as the .ksh, .csh, and .profile files. Refer to "User Environment and System Information Overview" in AIX 5L Version 5.2 System User's Guide: Operating System and Devices for discussions about determining and configuring your system environment.

The more command uses the TERM variable to determine terminal characteristics. If this variable is NULL or not set, the command uses the default terminal type. The /usr/share/lib/terminfo directory contains definitions for terminal characteristics.

By default, the more command window size is 2 lines less than what the system terminal is capable of. The command sets the default window size based on the LINES variable. Also, you can easily adjust the window size for each run of the command by adding the -n flag.

Use the MORE variable to customize the more command with your preferred configuration each time the system starts. This variable accepts more command flags.

Flags
-c  Prevents the screen from scrolling, which makes text easier to read as the more command writes to the screen. The system ignores the -c flag if the terminal cannot clear to the end of a line.
-d  Prints a message, appended to the More prompt at the bottom of the screen, about which keys continue, quit, and provide help for the more command. Displays error messages rather than ringing the terminal bell if an unrecognized command is used. This is helpful for inexperienced users.
-e  Exits automatically after displaying the last line of the last file.
-i  Searches for patterns without considering uppercase and lowercase.
-l  Pauses after detecting a page break in the input. If the -l flag is not used, the more command pauses to accept commands after any line containing a ^L (CTRL-L) character. Also, if a file begins with a FORMFEED, the screen is cleared before the file is printed.
Subcommands

The more command accepts subcommands when the command pauses and as parameters for the -p flag. Many subcommands take an optional integer, symbolized here by K, which you must enter before the
The **more** command uses the following subcommands:

- **h**
  - Displays a help screen that describes the **more** subcommands.
- **v**
  - Starts the **vi** editor, editing the current file in the current line.
- **r or ^L**
  - Refreshes the display.
- **R**
  - Refreshes the display and removes buffered input.
- **[K](Spacebar)**
  - Moves forward **K** lines when you press the spacebar. If you do not give a value for **K**, pressing the spacebar displays the next full screen by default. This spacebar subcommand is the same as **[K]f or [K]^F or [K]z**.
- **[K]f or [K]^F or [K]z**
  - Moves forward **K** lines, or one full screen if you do not give a value for **K**.
- **[K]b or [K]^B**
  - Moves backward **K** lines, or one full screen if you do not give a value for **K**.
- **[K]d or [K]^D**
  - Moves forward **K** lines, or half a screen if you do not give a value for **K**. If you give a value for **K**, the **more** command sets the **d** and **u** scroll size to **K** lines for the session.
- **[K]u or [K]^U**
  - Moves backward **K** lines, or half a screen if you do not give a value for **K**. If you give a value for **K**, the **more** command sets the **d** and **u** scroll size to **K** lines for the session.
- **[K]j or [K](Enter) or [K]^E**
  - Moves forward **K** lines, or one line if you do not give a value for **K**.
- **[K]k or [K]^Y**
  - Moves backward **K** lines, or one line if you do not give a value for **K**.
- **[K]g**
  - Moves to the beginning of the file, unless you give a line number for **K**. The default for **K** is line number 1.
- **[K]G**
  - Moves to the last line in the file, unless you give a line number for **K**. The default for **K** is the last line in the file.
- **[K]p or [K]%**
  - Moves to the point in the file that is **K** percent of the total file. The default for **K** is one percent, or the first line in the file.
- **ma-z**
  - Marks the current position in the file with the specified letter. (Single quote) Moves to the position marked with the specified letter.
- **'a-z**
  - (Two single quotes) Returns to the position from which the last large movement (moving more than a page) command was run. If no such movements have been made, returns to the beginning of the file.
- **[K]//pattern**
  - (Slash) Searches forward, from the current position, for the specified occurrence of the specified pattern of characters. The default value for **K** is the first occurrence.
- **[K]//pattern**
  - (Slash, exclamation mark) Searches forward, from the current position, for the specified occurrence of a line that does not contain the specified pattern of characters. The default value for **K** is the first occurrence.
- **[K]?pattern**
  - (Question mark) Searches backward, from the current position, for the specified occurrence of the specified pattern of characters. The default value for **K** is the first occurrence.
- **[K]?!pattern**
  - (Question mark, exclamation mark) Searches backward, from the current position, for the specified occurrence of a line that does not contain the specified pattern of characters. The default value for **K** is the first occurrence.
Repeats the last search, specifying an occurrence of the pattern (or an occurrence not containing the pattern if the search subcommand included !). The default value for $K$ is the first occurrence.

Lists the file or files you named in the more command line.

Displays information about the current file:
- file name
- order of the file in the list of files
- current line number
- current position in the file, given as a percentage
- current byte number and total bytes to display.

Examines the specified file, provided you named it in the more command line.

Examines either the next file (if you do not give a value for $K$) or the file $K$ number of positions forward in the list of files you named in the more command line.

Examines either the previous file (if you do not give a value for $K$) or the file $K$ number of positions backward in the list of files you named in the more command line.

Displays the portion of the file that contains the specified tag. This subcommand works only on files containing tags created with the ctags command. The :t subcommand is the interactive version of the -t flag.

Exits the more command.

Starts the specified command in a new shell.

Exit Status
This command returns the following exit values:

0 Successful completion.

>0 An error occurred.

Examples
1. To view a file named myfile, enter:
   `more myfile`
2. To view output from the nroff command, enter:
   `ls -l | more`
3. To view each file starting at its last screen, enter:
   `more -p 6 file1 file2`
4. To view each file with the 100th line at the current position, enter:
   `more -p 100 file1 file2`
   Typically, the current position in a more command display is the third line on the screen. In this example, the first line on the screen is the 98th line in the file.
5. To view each file starting with the first line that contains the foo string, enter:
   `more -p /foo file1 file2`
   The more command displays the line in the current position, the third line on the screen.
Files

 Files
 /usr/share/lib/terminfo Indicates the terminal information database.

Related Information
The [cat][csh][ctags][ksh][pg][script] command.

The [environment][terminfo] file.


mosy Command

Purpose
Converts the ASN.1 definitions of Structure and Identification of Management Information (SMI) and Management Information Base (MIB) modules into objects definition files for the [snmpinfo][snmpd] command.

Syntax
mosy [-o] OutputFile [-s] InputFile ...

Description
The mosy command reads in the ASN.1 definitions of SMI and MIB modules and produces objects definition files in specific formats. The resulting objects definition files are used by the [snmpinfo] command.

The InputFile parameter files are required to be in the [smi.my][mibII.my] format. Sample files are the /usr/samples/snmpd/smi.my and /usr/samples/snmpd/mibII.my files. See the [smi.my] and the [mibII.my] files for information on the required format of the file specified by the InputFile parameter.

The mosy -o command is used to create the objects definition file specified by the OutputFile parameter for the [snmpinfo] command. This file is normally the /etc/mib.defs file.

The objects definition file can be created with one pass of the mosy compiler if the [smi.my] and [mibII.my] files are both specified as InputFile parameters. The [smi.my] file must precede the [mibII.my] file on the command line.
The **mosy** **-o** command can also be used to create subfiles. If subfiles are created separately from the SMI and MIB modules, you must concatenate the various subfiles before the **snmpinfo** command can successfully use the resultant **mib.defs** file. The SMI subfile must be at the top of the final objects definition file.

You can add objects definitions for experimental MIB modules or private-enterprise-specific MIB modules to the **/etc/mib.defs** file, but you must first obtain the private MIB module from the vendor that supports those MIB variables.

To update the **/etc/mib.defs** file to incorporate a vendor's private or experimental MIB objects definitions, create a subfile and then concatenate that subfile to the existing MIB II **/etc/mib.defs** file. See example 3.

### Flags

**-o OutputFile**

Defines the path and file name of the MIB objects definition file for the **snmpinfo** command. There is no default path and file name for this flag. If this flag is not specified, the objects definition file is not created.

**-s**

Suppresses the conversion verification messages. If this flag is not specified, the conversion verification messages are printed to standard output.

### Parameters

**InputFile**

Defines the ASN.1 object definitions module for input to the **mosy** compiler. This file can be formatted according to either the **smi.my** or **mibII.my** file format.

### Examples

1. To create an objects definition file for use by the **snmpinfo** command with one pass of the **mosy** command, enter:

   ```bash
   mosy -o /etc/mib.defs /usr/samples/snmpd/smi.my
   /usr/samples/snmpd/mibII.my
   ```

   In this example, **/usr/samples/snmpd/smi.my** and **/usr/samples/snmpd/mibII.my** are both specified as input files and the resultant objects definition file is the **/etc/mib.defs** file.

2. To create objects definition subfiles, enter:

   ```bash
   mosy -o /tmp/smi.obj /usr/samples/snmpd/smi.my
   mosy -o /tmp/mibII.obj /usr/samples/snmpd/mibII.my
   cat /tmp/smi.obj /tmp/mibII.obj > /etc/mib.defs
   ```

   In this example, the first command creates an SMI objects file, **/tmp/smi.obj**, from the **/usr/samples/snmpd/smi.my** file. The second command creates the MIB objects definition file, **/tmp/mibII.obj**, from the **/usr/samples/snmpd/mibII.my** file. The final command concatenates the subfiles, placing the SMI objects definition file first in the resultant **/etc/mib.defs** file.

3. To add private enterprise specific MIB objects definitions to an existing **/etc/mib.defs** file for use by the **snmpinfo** command, enter:

   ```bash
   mosy -o /tmp/private.obj /tmp/private.my
   cat /etc/mib.defs /tmp/private.obj > /tmp/mib.defs
   mv /tmp/mib.defs /etc/mib.defs
   ```

   In this example, the first command creates the **/tmp/private.obj** objects definition file. The second command concatenates the **/etc/mib.defs** MIB objects definition file with the **/tmp/private.obj** private MIB file and places the concatenated contents into the **/tmp/mib.defs** temporary MIB objects definition file. The final command moves the temporary file to the **/etc/mib.defs** file for use by the **snmpinfo** command.
Files

/etc/mib.defs

/usr/samples/snmpd/smi.my

/usr/samples/snmpd/mibII.my

 Defines the Management Information Base (MIB) variables the SNMP agent should recognize and handle. The format of the /etc/mib.defs file is required by the snmpinfo command.

 Defines the ASN.1 definitions by which the SMI is defined as in RFC 1155.

 Defines the ASN.1 definitions for the MIB II variables as defined in RFC 1213.

Related Information

The snmpinfo command.

Understanding the Management Information Base (MIB) Variables in AIX 5L Version 5.2 Communications Programming Concepts.

mount Command

Purpose

Makes a file system available for use.

Syntax

mount [ [ -f ] [ -n Node ] [ -o Options ] [ -p ] [ -r ] [ -v VfsName ] ] [ -t Type ] [ Device | Node:Directory ] Directory [ [ -a ] [ -l ] [ -t ] [ -V ] [ generic_options ] special_mount_points ]

Description

The mount command instructs the operating system to make a file system available for use at a specified location (the mount point). In addition, you can use the mount command to build other file trees made up of directory and file mounts. The mount command mounts a file system expressed as a device using the Device or Node:Directory parameter on the directory specified by the Directory parameter. After the mount command has finished, the directory specified becomes the root directory of the newly mounted file system.

Only users with root authority or are members of the system group and have write access to the mount point can issue file or directory mounts. The file or directory may be a symbolic link. The mount command uses the real user ID, not the effective user ID, to determine if the user has appropriate access. System group members can issue device mounts, provided they have write access to the mount point and those mounts specified in the /etc/filesystems file. Users with root user authority can issue any mount command.

Users can mount a device provided they belong to the system group and have appropriate access. When mounting a device, the mount command uses the Device parameter as the name of the block device and the Directory parameter as the directory on which to mount the file system.

If you enter the mount command without flags, the command displays the following information for the mounted file systems:

• the node (if the mount is remote)
• the object mounted
• the mount point
• the virtual-file-system type
• the time mounted
• any mount options

If you specify only the Directory parameter, the mount command takes it to be the name of the directory or file on which a file system, directory, or file is usually mounted (as defined in the /etc/filesystems file). The mount command looks up the associated device, directory, or file and mounts it. This is the most convenient way of using the mount command, because it does not require you to remember what is normally mounted on a directory or file. You can also specify only the device. In this case, the command obtains the mount point from the /etc/filesystems file.

The /etc/filesystems file should include a stanza for each mountable file system, directory, or file. This stanza should specify at least the name of the file system and either the device on which it resides or the directory name. If the stanza includes a mount attribute, the mount command uses the associated values. It recognizes five values for the mount attributes: automatic, true, false, removable, and readonly.

The mount all command causes all file systems with the mount=true attribute to be mounted in their normal places. This command is typically used during system initialization, and the corresponding mounts are referred to as automatic mounts.

The mount command can also be used to access a snapshot of a JFS2 file system as a directory tree. The snapshot on Device is mounted read-only at Directory. A snapshot can only be mounted once. When mounting a JFS2 file system with snapshots, the snapshots are activated.

You can use the File Systems application in Web-based System Manager (wsm) to run this command. You could also use the System Management Interface Tool (SMIT) smit mount fast path to run this command.

Note: If the mount command encounters a Journaled File System (JFS) or Enhanced Journaled File System (JFS2) which was not unmounted before reboot, a replay of any JFS or JFS2 log records is attempted. In order to move a compatible JFS filesystem to a system running an earlier release of the operating system, the filesystem must always be unmounted cleanly prior to its movement. Failure to unmount first may result in an incompatible JFS log device. If the movement results in an unknown log device, the filesystem should be returned to the system running the latter operating system release, and fsck should be run on the filesystem.

Note: If the cdromd CD and DVD automount daemon is enabled, then those devices will be automatically mounted as specified in the /etc/cdromd.conf file. Use the cdmount or cdeject command to unmount an automounted CD or DVD. Use stopsrc -s cdromd to disable the CD/DVD automount daemon.

Note: For CacheFS, the remote filesystem that is to be cached locally must be exported such that the root ID of the local system is not remapped on the remote host to nobody (or the ID that the remote host uses as the anonymous user). For example, if host A were to export a filesystem /F, which would be mounted with CacheFS on host B, then the /etc/exports on host A would need to have an entry similar to:

/F -rw,root=B
or
/F -ro,root=B

depending on the mount options used for the local CacheFS mount.

Flags

-a     | Mounts all file systems in the /etc/filesystems file with stanzas that contain the true mount attribute.
all    | Same as the -a flag.
-f     | Requests a forced mount during system initialization to enable mounting over the root file system.
-n Node  Specifies the remote node that holds the directory to be mounted.
File System Specific Options

- `Options` Specifies options. Options entered on the command line should be separated only by a comma. The following file system-specific options do not apply to all virtual file system types:

  `bsy` Prevents the mount operation if the directory to be mounted over is the current working directory of a process.

  `cio` Specifies the file system to be mounted for concurrent readers and writers. I/O on files in this filesystem will behave as if they had been opened with O_CIO specified in the `open()` system call. Using this option will prevent access in any manner other than CIO. It is impossible to use cached I/O on a filesystem mounted with the cio option. This means that mapping commands such as `mmap()` and `shmat()` will fail with EINVAL when used on any file in a filesystem mounted with the cio option. One side-effect of this is that it is impossible to run binaries out of a cio mounted filesystem, since the loader may use `mmap()`.

  `dio` Specifies that I/O on the filesystem will behave as if all the files had been opened with O_DIRECT specified in the `open()` system call.

  `fmode=octal` Specifies the mode for a file and directory. The default is 755.

  `gid=gid` Specifies the GID that is assigned to files in the mount. The default is bin.

  `log=LVName` Specifies the full path name of the filesystem logging logical volume name where the following file-system operations are logged.

  `nocase` Turns off case mapping. This is useful for cdroms using the ISO 9660:1998/HSG standard.

  `nodev` Specifies that you cannot open devices from this mount. This option returns a value of ENXIO if a failure occurs.

  `nosuid` Specifies that execution of setuid and setgid programs by way of this mount is not allowed. This option returns a value of EPERM if a failure occurs.

  `rbr` Mount filesystem with the release-behind-when-reading capability. When sequential reading of a file in this filesystem is detected, the real memory pages used by the file will be released once the pages are copied to internal buffers. **Note:** When rbr is specified, the D_RB_READ flag is ultimately set in the _devflags field in the _pdtentry structure.

  `rbw` Mount filesystem with the release-behind-when-writing capability. When sequential writing of a file in this filesystem is detected, the real memory pages used by the file will be released once the pages written to disk. **Note:** When rbw is specified, the D_RB_WRITE flag is set.

  `rbrw` Mount filesystem with both release-behind-when-reading and release-behind-when-writing capabilities. **Note:** If rbrw is specified, both the D_RB_READ and the D_RB_WRITE flags are set.

  `ro` Specifies that the mounted file is read-only. The default value is rw.

  `rw` Specifies that the mounted file is read/write accessible. rw is the default value.

  `snapshot` Specifies the Device to be mounted is a snapshot. The snapped file system for the specified snapshot must already be mounted or an error message will display.

  `snapto=snapshot` Specifies the location to start a snapshot with the value of snapshot when mounting the specified JFS2 file system.
File System Specific Options (continued)

- o Options Specifies options. Options entered on the command line should be separated only by a comma. The following file system-specific options do not apply to all virtual file system types:

  upcase Changes case mapping from default lowercase to uppercase. This is useful for cdroms using the ISO 9660:1998/HSG standard.

  uid=uid Specifies the UID that is assigned to files in the mount, the default is bin.

  wrkgrp=workgroup Specifies the workgroup that the SMB server belongs.

NFS Specific Options

- o Options Specifies options. Options you enter on the command line should be separated only by a comma, not a comma and a space. The following NFS-specific options do not apply to all virtual file system types:

  acdirmax=n Holds cached attributes for no more than n seconds after directory update. The default is 60 seconds.

  acdirmin=n Holds cached attributes for at least n seconds after directory update. The default is 30 seconds.

  acl Requests using the Access Control List RPC program for this NFS mount. If the acl option is used, the ACL RPC program is used only if the NFS server provides it. The default is noacl. This option only applies to AIX 4.2.1 or later.

  acregmax=n Holds cached attributes for no longer that n seconds after file modification. The default is 60 seconds.

  acregmin=n Holds cached attributes for at least n seconds after file modification. The default is 3 seconds.

  actimeo=n Sets minimum and maximum times for regular files and directories to n seconds. If this option is set, it overrides any settings for the acregmin, acregmax, acdirmin, and acdirmax options.

  bg Attempts mount in background if first attempt is unsuccessful. The default value is fg.

  fastattr Bypasses the requirement that files currently being written will be sent to the server before the attributes of the file is read. This option is to be used with caution, since it will cause the client to assume that the file data that has not yet reached the server will be written without problem. In case of write errors, the client and server will have different opinions on what the size of the file really is. Likewise, a client will not be aware of attribute changes to the file being made by another client, so this option must not be used in environments where two clients are writing to the same files.

  fg Attempts mount in foreground if first attempt is unsuccessful. fg is the default value.

  grpdir Directs any file or directory created on the file system to inherit the group ID of the parent directory.

  hard Retries a request until server responds. The option is the default value.

  intr Allows keyboard interrupts on hard mounts.
**llock** Requests that files lock locally at the NFS client. NFS network file locking requests are not sent to the NFS server if the **llock** option is used. This option only applies to AIX 4.2.1 or later.

**maxgroups=n** Indicates that NFS RPC calls using **AUTH_UNIX** may include up to n member groups of information. Using this option to increase the number of member groups beyond the RPC protocol standard of 16 will only work against servers that support more than 16 member groups. Otherwise, the client will experience errors.

Values below 16 or greater than 64 will be ignored. By default, the protocol standard maximum of 16 is adhered to. AIX NFS servers will accept and process **AUTH_UNIX** credentials with up to 64 groups starting with AIX 5L Version 5.2 with the 5200-01 Recommended Maintenance package. The actual number of member groups sent by the NFS client is dependent on the number of groups the involved user is a member of, and may be limited by the length of the NFS client's hostname (which is included in the **AUTH_UNIX** information).

**noac** Specifies that the **mount** command performs no attribute or directory caching. If you do not specify this option, the attributes (including permissions, size, and timestamps) for files and directories are cached to reduce the need to perform over-the-wire NFSPROC_GETATTR Remote Procedure Calls (RPCs). The NFSPROC_GETATTR RPC enables a client to prompt the server for file and directory attributes. The acrregmin, acrregmax, acdirmin, and acdirmax options control the length of time for which the cached values are retained.

**noacl** Specifies not to use the Access Control List RPC program for this NFS mount request. The default is **noacl**. This option only applies to AIX 4.2.1 or later.

**nointr** Specifies no keyboard interrupts allowed on hard mounts. This option only applies to AIX 4.2.1 or later.

**port=n** Sets server Internet Protocol (IP) port number to n. The default value is the 2049.

**posix** Requests that pathconf information be exchanged and made available on an NFS Version 2 mount. Requires a mount Version 2 rpc.mountd at the NFS server.

This option only applies to AIX 4.2.1 or later.

**proto=[udpltcp]** Specifies the transport protocol. The default transport protocol used for AIX 4.2.1 NFS mounts is **udp** if available on the server. For AIX 4.3 and later the default is **tcp**. Use the **proto=[udpltcp]** option to override the default. This option only applies to AIX 4.2.1 or later.

**retrans=n** Sets the number of NFS transmissions to n. The default value is 5.

**retry=n** Sets the number of times the mount is attempted to n; the default value is 1000. When the retry value is 0, the system makes 10,000 attempts.

**rsize=n** Sets the read buffer size to n bytes. The default value is 8192. For AIX 4.2.1 only, the default value is 32768 when using Version 3 of the NFS protocol.

**secure** Specifies that the **mount** command uses Data Encryption Standard (DES) for NFS transactions.

**shortdev** Specifies that you are mounting a file system from a host that does not support 32-bit device special files. This option does not apply to AIX 4.2.1 or later.

**soft** Returns an error if the server does not respond. The default value is **hard**.
timeo=n
Sets the Network File System (NFS) time-out period to \( n \) tenths of a second. For TCP mounts, default timeout is 100, which equals 10 seconds. For UDP mounts, default timeout is 11, which equals 1.1 seconds, but varies depending on the NFS operation taking place. For UDP mounts, the timeout will increase for each failed transmission, with a maximum value of 20 seconds. Each transmission will be attempted twice, after which the time-out value is updated.

vers=[2|3]
Specifies NFS version. The default is the version of NFS protocol used between the client and server and is the highest one available on both systems. If the NFS server does not support NFS Version 3, the NFS mount will use NFS Version 2. Use the vers=[2|3] option to select the NFS version. This option only applies to AIX 4.2.1 or later.

wsize=n
Sets the write buffer size to \( n \) bytes. The default value is 8192. For AIX 4.2.1 only, the default value is 32768 when using Version 3 of the NFS protocol.

-p
Mounts a file system as a removable file system. While open files are on it, a removable mounted file system behaves the same as a normally mounted file system. However, when no files are open (and no process has a current directory on the file system), all of the file system disk buffers in the file system are written to the medium, and the operating system forgets the structure of the file system.

-r
Mounts a file system as a read-only file system, regardless of its previous specification in the /[etc/filesystems] file.

-t Type
Mounts all stanzas in the [etc/filesystems] file that contain the type=Type attribute and are not mounted. The Type parameter specifies the name of the group.

-v VfsName
Specifies that the file system is defined by the VfsName parameter in the [etc/vfs] file.

**CacheFS Specific Options**
The CacheFS-specific version of the mount command mounts a cached file system; if necessary, it NFS-mounts its back file system. It also provides a number of CacheFS-specific options for controlling the caching process.

To mount a CacheFS file system, use the mount command with the -V flag followed by the argument. The following mount flags are available.

The following arguments to the -o flag are specifically for CacheFS mounts. Options you enter on the command line should be separated only by a comma, not a comma and a space.

**Note:** The backfstype argument must be specified.

-o
Specifies options.

acdirmax=n
Specifies that cached attributes are held for no more than \( n \) seconds after directory update. Before \( n \) seconds, CacheFS checks to see if the directory modification time on the back file system has changed. If it has, all information about the directory is purged from the cache and new data is retrieved from the back file system. The default value is 60 seconds.

acdirmin=n
Specifies that cached attributes are held for at least \( n \) seconds after directory update. After \( n \) seconds, CacheFS checks to see if the directory modification time on the back file system has changed. If it has, all information about the directory is purged from the cache and new data is retrieved from the back file system. The default value is 30 seconds.
acregmax=\(n\)
Specifies that cached attributes are held for no more than \(n\) seconds after file modification. After \(n\) seconds, all file information is purged from the cache. The default value is 30 seconds.

acregmin=\(n\)
Specifies that cached attributes are held for at least \(n\) seconds after file modification. After \(n\) seconds, CacheFS checks to see if the file modification time on the back file system has changed. If it has, all information about the file is purged from the cache and new data is retrieved from the back file system. The default value is 30 seconds.

actimeo=\(n\)
Sets acregmin, acregmax, acdirmin, and acdirmax to \(n\).

backfstype=\(\text{file\_system\_type}\)
The file system type of the back file system (for example, nfs).

backpath=\(\text{path}\)
Specifies where the back file system is already mounted. If this argument is not supplied, CacheFS determines a mount point for the back file system.

cachedir=\(\text{directory}\)
The name of the cache directory.

cacheid=\(\text{ID}\)
ID is a string specifying a particular instance of a cache. If you do not specify a cache ID, CacheFS will construct one.

demandconst
Enables maximum cache consistency checking. By default, periodic consistency checking is enabled. When you enable demandconst, it checks on every read and write.

Note: If this option is used the first time a specific CacheFS is mounted, then the option must also be specified for subsequent mounts. There is state information stored in the cache control files that enforces consistent use of this option.

local_access
Causes the front file system to interpret the mode bits used for access checking instead of having the back file system verify access permissions. Do not use this argument with secure NFS.

noconst
Disables cache consistency checking. By default, periodic consistency checking is enabled. Specify noconst only when you know that the back file system will not be modified. Trying to perform cache consistency check using cfadmin-s will result in error. demandconst and noconst are mutually exclusive.

Note: If this option is used the first time a specific CacheFS is mounted, then the option must also be specified for subsequent mounts. There is state information stored in the cache control files that enforces consistent use of this option.

purge
Purge any cached information for the specified file system.

Note: If this option is used the first time a specific CacheFS is mounted, then the option must also be specified for subsequent mounts. There is state information stored in the cache control files that enforces consistent use of this option.

rw | ro
Read-write (default) or read-only.

suid | nosuid
Allow (default) or disallow set-uid execution
Write-around | non-shared

Write modes for CacheFS. The write-around mode (the default) handles writes the same as NFS does; that is, writes are made to the back file system, and the affected file is purged from the cache. You can use the non-shared mode when you are sure that no one else will be writing to the cached file system.

**Note:** If this option is used the first time a specific CacheFS is mounted, then the option must also be specified for subsequent mounts. There is state information stored in the cache control files that enforces consistent use of this option.

-V

Mounts a CacheFS file system.

**Examples**

1. To list the mounted file systems, enter:

   ```
   mount
   ```

   This command produces output similar to the following:

   ```
   node mounted mounted vfs date options over
   ---- -------- -------- ------ ------ -------
   /dev/hd0 / jfs Dec 17 08:04 rw, log =/dev/hd8
   /dev/hd3 /tmp jfs Dec 17 08:04 rw, log =/dev/hd8
   /dev/hd1 /home jfs Dec 17 08:06 rw, log =/dev/hd8
   /dev/hd2 /usr jfs Dec 17 08:06 rw, log =/dev/hd8
   sue /home/local/src /usr/code nfs Dec 17 08:06 ro, log =/dev/hd8
   ```

   For each file system, the `mount` command lists the node name, the device name, the name under which it is mounted, the virtual-file-system type, the date and time it was mounted, and its options.

2. To mount all default file systems, enter:

   ```
   mount all
   ```

   This command sequence mounts all standard file systems in the `/etc/filesystems` file marked by the `mount=true` attribute.

3. To mount a remote directory, enter:

   ```
   mount -n nodeA /home/tom.remote /home/tom.local
   ```

   This command sequence mounts the `/home/tom.remote` directory located on `nodeA` onto the local `/home/tom.local` directory. It assumes the default `VfsName` parameter=`remote`, which must be defined in the `etc/vfs` file.

4. To mount a file or directory from the `/etc/filesystems` file with a specific type, enter:

   ```
   mount -t remote
   ```

   This command sequence mounts all files or directories in the `etc/filesystems` file that have a stanza that contains the `type=remote` attribute.

5. To CacheFS-mount the file system which is already NFS-mounted on `/usr/abc`, enter:

   ```
   mount -V cachefs -o backfstype=nfs,backpath=/usr/abc,
   cachedir=/cache1 server1:/user2/xyz
   ```

   The lines similar to the following appear in the `etc/mnttab` file after the mount command is executed:

   ```
   server1:/user2 /usr/abc nfs
   /usr/abc /cache1/xyz cachefs backfstype=nfs
   ```

6. To mount a snapshot, type:

   ```
   mount -o snapshot /dev/snapsb /home/janet/snapsb
   ```
This command mounts the snapshot contained on the /dev/snapsb device onto the /home/janet/snapsb directory.

7. To mount a file system and create a snapshot, type:

   mount -o snapto=/dev/snapsb /dev/sb /home/janet/sb

   This command mounts the file system contained on the /dev/sb device onto the /home/janet/sb directory and creates a snapshot for the file system on the /dev/snapsb device.

8. To access files on an SMB server as a local filesystem, type:

   mount -v cifs -n pezman/user1/pass1 -o uid=201,fmode=750 /home /mnt

   This command mounts the file system contained on the /dev/sb device onto the /home/janet/sb directory and creates a snapshot for the file system on the /dev/snapsb device.

Files

/etc/filesystems     Lists the known file systems and defines their characteristics.
/etc/vfs             Contains descriptions of virtual-file-system types.

Related Information
The cdcheck command, cdeject command, cdmount command, cdromd command, cdumount command, cdutil command, nfso command, umount command.

The mntctl subroutine, mount subroutine, umount subroutine.

The filesystems file, vfs file.

For information on installing the Web-based System Manager, see Chapter 2: Installing Web-based System Manager in AIX 5L Version 5.2 Web-based System Manager Administration Guide.


mountd Daemon

Purpose
Answers requests from clients for file system mounts.

Syntax
/usr/sbin/rpc.mountd [-n] [-N]

Description
The mountd daemon is a Remote Procedure Call (RPC) that answers a client request to mount a file system. The mountd daemon finds out which file systems are available by reading the /etc/xtab file.

In addition, the mountd daemon provides a list of currently mounted file systems and the clients on which they are mounted. You can display this list by using the showmount command.

The mountd daemon listens for requests on the port specified in the /etc/services file for the service mountd. If the /etc/services file does not specify a port, one will be chosen when the daemon starts. For example, adding the lines:

   mountd    6666/tcp
   mountd    6666/udp
will cause mountd to listen for requests on port 6666.

**Examples**
The mountd daemon is started from the `/etc/rc.nfs` file. The mountd daemon can be started and stopped by the following System Resource Controller (SRC) commands:

```
startsrc -s rpc.mountd
stopsrc -s rpc.mountd
```

To change the parameters passed to the mountd daemon, use the chssys command. For example:

```
chssys -s rpc.mountd -a Argument
```

The change will not take effect until the daemon is restarted.

**Flags**

- `-n`  Allows clients that use older versions of NFS to mount file systems. This option makes the system less secure. It is the default.
- `-N`  Deny mount requests originating from non-privileged ports. This is the opposite of using the `-n` flag, and is not enabled by default.

**Files**

- `/etc/exports`  Lists the directories that the server can export.
- `/etc/inetd.conf`  Defines how the inetd daemon handles Internet service requests.
- `/etc/xtab`  Lists currently exported directories.
- `/etc/services`  Defines the sockets and protocols used for Internet services. Contains information about the known services used in the DARPA Internet network.

**Related Information**
The chssys command, mount command, showmount command.

The nfsd daemon, portmap daemon.


List of NFS Commands

**mpcfg Command**

**Purpose**

Manages remote maintenance service information.
Syntax

Display Service Information
mpcfg [-d] { -f | -m | -p | -S }

Change Service Information
mpcfg [-c] { -f | -m | -p | -S | -w } Index Value...

Save or Restore Service Information
mpcfg { -r | -s }

Description
The mpcfg command enables a user with root authority to manage service information consisting of the service support and diagnostic flags (-S and -f flags), the modem and site configuration (-m flag), and the remote support phone numbers (-p flag).

The mpcfg command works only on multiprocessor systems with Micro Channel I/O for AIX 5.1 and earlier. For IBM systems, this includes the IBM 7012 Model G Series, the IBM 7013 Model J Series, and the IBM 7015 Model R Series for AIX 5.1 and earlier.

You can perform the following actions:
• Display (-d flag) the service information
• Change (-c flag) the service information
• Save (-s flag) the service information in the /etc/lpp/diagnostics/data/bump file
• Restore (-r flag) the service information to values read in the /etc/lpp/diagnostics/data/bump file.

Note: Generally the mpcfg command is not used directly but through the diag command.

Flags
-c Changes values of service information. The values that you want to modify are identified first by the flag -f, -m, -p or -S, and then by their index (Index parameter) within this category. The new value to assign (Value parameter) directly follows this index in the command. You can specify only one flag among -f, -m, -p, -S but several pairs "Index Value". The flag -w allows to change maintenance passwords.
-d Displays the values of service information according to the -f, -m, -p and -S flags set in the command. These values are displayed associated with their corresponding indexes and names.
-r Reads the service information in the /etc/lpp/diagnostics/data/bump file, and restores it in the non volatile memory (NVRAM).
-s Saves the service information in the /etc/lpp/diagnostics/data/bump file.
-f Indicates that the action (display or change) will be applied to the diagnostic flags.
-m Indicates that the action (display or change) will be applied to the modem and site configuration.
-p Indicates that the action (display or change) will be applied to the remote support phone numbers.
-S Indicates that the action (display or change) will be applied to the service support flags.
-w Indicates that the change will be applied to a password.

Security
Access Control: only the root user can run this command.

Examples
1. To display the modem and site configuration, enter the following command:

mpcfg -d m
This produces output similar to the following:

<table>
<thead>
<tr>
<th>Index</th>
<th>Name</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Modem Parameters File Name</td>
<td>/usr/share/modems/plextel</td>
</tr>
<tr>
<td>2</td>
<td>Service Line Speed</td>
<td>2400</td>
</tr>
<tr>
<td>3</td>
<td>Protocol Inter Data Block Delay</td>
<td>15</td>
</tr>
<tr>
<td>4</td>
<td>Protocol Time Out</td>
<td>30</td>
</tr>
<tr>
<td>5</td>
<td>Retry Number</td>
<td>2</td>
</tr>
<tr>
<td>6</td>
<td>Customer ID</td>
<td>xyz</td>
</tr>
<tr>
<td>7</td>
<td>Login ID</td>
<td>abcd</td>
</tr>
<tr>
<td>8</td>
<td>Password ID</td>
<td>%qw!as</td>
</tr>
</tbody>
</table>

2. To assign the new value 22114433 to the first remote support phone number, enter the following command:

   >mpcfg -c -p 1 22114433

3. To save the service information in the /etc/lpp/diagnostics/data/bump file, enter the following command:

   mpcfg -s

4. To restore the service information from the /etc/lpp/diagnostics/data/bump file to NVRAM, enter the following command:

   mpcfg -r

Files

/usr/sbin/mpcfg Contains the mpcfg command.
/etc/lpp/diagnostics/data/bump Contains the service support and diagnostic flags, remote support phone numbers, and modem and site configuration.

mpcstat Command

Purpose
Displays operational information about a Multi-Protocol Over ATM (MPOA) Client.

Syntax

mpcstat [-a -c -e -i -m -r -s -t -v] [Device_Name]

Description
This command displays Multi-Protocol Over ATM (MPOA) Client operational information gathered by a specified MPOA Client device. If a MPOA Client (MPC) device name is not entered, information for the available MPC appear. You can use the flags to narrow down your search to specify specific categories of information such as Configuration, Egress Cache Entries, Ingress Cache Entries, MPOA Servers, Shortcut Virtual Connections, and Statistics, or you can elect to have all of the information categories display.

You can also toggle debug tracing on or off and reset statistics counters.

Parameters

Device_Name The name of the MPOA Client device name, for example, mpc0.
Flags

- `a` Requests that all of the MPOA Client information appear. Note that this flag does not reset statistics counters or toggle trace. If a flag is not entered, the `-a` flag is the default flag.

- `c` Requests the configuration

- `e` Requests the egress (incoming) cache

- `i` Requests the ingress (outgoing) cache.

- `m` Requests the list of MPOA Servers in use.

- `r` Resets the statistics counters after reading.

- `s` Requests the statistics counters.

- `t` Toggles full debug trace on or off.

- `v` Requests the list of Shortcut Virtual Connections.

The following information appears for all valid calls and contains the following fields:

**Device Name**

Displays the device name of the MPOA Client.

**MPC State**

Displays the current state of the MPOA Client.

**Example States:**

<table>
<thead>
<tr>
<th>State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Idle</td>
<td>Registering with the ELAN.</td>
</tr>
<tr>
<td>Initializing</td>
<td>Registering with the switch.</td>
</tr>
<tr>
<td>Operational</td>
<td>Fully operational.</td>
</tr>
<tr>
<td>Network Down</td>
<td>Network is currently unavailable.</td>
</tr>
</tbody>
</table>

**MPC Address**

Displays the MPOA Client’s 20-byte ATM address for a specific ATM adapter port device name. The adapter port device name is also displayed.

**Elapsed Time**

Displays the real time period which has elapsed since statistics were last reset.

**MPC Configuration**

Selected with the `-a` or `-c` flags. Displays the network administrator’s pre-configured attributes for the MPOA Client, or the values provided by a Lan Emulation Configuration Server (LECS).

**MPC Egress Cache**

Selected with the `-a` or `-e` flags. Displays the current egress cache entries. Included are the state of the entry, its Level-3 address, and ATM shortcut address, as well as additional descriptive values associated with each entry.

**Example States**

<table>
<thead>
<tr>
<th>State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Active</td>
<td>Has active shortcut connection.</td>
</tr>
<tr>
<td>Purging</td>
<td>Purging the egress MPOA Server entry.</td>
</tr>
<tr>
<td>DP Purge</td>
<td>Purging the remote MPOA Client data plane.</td>
</tr>
<tr>
<td>Inactive</td>
<td>No current activity on shortcut connection.</td>
</tr>
</tbody>
</table>

**MPC Ingress Cache:**

Selected with the `-a` or `-i` flags. Displays the current ingress cache entries. Included are the state of the entry, its Level-3 address, and ATM shortcut address, as well as additional descriptive values associated with each entry.

**Example States**

<table>
<thead>
<tr>
<th>State</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow Detect</td>
<td>Waiting for packet threshold to enable shortcut.</td>
</tr>
<tr>
<td>Resolution</td>
<td>Packet threshold reached, resolving shortcut.</td>
</tr>
<tr>
<td>Hold Down</td>
<td>Shortcut resolution failed, waiting for retry.</td>
</tr>
<tr>
<td>Resolved</td>
<td>Shortcut resolution sequence complete.</td>
</tr>
</tbody>
</table>
MPOA Server List
Selected with the -a or -m flags. Displays a list of MPOA Servers currently known by this MPC. Included in each entry are the name of the LE Client that identified the MPS, the MPS ATM address, and the MPS LAN MAC address.

MPC Statistics
Selected with the -a or -s flags. Displays the current Transmit, Receive, and General statistics for this MPOA Client.

Shortcut Virtual Connection
Selected with the -a or -v flags. Displays the current list of shortcut virtual circuits in use by the MPOA client. Included are virtual path and channel values, VC state, ATM device name, as well as additional descriptive values associated with each entry.

Example States:
Idle Call idle.
Signalling Call placed but not established.
Operational Call connected; data path valid.
Released Call released.
Retry Temporary call failure; will be retried.
Hold Down Call failure; will be suspended for hold down period.

Exit Status
If an invalid Device_Name is specified, this command produces error messages stating that it could not connect to the device. Note that MPOA is a protocol extension to the ATM LAN Emulation protocol, and must have a corresponding and available LE Client to be operational. Examples of an invalid device error message are:
MPCSTAT: Device is not an MPOA device.
MPCSTAT: No LEC device with MPOA enabled.
MPCSTAT: Device is not available.

Related Information
The atmstat command, entstat command, lecstat command, and tokstat command.

mrouted Daemon

Purpose
Forwards a multicast datagram. This daemon only applies to AIX 4.2.1 or later.

Syntax
/usr/sbin/mrouted [ -P ] [ -C Config_File ] [ -D Debug_Level ]

Description
The mrouted daemon is an implementation of the Distance Vector Multicast Routing Protocol (DVMRP), an earlier version of which is specified in RFC 1075. It maintains topological knowledge using a distance vector routing protocol (like RIP, described in RFC 1058), on which it implements a multicast datagram forwarding algorithm called Reverse Path Multicasting.

The mrouted daemon forwards a multicast datagram along a shortest (reverse) path tree rooted at the subnet on which the datagram originates. The multicast delivery tree may be thought of as a broadcast delivery tree that has been pruned back so that it does not extend beyond those subnetworks that have members of the destination group. Hence, datagrams are not forwarded along those branches that have no listeners of the multicast group. The IP time-to-live of a multicast datagram can be used to limit the range of multicast datagrams.
To support multicasting among subnets that are separated by (unicast) routers that do not support IP multicasting, the mrouted daemon includes support for tunnels, which are virtual point-to-point links between pairs of the mrouted daemons located anywhere in an Internet. IP multicast packets are encapsulated for transmission through tunnels, so that they look like typical unicast datagrams to intervening routers and subnets. The encapsulation is added on entry to a tunnel, and stripped off on exit from a tunnel. By default, the packets are encapsulated using the IP-in-IP protocol (IP protocol number 4). Older versions of the mrouted tunnel use IP source routing, which puts a heavy load on some types of routers. This version does not support IP source-route tunneling.

The tunneling mechanism allows the mrouted daemon to establish a virtual Internet, for the purpose of multicasting only, which is independent of the physical Internet and which may span multiple Autonomous Systems. This capability is intended for experimental support of Internet multicasting only, pending widespread support for multicast routing by the regular (unicast) routers. The mrouted daemon suffers from the well-known scaling problems of any distance-vector routing protocol and does not support hierarchical multicast routing.

The mrouted daemon automatically configures itself to forward on all multicast-capable interfaces (that is, interfaces that have the IFF_MULTICAST flag set, excluding the loopback interface), and it finds other mrouted daemons directly reachable using those interfaces.

The mrouted daemon does not initiate execution if it has fewer than two enabled virtual interfaces, where a virtual interface (Vif) is either a physical multicast-capable interface or a tunnel. It logs a warning if all of its virtual interfaces are tunnels; such an mrouted daemon’s configuration would be better replaced by more direct tunnels.

The mrouted daemon handles multicast routing only; there might be unicast-routing software running on the same machine as the mrouted daemon. With the use of tunnels, it is unnecessary for the mrouted daemon to have access to more than one physical subnet to perform multicast forwarding.
Flags

-\texttt{-c Config\_File} \hspace{1em} Starts the \texttt{mrouted} command using an alternate configuration file specified by the \texttt{Config\_File} variable.

There are five types of configuration entries:

\begin{verbatim}
phyint local-addr [disable] [metric m] [threshold t] [rate_limit b]
[boundary (boundary-name|scoped-addr/mask-len)] [altnet
network/mask-len]
\end{verbatim}

\begin{verbatim}
tunnel local-addr remote-addr
 [ metric m ]
 [ threshold t ]
 [ rate_limit b ]

[ boundary
 ( boundary-name
 | scoped-addr
 / mask-len
 )]
\end{verbatim}

\begin{verbatim}
cache_lifetime ct
pruning off
 |
on
name boundary-name scoped-addr
 / mask-len
\end{verbatim}

See [mrouted.conf File] in AIX 5L Version 5.2 Files Reference for more information.

-\texttt{-d} \hspace{1em} Sets the debug level. If no -d option is given, or if the debug level is specified as 0, the mrouted daemon detaches from the invoking terminal. Otherwise, it remains attached to the invoking terminal and responsive to signals from that terminal. If -d is given with no argument, the debug level defaults to 2. Regardless of the debug level, the mrouted daemon always writes warning and error messages to the system log daemon. Non-zero debug levels have the following effects:

\begin{enumerate}
\item [level 1] All syslog'ed messages are also printed to stderr.
\item [level 2] All level 1 messages plus notifications of significant events are printed to stderr.
\item [level 3] All level 2 messages plus notifications of all packet arrivals and departures are printed to stderr.
\end{enumerate}

Upon startup, the mrouted daemon writes its pid to the file /etc/mrouted.pid.

-\texttt{-p} \hspace{1em} Turns off pruning. Default is pruning enabled.

Signals

The following signals can be sent to the mrouted daemon:

\begin{itemize}
\item [HUP] Restarts the mrouted daemon. The configuration file is reread every time this signal is evoked.
\item [INT] Terminates execution gracefully (that is, by sending good-bye messages to all neighboring routers).
\item [TERM] Same as INT.
\item [USR1] Dumps the internal routing tables to /usr/tmp/mrouted.dump.
\end{itemize}
USR2  Dumps the internal cache tables to /usr/tmp/mrouted.cache.
QUIT  Dumps the internal routing tables to stderr (if the mrouted daemon was invoked with a nonzero debug level).

For convenience in sending signals, the mrouted daemon writes its pid to /etc/mrouted.pid on startup.

Examples

1. To display routing table information, type:
   kill -USR1 *cat /etc/mrouted.pid*

   This produces the following output:

   Virtual Interface Table
   Vif Local-Address  Metric  Thresh  Flags
   0 36.2.0.8  subnet: 36.2  1  1  querier
   groups: 224.0.2.1  224.0.0.4
   pkts in: 3456
   pkts out: 2322323
   1 36.11.0.1  subnet: 36.11  1  1  querier
   groups: 224.0.2.1  224.0.1.0  224.0.0.4
   pkts in: 345
   pkts out: 3456
   2 36.2.0.8  tunnel: 36.8.0.77  3  1
   peers: 36.8.0.77 (2.2)
   boundaries: 239.0.1  239.1.2
   pkts in: 34545433
   pkts out: 234342
   3 36.2.0.8  tunnel: 36.6.8.23  3  16

   Multicast Routing Table (1136 entries)
   Origin-Subnet  From-Gateway  Metric  Tmr  In-Vif  Out-Vifs
   36.2  1  45  0  1* 2 3*
   36.8  36.8.0.77  4  15  2  0* 1* 3*
   36.11  1  20  1  0* 2 3*

   In this example, there are four virtual interfaces connecting to two subnets and two tunnels. The Vif 3 tunnel is not in use (no peer address). The Vif 0 and Vif 1 subnets have some groups present; tunnels never have any groups. This instance of the mrouted daemon is the one responsible for sending periodic group membership queries on the Vif 0 and Vif 1 subnets, as indicated by the querier flags. The list of boundaries indicate the scoped addresses on that interface. A count of the no. of incoming and outgoing packets is shown at each interface.

   Associated with each subnet from which a multicast datagram can originate is the address of the previous hop router (unless the subnet is directly connected), the metric of the path back to the origin, the amount of time since an update for this subnet was last received, the incoming virtual interface for multicasts from that origin, and a list of outgoing virtual interfaces. The * (asterisk) means that the outgoing virtual interface is connected to a leaf of the broadcast tree rooted at the origin, and a multicast datagram from that origin will be forwarded on that outgoing virtual interface only if there are members of the destination group on that leaf.

   The mrouted daemon also maintains a copy of the kernel forwarding cache table. Entries are created and deleted by the mrouted daemon.
2. To display cache table information, type:

```bash
kill -USR2 *cat /etc/mrouted.pid*
```

This produces the following output:

```
Multicast Routing Cache Table (147 entries)
Origin Mcast-group CTmr Age Ptmr IVif Forwvifs
13.2.116/22 224.2.127.255 3m 2m -  0  1
>13.2.116.19
>13.2.116.196
138.96.48/21 224.2.127.255 5m 2m -  0  1
>138.96.48.108
128.9.160/20 224.2.127.255 3m 2m -  0  1
>128.9.160.45
198.106.194/24 224.2.135.190 9m 28s 9m 0P
>198.106.194.22
```

Each entry is characterized by the origin subnet number and mask and the destination multicast group. The CTmr field indicates the lifetime of the entry. The entry is deleted from the cache table when the timer decrements to zero. The Age field is the time since this cache entry was originally created. Because cache entries get refreshed if traffic is flowing, routing entries can grow very old. The Ptmr field is a hyphen if no prune was sent upstream or the amount of time until the upstream prune will time out. The IVif field indicates the incoming virtual interface for multicast packets from that origin. Each router also maintains a record of the number of prunes received from neighboring routers for a particular source and group. If there are no members of a multicast group on any downward link of the multicast tree for a subnet, a prune message is sent to the upstream router. They are indicated by a P after the virtual interface number. The Forwvifs field shows the interfaces along which datagrams belonging to the source group are forwarded. A p indicates that no datagrams are being forwarded along that interface. An unlisted interface is a leaf subnet with are no members of the particular group on that subnet. A b on an interface indicates that it is a boundary interface, that is, traffic will not be forwarded on the scoped address on that interface. An additional line with a > (greater-than sign) as the first character is printed for each source on the subnet. There can be many sources in one subnet.

**Files**

- `/etc/mrouted.conf` Contains the configuration information for the `mrouted` daemon.
- `/usr/tmp/mrouted.dump` Contains the internal routing tables for the `mrouted` daemon.
- `/etc/mrouted.pid` Contains the process ID for the `mrouted` daemon.
- `/usr/tmp/mrouted.cache` Contains the internal cache tables for the `mrouted` daemon.

**Related Information**

- `/etc/mrouted.conf` File in *AIX 5L Version 5.2 Files Reference*.

---

**msgchk Command**

**Purpose**
Checks for messages.

**Syntax**

```
msgchk [ User ... ]
```

**Description**

The `msgchk` command checks mail drops for messages. The `msgchk` command reports whether the mail drop for the specified user contains messages and indicates if the user has already seen these messages. By default, the `msgchk` command checks the mail drop for the current user.


**Flags**

- **help** Lists the command syntax, available switches (toggles), and version information.

  **Note:** For the Message Handler (MH), the name of this flag must be fully spelled out.

**Examples**

1. To check to see if you have any new messages, enter:
   
   ```
   msgchk
   ```

   If you have new messages, the system responds with a message similar to the following:
   
   You have new Internet mail waiting

   If you have no messages, the system responds with a message similar to the following:
   
   You don't have any mail waiting

2. To check to see if user karen on your local system has any new messages, enter:
   
   ```
   msgchk karen
   ```

   In this example, if user karen on your local system has new messages, the system responds with a message similar to the following:
   
   karen has new Internet mail waiting

   If user karen on your local system has no messages, the system responds with a message similar to the following:
   
   karen doesn't have any mail waiting

**Files**

- **$HOME/.mh_profile** Contains the user's MH profile.
- **/etc/mh/mtstailor** Contains the MH tailor file.
- **/var/spool/Mail/$USER** Defines the location of the mail drop.
- **/usr/bin/msgchk** Contains the **msgchk** command.

**Related Information**

The **inc** command.

The **mh_alias** file format, **mh_profile** file format.

[Mail Overview](AIX 5L Version 5.2 System User's Guide: Communications and Networks).

**msh Command**

**Purpose**

Creates a Message Handler (MH) shell.

**Syntax**

```
  msh [ File ] [ -prompt String ] [ -notopcur | -topcur ]
```
Description

The msh command creates an MH shell for use with messages that are packed in a file. By default, this command looks for the msgbox file in the current directory. Within the MH shell, you can use the following MH commands:

<table>
<thead>
<tr>
<th>ali</th>
<th>burst</th>
<th>comp</th>
<th>dist</th>
</tr>
</thead>
<tbody>
<tr>
<td>folder</td>
<td>forw</td>
<td>inc</td>
<td>mark</td>
</tr>
<tr>
<td>mhmile</td>
<td>msgchk</td>
<td>next</td>
<td>packf</td>
</tr>
<tr>
<td>pick</td>
<td>prev</td>
<td>refile</td>
<td>repl</td>
</tr>
<tr>
<td>rmrm</td>
<td>scan</td>
<td>send</td>
<td>show</td>
</tr>
<tr>
<td>sortm</td>
<td>whatnow</td>
<td>whom</td>
<td></td>
</tr>
</tbody>
</table>

These commands operate with limited functionality in the MH shell. To see how a command operates in the MH shell, enter the command name followed by the -help flag. Entering help or a ? (question mark) displays a list of the MH commands you can use.

To leave the msh shell, press the Ctrl-D key sequence or enter quit.

Flags

- **-help**
  Lists the command syntax, available switches (toggles), and version information.
  
  **Note:** For MH, the name of this flag must be fully spelled out.

- **-notopcur**
  Makes the current message track the center line of the vmh scan window when the msh command is started using the vmh command. This flag is the default.

- **-prompt String**
  Prompts for the msh commands with the specified string.

- **-topcur**
  Makes the current message track the top line of the vmh scan window when the msh command is started using the vmh command.

Profile Entries

The following entries are found in the UserMhDirectory/.mh_profile file:

<table>
<thead>
<tr>
<th>fileproc:</th>
<th>Specifies the program used to refile messages.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Msg-Protect:</td>
<td>Sets the protection level for your new message files.</td>
</tr>
<tr>
<td>Path:</td>
<td>Specifies the user's MH directory.</td>
</tr>
<tr>
<td>showproc:</td>
<td>Specifies the program used to show messages.</td>
</tr>
</tbody>
</table>

Examples

1. To start an msh shell, enter:

   msh

   If the msgbox file exists in the current directory, the system responds with a message similar to the following:

   Reading ./msgbox, currently at message 1 of 10

   Then, the system prompt appears as follows:

   (msh)

   In this example, the current message is message 1 in the msgbox file. You can now enter a modified subset of MH commands.

2. To start an msh shell to manipulate the messages stored in the meetings file, enter:

   msh meetings
Files

$HOME/.mh_profile    Specifies the user’s MH profile.
/etc/mh/mhmtailor    Contains the MH tailor file.
/usr/bin/msh         Contains the msh command.

Related Information

The ali command, burst command, comp command, dist command, folder command, forw command, inc command, mark command, mhmail command, msgchk command, next command, pack command, pick command, prev command, refile command, repl command, rmm command, scan command, send command, show command, sortm command, vmh command, whatnow command, whom command.

The mh_alias file format, mh_profile file format.

Mail Overview in AIX 5L Version 5.2 System User’s Guide: Communications and Networks.

---

**mt Command** (BSD)

**Purpose**

Gives subcommands to streaming tape device.

**Syntax**

```
mt [ -f TapeName ] Subcommand [ Count ]
```

**Description**

The mt command gives subcommands to a streaming tape device. If you do not specify the -f flag with the TapeName parameter, the TAPE environment variable is used. If the environment variable does not exist, the mt command uses the /dev/rmt0.1 device. The TapeName parameter must be a raw (not block) tape device. You can specify more than one operation with the Count parameter.

**Subcommands**

- **eof, weof**: Writes the number of end-of-file markers specified by the Count parameter at the current position on the tape.
- **fsf**: Moves the tape forward the number of files specified by the Count parameter and positions it to the beginning of the next file.
- **bsf**: Moves the tape backwards the number of files specified by the Count parameter and positions it to the beginning of the last file skipped. If using the bsf subcommand would cause the tape head to move back past the beginning of the tape, then the tape will be rewound, and the mt command will return EIO.
- **fsr**: Moves the tape forward the number of records specified by the Count parameter.
- **bsr**: Moves the tape backwards the number of records specified by the Count parameter.
- **rewoff1, rewind**: Rewinds the tape. The Count parameter is ignored.
- **status**: Prints status information about the specified tape device. The output of the status command may change in future implementations.

**Flag**

- **-f TapeName**: Specifies the TapeName parameter.
Examples
1. To rewind the rmt1 tape device, enter:
   `mt -f /dev/rmt1 rewind`
2. To move forward two files on the default tape device, enter:
   `mt fsf 2`
3. To write two end-of-file markers on the tape in the /dev/rmt0.6 file, enter:
   `mt -f /dev/rmt0.6 weof 2`

Exit Status
0 Indicates a successful completion.
>0 Indicates an error occurred.

Files
/dev/rmt/n.n  Specifies the raw streaming tape interface.
/usr/bin/mt  Contains the mt command file.

Related Information
The `tctl` command.
The `environment` file, `rmt` special file.
The `ioctl` subroutine.


mtrace Command

Purpose
Prints a multicast path from a source to a receiver.

Syntax

Description
A trace query is passed hop-by-hop along the path from the receiver to the source, collecting hop addresses, packet counts, and routing error conditions along the path, and then the response is returned to the requestor. The default receiver is the host running the mtrace command, and the default group is 0.0.0.0.

Note: The mtrace command is intended for use in network testing, measurement, and management. Because the mtrace command heavily loads on the network, avoid using the mtrace command during typical operations or from automated scripts. It should be used primarily or with manual fault isolation. If the -g flag is specified, the source defaults to the host running mtrace and the receiver defaults to the router being addressed.

By default, the mtrace command first attempts to trace the full reverse path, unless the number of hops to trace is explicitly set with the -m flag. If there is no response within a 3-second timeout interval (changed
with the \texttt{-w} flag), an * (asterisk) is printed and the probing switches to hop-by-hop mode. Trace queries are
issued starting with a maximum hop count of one and increasing by one until the full path is traced or no
response is received. At each hop, multiple probes are sent (default is three, changed with \texttt{-q} flag). The
first half of the attempts (default is two) are made with the reply address set to standard multicast address,
\texttt{mtrace.mcast.net} (224.0.1.32) with the \texttt{ttl} set to 32 more than what is needed to pass the thresholds seen
so far along the path to the receiver. For each additional attempt, the \texttt{ttl} is increased by another 32 each
time up to a maximum of 192. Because the desired router may not be able to send a multicast reply, the
remainder of the attempts request that the response be sent via unicast to the host running the \texttt{mtrace}
command.

Alternatively, the multicast \texttt{ttl} can be set explicitly with the \texttt{-t} flag, the initial multicast attempts can be
forced to use unicast instead with the \texttt{-U} flag, the final unicast attempts can be forced to use multicast
instead with the \texttt{-M} flag, or if you specify \texttt{-UM}, the \texttt{mtrace} command will first attempt using unicast and
then multicast. For each attempt, if no response is received within the timeout, an * (asterisk) is printed.
After the specified number of attempts have failed, the \texttt{mtrace} command will try to query the next hop
router with a \texttt{DVMRP\_ASK\_NEIGHBORS2} request to see what kind of router it is. The \texttt{mtrace} command
will try to query three (changed with the \texttt{-e} flag) hops past a non-responding router. Even though the
\texttt{mtrace} command is incapable of sending a response, it might be capable of forwarding the request.

\textbf{Flags}

- \texttt{-g gateway} Sends the trace query via unicast directly to the multicast router \texttt{gateway} rather than multicasting
the query. This must be the last-hop router on the path from the intended source to the receiver.

- \texttt{-i if\_addr} Uses \texttt{if\_addr} as the local interface address (on a multi-homed host) for sending the trace query and
as the default for the receiver and the response destination.

- \texttt{-l} Loops indefinitely printing packet rate and loss statistics for the multicast path every 10 seconds
(see \texttt{-S stat\_int}).

- \texttt{-m max\_hops} Sets \texttt{max\_hops} to the maximum number of hops that will be traced from the receiver to the source.
The default is 32 hops and infinity for the DVMRP routing protocol.

- \texttt{-M} Always requests the response using multicast rather than attempting unicast for the last half of the
tries.

- \texttt{-n} Prints hop addresses numerically rather than symbolically and numerically (saves a name server
address-to-name lookup for each router found on the path).

- \texttt{-p} Listens passively for multicast responses from traces initiated by others. This works best when run
on a multicast router.

- \texttt{-q nqueries} Sets the maximum number of query attempts for any hop to \texttt{nqueries}. The default is 3.

- \texttt{-r resp\_dest} Sends the trace response to \texttt{dhost} rather than to the host on which the \texttt{mtrace} command is being
run, or to a multicast address other than the one registered for this purpose (224.0.1.32).

- \texttt{-s} Prints a short form output including only the multicast path and not the packet rate and loss
statistics.

- \texttt{-S stat\_int} Changes the interval between statistics gathering traces to \texttt{stat\_int} seconds (default 10 seconds).

- \texttt{-t \texttt{ttl}} Sets the \texttt{ttl} (time-to-live, or number of hops) for multicast trace queries and responses. The default
is 127, except for local queries to the all routers multicast group that use the \texttt{ttl} of 1.

- \texttt{-U} Forces initial multicast attempts to use unicast instead.

- \texttt{-w wait} Sets the time to wait for a trace response to \texttt{wait} seconds (default 3 seconds).

\textbf{Parameters}

\texttt{source} Specifies the host for which the multicast path from a particular receiver is sought. This is a
required parameter.
receiver Specifies the host from which the multicast path is sought for a particular source. Default is the host in which the mtrace command is running. This is an optional parameter.

group Specifies the multicast group. This is an optional parameter.

Examples
In the following example, the two machines, 10.27.41.57 and 10.27.40.20, are on two different subnets separated by a router having two interfaces, 10.27.40.11 and 10.27.41.11. To find the multicast path from 10.27.40.20 to 10.27.41.57, type the following:

```
# mtrace 10.27.41.57 224.2.0.1
```

The following is displayed:

```
Mtrace from 10.27.41.57 to 10.27.40.20 via group 224.2.0.1
Querying full reverse path...
  0  ? (10.27.40.20)
-1  ? (10.27.40.11)  DVMRP  thresh^ 1
-2  ? (10.27.41.57)
Round trip time 1 ms; total ttl of 2 required.
```

Waiting to accumulate statistics... Results after 10 seconds:

<table>
<thead>
<tr>
<th>Source</th>
<th>Response Dest</th>
<th>Overall</th>
<th>Packet Statistics For Traffic From 10.27.41.57 224.0.1.32 224.2.0.1</th>
<th>Rate</th>
<th>Lost/Sent = Pct Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.27.41.57</td>
<td>224.0.1.32</td>
<td>Packet</td>
<td>10.27.41.57 To 224.2.0.1</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>v</td>
<td>rtt 1 ms</td>
<td>Rate</td>
<td>Lost/Sent = Pct Rate</td>
</tr>
<tr>
<td>10.27.41.11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10.27.40.11</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10.27.40.20</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Related Information
Related commands include the mrouted daemon and the traceroute command.

mv Command

Purpose
Moves files.

Syntax

**To Move and Rename a File**

```
mv [E forcelignore|warn] ] [i | f ] SourceFile TargetFile
```

**To Move and Rename a Directory**

```
mv [E forcelignore|warn] ] [i | f ] SourceDirectory ... TargetDirectory
```

**To Move Files to a Directory Maintaining Original File Names**

```
mv [E forcelignore|warn] ] [i | f ] SourceFile ... TargetDirectory
```

Description

Attention: The mv command can overwrite many existing files unless you specify the -i flag. The -i flag prompts you to confirm before it overwrites a file. If both the -i and -f flags are specified in combination, the last flag specified takes precedence.
The **mv** command moves files and directories from one directory to another or renames a file or directory. If you move a file or directory to a new directory, it retains the base file name. When you move a file, all links to other files remain intact, except when you move it to a different file system. When you move a directory into an existing directory, the directory and its contents are added under the existing directory.

When you use the **mv** command to rename a file or directory, the *TargetDirectory* parameter can specify either a new file name or a new directory path name.

If moving the file would overwrite an existing file that does not have write-permission set and if standard input is a workstation, the **mv** command displays the file-permission code and reads a line from standard input. If that line begins with a *y* or the locale’s equivalent of a *y*, the **mv** command moves the file. If the response is anything other than a *y*, the **mv** command does nothing to that file and continues with the next specified file. The file-permission code displayed may not fully represent the access permission if the *TargetFile* is associated with an ACL. When the parent directory of the *SourceFile* is writable and has the sticky bit set, one or more of the following conditions are true:

- The user must own the file.
- The user must own the directory
- The user must be a privileged user.
- The file must be writable by the user.

This warning message and prompt for input can be overridden by using the **-f** option.

You can use the **mv** command to move files within the same file system or between file systems. Whether you are working in one file system or across file systems, the **mv** command copies the file to the target and deletes the original file. The **mv** command preserves in the new file the time of the most recent data modification, the time of the most recent access, the user ID, the group ID, and the file mode of the original file. For symbolic links, the **mv** command preserves only the owner and group of the link itself.

If it is unable to preserve the owner and group ID, the **mv** command clears S_ISUID and S_ISGID bits in the target. The **mv** command prints a diagnostic message to stderr if it is unable to clear these bits, though the exit code is not affected.

The **mv** command modifies either the source file or the destination path if the command is prematurely terminated.

**Note:** The **mv** command supports the **—** (dash, dash) parameter as a delimiter that indicates the end of the flags.

**Flags**

**Attention:** The **mv** command can overwrite many existing files unless you specify the **-i** flag. The **-i** flag prompts you to confirm before it overwrites a file. If both the **-f** and **-i** flags are specified in combination, the last flag specified takes precedence.

- **-E** The **-E** option requires one of the following arguments. If you omit the **-E** option, **warn** is the default behavior.
  - **force** Fails the **mv** operation on a file if the fixed extent size or space reservation of the file cannot be preserved.
  - **ignore** Ignores any errors in preserving extent attributes.
  - **warn** Issues a warning if the space reservation or the fixed extent size of the file cannot be preserved.
- **-f** Does not prompt you before overwriting an existing file.
- **-i** Prompts you before moving a file or directory to an existing path name by displaying the name of the file followed by a question mark. If you answer with a line starting with *y* or the locale’s equivalent of a *y*, the move continues. Any other reply prevents the move from occurring.
Examples

1. To rename a file, enter:
   `mv appendix apndx.a`
   This command renames `appendix` to `apndx.a`. If a file named `apndx.a` already exists, its old contents are replaced with those of `appendix`.

2. To move a directory, enter:
   `mv book manual`
   This command moves all files and directories under `book` to the directory named `manual`, if `manual` exists. Otherwise, the directory `book` is renamed `manual`.

3. To move a file to another directory and give it a new name, enter:
   `mv intro manual/chap1`
   This command moves `intro` to `manual/chap1`. The name `intro` is removed from the current directory, and the same file appears as `chap1` in the directory `manual`.

4. To move a file to another directory, keeping the same name, enter:
   `mv chap3 manual`
   This command moves `chap3` to `manual/chap3`

   **Note:** Examples 1 and 3 name two files, example 2 names two existing directories, and example 4 names a file and a directory.

5. To move several files into another directory, enter:
   `mv chap4 jim/chap5 /home/manual`
   This command moves the `chap4` file to the `/home/manual/chap4` file directory and the `jim/chap5` file to the `/home/manual/chap5` file.

6. To use the `mv` command with pattern-matching characters, enter:
   `mv manual/* .`
   This command moves all files in the `manual` directory into the current directory . (period), retaining the names they had in `manual`. This move also empties `manual`. You must type a space between the asterisk and the period.

   **Note:** Pattern-matching characters expand names of existing files only. For example, the command `mv intro man*/chap1` does not work if the file `manual/chap1` does not exist.

Exit Status

0   All input files were moved successfully.
>0  An error occurred.

Files

`/usr/bin/mv` Contains the `mv` command.

Related Information

The `chmod` command, `ln` command, `rm` command.

The `rename` subroutine.


mvdir Command

Purpose
Moves (renames) a directory.

Syntax
mvdir Directory1 Directory2

Description
The mvdir command renames directories within a file system. To use the mvdir command, you must have write permission to Directory1 and Directory2 as well as in the parent directories.

The Directory1 parameter must name an existing directory. If Directory2 does not exist, Directory1 is moved to Directory2. If Directory2 exists, Directory1 becomes a subdirectory of Directory2. Neither directory can be a subset of the other.

The mvdir Command can also be used to move or rename files. If the Directory1 parameter is an existing file name and the Directory2 parameter is an existing directory name, the file specified by Directory1 is moved to the directory specified by Directory2. If the Directory1 parameter is an existing file name and the Directory2 parameter does not yet exist, Directory2 replaces the file name Directory1. If both are existing file names, the file specified by Directory1 is renamed Directory2, and the existing Directory2 is removed.

The mv command provides the same functionality as the mvdir command.

Example
To rename or move a directory to another location, enter:

mvdir appendixes manual

If manual does not exist, this renames the appendixes directory to manual.

If a directory named manual already exists, this moves appendixes and its contents to manual/appendixes. In other words, appendixes becomes a subdirectory of manual.

Files

/usr/sbin/mvdir Contains the mvdir command.

Related Information
The mkdir command, mv command.

The rename subroutine.


mvfilt Command

Purpose
Moves a filter rule.

Syntax
```
```

Description
Use the `mvfilt` command to change the position of a filter rule in the filter rule table.

Flags
- **-v**  
  IP version of the filter rule. The value 4 specifies IP version 4 and the value 6 specifies IP version 6.
- **-p**  
  Filter rule ID. It specifies the previous position of the rule in the filter rule table. For IP version 4, the value of 1 is invalid since the first filter rule is unmoveable.
- **-n**  
  Filter rule ID. It specifies the new position of the rule in the filter rule table after the move. For IP version 4, the value of 1 is invalid since the first filter rule is reserved and thus is unmoveable.

mvt Command

Purpose
Typesets English-language view graphs and slides.

Syntax
```
mvt [-a] [-c] [-e] [-g] [-p] [-t] [-z] [-T Name] [-D Destination] [ File ... ]
```

Description
The `mvt` command typesets its input with the `mv` macro package for view graphs and slides in a manner similar to the `mmt` command. The `mvt` command has flags to specify preprocessing by the `tbl`, `eqn`, `pic`, `cw`, and `grap` commands. The flags you select determine which pipelines, flags, and parameters are generated for the `troff` command and the macro package.

The `mvt` command, unlike the `troff` command, automatically pipes its output to a postprocessor, unless specifically requested not to do so. The user should not specify a postprocessor when using the `mvt` command. The path that the `mvt` command takes is as follows:
1. The -z flag (no postprocessor is used).
2. The -TName flag.
3. The TYPESETTER environment variable is read.
4. The default is set to `ibm3816`.

*File* specifies the file that the `mvt` command formats.

Flags
Flags can occur in any order, but they must be displayed before the *File* parameter. If no file is specified, the `mvt` command prints a list of its flags.

- **-a**  
  Displays readable `troff` output to the terminal.
- **-c**  
  Calls the `cw` command.
-e       Calls the eqn command; also causes the eqn command to read the 
        /usr/share/lib/pub/eqnchar file.
-g       Calls the grap command, which in turn calls the pic command.
-p       Calls the pic command.
-t       Calls the tbl command.
-z       Calls no output filter (or postprocessor) to process or redirect the output of the troff 
        command.
-D       Directs the output to the specified device destination. Supported value for the Destination 
        variable is 4014, which is the Tektronix 4014 terminal by way of the tc command.
-T       Creates output for the troff device as specified by the Name variable. The output is sent 
        through the appropriate postprocessor. The default is ibm3816.
-       Forces input to be read from standard input.

Any other parameters or flags that you give the mvt command (such as the -a flag) are passed to the troff 
command.

The mvt command reads standard input when you specify the - (minus) flag instead of the File parameter.

Use the -oList flag of the troff command to specify ranges of pages to be output.

    Note: If you call the mvt command with one or more of the -e, -c, -t, -p, -g, or - flags, together with 
    the -oList flag of the troff command, you may receive a broken pipe message. This occurs if you do 
    not specify the last page of the document in the List variable. This broken pipe message is not an 
    indication of any problem and can be ignored.

Environment Variables

TYPESETTER Contains information about a particular printing device.

Files

/usr/share/lib/pub/eqnchar Contains special character definitions.

Related Information

The cw command, eqn command, grap command, mmt command, pic command, tbl command, tc 
command, troff command.

The eqnchar file format.

The article mv Macro Package for the mvt and troff Commands in the troff Command.

mwm Command

Purpose

Runs the AIXwindows Window Manager (MWM).

Syntax

  mwm [-display Host:Display:ScreenID] [-xrm ResourceString] [-multiscreen -name Name] [-screens Name [ 
  Name ... ] ]
Description

The `mwm` command runs the AIxwindows Window Manager (MWM) and is often started by a display or session manager. The AIxwindows Window Manager (MWM) is an X Window System client that provides window management functionality and some session management functionality. It provides functions that facilitate control (by the user and the programmer) of elements of window states such as placement, size, icon or normal display, and input-focus ownership. It also provides session management functions such as stopping a client.

The appearance and behavior of the window manager can be altered by changing the configuration of specific resources. Resources are defined under [X Defaults](#).

By default, the `mwm` command manages only the single screen specified by the `-display` option or the DISPLAY environment variable (by default, screen 0). If the `-multiscreen` option is specified or if the multiScreen resource is True, the `mwm` command tries to manage all the screens on the display.

When the `mwm` command is managing multiple screens, the `-screens` option can be used to give each screen a unique resource name. The names are separated by blanks, for example, `-screens mwm0 mwm1`. If there are more screens than names, resources for the remaining screens are retrieved using the first name. By default, the screen number is used for the screen name.

For information on windows, icons, resources, events, button and key bindings, menus, and variables, see the following sections:

- Windows
- Icons
- Icon Box
- Component Appearance Resources
- General Appearance and Behavior Resources
- Client-Specific Resources
- Window Manager Event Specification
- Button Bindings
- Key Bindings
- Menu Panes
- Environment
- Related Information

Flags

- `-display Host:Display:ScreenID` Specifies the display to use. The `-display` option has the following parameters:
  
  _Host_ Specifies the host name of a valid system on the network. Depending on the situation, this could be the host name of the user or the host name of a remote system.

  _Display_ Specifies the number (usually 0) of the display on the system on which the output is to be displayed.

  _ScreenID_ Specifies the number of the screen where the output is to be displayed. This number is 0 for single-screen systems.

- `-xrm ResourceString` Enables the named resources when starting the `mwm` command.

- `-multiscreen` Causes the `mwm` command to manage all screens on the display. The default is to manage only a single screen.

- `-name Name` Causes the `mwm` command to retrieve its resources using the specified name, as in `Name*Resource`. 
-screens Name [Name [...]]

Specifies the resource names to use for the screens managed by MWM. If MWM is managing a single screen, only the first name in the list is used. If multiple screens are being managed, the names are assigned to the screens in order, starting with screen 0. For example, screen 0 gets the first name and screen 1 gets the second name.

Windows

Default window manager window frames have the following distinct components with associated functions:

title area

In addition to displaying the client’s title, the title area is used to move the window. To move the window, place the pointer over the title area, press button 1 and drag the window to a new location. A wire frame is moved during the drag to indicate the new location. When the button is released, the window is moved to the new location.

title bar

The title bar includes the title area, the Minimize button, the Maximize button, and the Window Menu button. In shaped windows, such as round windows, the title bar floats above the window.

Minimize button

To turn the window into an icon, click button 1 on the Minimize button (the frame box with a small square in it).

Maximize button

To make the window fill the screen (or enlarge to the largest size allowed by the configuration files), click button 1 on the Maximize button (the frame box with a large square in it).

Window Menu button

The Window Menu button is the frame box with a horizontal bar in it. To pull down the window menu, press button 1. While pressing the button, drag the pointer on the menu to your selection and release the button when your selection is highlighted. Pressing button 3 in the title bar or resize border handles also posts the window menu. Alternately, you can click button 1 to pull down the menu and keep it posted; then position the pointer and select. You can also post the window menu by pressing the Shift+Esc or Alt+Space key sequence. Double-clicking button 1 with the pointer on the Window Menu button closes the window. The following table lists the contents of the window menu:

Default Window Menu

<table>
<thead>
<tr>
<th>Selection</th>
<th>Accelerator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Restore</td>
<td>Alt+F5</td>
<td>Restores the window to its size before minimizing or maximizing.</td>
</tr>
<tr>
<td>Move</td>
<td>Alt+F7</td>
<td>Allows the window to be moved with keys or mouse.</td>
</tr>
<tr>
<td>Size</td>
<td>Alt+F8</td>
<td>Allows the window to be resized.</td>
</tr>
<tr>
<td>Minimize</td>
<td>Alt+F9</td>
<td>Turns the window into an icon.</td>
</tr>
<tr>
<td>Maximize</td>
<td>Alt+F10</td>
<td>Makes the window fill the screen.</td>
</tr>
<tr>
<td>Lower</td>
<td>Alt+F3</td>
<td>Moves window to bottom of window stack.</td>
</tr>
<tr>
<td>Close</td>
<td>Alt+F4</td>
<td>Causes client to stop.</td>
</tr>
</tbody>
</table>

resize border handles

To change the size of a window, move the pointer over a resize border handle (the cursor changes), press button 1, and drag the window to a new size. When the button is released, the window is resized. While dragging is being done, a rubber-band outline is displayed to indicate the new window size.

matte

An optional matte decoration can be added between the client area and the window frame. A matte is not actually part of the window frame. There is no functionality associated with a matte.

Icons

Icons are small graphic representations of windows. A window can be iconified (minimized) using the Minimize button on the window frame. Icons provide a way to reduce clutter on the screen.

Pressing the left mouse button when the pointer is over an icon causes the icon’s window menu to open. Releasing the button (press + release without moving mouse = click) causes the menu to stay posted. The menu contains the following selections:
**Icon Window Menu**

<table>
<thead>
<tr>
<th>Selection</th>
<th>Accelerator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Restore</td>
<td>Alt+F5</td>
<td>Opens the associated window.</td>
</tr>
<tr>
<td>Move</td>
<td>Alt+F7</td>
<td>Allows the icon to be moved with keys.</td>
</tr>
<tr>
<td>Size</td>
<td>Alt+F8</td>
<td>Inactive (not an option for icons).</td>
</tr>
<tr>
<td>Minimize</td>
<td>Alt+F9</td>
<td>Inactive (not an option for icons).</td>
</tr>
<tr>
<td>Maximize</td>
<td>Alt+F10</td>
<td>Opens the associated window and makes it fill the screen.</td>
</tr>
<tr>
<td>Lower</td>
<td>Alt+F3</td>
<td>Moves icon to bottom of icon stack.</td>
</tr>
<tr>
<td>Close</td>
<td>Alt+F4</td>
<td>Removes client from window manager management.</td>
</tr>
</tbody>
</table>

Pressing button 3 over an icon also causes the icon’s window menu to open. To make a menu selection, drag the pointer over the menu and release button 3 when the desired item is highlighted.

Double-clicking button 1 on an icon calls the `f.restore_and_raise` function and restores the icon’s associated window to its previous state. For example, if a maximized window is iconified, double-clicking button 1 restores it to its maximized state. Double-clicking button 1 on the icon box’s icon opens the icon box and allow access to the contained icons. (Double-clicking a mouse button is a quick way to perform a function.) Pressing the Shift+Esc key sequence or the pop-up Menu key causes the icon window menu of the currently selected icon to open.

**Icon Box**

When icons begin to clutter the screen, they can be packed into an icon box. (To use an icon box, the window manager must be started with the icon box configuration already set.) The icon box is a window manager window that holds client icons. It includes one or more scroll bars when there are more window icons than the icon box can show at the same time.

Icons in the icon box can be manipulated with the mouse. The following button action descriptions summarize the behavior of this interface. Button actions apply whenever the pointer is on any part of the icon. Double-clicking an icon in the icon box calls the `f.restore_and_raise` function.

<table>
<thead>
<tr>
<th>Button Action</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Button 1 click</td>
<td>Selects the icon.</td>
</tr>
<tr>
<td>Button 1 double-click</td>
<td>Normalizes (opens) the associated window.</td>
</tr>
<tr>
<td>Button 1 double-click</td>
<td>Raises an already open window to the top of the stack.</td>
</tr>
<tr>
<td>Button 1 drag</td>
<td>Moves the icon.</td>
</tr>
<tr>
<td>Button 3 press</td>
<td>Causes the menu for that icon to open.</td>
</tr>
<tr>
<td>Button 3 drag</td>
<td>Highlights items as the pointer moves across the menu.</td>
</tr>
</tbody>
</table>

Pressing mouse button 3 when the pointer is over an icon causes the menu for that icon to open.

**Icon Menu for Icon Box**

<table>
<thead>
<tr>
<th>Selection</th>
<th>Accelerator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Restore</td>
<td>Alt+F5</td>
<td>Opens the associated window (if not already open).</td>
</tr>
<tr>
<td>Move</td>
<td>Alt+F7</td>
<td>Allows the icon to be moved with keys.</td>
</tr>
<tr>
<td>Size</td>
<td>Alt+F8</td>
<td>Inactive.</td>
</tr>
<tr>
<td>Minimize</td>
<td>Alt+F9</td>
<td>Inactive.</td>
</tr>
<tr>
<td>Maximize</td>
<td>Alt+F10</td>
<td>Opens the associated window (if not already open) and maximizes its size.</td>
</tr>
</tbody>
</table>
Icon Menu for Icon Box

<table>
<thead>
<tr>
<th>Selection</th>
<th>Accelerator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lower</td>
<td>Alt+F3</td>
<td>Inactive.</td>
</tr>
<tr>
<td>Close</td>
<td>Alt+F4</td>
<td>Removes client from window manager management.</td>
</tr>
</tbody>
</table>

To pull down the window menu for the icon box itself, press button 1 with the pointer over the menu button for the icon box. The window menu of the icon box differs from the window menu of a client window: The Close selection is replaced with the PackIcons (Shift+Alt+F7) selection. When selected, the PackIcons option packs the icons in the box to achieve neat rows with no empty slots.

You can also post the window menu by pressing the Shift+Esc or Alt+Space key sequence. Pressing the pop-up Menu key causes the icon window menu of the currently selected icon to open.

**Input Focus**

The mwm command supports (by default) a keyboard input focus policy of *explicit selection*. This means when a window is selected to get keyboard input, it continues to get keyboard input until the window is withdrawn from window management, another window is explicitly selected to get keyboard input, or the window is iconified. Several resources control the input focus. The client window with the keyboard input focus has the active window appearance with a visually distinct window frame.

The following table and key action descriptions summarize the keyboard input focus selection behavior:

<table>
<thead>
<tr>
<th>Button Action</th>
<th>Object</th>
<th>Function Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Button 1 press</td>
<td>Window or window frame</td>
<td>Keyboard focus selection</td>
</tr>
<tr>
<td>Button 1 press</td>
<td>Icon</td>
<td>Keyboard focus selection</td>
</tr>
</tbody>
</table>

**Key Action**

<table>
<thead>
<tr>
<th>Function Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alt+Tab</td>
</tr>
<tr>
<td>Moves the input focus to next window in the window stack.</td>
</tr>
<tr>
<td>Alt+Shift+Tab</td>
</tr>
<tr>
<td>Moves the input focus to the previous window in the window stack (available only in explicit focus mode).</td>
</tr>
</tbody>
</table>

**Window Stacking**

There are two types of window stacks: global window stacks and an application’s local family window stack.

The global stacking order of windows can be changed as a result of setting the keyboard input focus, iconifying a window, or performing a window manager window stacking function. When keyboard focus policy is explicit the default value of the *focusAutoRaise* resource is True. This causes a window to be raised to the top of the stack when it receives input focus, for example, by pressing button 1 on the title bar. The key actions defined in the preceding list raise the window receiving focus to the top of the stack.

In pointer mode, the default value of the *focusAutoRaise* resource is False; that is, the window stacking order is not changed when a window receives keyboard input focus. The following key actions can be used to cycle through the global window stack:

<table>
<thead>
<tr>
<th>Key Action</th>
<th>Function Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alt+Esc</td>
<td>Places top window on bottom of stack.</td>
</tr>
<tr>
<td>Alt+Shift+Esc</td>
<td>Places bottom window on top of stack.</td>
</tr>
</tbody>
</table>
By default, a window’s icon is placed on the bottom of the stack when the window is iconified; however, the default can be changed by the `lowerOnIconify` resource.

Transient windows (secondary windows such as dialog boxes) stay above their parent windows by default. However, an application’s local family stacking order can changed to allow a transient window to be placed below its parent top-level window. The following parameter values show the modification of the stacking order for the `f.lower` function:

- `f.lower` Lower the transient window within the family (staying above the parent) and lowers the family in the global window stack.
- `f.lower [within]` Lower the transient window within the family (staying above the parent) but does not lower the family in the global window stack.
- `f.lower [freeFamily]` Lowers the window separate from its family stack (below the parent), but does not lower the family in the global window stack.

The `within` and `freeFamily` parameter values can also be used with the `f.raise` and `f.raise_lower` functions.

**X Defaults**

The `mwm` command is configured from its resource database. This database is built from the following sources. They are listed in order of precedence.

1. `mwm` command line options
2. `XENVIRONMENT` variable or `$HOME/Xdefaults-host`
3. `RESOURCE_MANAGER` root window property or `$HOME/.Xdefaults`
4. `$HOME/Mwm`
5. `/usr/lib/X11/app-defaults/Mwm`

The `/usr/lib/X11/app-defaults/Mwm` and `$HOME/Mwm` file names represent customary locations for these files. The actual location of the systemwide class resource file might depend on the `XFILESEARCHPATH` environment variable and the current language environment. The actual location of the user-specific class resource file might depend on the `XUSERFILESEARCHPATH` and `XAPPLRESDIR` environment variables and the current language environment.

Entries in the resource database can refer to other resource files for specific types of resources. These include files that contain bitmaps, fonts, and `mwm`-specific resources such as menus and behavior specifications (for example, button and key bindings).

`Mwm` is the resource class name of the `mwm` command and `mwm` is the resource name used by the `mwm` command to look up resources. (For looking up resources of multiple screens, the `-screens` command-line option specifies resource names such as `mwm_b+w` and `mwm_color`.) In the following discussion of resource specification, “Mwm” and “mwm” (and the aliased `mwm` resource names) can be used interchangeably, but “mwm” takes precedence over “Mwm”. The `mwm` command uses the following types of resources:

- **component appearance resource set** These resources specify appearance attributes of window manager user-interface components. They can be applied to the appearance of window manager menus, feedback windows (for example, the window reconfiguration feedback window), client window frames, and icons.

- **frame and icon component resource set** This subset of component appearance resources specifies attributes that are specific to frame and icon components.
Resource identifiers can be either a resource name (for example, foreground) or a resource class (for example, Foreground). If the value of a resource is a file name and if the file name is prefixed by the ~/ (tilde followed by a slash) characters, it is relative to the path contained in the HOME environment variable (generally the user’s home directory).

### Component Appearance Resources

The syntax for specifying component appearance resources that apply to window manager icons, menus, and client window frames is as follows:

**Mwm**`*ResourceID`

For example, **Mwm**`*foreground` is used to specify the foreground color for the **mwm** command menus, icons, client window frames, and feedback dialogs.

The syntax for specifying component appearance resources that apply to a particular **mwm** component is as follows:

**Mwm**`*[Menu|Icon|Client|Feedback]*` `*ResourceID`

If *Menu* is specified, the resource is applied only to **Mwm** menus; if *Icon* is specified, the resource is applied to icons; and if *Client* is specified, the resource is applied to client window frames. For example, **Mwm**`*Icon*` `*foreground` is used to specify the foreground color for the **mwm** command icons, **Mwm**`*Menu*` `*foreground` specifies the foreground color for the **mwm** command menus, and **Mwm**`*Client*` `*foreground` is used to specify the foreground color for the **mwm** command client window frames.

The appearance of the title area of a client window frame (including window management buttons) can be separately configured. The syntax for configuring the title area of a client window frame is as follows:

**Mwm**`*Client*` `*Title*` `*ResourceID`

For example, **Mwm**`*Client*` `*Title*` `*foreground` specifies the foreground color for the title area. Defaults for title area resources are based on the values of the corresponding client window frame resources.

The appearance of menus can be configured based on the name of the menu. The syntax for specifying menu appearance by name is as follows:

**Mwm**`*Menu*` `*MenuName*` `*ResourceID`

For example, **Mwm**`*Menu*` `*MyMenu*` `*foreground` specifies the foreground color for the menu named *MyMenu*.

The user can also specify resources for window manager menu components (the gadgets that comprise the menu). These may include, for example, a menu title, a title separator, one or more buttons, and separators. If a menu contains more than one instance of a class, such as multiple **PushButtonGadget** gadgets, the name of the first instance is **PushButtonGadget1**, the second is **PushButtonGadget2**, and so on. The following list identifies the naming conventions used for window manager menu components:
The following component appearance resources that apply to all window manager parts can be specified.

**Component Appearance Resource Set**

<table>
<thead>
<tr>
<th>Name</th>
<th>Class</th>
<th>Value Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>background</td>
<td>Background</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>backgroundPixmap</td>
<td>BackgroundPixmap</td>
<td>string(^2)</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>bottomShadowColor</td>
<td>Foreground</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>bottomShadowPixmap</td>
<td>BottomShadowPixmap</td>
<td>string(^2)</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>fontList</td>
<td>FontList</td>
<td>string(^3)</td>
<td>“fixed”</td>
</tr>
<tr>
<td>foreground</td>
<td>Foreground</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>saveUnder</td>
<td>SaveUnder</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>topShadowColor</td>
<td>Background</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>topShadowPixmap</td>
<td>TopShadowPixmap</td>
<td>string(^2)</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>background</td>
<td>Background</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>backgroundPixmap</td>
<td>BackgroundPixmap</td>
<td>string(^2)</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>bottomShadowColor</td>
<td>Foreground</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>bottomShadowPixmap</td>
<td>BottomShadowPixmap</td>
<td>string(^2)</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>fontList</td>
<td>FontList</td>
<td>string(^3)</td>
<td>“fixed”</td>
</tr>
<tr>
<td>foreground</td>
<td>Foreground</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>saveUnder</td>
<td>SaveUnder</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>topShadowColor</td>
<td>Background</td>
<td>color</td>
<td>varies(^1)</td>
</tr>
<tr>
<td>topShadowPixmap</td>
<td>TopShadowPixmap</td>
<td>string(^2)</td>
<td>varies(^1)</td>
</tr>
</tbody>
</table>

**Notes:**
1. The default is chosen based on the visual type of the screen.
2. Image name.

**background** (class *Background*)

Specifies the background color. Any legal X color can be specified. The default value is chosen based on the visual type of the screen.

**backgroundPixmap** (class *BackgroundPixmap*)

Specifies the background pixmap of the mwm decoration when the window is inactive (does not have the keyboard focus). The default value is chosen based on the visual type of the screen.

**bottomShadowColor** (class *Foreground*)

Specifies the bottom shadow color. This color is used for the lower and right bevels of the window manager decoration. Any legal X color can be specified. The default value is chosen based on the visual type of the screen.

**bottomShadowPixmap** (class *BottomShadowPixmap*)

Specifies the bottom shadow pixmap. This pixmap is used for the lower and right bevels of the window manager decoration. The default is chosen based on the visual type of the screen.
fontList (class FontList) Specifies the font used in the window manager decoration. The character encoding of the font needs to match the character encoding of the strings that are used. The default is the fixed value.

foreground (class Foreground) Specifies the foreground color. The default is chosen based on the visual type of the screen.

saveUnder (class SaveUnder) Controls the repainting of windows that are uncovered after being obscured. This resource indicates whether save unders are used for mwm components. For this to have any effect, save unders must be implemented by the X server. If save unders are implemented, the X server saves the contents of windows obscured by windows that have the save under attribute set. If the saveUnder resource is True, the mwm command sets the save under attribute on the window manager frame of any client that has it set. If the saveUnder resource is False, save unders is not used on any window manager frames. The default value is False.

topShadowColor (class Background) Specifies the top shadow color. This color is used for the upper and left bevels of the window manager decoration. The default is chosen based on the visual type of the screen.

topShadowPixmap (class TopShadowPixmap) Specifies the top shadow pixmap. This pixmap is used for the upper and left bevels of the window manager decoration. The default is chosen based on the visual type of the screen.

### Frame and Icon Component Resource Set

**Note:** Hyphens in the following table are for readability purposes only. Do not include hyphens within names in programs.

<table>
<thead>
<tr>
<th>Name</th>
<th>Class</th>
<th>Value Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>activeBackground</td>
<td>Background</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeBackground-Pixmap</td>
<td>BackgroundPixmap</td>
<td>string²</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeBottomShadow-Color</td>
<td>Foreground</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeBottomShadow-Pixmap</td>
<td>BottomShadow-Pixmap</td>
<td>string²</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeForeground</td>
<td>Foreground</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeTopShadowColor</td>
<td>Background</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeTopShadowPixmap</td>
<td>TopShadowPixmap</td>
<td>string²</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeBackgroundPixmap</td>
<td>BackgroundPixmap</td>
<td>string²</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeBottomShadowColor</td>
<td>Foreground</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeBottomShadowPixmap</td>
<td>BottomShadowPixmap</td>
<td>string²</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeForeground</td>
<td>Foreground</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeTopShadowColor</td>
<td>Background</td>
<td>color</td>
<td>varies¹</td>
</tr>
<tr>
<td>activeTopShadowPixmap</td>
<td>TopShadowPixmap</td>
<td>string²</td>
<td>varies¹</td>
</tr>
</tbody>
</table>

**Notes:**
1. The default is chosen based on the visual type of the screen.
2. Image name.

**activeBackground (class Background)** Specifies the background color of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.
activeBackgroundPixmap (class BackgroundPixmap)  Specifies the background pixmap of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.

activeBottomShadowColor (class Foreground)  Specifies the bottom shadow color of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.

activeBottomShadowPixmap (class BottomShadowPixmap)  Specifies the bottom shadow pixmap of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.

activeForeground (class Foreground)  Specifies the foreground color of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.

activeTopShadowColor (class Background)  Specifies the top shadow color of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.

activeTopShadowPixmap (class TopShadowPixmap)  Specifies the top shadow pixmap of the mwm decoration when the window is active (has the keyboard focus). The default is chosen based on the visual type of the screen.

General Appearance and Behavior Resources
The syntax for specifying general appearance and behavior resources is as follows:

Mwm*ResourceId

For example, Mwm*keyboardFocusPolicy specifies the window manager policy for setting the keyboard focus to a particular client window.

General Appearance and Behavior Resource Set

Note: Hyphens in the following table are for readability purposes only. Do not include hyphens within names in programs.

<table>
<thead>
<tr>
<th>Name</th>
<th>Class</th>
<th>Value Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>autoKeyFocus</td>
<td>AutoKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>autoRaiseDelay</td>
<td>AutoRaiseDelay</td>
<td>millisec.</td>
<td>500</td>
</tr>
<tr>
<td>bitmapDirectory</td>
<td>BitmapDirectory</td>
<td>directory</td>
<td>/usr/include/X11/bitmaps</td>
</tr>
<tr>
<td>buttonBindings</td>
<td>ButtonBindings</td>
<td>string</td>
<td>“DefaultButton-Bindings”</td>
</tr>
<tr>
<td>cleanText</td>
<td>CleanText</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>clientAutoPlace</td>
<td>ClientAutoPlace</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>colormapFocusPolicy</td>
<td>ColormapFocus-Policy</td>
<td>string</td>
<td>keyboard</td>
</tr>
<tr>
<td>configFile</td>
<td>ConfigFile</td>
<td>file</td>
<td>.mwmrc</td>
</tr>
<tr>
<td>deiconifyKeyFocus</td>
<td>DeiconifyKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>doubleClickTime</td>
<td>DoubleClickTime</td>
<td>millisec.</td>
<td>multiclick time</td>
</tr>
<tr>
<td>enableWarp</td>
<td>enableWarp</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>enforceKeyFocus</td>
<td>EnforceKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>Name</td>
<td>Class</td>
<td>Value Type</td>
<td>Default</td>
</tr>
<tr>
<td>--------------------</td>
<td>---------------------------</td>
<td>----------------</td>
<td>----------</td>
</tr>
<tr>
<td>fadeNormalIcon</td>
<td>FadeNormalIcon</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>feedbackGeometry</td>
<td>FeedbackGeometry</td>
<td>string</td>
<td>center on screen</td>
</tr>
<tr>
<td>frameBorderWidth</td>
<td>FrameBorderWidth</td>
<td>pixels</td>
<td>varies</td>
</tr>
<tr>
<td>iconAutoPlace</td>
<td>IconAutoPlace</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>iconBoxGeometry</td>
<td>IconBoxGeometry</td>
<td>string</td>
<td>6x1+0-0</td>
</tr>
<tr>
<td>iconBoxName</td>
<td>IconBoxName</td>
<td>string</td>
<td>iconbox</td>
</tr>
<tr>
<td>iconBoxSBDisplayPolicy</td>
<td>IconBoxSBDisplayPolicy</td>
<td>string</td>
<td>all</td>
</tr>
<tr>
<td>iconBoxTitle</td>
<td>IconBoxTitle</td>
<td>XmString</td>
<td>Icons</td>
</tr>
<tr>
<td>iconClick</td>
<td>IconClick</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>iconDecoration</td>
<td>IconDecoration</td>
<td>string</td>
<td>varies</td>
</tr>
<tr>
<td>iconImage-Maximum</td>
<td>IconImage-Maximum</td>
<td>wxh</td>
<td>50x50</td>
</tr>
<tr>
<td>iconImageMinimum</td>
<td>IconImageMinimum</td>
<td>wxh</td>
<td>16x16</td>
</tr>
<tr>
<td>iconPlacement</td>
<td>IconPlacement</td>
<td>string</td>
<td>left bottom</td>
</tr>
<tr>
<td>iconPlacementMargin</td>
<td>IconPlacementMargin</td>
<td>pixels</td>
<td>varies</td>
</tr>
<tr>
<td>interactivePlacement</td>
<td>InteractivePlacement</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>keyBindings</td>
<td>KeyBindings</td>
<td>string</td>
<td>&quot;DefaultKeyBindings&quot;</td>
</tr>
<tr>
<td>keyboardFocusPolicy</td>
<td>KeyboardFocusPolicy</td>
<td>string</td>
<td>explicit</td>
</tr>
<tr>
<td>limitResize</td>
<td>LimitResize</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>lowerOnIconify</td>
<td>LowerOnIconify</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>maximumMaximuSize</td>
<td>MaximumMaximuSize</td>
<td>wxh (pixels)</td>
<td>2X screen w&amp;h</td>
</tr>
<tr>
<td>moveOpaque</td>
<td>MoveOpaque</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>moveThreshold</td>
<td>MoveThreshold</td>
<td>pixels</td>
<td>4</td>
</tr>
<tr>
<td>multiScreen</td>
<td>MultiScreen</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>passButtons</td>
<td>PassButtons</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>PassSelectButton</td>
<td>PassSelectButton</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>positionIsFrame</td>
<td>PositionIsFrame</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>positionOnScreen</td>
<td>PositionOnScreen</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>quitTimeout</td>
<td>QuitTimeout</td>
<td>millisec.</td>
<td>1000</td>
</tr>
<tr>
<td>raiseKeyFocus</td>
<td>RaiseKeyFocus</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>resizeBorderWidth</td>
<td>ResizeBorder-Width</td>
<td>pixels</td>
<td>varies</td>
</tr>
<tr>
<td>resizeCursors</td>
<td>ResizeCursors</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>Name</td>
<td>Class</td>
<td>Value Type</td>
<td>Default</td>
</tr>
<tr>
<td>--------------------</td>
<td>----------------------------</td>
<td>------------</td>
<td>---------------</td>
</tr>
<tr>
<td>screens</td>
<td>Screens</td>
<td>string</td>
<td>varies</td>
</tr>
<tr>
<td>showFeedback</td>
<td>ShowFeedback</td>
<td>string</td>
<td>all</td>
</tr>
<tr>
<td>startupKeyFocus</td>
<td>StartupKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>transientDecoration</td>
<td>Transient-Decoration</td>
<td>string</td>
<td>menu title</td>
</tr>
<tr>
<td>transientFunctions</td>
<td>Transient-Functions</td>
<td>string</td>
<td>-minimize -maximize</td>
</tr>
<tr>
<td>useIconBox</td>
<td>UseIconBox</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>wMenuButtonClick</td>
<td>WMenuButtonClick</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>wMenuButtonClick2</td>
<td>WMenuButtonClick2</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>autoKeyFocus</td>
<td>AutoKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>autoRaiseDelay</td>
<td>AutoRaiseDelay</td>
<td>millisec.</td>
<td>500</td>
</tr>
<tr>
<td>buttonDirectory</td>
<td>BitmapDirectory</td>
<td>directory</td>
<td>/usr/include/X11/bitmaps</td>
</tr>
<tr>
<td>buttonBindings</td>
<td>ButtonBindings</td>
<td>string</td>
<td>&quot;DefaultButtonBindings&quot;</td>
</tr>
<tr>
<td>cleanText</td>
<td>CleanText</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>clientAutoPlace</td>
<td>ClientAutoPlace</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>colormapFocusPolicy</td>
<td>ColormapFocusPolicy</td>
<td>string</td>
<td>keyboard</td>
</tr>
<tr>
<td>configFile</td>
<td>ConfigFile</td>
<td>file</td>
<td>.mwmrc</td>
</tr>
<tr>
<td>deiconifyKeyFocus</td>
<td>DeiconifyKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>doubleClickTime</td>
<td>DoubleClickTime</td>
<td>millisec.</td>
<td>multiclick time</td>
</tr>
<tr>
<td>enableWarp</td>
<td>enableWarp</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>enforceKeyFocus</td>
<td>EnforceKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>fadeNormalIcon</td>
<td>FadeNormalIcon</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>feedbackGeometry</td>
<td>FeedbackGeometry</td>
<td>string</td>
<td>center on screen</td>
</tr>
<tr>
<td>frameBorderWidth</td>
<td>FrameBorderWidth</td>
<td>pixels</td>
<td>varies</td>
</tr>
<tr>
<td>iconAutoPlace</td>
<td>IconAutoPlace</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>iconBoxGeometry</td>
<td>IconBoxGeometry</td>
<td>string</td>
<td>6x1+0-0</td>
</tr>
<tr>
<td>iconBoxName</td>
<td>IconBoxName</td>
<td>string</td>
<td>iconbox</td>
</tr>
<tr>
<td>iconBoxSBDisplayPolicy</td>
<td>IconBoxSBDisplayPolicy</td>
<td>string</td>
<td>all</td>
</tr>
<tr>
<td>iconBoxTitle</td>
<td>IconBoxTitle</td>
<td>XmString</td>
<td>Icons</td>
</tr>
<tr>
<td>iconClick</td>
<td>IconClick</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>iconDecoration</td>
<td>IconDecoration</td>
<td>string</td>
<td>varies</td>
</tr>
<tr>
<td>iconImageMaximum</td>
<td>IconImageMaximum</td>
<td>wxh</td>
<td>50x50</td>
</tr>
<tr>
<td>iconImageMinimum</td>
<td>IconImageMinimum</td>
<td>wxh</td>
<td>16x16</td>
</tr>
<tr>
<td>Name</td>
<td>Class</td>
<td>Value Type</td>
<td>Default</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>----------------------------</td>
<td>------------</td>
<td>---------</td>
</tr>
<tr>
<td>iconPlacement</td>
<td>IconPlacement</td>
<td>string</td>
<td>left bottom</td>
</tr>
<tr>
<td>iconPlacementMargin</td>
<td>IconPlacementMargin</td>
<td>pixels</td>
<td>varies</td>
</tr>
<tr>
<td>interactivePlacement</td>
<td>InteractivePlacement</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>keyBindings</td>
<td>KeyBindings</td>
<td>string</td>
<td>&quot;DefaultKeyBindings&quot;</td>
</tr>
<tr>
<td>keyboardFocusPolicy</td>
<td>KeyboardFocusPolicy</td>
<td>string</td>
<td>explicit</td>
</tr>
<tr>
<td>limitResize</td>
<td>LimitResize</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>lowerOnIconify</td>
<td>LowerOnIconify</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>maximumMaximumSize</td>
<td>MaximumMaximumSize</td>
<td>wxh (pixels)</td>
<td>2X screen w&amp;h</td>
</tr>
<tr>
<td>moveOpaque</td>
<td>MoveOpaque</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>moveThreshold</td>
<td>MoveThreshold</td>
<td>pixels</td>
<td>4</td>
</tr>
<tr>
<td>multiScreen</td>
<td>MultiScreen</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>passButtons</td>
<td>PassButtons</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>PassSelectButton</td>
<td>PassSelectButton</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>positionIsFrame</td>
<td>PositionIsFrame</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>positionOnScreen</td>
<td>PositionOnScreen</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>quitTimeout</td>
<td>QuitTimeout</td>
<td>millisec.</td>
<td>1000</td>
</tr>
<tr>
<td>raiseKeyFocus</td>
<td>RaiseKeyFocus</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>resizeBorderWidth</td>
<td>ResizeBorderWidth</td>
<td>pixels</td>
<td>varies</td>
</tr>
<tr>
<td>resizeCursors</td>
<td>ResizeCursors</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>screens</td>
<td>Screens</td>
<td>string</td>
<td>varies</td>
</tr>
<tr>
<td>showFeedback</td>
<td>ShowFeedback</td>
<td>string</td>
<td>all</td>
</tr>
<tr>
<td>startupKeyFocus</td>
<td>StartupKeyFocus</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>transientDecoration</td>
<td>TransientDecoration</td>
<td>string</td>
<td>menu title</td>
</tr>
<tr>
<td>transientFunctions</td>
<td>TransientFunctions</td>
<td>string</td>
<td>-minimize -maximize</td>
</tr>
<tr>
<td>useIconBox</td>
<td>UseIconBox</td>
<td>True or False</td>
<td>False</td>
</tr>
<tr>
<td>wMenuButtonClick</td>
<td>WMenuButtonClick</td>
<td>True or False</td>
<td>True</td>
</tr>
<tr>
<td>wMenuButtonClick2</td>
<td>WMenuButtonClick2</td>
<td>True or False</td>
<td>True</td>
</tr>
</tbody>
</table>

**Note:** Abbreviations for value types:

w = width

h = height

millisec = milliseconds
autoKeyFocus (class AutoKeyFocus)  
Controls whether the focus is set to the previous window that had the focus. If the autoKeyFocus resource is given a value of True and a window with the keyboard input focus is withdrawn from window management or is iconified, the focus is set to the previous window that had the focus. If the value given is False, there is no automatic setting of the keyboard input focus. It is recommended that both the autoKeyFocus resource and the startupKeyFocus resource be set to a value of True to work with tear-off menus. The default value is True. This resource is available only when the keyboard input focus policy is set to the explicit value.

autoRaiseDelay (class AutoRaiseDelay)  
Specifies the amount of time in milliseconds (ms) that the mwm command waits before raising a window after it gets the keyboard focus. The default value of this resource is 500 (milliseconds). This resource is available only when the focusAutoRaise resource is True and the keyboard focus policy is the pointer value.

bitmapDirectory (class BitmapDirectory)  
Identifies a directory to be searched for bitmaps referenced by mwm resources. This directory is searched if a bitmap is specified without an absolute path name. The default value for this resource is /usr/include/X11/bitmaps. The /usr/include/X11/bitmaps directory represents the customary locations for this directory. The actual location of this directory may vary on some systems. If the bitmap is not found in the specified directory, the XBMLANGPATH environment variable is searched.

buttonBindings (class ButtonBindings)  
Identifies the set of button bindings for window management functions. The named set of button bindings is specified in the mwm resource description file. These button bindings are merged with the built-in default bindings. The default value for this resource is DefaultButtonBindings.

cleanText (class CleanText)  
Controls the display of window manager text in the client title and feedback windows. If the default value of True is used, the text is drawn with a clear (no stipple) background. This makes text easier to read on monochrome systems where a backgroundPixmap is specified. Only the stippling in the area immediately around the text is cleared. If False, the text is drawn directly on top of the existing background.

clientAutoPlace (class ClientAutoPlace)  
Determines the position of a window when the window does not have a user-specified position. With a value of True, windows are positioned with the top left corners of the frames offset horizontally and vertically. A value of False causes the currently configured position of the window to be used. In either case, the mwm command attempts to place the windows totally on-screen. The default value is True.

colormapFocusPolicy (class ColormapFocusPolicy)  
Indicates the colormap focus policy that is to be used. If the resource value is explicit, a colormap selection action is done on a client window to set the colormap focus to that window. If the value is pointer, the client window containing the pointer has the colormap focus. If the value is keyboard, the client window that has the keyboard input focus has the colormap focus. The default value for this resource is keyboard.
**configFile (class ConfigFile)**

Contains the path name for an mwm resource description file.

If the path name begins with the ~ characters, the mwm command considers it to be relative to the user’s home directory (as specified by the HOME environment variable). If the LANG environment variable is set, the mwm command looks for $HOME/$LANG/configFile. If that file does not exist or if LANG is not set, mwm looks for $HOME/configFile.

If the configFile path name does not begin with the ~ characters, mwm considers it to be relative to the current working directory.

If the configFile resource is not specified or if that file does not exist, the mwm command uses several default paths to find a configuration file. If the LANG environment variable is set, the mwm command looks for the configuration file first in the $HOME/$LANG/.mwmrc file. If that file does not exist or if the LANG environment variable is not set, the mwm command looks for the $HOME/mwmrc file. If the $HOME/.mwmrc file does not exist and if the LANG environment variable is set, the mwm command next looks for the /usr/lib/X11/$LANG/system.mwmrc file. If the /usr/lib/X11/$LANG/system.mwmrc file does not exist or if the LANG environment variable is not set, the mwm command looks for /usr/lib/X11/system.mwmrc.

**deiconifyKeyFocus (class DeiconifyKeyFocus)**

Determines whether a window receives the keyboard input focus when it is deiconified (normalized). The default value is True. This resource applies only when the keyboard input focus policy is set to the explicit value.

**doubleClickTime (class DoubleClickTime)**

Sets the maximum time (in ms) between the clicks (button presses) that make up a double-click. The default value of this resource is the display’s multicleck time.

**enableWarp (class EnableWarp)**

Causes the mwm command to warp the pointer to the center of the selected window during keyboard-controlled resize and move operations. Setting the value to False causes the mwm command to leave the pointer at its original place on the screen unless the user explicitly moves it with the cursor keys or pointing device. The default value of this resource is True.

**enforceKeyFocus (class EnforceKeyFocus)**

Determines whether the keyboard input focus is always explicitly set to selected windows even if there is an indication that they are globally active input windows. (An example of a globally active window is a scroll bar that can be operated without setting the focus to that client.) If the resource is False, the keyboard input focus is not explicitly set to globally active windows. The default value is True.

**fadeNormalIcon (class FadeNormalIcon)**

Determines whether an icon is unavailable whenever it is normalized (its window is opened). The default value is False.

**feedbackGeometry (class FeedbackGeometry)**

Sets the position of the move and resize feedback window. If this resource is not specified, the default is to place the feedback window at the center of the screen. The value of the resource is a standard window geometry string with the following syntax:

`[=][(+)-XOffset(+)-YOffset]`

**frameBorderWidth (class FrameBorderWidth)**

Specifies the width in pixels of a client window frame border without resize handles. The border width includes the three-dimensional (3-D) shadows. The default value is based on the size and resolution of the screen.
iconAutoPlace (class IconAutoPlace) Indicates whether the window manager arranges icons in a particular area of the screen or places each icon where the window was when it was iconified. The True value indicates that icons are arranged in a particular area of the screen determined by the iconPlacement resource. The False value indicates that an icon is placed at the location of the window when it is iconified. The default is True.

iconBoxGeometry (class IconBoxGeometry) Indicates the initial position and size of the icon box. The value of the resource is a standard window geometry string with the following syntax:

\[XOffset\{+-}\] Width x Height\{+-\} YOffset\]

If the offsets are not provided, the iconPlacement policy is used to determine the initial placement. The units for width and height are columns and rows.

The actual screen size of the icon box window depends on the iconImageMaximum (size) and iconDecoration resources. The default value for size is (6 times iconWidth + padding) wide by (1 times iconHeight + padding) high. The default value of the location is +0 -0.

iconBoxName (class IconBoxName) Specifies the name that is used to look up icon box resources. The default name is iconbox.

iconBoxSBDisplayPolicy (class IconBoxSBDisplayPolicy) Specifies the scroll bar display policy of the window manager in the icon box. The resource has the following three possible values: all, vertical, and horizontal. The default value, all, causes both vertical and horizontal scroll bars always to be displayed. The vertical value causes a single vertical scroll bar to be displayed in the icon box and sets the orientation of the icon box to horizontal (regardless of the iconBoxGeometry specification). The horizontal value causes a single horizontal scroll bar to be displayed in the icon box and sets the orientation of the icon box to vertical (regardless of the iconBoxGeometry specification).

iconBoxTitle (class IconBoxTitle) Specifies the name that is used in the title area of the icon box frame. The default value is Icons.

iconClick (class IconClick) Specifies whether the system menu is posted and remains posted when an icon is clicked. The default value is True.

iconDecoration (class IconDecoration) Specifies the general icon decoration. The resource value is label (only the label part is displayed) or image (only the image part is displayed) or label image (both the label and image parts are displayed). A value of activelabel can also be specified to get a label (not truncated to the width of the icon) when the icon is selected. The default icon decoration for icon box icons is that each icon has a label part and an image part (label image). The default icon decoration for standalone icons is that each icon has an active label part, a label part, and an image part (activelabel, label, and image).

iconImageMaximum (class IconImageMaximum) Specifies the maximum size of the icon image. The resource value is Width x Height (for example, 64x64). The maximum supported size is 128x128. The default value of this resource is 50x50.

iconImageMinimum (class IconImageMinimum) Specifies the minimum size of the icon image. The resource value is Width x Height (for example, 32x50). The minimum supported size is 16x16. The default value of this resource is 16x16.

iconPlacement (class IconPlacement) Specifies the icon placement scheme to be used. The resource value has the following syntax:

PrimaryLayout SecondaryLayout [Tight]

The layout values are described as one of the following:
**top**  Lays out the icons from top to bottom.

**bottom**  Lays out the icons from bottom to top.

**left**  Lays out the icons from left to right.

**right**  Lays out the icons from right to left.

A horizontal (vertical) layout value must not be used for both the **PrimaryLayout** and the **SecondaryLayout** (for example, do not use top for the **PrimaryLayout** and bottom for the **SecondaryLayout**). The **PrimaryLayout** indicates at the time an icon placement is done whether the icon is placed in a row or a column and the direction of placement. The **SecondaryLayout** indicates where to place new rows or columns.

For example, the top right value indicates that icons should be placed top to bottom on the screen and that columns should be added from right to left on the screen. The default placement is the left bottom value (icons are placed from left to right on the screen, with the first row on the bottom of the screen, and new rows added from the bottom of the screen to the top of the screen). A tight value places icons with zero spacing between icons. This value is useful for aesthetic reasons, as well as for terminals with small screens.

The following is a list of options for **iconPlacement** values:

**Icon Placement**

**Appropriate Scheme**

From left to right across the top of the screen, new rows below
   Left top

From right to left across the top of the screen, new rows below
   Right top

From left to right across the bottom of the screen, new rows above
   Left bottom

From right to left across the bottom of the screen, new rows above
   Right bottom

From bottom to top along the left of the screen, new columns to right
   Bottom left

From bottom to top along the right of the screen, new columns to left
   Bottom right

From top to bottom along the left of the screen, new columns to right
   Top left

From top to bottom along the right of the screen, new columns to left
   Top right

**iconPlacementMargin** (class IconPlacementMargin)

Sets the distance between the edge of the screen and the icons that are placed along the edge of the screen. The value should be greater than or equal to 0. A default value is used if the value specified is invalid. The default value for this resource is equal to the space between icons as they are placed on the screen (this space is based on maximizing the number of icons in each row and column).

**interactivePlacement** (class InteractivePlacement)

Controls the initial placement of new windows on the screen. If the value is True, the pointer shape changes before a new window is placed on the screen to indicate to the user that a position needs to be selected for the upper-left corner of the window. If the value is False, windows are placed according to the initial window configuration attributes. The default value of this resource is False.
**keyBindings (class KeyBindings)**

Identifies the set of key bindings for window management functions. If specified, these key bindings replace the built-in default bindings. The named set of key bindings is specified in mwm resource description file. The default value for this resource is DefaultKeyBindings.

**keyboardFocusPolicy (class KeyboardFocusPolicy)**

Determines the keyboard focus policy. If set to the pointer value, the keyboard focus policy has the keyboard focus set to the client window that contains the pointer (the pointer could also be in the client window decoration that the mwm command adds). If set to the explicit value, the policy is to have the keyboard focus set to a client window when the user presses the left mouse button with the pointer on the client window or any part of the associated mwm decoration.

The default value for this resource is explicit.

**limitResize (class LimitResize)**

Determines whether the user is allowed to resize a window to greater than the maximum size. If this resource is True, the user is not allowed to resize a window to greater than the maximum size. The default value for this resource is True.

**lowerOnIconify (class LowerOnIconify)**

Determines whether a window icon is displayed on the bottom of the window stack when the window is iconified (minimized). A value of False places the icon in the stacking order at the same place as its associated window. The default value of this resource is True.

**maximumMaximumSize (class MaximumMaximumSize)**

Limits the maximum size of a client window as set by the user or client. The resource value is Width x Height (for example, 1024x1024) where the width and height are in pixels. The default value of this resource is twice the screen width and height.

**moveOpaque (class MoveOpaque)**

Controls whether the actual window is moved or a rectangular outline of the window is moved. A default value of False displays a rectangular outline on move operations.

**moveThreshold (class MoveThreshold)**

Controls the sensitivity of dragging operations that move windows and icons. The value of this resource is the number of pixels that the locator is moved with a button down before the move operation is initiated. This is used to prevent window and icon movement when you click or double-click and there is unintentional pointer movement with the button down. The default value of this resource is 4 (pixels).

**multiScreen (class MultiScreen)**

Determines whether the mwm command manages all the screens on the display. If False, the mwm command manages only a single screen. The default value is False.

**passButtons (class PassButtons)**

Indicates whether button press events are passed to clients after they are used to do a window manager function in the client context. If the resource value is False, the button press is not passed to the client. If the value is True, the button press is passed to the client window. The window manager function is done in either case. The default value for this resource is False.

**passSelectButton (class PassSelectButton)**

Indicates whether to pass the select button press events to clients after they are used to do a window manager function in the client context. If the resource value is False, the button press is not passed to the client. If the value is True, the button press is passed to the client window. The window manager function is done in either case. The default value for this resource is True.

**positionIsFrame (class PositionIsFrame)**

Indicates how client window position information (from the WM_NORMAL_HINTS property and from configuration requests) is to be interpreted. If the resource value is True, the information is interpreted as the position of the MWM client window frame. If the value is False, it is interpreted as being the position of the client area of the window. The default value of this resource is True.
**positionOnScreen** (class PositionOnScreen) Indicates that windows should initially be placed (if possible) so that they are not clipped by the edge of the screen (if the resource value is True). If a window is larger than the size of the screen, at least the upper-left corner of the window is on-screen. If the resource value is False, windows are placed in the requested position even if totally off-screen. The default value of this resource is True.

**quitTimeout** (class QuitTimeout) Specifies the amount of time in milliseconds that the mwm command waits for a client to update the WM_COMMAND property after the mwm command has sent the WM_SAVE_YOURSELF message. This protocol is used only for those clients that have a WM_SAVE_YOURSELF atom and no WM_DELETE_WINDOW atom in the WM_PROTOCOLS client window property. The default value of this resource is 1000 (milliseconds). See the f.kill function for additional information.

**raiseKeyFocus** (class RaiseKeyFocus) Specifies whether a window raised by means of the f.normalize_and_raise function also receives the input focus. The default value of this resource is False. This resource is available only when the keyboard input focus policy is set to the explicit value.

**resizeBorderWidth** (class ResizeBorderWidth) Specifies the width (in pixels) of a client window frame border with resize handles. The specified border width includes the 3-D shadows. The default value is based on the size and resolution of the screen.

**resizeCursors** (class ResizeCursors) Indicates whether the resize cursors are always displayed when the pointer is in the window size border. If True, the cursors are shown; otherwise, the window manager cursor is shown. The default value is True.

**screens** (class Screens) Specifies the resource names to use for the screens managed by the mwm command. If the mwm command is managing a single screen, only the first name in the list is used. If the mwm command is managing multiple screens, the names are assigned to the screens in order, starting with screen 0. For example, screen 0 gets the first name and screen 1 gets the second name. Examples of default screen names are 0 and 1.
showFeedback (class ShowFeedback) Controls when feedback information is displayed. It controls both window position and size feedback during move or resize operations and initial client placement. It also controls window manager message and dialog boxes.

The value for this resource is a list of names of the feedback options to be enabled or disabled; the names must be separated by a space. If an option is preceded by a minus sign, that option is excluded from the list. The sign of the first item in the list determines the initial set of options. If the sign of the first option is - (a minus sign), the mwm command assumes all options are present and starts subtracting from that set. If the sign of the first decoration is + (a plus sign) or is not specified, the mwm command starts with no options and builds up a list from the resource.

The names of the feedback options are as follows:

all Shows all feedback (default value).
behavior Confirms the behavior switch.
killed Confirms on receipt of the KILL signal.
move Shows position during the move.
one Shows no feedback.
placement Shows position and size during initial placement.
quit Confirms quitting MWM.
resize Shows size during resize.
restart Confirms MWM restart.

The following command line illustrates the syntax for the showFeedback resource:

Mwm*showFeedback: placement resize behavior restart

This resource specification provides feedback for initial client placement and resize, and it enables the dialog boxes to confirm the restart and set behavior functions. It disables feedback for the move function.

The default value for this resource is the all value.

class StartupKeyFocus

determines whether a window gets the keyboard input focus when the window is mapped (that is, initially managed by the window manager). It is recommended that both the autoKeyFocus resource and the startupKeyFocus resource be set to a value of True to work with tear-off menus. The default value is True. This resource is available only when the keyboard input focus policy is set to the explicit value.

startupKeyFocus (class StartupKeyFocus)
transientDecoration (class TransientDecoration)

Controls the amount of decoration that Mwm puts on transient windows. The decoration specification is exactly the same as for the clientDecoration (client-specific) resource. Transient windows are identified by the WM_TRANSIENT_FOR property, which is added by the client to indicate a relatively temporary window. The default value for this resource is the menu title value (that is, transient windows have resize borders and a title bar with a window menu button).

An application can also specify which decorations the window manager should apply to its windows. If it does so, the window manager applies only those decorations indicated by both the application and the transientDecoration resource. Otherwise, the window manager applies only the decorations indicated by the transientDecoration resource.

transientFunctions (class TransientFunctions)

Indicates which window management functions are applicable (or not applicable) to transient windows. The function specification is exactly the same as for the clientFunctions (client-specific) resource. The default value for this resource is -minimize -maximize.

An application can also specify which functions the window manager should apply to its windows. If it does so, the window manager applies only those functions indicated by both the application and the transientFunctions resource. Otherwise, the window manager applies only the functions indicated by the transientFunctions resource.

useIconBox (class UseIconBox)

Determines whether icons are placed in an icon box. If this resource is given a value of True, icons are placed in an icon box. When an icon box is not used, the icons are placed on the root window (default value).

wMenuButtonClick (class WMenuButtonClick)

Indicates whether the window menu is posted and remains posted after a click of the mouse when the pointer is over the Window Menu button. If the value given this resource is True, the menu remains posted. True is the default value for this resource.

wMenuButtonClick2 (class WMenuButtonClick2)

Indicates whether a double-click action on the Window Menu button performs an f.kill function. When this resource is given the default value of True, a double-click action on the Window Menu button performs an f.kill function.

Client-Specific Resources

The syntax for specifying client-specific resources is as follows:

Mwm*ClientNameOrClass*ResourceId

For example, Mwm*mterm*windowMenu is used to specify the window menu to be used with mterm clients.

The syntax for specifying client-specific resources for all classes of clients is as follows:

Mwm*ResourceId

Specific client specifications take precedence over the specifications for all clients. For example, Mwm*windowMenu is used to specify the window menu to be used for all classes of clients that do not have a window menu specified.

The syntax for specifying resource values for windows that have an unknown name and class (that is, windows that do not have a WM_CLASS property associated with them) is as follows:

Mwm*defaults*ResourceId
For example, `Mwm*defaults*iconImage` is used to specify the icon image to be used for windows that have an unknown name and class.

**Client-Specific Resource Set**

**Note:** Hyphens in the following table are for readability purposes only. Do not include hyphens within names in programs.

<table>
<thead>
<tr>
<th>Name</th>
<th>Class</th>
<th>Value Type</th>
<th>Default</th>
</tr>
</thead>
<tbody>
<tr>
<td>clientDecoration</td>
<td>ClientDecoration</td>
<td>all</td>
<td></td>
</tr>
<tr>
<td>clientFunctions</td>
<td>ClientFunctions</td>
<td>string</td>
<td>all</td>
</tr>
<tr>
<td>focusAutoRaise</td>
<td>FocusAutoRaise</td>
<td>True or False</td>
<td>varies</td>
</tr>
<tr>
<td>iconImage</td>
<td>IconImage</td>
<td>pathname (image)</td>
<td></td>
</tr>
<tr>
<td>iconImageBackground</td>
<td>Background</td>
<td>color</td>
<td>icon background</td>
</tr>
<tr>
<td>iconImageBottomShadowColor</td>
<td>Foreground</td>
<td>color</td>
<td>icon bottom shadow</td>
</tr>
<tr>
<td>iconImageBottomShadowPixmap</td>
<td>BottomShadowPixmap</td>
<td>color</td>
<td>icon bottom shadow pixmap</td>
</tr>
<tr>
<td>iconImageForeground</td>
<td>Foreground</td>
<td>color</td>
<td>varies</td>
</tr>
<tr>
<td>iconImageTopShadowColor</td>
<td>Background</td>
<td>color</td>
<td>icon top shadow color</td>
</tr>
<tr>
<td>iconImageTopShadowPixmap</td>
<td>TopShadowPixmap</td>
<td>color</td>
<td>icon top shadow pixmap</td>
</tr>
<tr>
<td>matteBackground</td>
<td>Background</td>
<td>color</td>
<td>background</td>
</tr>
<tr>
<td>matteBottomShadowColor</td>
<td>Foreground</td>
<td>color</td>
<td>bottom shadow color</td>
</tr>
<tr>
<td>matteBottomShadowPixmap</td>
<td>BottomShadowPixmap</td>
<td>color</td>
<td>bottom shadow pixmap</td>
</tr>
<tr>
<td>matteForeground</td>
<td>Foreground</td>
<td>color</td>
<td>foreground</td>
</tr>
<tr>
<td>matteTopShadowColor</td>
<td>Background</td>
<td>color</td>
<td>top shadow color</td>
</tr>
<tr>
<td>matteTopShadowPixmap</td>
<td>TopShadowPixmap</td>
<td>color</td>
<td>top shadow pixmap</td>
</tr>
<tr>
<td>matteWidth</td>
<td>MatteWidth</td>
<td>pixels</td>
<td>0</td>
</tr>
<tr>
<td>maximumClientSize</td>
<td>MaximumClientSize</td>
<td>wxh, vertical, horizontal</td>
<td>fill the screen</td>
</tr>
<tr>
<td>useClientIcon</td>
<td>UseClientIcon</td>
<td>True or False</td>
<td>F</td>
</tr>
<tr>
<td>usePPosition</td>
<td>UsePPosition</td>
<td>string</td>
<td>nonzero</td>
</tr>
<tr>
<td>windowMenu</td>
<td>WindowMenu</td>
<td>string</td>
<td>DefaultWindowMenu</td>
</tr>
</tbody>
</table>
clientDecoration (class ClientDecoration)

Controls the amount of window frame decoration. The resource is specified as a list of decorations to specify their inclusion in the frame. If a decoration is preceded by - (a minus sign), that decoration is excluded from the frame. The sign of the first item in the list determines the initial amount of decoration. If the sign of the first decoration is a minus sign, the mwm command assumes all decorations are present and starts subtracting from that set. If the sign of the first decoration is plus (or not specified), the mwm command starts with no decoration and builds up a list from the resource.

An application can also specify which decorations the mwm command should apply to its windows. If it does so, the mwm command applies only those decorations indicated by both the application and the clientDecoration resource. Otherwise, the mwm command applies the decorations indicated by the clientDecoration resource. Following is a list of window frame decorations:

- **all**: Specifies to include all decorations (default value).
- **border**: Specifies the window border.
- **maximize**: Specifies the Maximize button (includes title bar).
- **minimize**: Specifies the Minimize button (includes title bar).
- **none**: Specifies no decorations.
- **resizeh**: Specifies the border resize handles (includes border).
- **menu**: Specifies the Window Menu button (includes title bar).
- **title**: Specifies the title bar (includes border).

Following are examples of window frame decoration commands:

```
Mwm*XClock.clientDecoration: -resizeh -maximize
```

This removes the resize handles and Maximize button from XClock windows.

```
Mwm*XClock.clientDecoration: menu minimize border
```

This removes the resize handles and Maximize button from XClock windows. Note that either menu or minimize implies title.
clientFunctions (class ClientFunctions)
Indicates which mwm functions are applicable (or not applicable) to the client window. The value for the resource is a list of functions. If the first function in the list has - (a minus sign) in front of it, the mwm command starts with all functions and subtracts from that set. If the first function in the list has a + (plus sign) in front of it, the mwm command starts with no functions and builds up a list. Each function in the list must be preceded by the appropriate + (plus) or - (minus) sign and separated from the next function by a space.

An application can also specify which functions the mwm command should apply to its windows. If it does so, the mwm command applies only those functions indicated by both the application and the clientFunctions resource. Otherwise, the mwm command applies the functions indicated by the clientFunctions resource.

Following is a list of functions available for this resource:

- **all** Specifies to include all functions (default value).
- **none** Specifies no functions.
- **resize** Specifies f.resize.
- **move** Specifies f.move.
- **minimize** Specifies f.minimize.
- **maximize** Specifies f.maximize.
- **close** Specifies f.kill.

focusAutoRaise (class FocusAutoRaise)
Determines whether clients are raised when they get the keyboard input focus. If the value is False, the stacking of windows on the display is not changed when a window gets the keyboard input focus. The default value is True when the keyboardFocusPolicy is the explicit value and False when the keyboardFocusPolicy is the pointer value.

iconImage (class IconImage)
Specifies an icon image for a client (for example, Mwm*myclock*iconImage). The resource value is a path name for a bitmap file. The value of the (client-specific) useClientIcon resource is used to determine whether user-supplied icon images are used instead of client-supplied icon images. The default value is to display a built-in window manager icon image.

iconImageBackground (class Background)
Specifies the background color of the icon image that is displayed in the image part of an icon. The default value of this resource is the icon background color (that is, specified by Mwm*background or Mwm*icon*background).

iconImageBottomShadowColor (class Foreground)
Specifies the bottom shadow color of the icon image that is displayed in the image part of an icon. The default value of this resource is the icon bottom shadow color (that is, specified by Mwm*icon*bottomShadowColor).

iconImageBottomShadowPixmap (class BottomShadowPixmap)
Specifies the bottom shadow pixmap of the icon image that is displayed in the image part of an icon. The default value of this resource is the icon bottom shadow pixmap (that is, specified by Mwm*icon*bottomShadowPixmap).
iconImageForeground (class Foreground)  
Specifies the foreground color of the icon image that is displayed in the image part of an icon. The default value of this resource varies depending on the icon background.

iconImageTopShadowColor (class Background)  
Specifies the top shadow color of the icon image that is displayed in the image part of an icon. The default value of this resource is the icon top shadow color (that is, specified by Mwm*icon*topShadowColor).

iconImageTopShadowPixmap (class TopShadowPixmap)  
Specifies the top shadow pixmap of the icon image that is displayed in the image part of an icon. The default value of this resource is the icon top shadow pixmap (that is, specified by Mwm*icon*topShadowPixmap).

matteBackground (class Background)  
Specifies the background color of the matte when the matteWidth resource is a positive value. The default value of this resource is the client background color (that is, specified by Mwm*client*background or Mwm*client*background).

matteBottomShadowColor (class Foreground)  
Specifies the bottom shadow color of the matte when the matteWidth resource is a positive value. The default value of this resource is the client bottom shadow color (that is, specified by Mwm*bottomShadowColor or Mwm*client*bottomShadowColor).

matteBottomShadowPixmap (class BottomShadowPixmap)  
Specifies the bottom shadow pixmap of the matte when the matteWidth resource is a positive value. The default value of this resource is the client bottom shadow pixmap (that is, specified by Mwm*bottomShadowPixmap or Mwm*client*bottomShadowPixmap).

matteForeground (class Foreground)  
Specifies the foreground color of the matte when the matteWidth resource is a positive value. The default value of this resource is the client foreground color (that is, specified by Mwm*foreground or Mwm*client*foreground).

matteTopShadowColor (class Background)  
Specifies the top shadow color of the matte when the matteWidth resource is a positive value. The default value of this resource is the client top shadow color (that is, specified by Mwm*topShadowColor or Mwm*client*topShadowColor).

matteTopShadowPixmap (class TopShadowPixmap)  
Specifies the top shadow pixmap of the matte when the matteWidth resource is a positive value. The default value of this resource is the client top shadow pixmap (that is, specified by Mwm*topShadowPixmap or Mwm*client*topShadowPixmap).

matteWidth (class MatteWidth)  
Specifies the width of the optional matte. The default value is 0, which effectively disables the matte.

maximumClientSize (class MaximumClientSize)  
Indicates the client size to be used when an application is maximized. The resource value is specified WidthxHeight. The width and height are interpreted in the units that the client uses (for example, this is generally characters for terminal emulators). Alternately, the vertical or horizontal value can be specified to indicate the direction in which the client maximizes. If this resource is not specified, the maximum size from the WM_NORMAL_HINTS property is used if set. Otherwise, the default value is the size where the client window with window management borders fills the screen. When the maximum client size is not determined by the maximumClientSize resource, the maximumMaximumSize resource value is used as a constraint on the maximum size.
useClientIcon (class UseClientIcon) Determines whether a client-supplied icon image takes precedence over a user-supplied icon image. The default value is False, giving the user-supplied icon image higher precedence than the client-supplied icon image.

usePPosition (class UsePPosition) Specifies whether the window manager honors the program-specified position PPosition specified in the WM_NORMAL_HINTS property in the absence of a user-specified position. Setting this resource to On causes the mwm command to always honor the program-specified position. Setting this resource to Off causes the mwm command to always ignore the program-specified position. Setting this resource to the default value of nonzero causes the mwm command to honor program-specified positions other than (0,0).

windowMenu (class WindowMenu) Indicates the name of the menu pane that is posted when the window menu is opened (usually by pressing button 1 on the Window Menu button on the client window frame). Menu panes are specified in the mwm resource description file. Window menus can be customized on a client class basis by specifying resources of the form Mwm*ClassNameOrClass*windowMenu (See mwm Resource Description File Syntax for more information.) The default value of this resource is DefaultWindowMenu.

Resource Description File

The mwm resource description file is a supplementary resource file that contains resource descriptions that are referred to by entries in the defaults files (.Xdefaults, app-defaults/Mwm). It contains descriptions of resources that are to be used by the mwm command and that cannot be easily encoded in the defaults files (a bitmap file is an analogous type of resource description file). A particular mwm resource description file can be selected using the configFile resource.

The following types of resources can be described in the mwm resource description file:

buttons Window manager functions can be bound (associated) with button events.
keys Window manager functions can be bound (associated) with key press events.
menus Menu panes can be used for the window menu and other menus posted with key bindings and button bindings.

mwm Resource Description File Syntax

The mwm resource description file is a standard text file that contains items of information separated by blanks, tabs, and new-line characters. Blank lines are ignored. Items or characters can be quoted to avoid special interpretation (for example, the # (comment character) can be quoted to prevent it from being interpreted as the comment character). A quoted item can be contained in " " (double quotation marks). Single characters can be quoted by preceding them with the \ (backslash). All text from an unquoted # (comment character) to the end of the line is regarded as a comment and is not interpreted as part of a resource description. If an ! (exclamation mark) is the first character in a line, the line is regarded as a comment. If a line ends in a \ (backslash), the next line is considered a continuation of that line.

Window manager functions can be accessed with button and key bindings and with window manager menus. Functions are indicated as part of the specifications for button and key binding sets and for menu panes. The function specification has the following syntax:

Function = FunctionName [FunctionArguments]
FunctionName = Window Manager Function
FunctionArguments = {QuotedItem | UnquotedItem}
The following functions are supported. If a function is specified that is not one of the supported functions, it is interpreted by the mwm command as the f.nop function.

f.beep
Causes a beep.

f.circle_down [Icon | Window]
Causes the window or icon that is on the top of the window stack to be put on the bottom of the window stack (so that it no longer obscures any other window or icon). This function affects only those windows and icons that obscure other windows and icons or that are obscured by other windows and icons. Secondary windows (that is, transient windows) are restacked with their associated primary window. Secondary windows always stay on top of the associated primary window and there can be no other primary windows between the secondary windows and their primary window. If an icon function argument is specified, the function applies only to icons. If a Window function argument is specified, the function applies only to windows.

f.circle_up [Icon | Window]
Raises the window or icon on the bottom of the window stack (so that it is not obscured by any other windows). This function affects only those windows and icons that obscure other windows and icons or that are obscured by other windows and icons. Secondary windows (that is, transient windows) are restacked with their associated primary window. If an icon function argument is specified, the function applies only to icons. If a Window function argument is specified, the function applies only to windows.

f.exec or !
Causes the command to be run (using the value of the MWM SHELL environment variable if it is set; otherwise, the value of the SHELL environment variable if it is set; otherwise, /usr/bin/sh is used). The ! notation can be used in place of the f.exec function name.

f.focus_color
Sets the colormap focus to a client window. If this function is done in a root context, the default colormap (set up by the X Window System client for the screen where MWM is running) is installed and there is no specific client window colormap focus. This function is treated as f.nop if colormapFocusPolicy is not set to the explicit value.

f.focus_key
Sets the keyboard input focus to a client window or icon. This function is treated as f.nop if keyboardFocusPolicy is not set to the explicit value or the function is run in a root context.

f.kill
Stops a client. If the WM_DELETE_WINDOW protocol is set up, the client is sent a client message event indicating that the client window needs to be deleted. If the WM_SAVE_YOURSELF protocol is set up and the WM_DELETE_WINDOW protocol is not set up, the client is sent a client message event indicating that the client needs to prepare to be stopped. If the client does not have the WM_DELETE_WINDOW or WM_SAVE_YOURSELF protocol set up, this function causes a client’s X connection to be stopped (usually resulting in the end of the client).

See the description of the quitTimeout resource.
f.lower [-Client | within | freeFamily]  
Lowers a client window to the bottom of the window stack (where it obscures no other window). Secondary windows (that is, transient windows) are restacked with their associated primary window. The Client argument indicates the name or class of a client to lower. If the Client argument is not specified, the context, in which the function was started, indicates the window or icon to lower.

f.maximize  
Causes a client window to be displayed with its maximum size.

f.menu  
Associates a cascading (pull-right) menu with a menu pane entry or a menu with a button or key binding. The menu_name function argument identifies the menu to be used.

f.minimize  
Causes a client window to be iconified (minimized). When a window is minimized and no icon box is used, its icon is placed on the bottom of the window stack (so that it obscures no other windows). If an icon box is used, the client's icon changes to its iconified form inside the icon box. Secondary windows (that is, transient windows) are minimized with their associated primary window. There is only one icon for a primary window and all its secondary windows.

f.move  
Causes a client window to be interactively moved.

f.next_cmap  
Installs the next colormap in the list of colormaps for the window with the colormap focus.

f.next_key [ Icon | Window | Transient ]  
Sets the keyboard input focus to the next window or icon in the set of windows and icons managed by the window manager (the ordering of this set is based on the stacking of windows on the screen). This function is treated as f.nop if keyboardFocusPolicy is not the explicit value.

The keyboard input focus is moved only to windows that do not have an associated secondary window that is application-modal. If the Transient argument is specified, transient (secondary) windows are crossed (otherwise, if only the Window argument is specified, traversal is done only to the last focused window in a transient group). If an Icon function argument is specified, the function applies only to icons. If a Window function argument is specified, the function applies only to windows.

f.nop  
Does nothing. If a function is specified in a type of resource where it is not supported or is started in a context that does not apply, the function is treated as f.nop.

f.normalize  
Causes a client window to be displayed with its normal size. Secondary windows (that is, transient windows) are placed in their normal state along with their associated primary window.

f.normalize_and_raise  
Causes the corresponding client window to be displayed with its normal size and raised to the top of the window stack. Secondary windows (that is, transient windows) are placed in their normal state along with their associated primary window.

f.pack_icons  
Causes icons to be packed into the icon grid. This function is used to relay out icons (based on the layout policy being used) on the root window or in the icon box.
f.pass_keys

Enables or disables (toggles) processing of key bindings for window manager functions. When it disables key binding processing, all keys are passed on to the window with the keyboard input focus and no window manager functions are started. If the f.pass_keys function is started with a key binding to disable key-binding processing, the same key binding can be used to enable key-binding processing.

f.post_wmenu

Posts the window menu. If a key is used to post the window menu and the Window Menu button is present, the window menu is automatically placed with its top-left corner at the bottom-left corner of the Window Menu button for the client window. If no Window Menu button is present, the window menu is placed at the top-left corner of the client window.

f.prev_cmap

Installs the previous colormap in the list of colormaps for the window with the colormap focus.

f.prev_key [Icon | Window | Transient]

Sets the keyboard input focus to the previous window or icon in the set of windows and icons managed by the window manager (the ordering of this set is based on the stacking of windows on the screen). This function is treated as f.nop if keyboardFocusPolicy is not the explicit value. The keyboard input focus is moved only to windows that do not have an associated secondary window that is application-modal. If the Transient argument is specified, transient (secondary) windows are crossed (otherwise, if only window is specified, traversal is done only to the last focused window in a transient group). If an Icon function argument is specified, the function applies only to icons. If a Window function argument is specified, the function applies only to windows.

f.quit_mwm

Stops the mwm command (but not the X Window System client).

f.raise [-Client | within | freeFamily]

Raises a client window to the top of the window stack (where it is obscured by no other window). Raises the secondary window (transient window or dialog box) within the client family. The arguments to this function are mutually exclusive. The Client argument indicates the name or class of a client to raise. If the Client argument is not specified, the context in which the function was started indicates the window or icon to raise. Specifying within raises the secondary window within the family but does not raise the client family in the global window stack. Specifying freeFamily raises the window to the top of its local family stack and raises the family to the top of the global window stack.
f.raise_lower [within | freeFamily]

Raises a primary window to the top of the window stack if it is partially obscured by another window; otherwise, it lowers the window to the bottom of the window stack. The arguments to this function are mutually exclusive.

Specifying within raises a secondary window within the family (staying above the parent window), if it is partially obscured by another window in the application’s family; otherwise, it lowers the window to the bottom of the family stack. It has no effect on the global stacking order.

Specifying freeFamily raises the window to the top of its local family stack, if obscured by another window, and raises the family to the top of the global window stack; otherwise, it lowers the window to the bottom of its local family stack and lowers the family to the bottom of the global window stack.

f.refresh

Causes all windows to be redrawn.

f.refresh_win

Causes a client window to be redrawn.

f.resize

Causes a client window to be interactively resized.

f.restart

Causes the mwm command to be restarted (effectively stopped and restarted).

f.restore

Restores the previous state of an icon’s associated window. If a maximized window is iconified, the f.restore function restores it to its maximized state. If a normalized window is iconified, the f.restore function restores it to its normalized state.

f.restore_and_raise

Restores the previous state of an icon’s associated window and raises the window to the top of the window stack. If a maximized window is iconified, the f.restore_and_raise function restores it to its maximized state and raises it to the top of the window stack. If a normalized window is iconified, the f.restore_and_raise function restores it to its normalized state and raises it to the top of the window stack.

f.screen [next | prev | back | ScreenNumber]

Causes the pointer to warp to a specific screen number or to the next, previous, or last visited screen. The arguments to this function are mutually exclusive. The following arguments are available:

ScreenNumber
Indicates the screen number to which the pointer is warped. Screens are numbered starting from screen 0.

next  
Warps the pointer to the next managed screen (skipping over any unmanaged screens).

prev  
Warps the pointer to the previous managed screen (skipping over any unmanaged screens).

back  
Warps the pointer to the last visited screen.

f.send_msg MessageNumber

Sends a client message of the _MOTIF_WM_MESSAGES type with the MessageType indicated by the MessageNumber function argument. The client message is sent only if MessageNumber is included in the client’s _MOTIF_WM_MESSAGES property. A menu item label is unavailable if the menu item is used to perform the f.send_msg function of a message that is not included in the client’s _MOTIF_WM_MESSAGES property.

f.separator

Causes a menu separator to be put in the menu pane at the specified location (the label is ignored).
f.set_behavior  
Causes the window manager to restart with the default behavior (if a custom behavior is configured) or revert to the custom behavior. By default this is bound to the Shift+Ctrl+Meta+! key sequence.

The Meta+Shift+Ctrl+! key sequence switches (that is, toggles) between the default and custom behaviors. When the user switches to the default MWM behavior, a number of mwm resources assume their default values and the mwm command restarts. When the user switches back to the custom behavior, the resource values that were changed to default values are reset with the custom values and the mwm command restarts.

When an f.set_behavior function is performed, the following user interaction occurs:
1. A system-modal dialog box is displayed prompting the user for confirmation of the f.set_behavior action.
2. The user can cancel the action at this point.
3. The window manager restarts.
4. The window manager applies the new (custom or default) configuration values.
5. Window manager components are mapped.

When the default MWM behavior is being set, default resource values are applied and, if specified, client properties that control window manager behavior are applied. This includes the _MOTIF_WM_HINTS and _MOTIF_WM_MENU properties. These properties might alter default MWM behavior, but it is done in a way that is consistent for all users.

f.title  
Inserts a title in the menu pane at the specified location.

**Function Contexts**
Each function may be constrained as to which resource types can specify the function (for example, menu pane) and also what context the function can be used in (for example, the function is done to the selected client window). The following are the function contexts:

- **root**  No client window or icon is selected as an object for the function.
- **window**  A client window is selected as an object for the function. This includes the window’s title bar and frame. Some functions are applied only when the window is in its normalized state (for example, f.maximize) or its maximized state (for example, f.normalize).
- **icon**  An icon is selected as an object for the function.

If a function’s context is specified as icon|window and the function is started in an icon box, the function applies to the icon box, not to the icons inside.

If a function is specified in a type of resource where it is not supported or is started in a context that does not apply, the function is treated as f.nop. The following table indicates the resource types and function contexts in which window manager functions apply:

<table>
<thead>
<tr>
<th>Function</th>
<th>Contexts</th>
<th>Resources</th>
</tr>
</thead>
<tbody>
<tr>
<td>f.beep</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.circle_down</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.circle_up</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>Function</td>
<td>Contexts</td>
<td>Resources</td>
</tr>
<tr>
<td>-----------------------</td>
<td>---------------------------</td>
<td>------------------</td>
</tr>
<tr>
<td>f.exec</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.focus_color</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.focus_key</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.kill</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.lower</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.maximize</td>
<td>icon, window (normal)</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.menu</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.minimize</td>
<td>window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.move</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.next_cmap</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.next_key</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.nop</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.normalize</td>
<td>icon, window (maximized)</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.normalize_and_raise</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.pack_icons</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.pass_keys</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.post_wmenu</td>
<td>root, icon, window</td>
<td>button, key</td>
</tr>
<tr>
<td>f.prev_cmap</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.prev_key</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.quit_mwm</td>
<td>root, icon, window</td>
<td>button, key, menu (root only)</td>
</tr>
<tr>
<td>f.raise</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.raise_lower</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.refresh</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.refresh_win</td>
<td>window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.resize</td>
<td>window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.restart</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.restore</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.restore_and_raise</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.screen</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.send_msg</td>
<td>icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.separator</td>
<td>root, icon, window</td>
<td>menu</td>
</tr>
<tr>
<td>f.set_behavior</td>
<td>root, icon, window</td>
<td>button, key, menu</td>
</tr>
<tr>
<td>f.title</td>
<td>root, icon, window</td>
<td>menu</td>
</tr>
</tbody>
</table>

**Window Manager Event Specification**

Events are indicated as part of the specifications for button and key-binding sets and for menu panes.

Button events have the following syntax:

Button = [ModifierList]<ButtonEventName>

ModifierList = Modifier Name (ModifierName)
All modifiers specified are interpreted as being exclusive (this means that only the specified modifiers can be present when the button event occurs). Following is a list that indicates the values that can be used for the `ModifierName` parameter. The Alt key is frequently labeled Extend or Meta. Alt and Meta can be used interchangeably in event specification.

<table>
<thead>
<tr>
<th>Modifier</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl</td>
<td>Control key</td>
</tr>
<tr>
<td>Shift</td>
<td>Shift key</td>
</tr>
<tr>
<td>Alt</td>
<td>Alt or Meta key</td>
</tr>
<tr>
<td>Meta</td>
<td>Meta or Alt key</td>
</tr>
<tr>
<td>Lock</td>
<td>Lock key</td>
</tr>
<tr>
<td>Mod1</td>
<td>Modifier1</td>
</tr>
<tr>
<td>Mod2</td>
<td>Modifier2</td>
</tr>
<tr>
<td>Mod3</td>
<td>Modifier3</td>
</tr>
<tr>
<td>Mod4</td>
<td>Modifier4</td>
</tr>
<tr>
<td>Mod5</td>
<td>Modifier5</td>
</tr>
</tbody>
</table>

Following is a list that indicates the values that can be used for the `ButtonEventName` parameter.

<table>
<thead>
<tr>
<th>Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Btn1Down</td>
<td>Button 1 press</td>
</tr>
<tr>
<td>Btn1Up</td>
<td>Button 1 release</td>
</tr>
<tr>
<td>Btn1Click</td>
<td>Button 1 press and release</td>
</tr>
<tr>
<td>Btn1Click2</td>
<td>Button 1 double click</td>
</tr>
<tr>
<td>Btn2Down</td>
<td>Button 2 press</td>
</tr>
<tr>
<td>Btn2Up</td>
<td>Button 2 release</td>
</tr>
<tr>
<td>Btn2Click</td>
<td>Button 2 press and release</td>
</tr>
<tr>
<td>Btn2Click2</td>
<td>Button 2 double click</td>
</tr>
<tr>
<td>Btn3Down</td>
<td>Button 3 press</td>
</tr>
<tr>
<td>Btn3Up</td>
<td>Button 3 release</td>
</tr>
<tr>
<td>Btn3Click</td>
<td>Button 3 press and release</td>
</tr>
<tr>
<td>Btn3Click2</td>
<td>Button 3 double click</td>
</tr>
<tr>
<td>Btn4Down</td>
<td>Button 4 press</td>
</tr>
<tr>
<td>Btn4Up</td>
<td>Button 4 release</td>
</tr>
<tr>
<td>Btn4Click</td>
<td>Button 4 press and release</td>
</tr>
<tr>
<td>Btn4Click2</td>
<td>Button 4 double click</td>
</tr>
<tr>
<td>Btn5Down</td>
<td>Button 5 press</td>
</tr>
<tr>
<td>Btn5Up</td>
<td>Button 5 release</td>
</tr>
<tr>
<td>Btn5Click</td>
<td>Button 5 press and release</td>
</tr>
<tr>
<td>Btn5Click2</td>
<td>Button 5 double click</td>
</tr>
</tbody>
</table>

Key events that are used by the window manager for menu mnemonics and for binding to window manager functions are single key presses; key releases are ignored. Key events have the following syntax:

```
Key = [ModifierList] <Key> KeyName
ModifierList = ModifierName [ModifierName]
```

All modifiers specified are interpreted as being exclusive (this means that only the specified modifiers can be present when the key event occurs). Modifiers for keys are the same as those that apply to buttons. The `KeyName` parameter is an X11 keysym name. Key symbol names can be found in the `keysymdef.h` file (remove the `XK_` prefix).

The key symbol names will be resolved to a single specific key code by the Window Manager during startup and will not change unless the Window Manager is restarted.
Button Bindings

The `buttonBindings` resource value is the name of a set of button bindings that are used to configure window manager behavior. A window manager function can be used when a button press occurs with the pointer over a framed client window, an icon, or the root window. The context for indicating where the button press applies is also the context for starting the window manager function when the button press is done (significant for functions that are context-sensitive).

Following is the button binding syntax:

```plaintext
Buttons BindingsSetName
    { Button Context Function
      Button Context Function
      .
    }
    Button Context Function
}
```

Following is the syntax for the context specification:

```plaintext
Context = Object[[Context]]
Object = root | icon | window | title | frame | border | app
```

The `Context` specification indicates where the pointer must be for the button binding to be effective. For example, a context of `window` indicates that the pointer must be over a client window or window management frame for the button binding to be effective. The `frame` context is for the window management frame around a client window (including the border and title bar), the `border` context is for the border part of the window management frame (not including the title bar), the `title` context is for the title area of the window management frame, and the `app` context is for the application window (not including the window management frame).

If an `f.nop` function is specified for a button binding, the button binding is not done.

Key Bindings

The `keyBindings` resource value is the name of a set of key bindings that are used to configure window manager behavior. A window manager function can be done when a particular key is pressed. The context in which the key binding applies is indicated in the key binding specification. The valid contexts are the same as those that apply to button bindings.

Following is the key binding syntax:

```plaintext
Keys BindingsSetName
    { Key Context Function
      Key Context Function
      .
      Key Context Function
    }
```

If an `f.nop` function is specified for a key binding, the key binding is not done. If an `f.post_wmenu` or `f.menu` function is bound to a key, the `mwm` command automatically uses the same key for removing the menu from the screen after it is open.

The `Context` specification syntax is the same as for button bindings. For key bindings, the `frame`, `title`, `border`, and `app` contexts are equivalent to the `window` context. The context for a key event is the window or icon that has the keyboard input focus (`root` if no window or icon has the keyboard input focus).
Menu Panes

Menus can be opened using the `f.post_wmenu` and `f.menu` window manager functions. The context for window manager functions that are done from a menu is `root`, `icon`, or `window`, depending on how the menu is opened. In the case of the window menu or menus opened with a key binding, the location of the keyboard input focus indicates the context. For menus opened using a button binding, the context of the button binding is the context of the menu.

Following is the menu pane specification syntax:

```
Menu MenuName
{
Label [Mnemonic] [Accelerator] Function
Label [Mnemonic] [Accelerator] Function
.
Label [Mnemonic] [Accelerator] Function
}
```

Each line in the `Menu` specification identifies the label for a menu item and the function to be completed if the menu item is selected. Optionally, a menu button mnemonic and a menu button keyboard accelerator can be specified. Mnemonics are functional only when the menu is posted and keyboard traversal applies.

The label can be a string or a bitmap file. The `Label` specification has the following syntax:

```
Label = Text | BitmapFile
BitmapFile = @FileName
Text = QuotedItem | UnquotedItem
```

The string encoding for labels must be compatible with the menu font that is used. Labels are not available for menu items that use the `f.nop` function, an invalid function, or a function that does not apply in the current context.

A `Mnemonic` specification has the following syntax:

```
Mnemonic = _Character
```

The first matching `Character` in the label is underlined. If there is no matching `Character` in the label, no mnemonic is registered with the window manager for that label. Although the `Character` must exactly match a character in the label, the mnemonic does not perform if any modifier (such as the Shift key) is pressed with the character key.

The `Accelerator` specification is a key event specification with the same syntax that is used for key bindings to window manager functions.

Environment

The `mwm` command does the following:

- Uses the `HOME` environment variable to specify the user’s home directory.
- Uses the `LANG` environment variable to specify the user’s choice of language for the `mwm` message catalog and the `mwm` resource description file.
- Uses the `XFILESEARCHPATH`, `USERFILESEARCHPATH`, `XAPPRSDIR`, `XENVIRONMENT`, `LANG`, and `HOME` environment variables to determine search paths for resource defaults files. The `mwm` command can also use the `XBMLANGPATH` environment variable to search for bitmap files.
- Reads the `HOME/.motifbind` file, if it exists, to install a virtual key bindings property on the root window.
- Uses the `MWMSHELL` environment variable (or `SHELL` if `MWMSHELL` is not set) to specify the shell to use when running commands through the `f.exec` function.
Exit Status
This command returns the following exit values:

0 Indicates successful completion.
>1 Indicates an error occurred.

Files
/usr/lib/X11/$LANG/system.mwmrc
/usr/lib/X11/system.mwmrc
/usr/lib/X11/app-defaults/Mwm
$HOME/Mwm
$HOME/.Xdefaults
$HOME/$LANG/.mwmrc
$HOME/.mwmrc
$HOME/.motifbind

Related Information
The X command in AIX 5L Version 5.2 Commands Reference.
Appendix. Notices

This information was developed for products and services offered in the U.S.A.

IBM may not offer the products, services, or features discussed in this document in other countries. Consult your local IBM representative for information on the products and services currently available in your area. Any reference to an IBM product, program, or service is not intended to state or imply that only that IBM product, program, or service may be used. Any functionally equivalent product, program, or service that does not infringe any IBM intellectual property right may be used instead. However, it is the user’s responsibility to evaluate and verify the operation of any non-IBM product, program, or service.

IBM may have patents or pending patent applications covering subject matter described in this document. The furnishing of this document does not give you any license to these patents. You can send license inquiries, in writing, to:

IBM Director of Licensing
IBM Corporation
North Castle Drive
Armonk, NY 10504-1785
U.S.A.

The following paragraph does not apply to the United Kingdom or any other country where such provisions are inconsistent with local law: INTERNATIONAL BUSINESS MACHINES CORPORATION PROVIDES THIS PUBLICATION "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. Some states do not allow disclaimer of express or implied warranties in certain transactions, therefore, this statement may not apply to you.

This information could include technical inaccuracies or typographical errors. Changes are periodically made to the information herein; these changes will be incorporated in new editions of the publication. IBM may make improvements and/or changes in the product(s) and/or the program(s) described in this publication at any time without notice.

Licensees of this program who wish to have information about it for the purpose of enabling: (i) the exchange of information between independently created programs and other programs (including this one) and (ii) the mutual use of the information which has been exchanged, should contact:

IBM Corporation
Dept. LRAS/Bldg. 003
11400 Burnet Road
Austin, TX 78758-3498
U.S.A.

Such information may be available, subject to appropriate terms and conditions, including in some cases, payment of a fee.

The licensed program described in this document and all licensed material available for it are provided by IBM under terms of the IBM Customer Agreement, IBM International Program License Agreement or any equivalent agreement between us.

For license inquiries regarding double-byte (DBCS) information, contact the IBM Intellectual Property Department in your country or send inquiries, in writing, to:
IBM may use or distribute any of the information you supply in any way it believes appropriate without incurring any obligation to you.

Information concerning non-IBM products was obtained from the suppliers of those products, their published announcements or other publicly available sources. IBM has not tested those products and cannot confirm the accuracy of performance, compatibility or any other claims related to non-IBM products. Questions on the capabilities of non-IBM products should be addressed to the suppliers of those products.

Any references in this information to non-IBM Web sites are provided for convenience only and do not in any manner serve as an endorsement of those Web sites. The materials at those Web sites are not part of the materials for this IBM product and use of those Web sites is at your own risk.

This information contains examples of data and reports used in daily business operations. To illustrate them as completely as possible, the examples include the names of individuals, companies, brands, and products. All of these names are fictitious and any similarity to the names and addresses used by an actual business enterprise is entirely coincidental.

COPYRIGHT LICENSE:

This information contains sample application programs in source language, which illustrates programming techniques on various operating platforms. You may copy, modify, and distribute these sample programs in any form without payment to IBM, for the purposes of developing, using, marketing or distributing application programs conforming to the application programming interface for the operating platform for which the sample programs are written. These examples have not been thoroughly tested under all conditions. IBM, therefore, cannot guarantee or imply reliability, serviceability, or function of these programs. You may copy, modify, and distribute these sample programs in any form without payment to IBM for the purposes of developing, using, marketing, or distributing application programs conforming to IBM's application programming interfaces.

Any performance data contained herein was determined in a controlled environment. Therefore, the results obtained in other operating environments may vary significantly. Some measurements may have been made on development-level systems and there is no guarantee that these measurements will be the same on generally available systems. Furthermore, some measurement may have been estimated through extrapolation. Actual results may vary. Users of this document should verify the applicable data for their specific environment.

Trademarks

The following terms are trademarks of International Business Machines Corporation in the United States, other countries, or both:

- AIX
- AIX 5L
- IBM

UNIX is a registered trademark of The Open Group in the United States and other countries.

Other company, product, or service names may be the trademarks or service marks of others.


Index

Special characters
/etc/filesystems file
listing entries
  using lsfs command 320

/etc/inittab file
listing records
  using lsitab command 323
making records
  using mkinitab command 512

/etc/rc.nfs
uncomments entry in
  using mkclient command 487

/etc/vfs file
listing entries
  using lsvfs command 397

A
account, user
  customizing
    using mkuser.sys command 597
accounting system
  displaying information about last commands
    run 148
  displaying previous logins and logouts 147
  performing periodic accounting
    using monacct command 611
  updating last log in records 150
acct/* commands
  lastlogin 150
  monacct 611

adapters and devices
  displaying
    using lsmodule command 333
add, command in lb_admin 151
address maps
  in ld command 172

AIXwindows Window Manager
  running
    using mwm command 650
  window management
    using mwm command 650
answering client requests
  for file system mounts
    using mountd daemon 630
appearance
  general
    using mwm command 659
appearance of window
  managing
    using mwm command 650
Appearance resource set
  using mwm command 659
attributes
  displaying characteristics of
    using lsattr command 283
attributes (continued)
  displaying possible values of
    using lsattr command 283
    of exported symbols
      in ld symbols 171

B
behavior
  general
    using mwm command 659
behavior of window
  managing
    using mwm command 650
Behavior resource set
  using mwm command 659
bibliography
  building inverted index for
    finding references in
      using lookbib command 219
binder
  using ld command 160
binding
  button
    using mwm command 682
  key
    using mwm command 683
boot image
  creating
    using mkboot command 474
boot record
  creating
    using mkboot command 474
button
  binding
    using mwm command 683

C
C programming language
  checking source code for problems 189
  reformatting programs 32
CacheFS directory
  mounting a
    using mkclient command 484
CD-ROM
  reporting statistics 97
clean, command in lb_admin 151
Client-Specific resource set
  using mwm command 671
code page
  converting one page encoding set to another 5
command
  mwm 650
commands
  elogevent 208

© Copyright IBM Corp. 1997, 2004

689
Commands (continued)

command response resource manager (ERRM)
elogevent 208
logevent 208

ikepd 21
installms 60
installp 49
invscoutd 79
join 116
kdb 120
keycomp 124
keyenvoy 127
ld 156
logevent 208
lppchk 248
lpr 255
ls 271
lsactdef 276
lsattr 283
lsaudrec 287
lscomg 297
lscondition 299
lscondresp 303
lscons 310
lsllpp 325
lslv 329
lsmcode 333
lsresponse 359
lsrpdomain 365
lsrpnod 368
lsrsrc 372
lsrsrcdef 377
lsensor 386
mail 419
Mail 419
mailq 432
mailx 419
make 435
mgmtsvr 457
mkboot 474
mkcfsmnt 484
mkclient 487
mkcomg 488
mkcondition 491
mkcondresp 496
mkfs 506
mkitab 512
mklv 520
mknfs 531
mknfsmnt 533
mkptrsv 549
mkresponse 559
mkrole 563
mkrpdomain 565
mkrsrc 568
mksensor 579
mkssys 584
mkstr 587
mkuser 594
mkuser.sys 597
mkvg 598

Commands, mtrace 643
component
appearance resources
using mwm command 656
Component Appearance resource set
using mwm command 657
computer languages
C
lexical analyzer 180
configuring system
to run NFS
using mknfs command 531
connections
displaying for a device
using lsconn command 308
console device
writing name to standard output
using lscons command 310
CPU
statistics 97
creating
boot image, boot record, and service record
using mkboot command 474
error message file
using mkstr command 587
file system
using mkfs command 506
logical volume
using mklv command 520
new user account
using mkuser command 594
volume group
using mkvg command 598
customizing
new user account
using mkuser.sys command 597

D
daemon
launching Inventory Scout server
using invscoutd command 79
daemons
llbd 197
lockd 205
monitord 613
mountd 630
mrouted 635
datagram, multicast
forwarding a
using mrouted daemon 635
default window menu
using mwm command
using mwm command 652
delete, command in "lb_admin" 152
description file
creating
using makedev command 445
using mwm command 675
device
  adding to the system
    using mkdev command 498
Device Configuration Database 343
  listing acceptable devices from
    using lsparent command 343
device stanza name
  displaying the
    using lsquedev command 355
devices
  displaying characteristics of
    using lsdev command 311
devices and adapters
  displaying
    using lsmcode command 333
diagnostic messages
  issuing by optional programs 76
directories
  creating 500
    creating a hierarchy 502
  displaying contents
    using ls command 271
  moving 645, 648
  renaming 648
directory
  creating a lost and found
    using mklost+found command 519
  mounting
    using mount command 621
  using infocntr command 42
disks
  reporting statistics 97
displaying
  directory contents
    using ls command 271
displaying system images
  for examining a dump
    using kdb command 120
displays
  files
    one screen at a time 614
  listing currently available on the system
    using lsdisp command 317
document
  typesetting
    using mmt command 606
dump, examining
  displaying system images
    using kdb command 120

E
elogevent command 208
elogevent script 208
encryption
  generating an encryption key
    using makekey command 445
enqueueing
print jobs
  using lpr command 255
environment variable
  using mwm command 684
environment variables
  in ld command 173
ERRM
  event information
    logging 208
ERRM commands
  elogevent 208
  logevent 208
ERRM scripts
  elogevent 208
  logevent 208
error message file
  creation
    using mkstr command 587
error messages
  issuing by optional programs 76
event information
  logging 208
event response resource manager (ERRM)
  commands
    elogevent 208
    logevent 208
  event information
    logging 208
  scripts
    elogevent 208
    logevent 208
  event specification
    using mwm command 681

F
fields, data
  joining from two files
    using join command 116
file
  creating a special
    using mknod command 536
  finding lines in a sorted
    using look command 218
  resource description
    using mwm command 675
  writing inode information 113
file system
  constructing a prototype
    using mkproto command 542
  making
    using mkfs command 506
  making available for use
    using mount command 621
file system mounts
  answering client requests
    using mountd daemon 630
files
  archive
    using ld command 157
  creating links 198
  displaying
    one screen at a time 614
files (continued)
import and export
using ld command 157
joining the data fields of two
using join command 116
linking object
using ld command 156
moving 645
files
installing, associated with keywords or fixes 63
fixes
installing filesets associated with 63
flags
ignored and unsupported
using ld command 158
folder
incorporating new mail into 30
folders
printing full path names of
using mhpath command 462
font code
adding to the system
using mkfont command 504
font files
creating fonts.dir file
using mkfontdir command 505
font set
changing
using managefonts command 450
updating
using managefonts command 450
fonts
listing the fonts available for use
using lsfont command 319
Frame Component resource set
using mwm command 658

G
games
number-guessing game 614
garbage collection
using ld command 158
global location broker
administrative tools for 150
groups
creating new groups
using mkgroup command 509
displaying attributes of
using lsgroup command 321

H
help
using files, editors, and macros 176
hipid 6
hipindent 32
hipiostat 97
hipipcs 103
hipline 176
hiplord 220
hiplmkvdata 600

I
I/O
reporting statistics 97
IBM 3812 Model 2 Pageprinter
postprocessing troff command output for 1
IBM 3816 Pageprinter
postprocessing troff command output for 2
ibm3812 command 1
ibm3816 command 2
ibm5587G command 4
IBM5587G printer
postprocessing troff command output for 4
ibm558h-T Command 3
icon
box menu
using mwm command 653
putting in icon box
using mwm command 653
window menu
using mwm command 652
Icon Component resource set
using mwm command 658
iconifying window
using mwm command 652
iconv command 5
id command 6
idinstal 42
idlns 200
idprocess 102
ifconfig command 8
IKE database
retrieves, updates, deletes, imports, and exports
information
using ikedb command 21
ikedb command 21
IMAP commands
imap 25
imapd daemon 25
impilt command 26
import and export file format
-bl: and -bE: Flags
using ld command 169
importing filter rules
from export files 26
importvg command 26
imptun command
adding exported tunnel definitions and filter rules 29
inc command 30
indent command 32
index
building inverted for bibliography 36
indxbib command 35
inetd daemon 37
infocmp command
  managing terminfo descriptions 39
infocntr command 42
inode numbers
  information about 113
input extension records
  listing
    using listX11input command 197
input focus
  keyboard
    using mwm command 654
input method keymap file
  compiling a keyboard mapping file into an
    using keycomp command 124
install applications
  Installation Assistant 46
install command 42
install command (BSD) 48
install procedure
  saving files changed during 72
install_all_updates command 44
install_assist command 46
install_mh command 47
installable software product
  verify files
    using lppchk command 248
Installation Assistant
  starting
    using the install_assist command 46
installing
  programs in compatible package
    using instalp command 49
installing a command 42
installms command 60
installp command 49
installp format
  creating software packages 514
instfix command 63
intermediary between user processes
  keyserv daemon
    using keyenvoy command 127
interprocess communication
  removing identifiers 102
  reporting status 103
inucp command 65
inudocm command 66
inurecv command
  description of 68
  performing archive operations for 70
  performing restore operations for 70
inurest command 70
inurid command
  removing installation information 71
inusave command
  description of 72
  performing archive operations for 70
  performing restore operations for 70
  recovering files saved by 68
inutoc command 75
inuumsg command 76
Inventory Scout server daemon
  launching
    using invscoutd command 79
inverted index
  building for bibliography 36
invscoutd command 79
iio command 85
iostat command 97
ip security export files
  import conversion 107
ip security packets list 108
ip security tracing buffer 108
ipcrm command 102
ips command 103
ipreport command 107
ipsec_convert command 107
ipsecstat command 108
ipsectrcbuf command 108
iptrace daemon 109
Isallqdev command 281
isC2host command 111
isCCHost command 112
Isconn command 308
Isdev command 311
Isnparent command
  listing acceptable connection types from
    using Isnparent command 343
istat command 113
J
  job control 114
  jobs command 114
  join command 116
  joining data fields
    from two files
      using join command 116
  joinvg command 119
K
  kdb command 120
  key
    binding
      using mwm command 683
  keyadd command 123
  keyboard mapping file
    into an input method keymap file, compiling
      using keycomp command 124
  keyboard maps
    listing
      using Iskbd command 324
  keycomp command 124
  keydelete command 126
  keyenvoy command 127
  keylist command 128
  keylogin command 130
  keypasswd command 131
  keyserv daemon 132
  intermediary between user processes
    using keyenvoy command 127
logical volume (continued)
  mirroring
    using mirrorvg command 468
login command 212
login name
displaying the current process
  using logname command 217
logins command 214
logname command 217
logoffs
displaying all previous 147
logout command 218
look command 218
lookbib command 219
lookup, command in lb_admin 152
lorder command 220
lp command 221
lpd command 238
lppchk command 248
lppmgr command 250
lpq command 252
lpr command 255
lprm command 260
lpstat command 263
lptest command 269
ls command 271
ls-seclapclntd 275
lsactdef command 276
lsallq command 280
lsarm command 282
lsattr command 283
lsaudrec command 287
lsauthent command
  authentication methods 291
lsC2admin command 292
lsCCadmin command 292
lsconf command 293
lsclass command 295
lscomg command 297
lscondition command 299
lscondresp command 303
lscons command 310
lsdisp command 317
lsfilt command
  listing filter rules 318
lsfont command 319
lsfs command 320
lsgroup command 321
lsitab command 323
lskbd command 324
lslicense command
  fixed and floating licenses
    listing number and status of 324
lsllpp command 325
lslv command 329
lsmaster command 332
lsnode command 333
lsnode command 335
lsnamsv command 337
lsnfsexp command 338
lsnfsmnt command 339
lsnim command 339
lsnlspath command 342
lspath command 345
lsprtsv command 349
lspv command 350
lsvj command 351
lsvj command 354
lsquedev
  command 355
lsquedev command 355
lsresource command
  displaying bus resources 356
lsresponse command 359
lsrole command 364
lsrpdomain command 365
lsrpnode command 368
lsset 371
lsrsrc command 372
lsrsrccdef command 377
lssavevg command 382
lssec
  listing attributes of
    security stanza files 384
lssec command 386
lssec command 392
lstun command
  listing tunnel definitions 394
lsuser command 395
lsvfs command 397
lsjv command 398
lsjvrt command 401
lsjmode command 404
  displaying current video mode 404
lsjmode command 408
lvmstat 411

M
m4 command
  preprocessing files 413
mach command 416
machstat command 417
macref command
  producing cross-reference listing of 418
macro file 418
mail
  incorporating into a folder 30
  receiving
    using mail command 419
    using Mail command 419
  sending 461
    using mail command 419
    using Mail command 419
mail command 419
Mail command 419
Mail commands
  mailq 432
  mailstats 434
mail queue
  printing contents of
    using mailq command 432
mail queue messages
printing list of
using mailq command 432
mail traffic statistics
displaying 434
mailbox directories, setting up 47
mailq command 432
mailstats command 434
mailx command 419
maintaining groups of programs
using make command 435
make command 435
makedbm command 441
makedepend command 442
makedev command 445
man command 447
man pages
displaying information online 447
manefonts command 450
mant command 452
manual
typesetting pages of
using mant command 452
mark command 453
memorandum macro 604
menu pane
window manager
using mwm command 683
mesg command 456
message facility commands
mknod command 536
message queues
removing identifiers 102
message sequences
creating
using mark command 453
displaying
using mark command 453
modifying
using mark command 453
message source file
preprocessing 477
messages
checking for
using msgchk command 639
permitting
using mesg command 456
printing full path names of
using mhnpath command 462
producing formatted lists of
using mhl command 459
refusing
using mesg command 456
mngtsvr command 457
MH
install_mh command 47
MH shell
creating
using msh command 640
mhl command 459
mhmail command 461
mhpah command 462
microcode and firmware levels of system
displaying
using lsmcode command 333
migratelp 464
migratepv command 465
mirrord command
controlling and monitoring
mirror monitors for maintenance 466
mirrorvg command 468
mkboot command 474
mkC2admin command 476
mknod command 477
mkCCadmin command 479
mkcd command 480
mkcssmmt command 484
mkclass command 485
mkclient command 487
mkcomg command 488
mkcondition command 491
mkcondresp command 496
mkdev command 498
mkdir command 500
mkdirhier command 502
mkfif0 command
making FIFO special files 502
mkfilt command
activating or deactivating filter rules 503
mkfont command 504
mkfontdir command 505
mkfs command 506
mkgroup command
description of 509
mkhosts command 511
mkinstallp command 514
mkitem command 512
mkkeyserv command 515
mklost+found command 519
mkiv command 520
mkvcopy command 525
mkmaster command 527
mknetid command 529
mknetid command 530
mkns command 531
mknfs command 532
mknfsxexp command 532
mknfsxmp command 533
mknod command 536
mkpasswd command 538
mpath command 540
mkproto command 542
mkptldap command 545
mkptlsv command 549
mkps command 552
mkqueue command 554
mkqueue command 555
mkramdisk command 557
mrresponse command 559
mkrole command 563
mrzrpdoma in command 565
mrzser 568
mkrsrc command 568

696 Commands Reference, Volume 3
password
reserving a terminal
using lock command 204
performing simply copy operations 65
physical volume
moving partitions from one volume to another volume
using migratepv command 465
plotter queue
listing device names with a queue
using lsallqdev command 281
policy
keyboard input
using mwm command 654
port
stopping all processes on a
using logout command 218
port 515
monitoring for print requests
using lpd command 238
print server
providing on a network
using lpd command 238
print services
configuring TCP/IP-based
configuring 549
printer
enqueuing
using lpr command 255
line
sending requests to a 221
printer queue
adding to the system
using mkque command 554
using mkqueuedev command 555
listing device names with a queue
using lsallqdev command 281
printing a multicast path
from a source to a receiver
using mtrace command 643
process
stopping 133
process groups 114
processes
canceling all 135
stopping all port
using logout command 218
program products
listing optional
using lsllp command 325
programs
creating make file dependencies 442
installing compatible
using instaltp command 49
provided a C preprocessor interface to the make
command 23
programs, groups of
maintaining
using make command 435
regenerating
using make command 435
programs, groups of (continued)
updating
using make command 435
Q
queue
displaying the stanza name for the
using lsallqdev command 355
displaying the stanza name of
using lsque command 354
queues
listing the names of all configured
using lsallq command 280
quit, command in lb_admin 152
R
RAM disk 557
read operations
copying from standard input 187
receiving mail
using mail command 419
records, makes
in /etc/inittab file
using mkkitab command 512
regenerating groups of programs
using make command 435
register, command in lb_admin 152
resource
client-specific
using mwm command 670
resource description file
using mwm command 675
resource set
Client-Specific 671
Component Appearance 657
Frame and Icon Component 658
General Appearance and Behavior 659
roles
creating new
using mkrole command 563
run-time linking
using ld command 169
S
screens
displaying text one screen at a time 614
scripts
elogevent 208
event response resource manager (ERRM)
elogevent 208
logevent 208
logevent 208
sending mail
using mail command 419
service record
creating
making mkboot command 474
| session | initiating a user using login command 212 |
| session management | stopping client using mwm command 650 |
| set_broker, command in lb_admin 152 | set_timeout, command in lb_admin 153 |
| shared objects | using ld command 157 |
| shells | Korn 144 standard 144 |
| slide | typesetting using mvt command 649 |
| smit | turning on/off maintenance of I/O history 98 |
| spool queue | examining using lpq command 252 |
| spooling queue | removing jobs from the line printer 260 |
| SRC | adding a definition to subsystem object class mkssys command examples 586 using mkssys command 584 adding a subserver object 582 mkserver command example 582 adding a subsystem notification method 537 mknotify command examples 537 |
| SRC configuration commands | mknotify 537 mkserver 582 mkssys 584 |
| standard input | reading 187 standard output writing 187 |
| stanza name, device | displaying the using lsquedev command 355 |
| storing public and private keys | 132 |
| subsystem | getting status using lssrc command 392 |
| subservers | adding a definition to SRC object class mkserver command example 582 |
| subsystem | getting status using lssrc command 392 |
| subsystems | adding a definition to SRC object class mkssys command examples 586 using mkssys command 584 adding a notification method 537 mknotify command examples 537 |
| system | adding a device to using mkdev command 498 |
| system images, displaying for examining a dump using kdb command 120 |
| system log | making entries in using the logger command 211 |
| system resource controller | 537, 582, 584 |

**T**

tape device

giving subcommands using mt command 642

TCP/IP

hosts

generating table file 511 setting required values 592 internet packets 109 name service displaying information 337 name services configuring 529 network interface interface, configuring 8 network options displaying, adding, and deleting 608 packet trace report generating 107 print services configuring 549 displaying information about 349 service management 37

TCP/IP commands

ifconfig 8 ipreport 107 lsamsv 337 lsprtsv 349 mkhosts 511 mknamsv 529 mkprtsv 549 mktcpip 592 mmtu 608

TCP/IP daemon

inetd 37 iptrace 109

TCP/IP smit commands

Isamsv 337 lsprtsv 349 mknamsv 529 mkprtsv 549 mktcpip 592

terminal

reserving 204

time management

creating reminders 177 timeout setting in lb_admin 153 tty reporting statistics 97 typesetting document using mmt command 606

Index 699
U
unregister, command in lb_admin 153
updating
  instructions
    getting copies of 66
  procedure
    saving files changed during 72
updating groups of programs
  using make command 435
use_broker
  command in lb_admin 153
user
  initiating a session for a
    using login command 212
user account
  creating
    using mkuser command 594
  customizing
    using mkuser.sys command 597
user database
  creating a version for searches
    using mkpasswd command 538
users
  displaying
    system ID 6
  displaying attributes
    using lsuser command 395
  displaying role attributes
    using lsrole command 364

V
verify files
  installable software product
    using lppchk command 248
view graph
  typesetting
    using mvt command 649
virtual printer
  displaying the attribute values of
    using lsvirprt command 401
  making
    using mkvirprt 601
virtual RAM disk
  making 557
volume group
  creating
    using mkvg command 598
  displaying information about
    using lsvg command 398
  displaying information about a physical volume
    using lspv command 351
  importing new definition 26

W
window
  default components 652
  default menu
    using mwm command 652
  (continued)
    iconifying
      using mwm command 652
    menu pane
      using mwm command 684
window manager
  event specification
    using mwm command 681
window stacking 654
Workload Manager (WLM)
  listing configurations 408
  lswlmconf command 408
X
x defaults 655
X Window System
  window management
    using mwm command 650
XCOFF
  modify 175
Y
ypbind daemon
  starting
    using mkclient command 487
Readers’ Comments — We’d Like to Hear from You

AIX 5L Version 5.2
Commands Reference, Volume 3, i - m

Publication No. SC23-4117-08

Overall, how satisfied are you with the information in this book?

<table>
<thead>
<tr>
<th>Overall satisfaction</th>
<th>Very Satisfied</th>
<th>Satisfied</th>
<th>Neutral</th>
<th>Dissatisfied</th>
<th>Very Dissatisfied</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

How satisfied are you that the information in this book is:

<table>
<thead>
<tr>
<th></th>
<th>Very Satisfied</th>
<th>Satisfied</th>
<th>Neutral</th>
<th>Dissatisfied</th>
<th>Very Dissatisfied</th>
</tr>
</thead>
<tbody>
<tr>
<td>Accurate</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Complete</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Easy to find</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Easy to understand</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Well organized</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Applicable to your tasks</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Please tell us how we can improve this book:

Thank you for your responses. May we contact you?  □ Yes  □ No

When you send comments to IBM, you grant IBM a nonexclusive right to use or distribute your comments in any way it believes appropriate without incurring any obligation to you.

Name

Company or Organization

Phone No.

Address