

OPEN-SOURCE EBOOK

++101 LINUX COMMANDS

BOBBY ILIEV

101 Linux commands Open-source eBook	15
Hacktoberfest	16
About me	17
Ebook PDF Generation Tool	19
Book Cover	20
License	21
The ls command	22
The cd command	24
The cat command	26
The tac command	29
The head command	31
The tail command	33
The pwd command	36
The touch Command	38
The cal Command	41
The bc command	44

The df command	47
The help command	50
Syntax	51
Options	52
Example	53
The factor command	54
Syntax	55
Options	56
Examples	57
The uname command	58
Syntax:	59
Examples	60
Options	61
The mkdir command	62
Syntax	63
Examples	64
Options	65
The gzip command	66
Usage	67
Compress a file	68
Decompress a file	69
Compress multiple files:	70
Decompress multiple files:	71

Compress a directory:	72
Decompress a directory:	73
Verbose (detailed) output while compressing:	74
The whatis command	75
The who command	76
The free command	77
Usage	78
Show memory usage	79
Show memory usage in human-readable form	80
The top/htop command	81
Comparison between top and htop:	82
Examples:	83
Syntax:	85
Additional Flags and their Functionalities:	86
The sl command	87
Installation	88
Syntax	89
The echo command	90
The finger command	92
The groups command	95

The man command	97
The passwd command	98
Example	99
The syntax of the passwd command is :	100
options	101
The w command	102
The whoami command	104
The history command	106
The login Command	107
Syntax	108
Flags and their functionalities	109
Examples	110
lscpu command	111
Options	112
The cp command	113
The mv command	116
The ps command	118
The kill command	120

The killall command	124
The env command	128
Syntax	129
Usage	130
Full List of Options	131
The printenv command	132
The hostname command	134
The nano command	135
The rm command	137
The ifconfig command	139
The ip command	143
The clear command	145
Example	146
Before:	147
After executing clear command:	148
The su command	149
Example :	150
The syntax of the su command is :	151
Options :	152

The wget command	153
Syntax	154
More options	155
The curl command	156
Example :	157
The syntax of the curl command is :	158
Options :	159
Installation:	160
The yes command	161
Options	162
The last command	163
The locate command	164
The iostat command	168
The sudo command	170
Examples	172
The apt command	173
The yum command	176
The zip command	178
The unzip command	180

The shutdown command	182
The dir command	184
The reboot Command	186
Syntax	187
The sort command	189
The paste command	192
The exit command	193
The diff/sdiff command	194
The tar command	196
The gunzip command	199
The hostnamectl command	201
Syntax	202
Example	203
The iptables Command	204
The netstat command	205
The lsof command	207

The bzip2 command	209
The service command	211
The vmstat command	212
The mpstat command	213
The ncdu Command	215
Example	216
Syntax	217
Additional Flags and their Functionalities:	218
The uniq command	219
The RPM command	221
Synopsis	223
Querying and Verifying Packages:	224
Installing, Upgrading, and Removing Packages:	225
Miscellaneous:	226
The scp command	228
The sleep command	230
Options	231
The split command	232

The stat command	235
The useradd command	237
The userdel command	239
The usermod command	241
The ionice command	244
Usage	245
A process can be of three scheduling classes:	246
Options	248
Examples	249
Conclusion	250
The du command	251
The ping command	253
The rsync command	255
Transfer Files from local server to remote	256
Transfer Files remote server to local	258
Transfer only missing files	259
Conclusion	260
The dig command	261
The whois command	266

The ssh command	269
The awk command	276
The crontab command	279
The xargs command	281
The nohup command	284
The pstree command	285
The tree command	287
The whereis command	289
The printf command	291
The cut command	297
The sed command	299
The vim command	302
The chown command	306
The find command	307
The rmdir command	309

The lsblk command	311
Summary	312
Examples	313
Syntax	315
Reading information given by lsblk	316
Reading information of a specific device	317
Useful flags for lsblk	318
Exit Codes	319
The cmatrix command	320
The chmod command	321
The grep command	323
The screen command	325
Restore a Linux Screen	326
Listing all open screen sessions	327
The nc command	328
The make command	331
The basename command	333
The banner command	336
The alias command	337

The which command	339
The date command	341
The mount command	345
The nice/renice command	347
The wc command	348
The tr command	350
The fdisk command	352
The Wait commands	354
Example	355
The zcat command	356
The fold command	357
The quota command	359
The aplay command	360
Syntax:	361
Options:	362
Examples :	363
The spd-say command	364

Syntax:	365
Options:	366
Examples :	368
 The xeyes command	369
 The parted command	370

This is an open-source eBook with 101 Linux commands that everyone should know. No matter if you are a DevOps/SysOps engineer, developer, or just a Linux enthusiast, you will most likely have to use the terminal at some point in your career.

This eBook is made possible thanks to [Hacktoberfest](#) and the open source community!

My name is Bobby Iliev, and I have been working as a Linux DevOps Engineer since 2014. I am an avid Linux lover and supporter of the open-source movement philosophy. I am always doing that which I cannot do in order that I may learn how to do it, and I believe in sharing knowledge.

I think it's essential always to keep professional and surround yourself with good people, work hard, and be nice to everyone. You have to perform at a consistently higher level than others. That's the mark of a true professional.

For more information, please visit my blog at <https://bobbyiliev.com>, follow me on Twitter [@bobbyiliev_](#) and [YouTube](#).

DigitalOcean is a cloud services platform delivering the simplicity developers love and businesses trust to run production applications at scale.

It provides highly available, secure, and scalable compute, storage, and networking solutions that help developers build great software faster.

Founded in 2012 with offices in New York and Cambridge, MA, DigitalOcean offers transparent and affordable pricing, an elegant user interface, and one of the largest libraries of open source resources available.

For more information, please visit <https://www.digitalocean.com> or follow [@digitalocean](#) on Twitter.

If you are new to DigitalOcean, you can get a free \$100 credit and spin up your own servers via this referral link here:

[Free \\$100 Credit For DigitalOcean](#)

The DevDojo is a resource to learn all things web development and web design. Learn on your lunch break or wake up and enjoy a cup of coffee with us to learn something new.

Join this developer community, and we can all learn together, build together, and grow together.

[Join DevDojo](#)

For more information, please visit <https://www.devdojo.com> or follow [@thedeveloper](#) on Twitter.

This ebook was generated by [Ibis](#) developed by [Mohamed Said](#).

Ibis is a PHP tool that helps you write eBooks in markdown.

The cover for this ebook was created by [Suhail Kakar](#).

MIT License

Copyright (c) 2020 Bobby Iliev

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

ls

The **ls** command lets you see the files and directories inside a specific directory (*current working directory by default*). It normally lists the files and directories in ascending alphabetical order.

1. To show the files inside your current working directory:

```
ls
```

2. To show the files and directory inside a specific Directory:

```
ls {Directory_Path}
```

```
ls [-OPTION] [DIRECTORY_PATH]
```

In this interactive tutorial, you will learn the different ways to use the **ls** command:

[The ls command by Tony](#)

Short Flag	Long Flag	Description
<code>-l</code>	<code>-</code>	Show results in long format
<code>-S</code>	<code>-</code>	Sort results by file size
<code>-t</code>	<code>-</code>	Sort results by modification time
<code>-r</code>	<code>--reverse</code>	Show files and directories in reverse order (<i>descending alphabetical order</i>)
<code>-a</code>	<code>--all</code>	Show all files, including hidden files (<i>file names which begin with a period .</i>)
<code>-A</code>	<code>--almost-all</code>	Shows all like <code>-a</code> but without showing <code>.</code> (current working directory) and <code>..</code> (parent directory)
<code>-d</code>	<code>--directory</code>	Instead of listing the files and directories inside the directory, it shows any information about the directory itself, it can be used with <code>-l</code> to show long formatted information
<code>-F</code>	<code>--classify</code>	Appends an indicator character to the end of each listed name, as an example: <code>/</code> character is appended after each directory name listed
<code>-h</code>	<code>--human-readable</code>	like <code>-l</code> but displays file size in human-readable unit not in bytes

Using the alias command it's possible to set persistent options for various commands, including ls. This alias command sets color to auto, lists in long format, and show human-readable file sizes

```
alias ls="ls --color=auto -lh"
```

This alias will be active only on the current session until it ends. For this alias to be active for all new sessions, add the command to your user rc file for example for bash : `~/.bashrc`

cd

The `cd` command is used to change the current working directory (*i.e., in which the current user is working*). The "cd" stands for "**c**hange **d**irectory" and it is one of the most frequently used commands in the Linux terminal.

The `cd` command is often combined with the `ls` command (see chapter 1) when navigating through a system, however, you can also press the `TAB` key two times to list the contents of the new directory you just changed to.

1. Change the current working directory:

```
cd <specified_directory_path>
```

2. Change the current working directory to the home directory:

```
cd ~
```

OR

```
cd
```

3. Change to the previous directory:

```
cd -
```

This will also echo the absolute path of the previous directory.

4. Change the current working directory to the system's root directory:


```
cd /
```

Adding a `..` as a directory will allow you to move "up" from a folder:

```
cd ..
```

This can also be done multiple times! For example, to move up three folders:

```
cd ../../../../
```

```
cd [OPTIONS] directory
```

Short flag	Long flag	Description
------------	-----------	-------------

- | | | |
|-----------------|---|---|
| <code>-L</code> | - | Follow symbolic links. By default, <code>cd</code> behaves as if the <code>-L</code> option is specified. |
| <code>-P</code> | - | Don't follow symbolic links. |

cat

The `cat` command allows us to create single or multiple files, to view the content of a file or to concatenate files and redirect the output to the terminal or files.

The "cat" stands for 'concatenate.' and it's one of the most frequently used commands in the Linux terminal.

1. To display the content of a file in terminal:

```
cat <specified_file_name>
```

2. To display the content of multiple files in terminal:

```
cat file1 file2 ...
```

3. To create a file with the cat command:

```
cat > file_name
```

4. To display all files in current directory with the same filetype:

```
cat *.<filetype>
```

5. To display the content of all the files in current directory:

```
cat *
```

6. To put the output of a given file into another file:

```
cat old_file_name > new_file_name
```

7. Use cat command with **more** and **less** options:

```
cat filename | more  
cat filename | less
```

8. Append the contents of file1 to file2:

```
cat file1 >> file2
```

9. To concatenate two files together in a new file:

```
cat file1_name file2_name merge_file_name
```

10. Some implementations of cat, with option -n, it's possible to show line numbers:

```
cat -n file1_name file2_name > new_numbered_file_name
```

```
cat [OPTION] [FILE]...
```

Short Flag	Long Flag	Description
-A	--show-all	equivalent to -vET
-b	--number-nonblank	number nonempty output lines, overrides -n
-e	-	equivalent to -vE
-T	-	Display tab separated lines in file opened with cat command.
-E	-	To show \$ at the end of each file.
-E	-	Display file with line numbers.
-n	--number	number all output lines
-s	--squeeze-blank	suppress repeated empty output lines
-u	-	(ignored)
-v	--show-nonprinting	use ^ and M- notation, except for LFD and TAB
-	--help	display this help and exit
-	--version	output version information and exit

tac

tac is a Linux command that allows you to view files line-by-line, beginning from the last line. (tac doesn't reverse the contents of each individual line, only the order in which the lines are presented.) It is named by analogy with **cat**.

1. To display the content of a file in terminal:

```
tac <specified_file_name>
```

2. This option attaches the separator before instead of after.

```
tac -b concat_file_name tac_example_file_name
```

3. This option will interpret the separator as a regular expression.

```
tac -r concat_file_name tac_example_file_name
```

4. This option uses STRING as the separator instead of newline.

```
tac -s concat_file_name tac_example_file_name
```

5. This option will display the help text and exit.

```
tac --help
```

6. This option will give the version information and exit.

```
tac --version
```

```
tac [OPTION]... [FILE]...
```

Short Flag	Long Flag	Description
-b	--before	attach the separator before instead of after
-r	--regex	interpret the separator as a regular expression
-s	--separator=STRING	use STRING as the separator instead of newline
-	--help	display this help and exit
-	--version	output version information and exit

head

The **head** command prints the first ten lines of a file.

Example:

```
head filename.txt
```

Syntax:

```
head [OPTION] [FILENAME]
```

Use the **-n** option with a number (should be an integer) of lines to display.

Example:

```
head -n 10 foo.txt
```

This command will display the first ten lines of the file **foo.txt**.

Syntax:

```
head -n <number> foo.txt
```

Short Flag	Long Flag	Description
-c	--bytes=[-]NUM	Print the first NUM bytes of each file; with the leading '-', print all but the last NUM bytes of each file
-n	--lines=[-]NUM	Print the first NUM lines instead of the first 10; with the leading '-', print all but the last NUM lines of each file
-q	--quiet, --silent	Never print headers giving file names
-v	--verbose	Always print headers giving file names
-z	--zero-terminated	Line delimiter is NUL, not newline
	--help	Display this help and exit
	--version	Output version information and exit

tail

The **tail** command prints the last ten lines of a file.

Example:

```
tail filename.txt
```

Syntax:

```
tail [OPTION] [FILENAME]
```

tail

Use the **-n** option with a number(should be an integer) of lines to display.

Example:

```
tail -n 10 foo.txt
```

This command will display the last ten lines of the file **foo.txt**.

It is possible to let tail output any new line added to the file you are looking into. So, if a new line is written to the file, it will immediately be shown in your output. This can be done using the **--follow** or **-f** option. This is especially useful for monitoring log files.

Example:

```
tail -f foo.txt
```

Syntax:

```
tail -n <number> foo.txt
```

Short Flag	Long Flag	Description
-c	--bytes=[+]NUM	Output the last NUM bytes; or use -c +NUM to output starting with byte NUM of each file
-f	--follow[={name descriptor}]	Output appended data as the file grows; an absent option argument means 'descriptor'
-F		Same as --follow=name --retry
-n	--lines=[+]NUM	Output the last NUM lines, instead of the last 10; or use -n +NUM to output starting with line NUM
	--max-unchanged-stats=N	with --follow=name, reopen a FILE which has not changed size after N (default 5) iterations to see if it has been unlinked or rename (this is the usual case of rotated log files); with inotify, this option is rarely useful
	--pid=PID	with -f, terminate after process ID, PID dies
-q	--quiet, --silent	Never output headers giving file names
``	--retry	keep trying to open a file if it is inaccessible

Short Flag	Long Flag	Description
-s	--sleep-interval=N	With -f, sleep for approximately N seconds (default 1.0) between iterations; with inotify and --pid=P, check process P at least once every N seconds
-v	--verbose	Always output headers giving file names
-z	--zero-terminated	Line delimiter is NUL, not newline
	--help	Display this help and exit
	--version	Output version information and exit

pwd

The **pwd** stands for Print Working Directory. It prints the path of the current working directory, starting from the root.

Example:

```
pwd
```

The output would be your current directory:

```
/home/your_user/some_directory
```

Syntax:

```
pwd [OPTION]
```

Tip: You can also check this by printing out the **\$PWD** variable:

```
echo $PWD
```

The output would be the same as of the **pwd** command.

Short Flag	Long Flag	Description
-L	--logical	If the environment variable \$PWD contains an absolute name of the current directory with no "." or ".." components, then output those contents, even if they contain symbolic links. Otherwise, fall back to default (-P) behavior.

Short Flag	Long Flag	Description
-P	--physical	Print a fully resolved name for the current directory, where all components of the name are actual directory names, and not symbolic links.
	--help	Display a help message, and exit.
	--version	Display version information, and exit.

By default, `pwd` behaves as if `-L` were specified.

touch

The **touch** command modifies a file's timestamps. If the file specified doesn't exist, an empty file with that name is created.

```
touch [OPTION]... FILE...
```

Short Flag	Long Flag	Description
------------	-----------	-------------

-

-c	--no-create	Do not create any files.
-d	STRING	Parse <i>STRING</i> and use it instead of the current time.

-

-h	--no-dereference	Affect each symbolic link instead of any referenced file (useful only on systems that can change the timestamps of a symlink). This option implies -c , nothing is created if the file does not exist.
----	------------------	---

-

-r	FILE	Use this file's times instead of the current time.
-t	STAMP	

-

		Use the numeric time <i>STAMP</i> instead of the current time. The format of <i>STAMP</i> is [[CC]YY]MMDDhhmm[.ss] .
--	--	---

-

--time	WORD	An alternate way to specify which type of time is set (e.g. <i>access</i> , <i>modification</i> , or <i>change</i>). This is equivalent to specifying -a or -m .
--------	------	---

- WORD is **access**, **atime**, or **use**: equivalent to **-a**.
- WORD is **modify** or **mtime**: equivalent to **-m**.

An alternate way to specify what type of time to set (as with **-a** and **-m**).

-
|--help|Display a help message, and exit.| |
-
|--version|Display version information, and exit.|

1. If **file.txt** exists, set all of its timestamps to the current system time. If **file.txt** doesn't exist, create an empty file with that name.

```
touch file.txt
```

2. If **file.txt** exists, set its times to the current system time. If it does not exist, do nothing.

```
touch -c file.txt
```

3. Change the *access* time of **file.txt**. The *modification* time is not changed. The *change* time is set to the current system time. If **file.txt** does not exist, it is created.

```
touch -a file.txt
```

4. Change the times of file **symboliclink**. If it's a symbolic link, change the times of the symlink, **NOT** the times of the referenced file.

```
touch -h symboliclink
```

5. Change the *access* and *modification* times of **file-b.txt** to match the times of **file-a.txt**. The *change* time will be set to the current system time. If **file-b.txt** does not exist, it is not created. Note, **file-a.txt** must already exist in this context.

```
touch -cr file-a.txt file-b.txt
```

6. Set the *access* time and *modification* time of **file.txt** to **February 1st** of the

current year. The *change* time is set to the current system time.

```
touch -d "1 Feb" file.txt
```


cal

The **cal** command displays a formatted calendar in the terminal. If no options are specified, cal displays the current month, with the current day highlighted.

```
cal [general options] [-jy] [[month] year]
```

Option	Description
-h	Don't highlight today's date.
-m month	Specify a month to display. The month specifier is a full month name (e.g., February), a month abbreviation of at least three letters (e.g., Feb), or a number (e.g., 2). If you specify a number, followed by the letter "f" or "p", the month of the following or previous year, respectively, display. For instance, -m 2f displays February of next year.
-y year	Specify a year to display. For example, -y 1970 displays the entire calendar of the year 1970.
-3	Display last month, this month, and next month.
-1	Display only this month. This is the default.
-A num	Display num months occurring after any months already specified. For example, -3 -A 3 displays last month, this month, and four months after this one; and -y 1970 -A 2 displays every month in 1970, and the first two months of 1971.
-B num	Display num months occurring before any months already specified. For example, -3 -B 2 displays the previous three months, this month, and next month.
-d YYYY-MM	Operate as if the current month is number MM of year YYYY.

1. Display the calendar for this month, with today highlighted.

```
cal
```

2. Same as the previous command, but do not highlight today.

```
cal -h
```

3. Display last month, this month, and next month.

```
cal -3
```

4. Display this entire year's calendar.

```
cal -y
```

5. Display the entire year 2000 calendar.

```
cal -y 2000
```

6. Same as the previous command.

```
cal 2000
```

7. Display the calendar for December of this year.

```
cal -m [December, Dec, or 12]
```

10. Display the calendar for December 2000.



cal 12 2000

bc

The **bc** command provides the functionality of being able to perform mathematical calculations through the command line.

1 . Arithmetic:

```
Input : $ echo "11+5" | bc
Output : 16
```

2 . Increment:

- **var -++** : Post increment operator, the result of the variable is used first and then the variable is incremented.
- **- ++var** : Pre increment operator, the variable is increased first and then the result of the variable is stored.

```
Input: $ echo "var=3;++var" | bc
Output: 4
```

3 . Decrement:

- **var --** : Post decrement operator, the result of the variable is used first and then the variable is decremented.
- **-- var** : Pre decrement operator, the variable is decreased first and then the result of the variable is stored.

```
Input: $ echo "var=3;--var" | bc
Output: 2
```

4 . Assignment:

- `var = value` : Assign the value to the variable
- `var += value` : similar to `var = var + value`
- `var -= value` : similar to `var = var - value`
- `var *= value` : similar to `var = var * value`
- `var /= value` : similar to `var = var / value`
- `var ^= value` : similar to `var = var ^ value`
- `var %= value` : similar to `var = var % value`

```
Input: $ echo "var=4;var" | bc
Output: 4
```

5 . Comparison or Relational:

- If the comparison is true, then the result is 1. Otherwise,(false), returns 0
- `expr1<expr2` : Result is 1, if `expr1` is strictly less than `expr2`.
- `expr1<=expr2` : Result is 1, if `expr1` is less than or equal to `expr2`.
- `expr1>expr2` : Result is 1, if `expr1` is strictly greater than `expr2`.
- `expr1>=expr2` : Result is 1, if `expr1` is greater than or equal to `expr2`.
- `expr1==expr2` : Result is 1, if `expr1` is equal to `expr2`.
- `expr1!=expr2` : Result is 1, if `expr1` is not equal to `expr2`.

```
Input: $ echo "6<4" | bc
Output: 0
```

```
Input: $ echo "2==2" | bc
Output: 1
```

6 . Logical or Boolean:

- `expr1 && expr2` : Result is 1, if both expressions are non-zero.
- `expr1 || expr2` : Result is 1, if either expression is non-zero.
- `! expr` : Result is 1, if `expr` is 0.

```
Input: $ echo "! 1" | bc
Output: 0
```

```
Input: $ echo "10 && 5" | bc
Output: 1
```

```
bc [ -hlwsqv ] [long-options] [ file ... ]
```

Note: This does not include an exhaustive list of options.

Short Flag	Long Flag	Description
-i	--interactive	Force interactive mode
-l	--mathlib	Use the predefined math routines
-q	--quiet	Opens the interactive mode for bc without printing the header
-s	--standard	Treat non-standard bc constructs as errors
-w	--warn	Provides a warning if non-standard bc constructs are used

1. The capabilities of **bc** can be further appreciated if used within a script. Aside from basic arithmetic operations, **bc** supports increments/decrements, complex calculations, logical comparisons, etc.
2. Two of the flags in **bc** refer to non-standard constructs. If you evaluate **100>50 | bc** for example, you will get a strange warning. According to the POSIX page for bc, relational operators are only valid if used within an **if**, **while**, or **for** statement.

df

The **df** command in Linux/Unix is used to show the disk usage & information. **df** is an abbreviation for "disk free".

df displays the amount of disk space available on the file system containing each file name argument. If no file name is given, the space available on all currently mounted file systems is shown.

```
df [OPTION]... [FILE]...
```

Short Flag	Long Flag	Description
-a	--all	Include pseudo, duplicate, inaccessible file systems.
-B	--block-size=SIZE	Scale sizes by SIZE before printing them; e.g., -BM prints sizes in units of 1,048,576 bytes; see SIZE format below.
-h	--human-readable	Print sizes in powers of 1024 (e.g., 1023M).
-H	--si	Print sizes in powers of 1000 (e.g., 1.1G).
-i	--inodes	List inode information instead of block usage.
-k	-	Like --block-size=1K .
-l	--local	Limit listing to local file systems.
-	--no-sync	Do not invoke sync before getting usage info (default).
-	--output[=FIELD_LIST]	Use the output format defined by FIELD_LIST , or print all fields if FIELD_LIST is omitted.
-P	--portability	Use the POSIX output format
-	--sync	Invoke sync before getting usage info.

Short Flag	Long Flag	Description
-	<code>--total</code>	Elide all entries insignificant to available space, and produce a grand total.
<code>-t</code>	<code>--type=TYPE</code>	Limit listing to file systems of type <code>TYPE</code> .
<code>-T</code>	<code>--print-type</code>	Print file system type.
<code>-x</code>	<code>--exclude-type=TYPE</code>	Limit listing to file systems not of type <code>TYPE</code> .
<code>-v</code>	-	Ignored; included for compatibility reasons.
-	<code>--help</code>	Display help message and exit.
-	<code>--version</code>	Output version information and exit.

1. Show available disk space **Action:** --- Output the available disk space and where the directory is mounted

Details: --- Outputted values are not human-readable (are in bytes)

Command:

```
df
```

2. Show available disk space in human-readable form **Action:** --- Output the available disk space and where the directory is mounted

Details: --- Outputted values ARE human-readable (are in GBs/MBs)

Command:

```
df -h
```

3. Show available disk space for the specific file system **Action:** --- Output the available disk space and where the directory is mounted

Details: --- Outputted values are only for the selected file system

Command:


```
df -hT file_system_name
```

4. Show available inodes **Action:** --- Output the available inodes for all file systems

Details: --- Outputted values are for inodes and not available space

Command:

```
df -i
```

5. Show file system type **Action:** --- Output the file system types

Details: --- Outputted values are for all file systems

Command:

```
df -T
```

6. Exclude file system type from the output **Action:** --- Output the information while excluding the chosen file system type

Details: --- Outputted values are for all file systems EXCEPT the chosen file system type

Command:

```
df -x file_system_type
```

help

The **help** command displays information about builtin commands. Display information about builtin commands.

If a **PATTERN** is specified, this command gives detailed help on all commands matching the **PATTERN**, otherwise the list of available help topics is printed.



```
$ help [-dms] [PATTERN ...]
```

Option Description

- d Output short description for each topic.
- m Display usage in pseudo-manpage format.
- s Output only a short usage synopsis for each topic matching the provided **PATTERN**.



```
$ help ls
```

factor

The **factor** command prints the prime factors of each specified integer **NUMBER**. If none are specified on the command line, it will read them from the standard input.



```
$ factor [NUMBER]...
```

OR:



```
$ factor OPTION
```

Option	Description
<code>--help</code>	Display this a help message and exit.
<code>--version</code>	Output version information and exit.

1. Print prime factors of a prime number.



```
$ factor 50
```

2. Print prime factors of a non-prime number.



```
$ factor 75
```

uname

The **uname** command lets you print out system information and defaults to outputting the kernel name.



\$ uname [OPTION]

1. Print out all system information.



```
$ uname -a
```

2. Print out the kernel version.



```
$ uname -v
```

Short Flag	Long Flag	Description
-a	--all	Print all information, except omit processor and hardware platform if unknown.
-s	--kernel-name	Print the kernel name.
-n	--nodename	Print the network node hostname.
-r	--kernel-release	Print the kernel release.
-v	--kernel-version	Print the kernel version.
-m	--machine	Print the machine hardware name.
-p	--processor	Print the processor type (non-portable).
-i	--hardware-platform	Print the hardware platform (non-portable).
-o	--operating-system	Print the operating system.

`mkdir`

The `mkdir` command in Linux/Unix is used to create a directory.



```
$ mkdir [-m=mode] [-p] [-v] [-Z=context] directory [directory  
...]
```

1. Make a directory named **myfiles**.

```
$ mkdir myfiles
```

2. Create a directory named **myfiles** at the home directory:

```
$ mkdir ~/myfiles
```

3. Create the **mydir** directory, and set its file mode (**-m**) so that all users (**a**) may read (**r**), write (**w**), and execute (**x**) it.

```
$ mkdir -m a=rwx mydir
```

You can also create sub-directories of a directory. It will create the parent directory first, if it doesn't exist. If it already exists, then it move further to create the sub-directories without any error message.

For directories, this means that any user on the system may view ("read"), and create/modify/delete ("write") files in the directory. Any user may also change to ("execute") the directory, for example with the **cd** command.

4. Create the directory **/home/test/src/python**. If any of the parent directories **/home**, **/home/test**, or **/home/test/src** do not already exist, they are automatically created.

```
$ mkdir -p /home/test/src/python
```


Short Flags	Long Flags	Descriptions
-m	--mode=MODE	Set file mode (as in chmod), not a=rwx - umask .
-p	--parents	No error if existing, make parent directories as needed.
-v	--verbose	Print a message for each created directory.
-Z	--context=CTX	Set the SELinux security context of each created directory to CTX.
-	--help	Display a help message and exit.
-	--version	Output version information and exit.

gzip

The `gzip` command in Linux/Unix is used to compress/decompress data.

Action: --- Compressing a file

Details: --- Reduce the size of the file by applying compression

Command:

```
gzip file_name
```

Action: --- Decompressing a file

Details: --- Restore the file's original form in terms of data and size

Command:

```
gzip -d archive_01.gz
```

Action: --- Compress multiple files

Details: --- Compress multiple files into multiple archives

Command:

```
gzip file_name_01 file_name_02 file_name_03
```

Action: --- Decompress multiple files

Details: --- Decompress multiple files from multiple archives

Command:

```
gzip -d archive_01.gz archive_02.gz archive_03.gz
```

Action: --- Compress all the files in a directory

Details: --- Compress multiple files under a directory in one single archive

Command:

```
gzip -r directory_name
```


Action: --- Decompress all the files in a directory

Details: --- Decompress multiple files under a directory from one single archive

Command:

```
gzip -dr directory_name
```

Action: --- Compress a file in a more verbose manner

Details: --- Output more information about the action of the command

Command:

```
gzip -v file_name
```

whatis

The **whatis** command is used to display one-line manual page descriptions for commands. It can be used to get a basic understanding of what a (unknown) command is used for.

1. To display what **ls** is used for:

```
whatis ls
```

2. To display the use of all commands which start with **make**, execute the following:

```
whatis -w make*
```

```
whatis [-OPTION] [KEYWORD]
```

Short Flag	Long Flag	Description
-d	--debug	Show debugging messages
-r	--regex	Interpret each keyword as a regex
-w	--wildcard	The keyword(s) contain wildcards

who

The **who** command lets you print out a list of logged-in users, the current run level of the system and the time of last system boot.

1. Print out all details of currently logged-in users

```
who -a
```

2. Print out the list of all dead processes

```
who -d -H
```

```
who [options] [filename]
```

Short Flag	Description
-r	prints all the current runlevel
-d	print all the dead processes
-q	print all the login names and total number of logged on users
-h	print the heading of the columns displayed
-b	print the time of last system boot

018-the-free-command.md

free

The **free** command in Linux/Unix is used to show memory (RAM/SWAP) information.

Action: --- Output the memory usage - available and used, as well as swap

Details: --- Outputted values are not human-readable (are in bytes)

Command:

 `free`

Action: --- Output the memory usage - available and used, as well as swap

Details: --- Outputted values ARE human-readable (are in GB / MB)

Command:



```
free -h
```


top/htop

top is the default command-line utility that comes pre-installed on Linux distributions and Unix-like operating systems. It is used for displaying information about the system and its top CPU-consuming processes as well as RAM usage.

htop is interactive process-viewer and process-manager for Linux and Unix-like operating system based on ncurses. If you take top and put it on steroids, you get htop.

Feature	top	htop
Type	Interactive system-monitor, process-viewer and process-manager	Interactive system-monitor, process-viewer and process-manager
Operating System	Linux distributions, macOS	Linux distributions, macOS
Installation	Built-in and is always there. Also has more adoption due to this fact.	Doesn't come preinstalled on most Linux distros. Manual installation is needed
User Interface	Basic text only	Colorful and nicer text-graphics interface
Scrolling Support	No	Yes, supports horizontal and vertical scrolling
Mouse Support	No	Yes
Process utilization	Displays processes but not in tree format	Yes, including user and kernel threads
Scrolling Support	No	Yes, supports horizontal and vertical scrolling
Mouse Support	No	Yes
Process utilization	Displays processes but not in tree format	Yes, including user and kernel threads
Network Utilization	No	No
Disk Utilization	No	No
Comments	Has a learning curve for some advanced options like searching, sending messages to processes, etc. It is good to have some knowledge of top because it is the default process viewer on many systems.	Easier to use and supports vi like searching with <code>/</code> . Sending messages to processes (kill, renice) is easier and doesn't require typing in the process number like top.

top

1. To display dynamic real-time information about running processes:

```
top
```

2. Sorting processes by internal memory size (default order - process ID):

```
top -o mem
```

3. Sorting processes first by CPU, then by running time:

```
top -o cpu -O time
```

4. Display only processes owned by given user:

```
top -user {user_name}
```

htop

1. Display dynamic real-time information about running processes. An enhanced version of **top**.

```
htop
```

2. displaying processes owned by a specific user:

```
htop --user {user_name}
```

3. Sort processes by a specified **sort_item** (use **htop --sort help** for

available options):



```
htop --sort {sort_item}
```



top [OPTIONS]



htop [OPTIONS]

Short Flag	Long Flag	Description
-a	-	Sort by memory usage.
-b	-	Batch mode operation. Starts top in 'Batch mode', which could be useful for sending output from top to other programs or to a file. In this mode, top will not accept input and runs until the iterations limit you've set with the '-n' command-line option or until killed.
-h	-	<code>top --user {user_name}</code> Only display processes owned by user.
-U	-user	Help.
-u	-	This is an alias equivalent to: -o cpu -O time.

sl

The **sl** command in Linux is a humorous program that runs a steam locomotive(sl) across your terminal.



Install the package before running.



```
sudo apt install sl
```


| sl

echo

The `echo` command lets you display the line of text/string that is passed as an argument

1. To Show the line of text or string passed as an argument:

```
echo Hello There
```

2. To show all files/folders similar to the `ls` command:

```
echo *
```

3. To save text to a file named `foo.bar`:

```
echo "Hello There" > foo.bar
```

4. To append text to a file named `foo.bar`:

```
echo "Hello There" >> foo.bar
```

```
echo [option] [string]
```

It is usually used in shell scripts and batch files to output status text to the screen or a file. The `-e` used with it enables the interpretation of backslash escapes

Option Description

<code>\b</code>	removes all the spaces in between the text
<code>\c</code>	suppress trailing new line with backspace interpreter '-e' to continue without emitting new line.
<code>\n</code>	creates new line from where it is used
<code>\t</code>	creates horizontal tab spaces
<code>\r</code>	carriage returns with backspace interpreter '-e' to have specified carriage return in output
<code>\v</code>	creates vertical tab spaces
<code>\a</code>	alert returns with a backspace interpreter '-e' to have sound alert
<code>-n</code>	omits echoing trailing newline .

finger

The **finger** displays information about the system users.

1. View detail about a particular user.

```
finger abc
```

Output

```
Login: abc                               Name: (null)
Directory: /home/abc                     Shell: /bin/bash
On since Mon Nov  1 18:45 (IST) on :0 (messages off)
On since Mon Nov  1 18:46 (IST) on pts/0 from :0.0
New mail received Fri May  7 10:33 2013 (IST)
Unread since Sat Jun  7 12:59 2003 (IST)
No Plan.
```

2. View login details and Idle status about an user

```
finger -s root
```

Output

Login	Name		Tty	Idle	Login Time
Office	Office	Phone			
root	root		*1	19d Wed 17:45	
root	root		*2	3d Fri 16:53	
root	root		*3	Mon 20:20	
root	root		*ta	2 Tue 15:43	
root	root		*tb	2 Tue 15:44	

```
finger [-l] [-m] [-p] [-s] [username]
```

Flag Description

- l Force long output format.
- m Match arguments only on user name (not first or last name).
- p Suppress printing of the .plan file in a long format printout.
- s Force short output format.

Default Format

The default format includes the following items:

Login name

Full username

Terminal name

Write status (an * (asterisk) before the terminal name indicates that write permission is denied)

For each user on the host, the default information list also includes, if known, the following items:

Idle time (Idle time is minutes if it is a single integer, hours and minutes if a : (colon) is present, or days and hours if a "d" is present.)

Login time

Site-specific information

Longer Format

A longer format is used by the `finger` command whenever a list of user's names is given. (Account names as well as first and last names of users are accepted.) This format is multiline, and includes all the information described above along with the following:

User's \$HOME directory

User's login shell

Contents of the `.plan` file in the user's \$HOME directory

Contents of the `.project` file in the user's \$HOME directory

groups

In Linux, there can be multiple users (those who use/operate the system), and groups (a collection of users). Groups make it easy to manage users with the same security and access privileges. A user can be part of different groups.

Important Points:

The **groups** command prints the names of the primary and any supplementary groups for each given username, or the current process if no names are given. If more than one name is given, the name of each user is printed before the list of that user's groups and the username is separated from the group list by a colon.

```
groups [username]
```

Example 1

Provided with a username

```
groups demon
```

In this example, username demon is passed with groups command and the output shows the groups in which the user demon is present, separated by a colon.

Example 2

When no username is passed then this will display the group membership for the current user:

```
groups
```

Here the current user is demon . So when we run the **groups** command without

arguments we get the groups in which demon is a user.

Example 3

Passing root with groups command:

A vertical orange bar is positioned to the left of the terminal output. The text "\$demon# groups" is displayed in a light blue monospaced font.

```
$demon# groups
```

Note: Primary and supplementary groups for a process are normally inherited from its parent and are usually unchanged since login. This means that if you change the group database after logging in, groups will not reflect your changes within your existing login session. The only options are -help and -version.

man

The **man** command is used to display the manual of any command that we can run on the terminal. It provides information like: DESCRIPTION, OPTIONS, AUTHORS and more.

1. Man page for printf:

```
man printf
```

2. Man page section 2 for intro:

```
man 2 intro
```

```
man [SECTION-NUM] [COMMAND NAME]
```

Short Flag Long Flag Description

-f	-	Return the sections of an command
-a	-	Display all the manual pages of an command
-k	-	Searches the given command with RegEx in all man pages
-w	-	Returns the location of a given command man page
-I	-	Searches the command manual case sensitive

passwd

In Linux, `passwd` command changes the password of user accounts. A normal user may only change the password for their own account, but a superuser may change the password for any account. `passwd` also changes the account or associated password validity period.



\$ passwd

passwd

```
$ passwd [options] [LOGIN]
```

-a, --all
This option can be used only with -S and causes show status for all users.

-d, --delete
Delete a user's password.

-e, --expire
Immediately expire an account's password.

-h, --help
Display help message and exit.

-i, --inactive
This option is used to disable an account after the password has been expired for a number of days.

-k, --keep-tokens
Indicate password change should be performed only for expired authentication tokens (passwords).

-l, --lock
Lock the password of the named account.

-q, --quiet
Quiet mode.

-r, --repository
change password in repository.

-S, --status
Display account status information.

W

The **W** command displays information about the users that are currently active on the machine and their [processes](#).

1. Running the **W** command without [arguments](#) shows a list of logged on users and their processes.

```
W
```

2. Show information for the user named *hope*.

```
W hope
```

```
finger [-l] [-m] [-p] [-s] [username]
```

Short Flag	Long Flag	Description
-h	--no-header	Don't print the header.
-u	--no-current	Ignores the username while figuring out the current process and cpu times. <i>(To see an example of this, switch to the root user with SU and then run both W and W -U.)</i>
-s	--short	Display abbreviated output <i>(don't print the login time, JCPU or PCPU times)</i> .

Short Flag	Long Flag	Description
<code>-f</code>	<code>--from</code>	Toggle printing the from (<i>remote hostname</i>) field. The default as released is for the from field to not be printed, although your system administrator or distribution maintainer may have compiled a version where the from field is shown by default.
<code>--help</code>	-	Display a help message, and exit.
<code>-V</code>	<code>--version</code>	Display version information, and exit.
<code>-o</code>	<code>--old-style</code>	Old style output (<i>prints blank space for idle times less than one minute</i>).
<code>user</code>	-	Show information about the specified the user only.

The [header](#) of the output shows (in this order): the current time, how long the system has been running, how many users are currently logged on, and the system [load](#) averages for the past 1, 5, and 15 minutes.

The following entries are displayed for each user:

- login name the [tty](#)
- name the [remote](#)
- [host](#) they are
- logged in from the amount of time they are logged in their
- [idle](#) time JCPU
- PCPU
- [command line](#) of their current process

The JCPU time is the time used by all processes attached to the tty. It does not include past background jobs, but does include currently running background jobs.

The PCPU time is the time used by the current process, named in the "what" field.

whoami

The **whoami** command displays the username of the current effective user. In other words it just prints the username of the currently logged-in user when executed.

To display your effective user id just type **whoami** in your terminal:

```
manish@godsmack:~$ whoami
# Output:
manish
```

Syntax:

```
whoami [-OPTION]
```

There are only two options which can be passed to it :

-help: Used to display the help and exit

Example:

```
whoami --help
```

Output:

Usage: whoami [OPTION]...

Print the user name associated with the current effective user ID.

Same as `id -un`.

`--help` display this help and exit

`--version` output version information and exit

--version: Output version information and exit

Example:

```
whoami --version
```

Output:

```
whoami (GNU coreutils) 8.32
```

```
Copyright (C) 2020 Free Software Foundation, Inc.
```

```
License GPLv3+: GNU GPL version 3 or later
```

```
<https://gnu.org/licenses/gpl.html>.
```

```
This is free software: you are free to change and redistribute  
it.
```

```
There is NO WARRANTY, to the extent permitted by law.
```

```
Written by Richard Mlynarik.
```

history

If you type `history` you will get a list of the last 500 commands used. This gives you the possibility to copy and paste commands that you executed in the past.

This is powerful in combination with `grep`. So you can search for a command in your command history.

1. If you want to search in your history for artisan commands you ran in the past.

```
history | grep artisan
```

2. If you only want to show the last 10 commands you can.

```
history 10
```

login

The `login` command initiates a user session.



```
$ login [-p] [-h host] [-H] [-f username|username]
```

Short Flag	Description
-f	Used to skip a login authentication. This option is usually used by the getty(8) autologin feature.
-h	Used by other servers (such as telnetd(8)) to pass the name of the remote host to login so that it can be placed in utmp and wtmp. Only the superuser is allowed use this option.
-p	Used by getty(8) to tell login to preserve the environment.
-H	Used by other servers (for example, telnetd(8)) to tell login that printing the hostname should be suppressed in the login: prompt.
--help	Display help text and exit.
-v	Display version information and exit.

To log in to the system as user abhishek, enter the following at the login prompt:



```
$ login: abhishek
```

If a password is defined, the password prompt appears. Enter your password at this prompt.

lscpu

lscpu in Linux/Unix is used to display CPU Architecture info. **lscpu** gathers CPU architecture information from **sysfs** and **/proc/cpuinfo** files.

For example :

```
manish@godsmack:~$ lscpu
Architecture:          x86_64
CPU op-mode(s):        32-bit, 64-bit
Byte Order:            Little Endian
CPU(s):                 4
On-line CPU(s) list:   0-3
Thread(s) per core:    2
Core(s) per socket:    2
Socket(s):              1
NUMA node(s):          1
Vendor ID:              GenuineIntel
CPU family:             6
Model:                  142
Model name:             Intel(R) Core(TM) i5-7200U CPU @
2.50GHz
Stepping:               9
CPU MHz:                700.024
CPU max MHz:            3100.0000
CPU min MHz:            400.0000
BogoMIPS:               5399.81
Virtualization:         VT-x
L1d cache:              32K
L1i cache:              32K
L2 cache:               256K
L3 cache:               3072K
NUMA node0 CPU(s):     0-3
```

-a, --all Include lines for online and offline CPUs in the output (default for -e). This option may only be specified together with option -e or -p. For example: **lsof -a**

-b, --online Limit the output to online CPUs (default for -p). This option may only be specified together with option -e or -p. For example: **lscpu -b**

-c, --offline Limit the output to offline CPUs. This option may only be specified together with option -e or -p.

-e, --extended [=list] Display the CPU information in human readable format. For example: **lsof -e**

For more info: use **man lscpu** or **lscpu --help**

cp

The **cp** is a command-line utility for copying files and directory. **cp** stands for copy. This command is used to copy files or group of files or directory. It creates an exact image of a file on a disk with different file name. The cp command requires at least two filenames in its arguments.

1. To copy the contents of the source file to the destination file.

```
cp sourceFile destFile
```

If the destination file doesn't exist then the file is created and the content is copied to it. If it exists then the file is overwritten.

2. To copy a file to another directory specify the absolute or the relative path to the destination directory.

```
cp sourceFile /folderName/destFile
```

3. To copy a directory, including all its files and subdirectories

```
cp -R folderName1 folderName2
```

The command above creates the destination directory and recursively copies all files and subdirectories from the source to the destination directory.

If the destination directory already exists, the source directory itself and its content are copied inside the destination directory.

4. To copy only the files and subdirectories but not the source directory

```
cp -RT folderName1 folderName2
```

The general syntax for the cp command is as follows:

```
cp [OPTION] SOURCE DESTINATION
cp [OPTION] SOURCE DIRECTORY
cp [OPTION] SOURCE-1 SOURCE-2 SOURCE-3 SOURCE-n DIRECTORY
```

The first and second syntax is used to copy Source file to Destination file or Directory. The third syntax is used to copy multiple Sources(files) to Directory.

Some useful options

1. **-i** (interactive) **i** stands for Interactive copying. With this option system first warns the user before overwriting the destination file. cp prompts for a response, if you press y then it overwrites the file and with any other option leave it uncopied.

```
$ cp -i file1.txt fileName2.txt
cp: overwrite 'file2.txt'? y
```

2. **-b**(backup) **-b**(backup): With this option cp command creates the backup of the destination file in the same folder with the different name and in different format.

```
$ ls
a.txt b.txt

$ cp -b a.txt b.txt

$ ls
a.txt b.txt b.txt~
```

3. **-f**(force) If the system is unable to open destination file for writing operation because the user doesn't have writing permission for this file then by using -f

option with cp command, destination file is deleted first and then copying of content is done from source to destination file.

```
$ ls -l b.txt
-r-xr-xr-x+ 1 User User 3 Nov 24 08:45 b.txt
```

User, group and others doesn't have writing permission.

Without -f option, command not executed

```
$ cp a.txt b.txt
cp: cannot create regular file 'b.txt': Permission denied
```

With -f option, command executed successfully

```
$ cp -f a.txt b.txt
```

Short Flag	Long Flag	Description
-i	--interactive	prompt before overwrite
-f	--force	If an existing destination file cannot be opened, remove it and try again
-b	-	Creates the backup of the destination file in the same folder with the different name and in different format.
-r or -R	--recursive	cp command shows its recursive behavior by copying the entire directory structure recursively.
-n	--no-clobber	do not overwrite an existing file (overrides a previous -i option)
-p	-	preserve the specified attributes (default: mode,ownership,timestamps), if possible additional attributes: context, links, xattr, all

mv

The **mv** command lets you **move one or more files or directories** from one place to another in a file system like UNIX. It can be used for two distinct functions:

- To rename a file or folder.
- To move a group of files to a different directory.

Note: *No additional space is consumed on a disk during renaming, and the mv command doesn't provide a prompt for confirmation*

```
mv [options] source (file or directory) destination
```

1. To rename a file called `old_name.txt`:

```
mv old_name.txt new_name.txt
```

2. To move a file called `essay.txt` from the current directory to a directory called `assignments` and rename it `essay1.txt`:

```
mv essay.txt assignments/essay1.txt
```

3. To move a file called `essay.txt` from the current directory to a directory called `assignments` without renaming it

```
mv essay.txt assignments
```

Short Flag	Long Flag	Description
-f	--force	Force move by overwriting destination file without prompt
-i	--interactive	Interactive prompt before overwrite
-u	--update	Move only when the source file is newer than the destination file or when the destination file is missing
-n	--no-clobber	Do not overwrite an existing file
-v	--verbose	Print source and destination files
-b	--backup	Create a Backup of Existing Destination File

ps

The **ps** command is used to identify programs and processes that are running on the system and the resources they are using. Its frequently **pipelined** with other commands like **grep** to search for a program/process or **less** so that the user can analyze the output one page at a time.

Let's say you have a program like openshot which is notorious for hogging system resources when exporting a video, and you want to close it, but the GUI has become unresponsive.

1. You want to find the PID of openshot and kill it.

```
ps aux | grep openshot  
kill - <openshot PID>
```

2. To Show all the running processes:

```
ps -A
```

ps [options]

When run without any options, it's useless and will print: **CMD** - the executable processes/(program) running, their **PID** - process ID, **TTY** - terminal type and **Time** - How long the process has utilized the CPU or thread.

If you are going to remember only one thing from this page let it be these three letter

aux: **a** - which displays all processes running, including those being run by other users. **u** - which shows the effective user of a process, i.e. the person whose file access permissions are used by the process. **x** - which shows processes that do not have a **TTY** associated with them.

Option	Description
a	Shows list all processes with a terminal (tty)
-A	Lists all processes. Identical to -e
-a	Shows all processes except both session leaders and processes not associated with a terminal
-d	Select all processes except session leaders
--deselect	Shows all processes except those that fulfill the specified conditions. Identical to -N
-e	Lists all processes. Identical to -A
-N	Shows all processes except those that fulfill the specified conditions. Identical to --deselect
T	Select all processes associated with this terminal. Identical to the -t option without any argument
r	Restrict the selection to only running processes
--help simple	Shows all the basic options
--help all	Shows every available options

Another useful command which give a realtime snapshot of the processes and the resources they are using about every ten seconds is **top**.

kill

`kill` command in Linux (located in `/bin/kill`), is a built-in command which is used to terminate processes manually. The `kill` command sends a signal to a process which terminates the process. If the user doesn't specify any signal which is to be sent along with `kill` command then default `TERM` signal is sent that terminates the process.

Signals can be specified in three ways:

- **By number (e.g. -5)**
- **With SIG prefix (e.g. -SIGkill)**
- **Without SIG prefix (e.g. -kill)**

```
kill [OPTIONS] [PID]...
```

1. To display all the available signals you can use below command option:

```
kill -l
```

2. To show how to use a *PID* with the *kill* command.

```
$kill pid
```

3. To show how to send signal to processes.

```
kill {-signal | -s signal} pid
```

4. Specify Signal:

- using numbers as signals

```
kill -9 pid
```

- using SIG prefix in signals

```
kill -SIGHUP pid
```

- without SIG prefix in signals

```
kill -HUP pid
```

The list of processes to be signaled can be a mixture of names and PIDs.

pid Each pid can be expressed in one of the following ways:

n where n is larger than 0. The process with PID n is signaled.

0 All processes in the current process group are signaled.

-1 All processes with a PID larger than 1 are signaled.

-n where n is larger than 1. All processes in process group n are signaled.

When an argument of the form '-n' is given, and it is meant to denote a process group, either a signal must be specified first, or the argument must be preceded by a '--' option, otherwise it will be taken as the signal to send.

name All processes invoked using this name will be signaled.

`-s, --signal signal`
The signal to send. It may be given as a name or a number.

`-l, --list [number]`
Print a list of signal names, or convert the given signal number to a name. The signals can be found in `/usr/include/linux/signal.h`.

`-L, --table`
Similar to `-l`, but it will print signal names and their corresponding numbers.

`-a, --all`
Do not restrict the command-name-to-PID conversion to processes with the same UID as the present process.

`-p, --pid`
Only print the process ID (PID) of the named processes, do not send any signals.

`--verbose`
Print PID(s) that will be signaled with kill along with the signal.

killall

killall sends a signal to **all** processes running any of the specified commands. If no signal name is specified, **SIGTERM** is sent. In general, **killall** command kills all processes by knowing the name of the process.

Signals can be specified either by name (e.g. **-HUP** or **-SIGHUP**) or by number (e.g. **-1**) or by option **-s**.

If the command name is not a regular expression (option **-r**) and contains a slash (/), processes executing that particular file will be selected for killing, independent of their name.

killall returns a zero return code if at least one process has been killed for each listed command, or no commands were listed and at least one process matched the **-u** and **-Z** search criteria. **killall** returns non-zero otherwise.

A **killall** process never kills itself (but may kill other **killall** processes).

1. Kill all processes matching the name **conky** with **SIGTERM**:

```
killall conky
# OR
killall -SIGTERM conky
# OR
killall -15 conky
```

I was able to kill Wine (which are Windows exe files running on Linux) applications this way too.

```
killall TQ.exe
```

2. List all the supported signals:

```
$ killall -l
HUP INT QUIT ILL TRAP ABRT BUS FPE KILL USR1 SEGV USR2 PIPE
ALRM TERM STKFLT
CHLD CONT STOP TSTP TTIN TTOU URG XCPU XFSZ VTALRM PROF WINCH
POLL PWR SYS
```

As for the numbers.

```
$ for s in $(killall -l); do echo -n "$s " && kill -l $s; done
HUP 1
INT 2
QUIT 3
ILL 4
TRAP 5
ABRT 6
BUS 7
FPE 8
KILL 9
USR1 10
SEGV 11
USR2 12
PIPE 13
ALRM 14
TERM 15
STKFLT 16
CHLD 17
CONT 18
STOP 19
TSTP 20
TTIN 21
TTOU 22
URG 23
XCPU 24
XFSZ 25
VTALRM 26
PROF 27
WINCH 28
POLL 29
PWR 30
SYS 31
```

3. Ask before killing, to prevent unwanted kills:

```
$ killall -i conky
Kill conky(1685) ? (y/N)
```

4. Kill all processes and wait until the processes die.

```
killall -w conky
```

5. Kill based on time:

```
# Kill all firefox younger than 2 minutes
killall -y 2m firefox
```

```
# Kill all firefox older than 2 hours
killall -o 2h firefox
```

```
killall [OPTION]... [--] NAME...
killall -l, --list
killall -V, --version
```

Short Flag	Long Flag	Description
-e	--exact	require an exact match for very long names
-I	--ignore-case	case insensitive process name match
-g	--process-group	kill process group instead of process
-y	--younger-than	kill processes younger than TIME
-o	--older-than	kill processes older than TIME
-i	--interactive	ask for confirmation before killing
-l	--list	list all known signal names
-q	--quiet	don't print complaints
-r	--regex	interpret NAME as an extended regular expression

Short Flag	Long Flag	Description
-s	--signal SIGNAL	send this signal instead of SIGTERM
-u	--user USER	kill only process(es) running as USER
-v	--verbose	report if the signal was successfully sent
-w	--wait	wait for processes to die
-n	--ns PID	match processes that belong to the same namespaces as PID
-Z	--context	REGEXP kill only process(es) having context (must precede other arguments)

[kill](#), [pidof](#)

env

The **env** command in Linux/Unix is used to either print a list of the current environment variables or to run a program in a custom environment without changing the current one.



```
env [OPTION]... [-] [NAME=VALUE]... [COMMAND [ARG]...]
```

1. Print out the set of current environment variables

```
env
```

2. Run a command with an empty environment

```
env -i command_name
```

3. Remove variable from the environment

```
env -u variable_name
```

4. End each output with NULL

```
env -0
```

Short Flag	Long Flag	Description
<code>-i</code>	<code>--ignore-environment</code>	Start with an empty environment
<code>-0</code>	<code>--null</code>	End each output line with NUL, not newline
<code>-u</code>	<code>--unset=NAME</code>	Remove variable from the environment
<code>-C</code>	<code>--chdir=DIR</code>	Change working directory to DIR
<code>-S</code>	<code>--split-string=S</code>	Process and split S into separate arguments. It's used to pass multiple arguments on shebang lines
<code>-v</code>	<code>--debug</code>	Print verbose information for each processing step
<code>-</code>	<code>--help</code>	Print a help message
<code>-</code>	<code>--version</code>	Print the version information

printenv

The **printenv** prints the values of the specified [environment VARIABLE\(s\)](#). If no [VARIABLE](#) is specified, print name and value pairs for them all.

1. Display the values of all environment variables.

```
printenv
```

2. Display the location of the current user's [home directory](#).

```
printenv HOME
```

3. To use the **--null** command line option as the terminating character between output entries.

```
printenv --null SHELL HOME
```

*NOTE: By default, the **printenv** command uses newline as the terminating character between output entries.*

```
printenv [OPTION]... PATTERN...
```

Short Flag Long Flag Description

-0	--null	End each output line with 0 byte rather than newline .
--help	-	Display a help message, and exit.

hostname

hostname is used to display the system's DNS name, and to display or set its hostname or NIS domain name.

```
hostname [-a|--alias] [-d|--domain] [-f|--fqdn|--long] [-A|--all-fqdns] [-i|--ip-address] [-I|--all-ip-addresses] [-s|--short] [-y|--yp|--nis]
```

1. **hostname -a, hostname --alias** Display the alias name of the host (if used). This option is deprecated and should not be used anymore.
2. **hostname -s, hostname --short** Display the short host name. This is the host name cut at the first dot.
3. **hostname -V, hostname --version** Print version information on standard output and exit successfully.

Run below command to view the complete guide to **hostname** command.

```
man hostname
```

nano

The **nano** command lets you create/edit text files.

Nano text editor is pre-installed on macOS and most Linux distros. It's an alternative to **vi** and **vim**. To check if it is installed on your system type:

```
nano --version
```

If you don't have **nano** installed you can do it by using the package manager:

Ubuntu or Debian:

```
sudo apt install nano
```

1. Open an existing file, type **nano** followed by the path to the file:

```
nano /path/to/filename
```

2. Create a new file, type **nano** followed by the filename:

```
nano filename
```

3. Open a file with the cursor on a specific line and character use the following syntax:



```
nano +line_number,character_number filename
```

Shortcut	Description
----------	-------------

Ctrl + S	Save current file
----------	-------------------

Ctrl + O	Offer to write file ("Save as")
----------	---------------------------------

Ctrl + X	Close buffer, exit from nano
----------	------------------------------

Ctrl + K	Cut current line into cutbuffer
----------	---------------------------------

Ctrl + U	Paste contents of cutbuffer
----------	-----------------------------

Alt + 6	Copy current line into cutbuffer
---------	----------------------------------

Alt + U	Undo last action
---------	------------------

Alt + E	Redo last undone action
---------	-------------------------

rm

rm which stands for "remove" is a command used to remove (*delete*) specific files. It can also be used to remove directories by using the appropriate flag.

```
rm filename.txt
```

```
rm [OPTION] [FILE|DIRECTORY]
```

Short Flag	Long Flag	Description
-f	--force	Ignore nonexistence of files or directories, never prompt
-i	-	Prompt before every removal
-I	-	Prompt once before removal of more than 3 files, or when removing recursively
-d	--dir	remove empty directories
-v	--verbose	explain what is being done
-r or -R	--recursive	remove directories and their contents recursively
-	--help	Display help then exit
-	--version	First, Print version Information, Then exit
-	--no-preserve-root	do not treat / specially
-	-preserve-root[=all]	do not remove / (default) with 'all', reject any command line argument on a separate device from its parent

Short Flag	Long Flag	Description
-	<code>--interactive[=WHEN]</code>	prompt according to WHEN, never, once <code>-I</code> , or always <code>-i</code> , without WHEN, prompt always
-	<code>--one-file-system</code>	when removing a hierarchy recursively, skip any directory that is on a file system different from that of the corresponding command line argument0

IMPORTANT NOTICE:

1. `rm` doesn't remove directories by default, so use `-r`, `-R`, `--recursive` options to remove each listed directory, along with all of its contents.
2. To remove a file whose name starts with `-` such as `-foo`, use one of the following commands:
 - `rm -- -foo`
 - `rm ./-foo`
3. To ensure that files/directories being deleted are truly unrecoverable, consider using the `shred` command.

ifconfig

ifconfig is used to configure the kernel-resident network interfaces. It is used at boot time to set up interfaces as necessary. After that, it is usually only needed when debugging or when system tuning is needed.

If no arguments are given, **ifconfig** displays the status of the currently active interfaces. If a single interface argument is given, it displays the status of the given interface only; if a single -a argument is given, it displays the status of all interfaces, even those that are down. Otherwise, it configures an interface.

```
ifconfig [-v] [-a] [-s] [interface]
ifconfig [-v] interface [atype] options
```

1. To display the currently active interfaces:

```
ifconfig
```

2. To show all interfaces which are currently active, even if down:

```
ifconfig -a
```

3. To show all the error conditions:

```
ifconfig -v
```

4. To show a short list:

```
ifconfig -s
```

5. To display details of the specific network interface (say `eth0`):

```
ifconfig eth0
```

6. To activate the driver for a interface (say `eth0`):

```
ifconfig eth0 up
```

7. To deactivate the driver for a interface (say `eth0`):

```
ifconfig eth0 down
```

8. To assign a specific IP address to a network interface (say `eth0`):

```
ifconfig eth0 10.10.1.23
```

9. To change MAC(Media Access Control) address of a network interface (say `eth0`):

```
ifconfig eth0 hw ether AA:BB:CC:DD:EE:FF
```

10. To define a netmask for a network interface (say `eth0`):

```
ifconfig eth0 netmask 255.255.255.224
```

11. To enable promiscuous mode on a network interface (say `eth0`):

```
ifconfig eth0 promisc
```

In normal mode, when a packet is received by a network card, it verifies that it belongs to itself. If not, it drops the packet normally. However, in the promiscuous mode, it accepts all the packets that flow through the network card.

12. To disable promiscuous mode on a network interface (say `eth0`):

```
ifconfig eth0 -promisc
```

13. To set the maximum transmission unit to a network interface (say `eth0`):

```
ifconfig eth0 mtu 1000
```

The MTU allows you to set the limit size of packets that are transmitted on an interface. The MTU is able to handle a maximum number of octets to an interface in one single transaction.

14. To add additional IP addresses to a network interface, you can configure a network alias to the network interface:

```
ifconfig eth0:0 10.10.1.24
```

Please note that the alias network address is in the same subnet mask of the network interface. For example, if your `eth0` network ip address is `10.10.1.23`, then the alias ip address can be `10.10.1.24`. Example of an invalid IP address is `10.10.2.24` since the interface subnet mask is `255.255.255.224`

15. To remove a network alias:

```
ifconfig eth0:0 down
```

Remember that for every scope (i.e. same net with address/netmask combination) all aliases are deleted, if you delete the first alias.

Run below command to view the complete guide to **ifconfig** command.

```
man ifconfig
```

ip

The **ip** command is present in the net-tools which is used for performing several network administration tasks. IP stands for Internet Protocol. This command is used to show or manipulate routing, devices, and tunnels. It can perform tasks like configuring and modifying the default and static routing, setting up tunnel over IP, listing IP addresses and property information, modifying the status of the interface, assigning, deleting and setting up IP addresses and routes.

1. To assign an IP Address to a specific interface (eth1) :

```
ip addr add 192.168.50.5 dev eth1
```

2. To show detailed information about network interfaces like IP Address, MAC Address information etc. :

```
ip addr show
```

```
ip [ OPTIONS ] OBJECT { COMMAND | help }
```

Flag Description

- a Display and modify IP Addresses
- l Display and modify network interfaces
- r Display and alter the routing table

Flag Description

- n Display and manipulate neighbor objects (ARP table)
- ru Rule in routing policy database.
- S Output more information. If the option appears twice or more, the amount of information increases
- f Specifies the protocol family to use
- r Use the system's name resolver to print DNS names instead of host addresses
- C To configure color output

clear

In linux, the `clear` command is used to clear terminal screen.



\$ clear



```
$ echo Hello World  
Hello World
```

```
$ clear
```

\$

Screenshot:

```
devdojo@bobbyiliev:~/101-linux-commands-ebook$ ls -l
total 20
-rw-r--r-- 1 devdojo devdojo 1068 Oct  1 13:31 LICENSE
-rw-r--r-- 1 devdojo devdojo 9806 Oct  1 13:31 README.md
drwxr-xr-x 3 devdojo devdojo 4096 Oct  1 13:31 ebook
devdojo@bobbyiliev:~/101-linux-commands-ebook$ clear
```

After running the command your terminal screen will be clear:

```
devdojo@bobbyiliev:~/101-linux-commands-ebook$
```

su

In linux, **su** allows you to run commands with a substitute user and group ID.

When called without arguments, **su** defaults to running an interactive shell as root.

```
$ su
```

In case that you wanted to switch to a user called **devdojo**, you could do that by running the following command:

```
$ su devdojo
```

su

```
$ su [options] [-] [<user>[<argument>...]]
```

```
-m, -p    --> do not reset environment variables
-w        --> do not reset specified variables
-g        --> specify the primary group
-G        --> specify a supplemental group
-l        --> make the shell a login shell
-f        --> pass -f to the shell (for csh or tcsh)
-s        --> run <shell> if /etc/shell allows it
-p        --> create a new pseudo terminal
-h        --> display this help
-v        --> display version
```


wget

The **wget** command is used for downloading files from the Internet. It supports downloading files using HTTP, HTTPS and FTP protocols. It allows you to download several files at once, download in the background, resume downloads, limit the bandwidth, mirror a website, and much more.

The **wget** syntax requires you to define the downloading options and the URL the to be downloaded file is coming from.

```
$ wget [options] [URL]
```

In this example we will download the Ubuntu 20.04 desktop iso file from different sources. Go over to your terminal or open a new one and type in the below **wget**. This will start the download. The download may take a few minutes to complete.

1. Starting a regular download

```
wget  
https://releases.ubuntu.com/20.04/ubuntu-20.04.3-desktop-amd64  
.iso
```

2. You can resume a download using the **-C** option

```
wget -c  
https://mirrors.piconets.webwerks.in/ubuntu-mirror/ubuntu-rele  
ases/20.04.3/ubuntu-20.04.3-desktop-amd64.iso
```

3. To download in the background, use the **-b** option

```
wget -b  
https://mirrors.piconets.webwerks.in/ubuntu-mirror/ubuntu-rele  
ases/20.04.3/ubuntu-20.04.3-desktop-amd64.iso
```

On top of downloading, **wget** provides many more features, such as downloading multiple files, downloading in the background, limiting download bandwidth and resuming stopped downloads. View all **wget** options in its man page.



```
man wget
```

Short Flag	Description
-v	prints version of the wget available on your system
-h	print help message displaying all the possible options
-b	This option is used to send a process to the background as soon as it starts.
-t	This option is used to set number of retries to a specified number of times
-c	This option is used to resume a partially downloaded file

curl

In linux, **curl** is a tool to transfer data from or to a server, using one of the supported protocols(DICT, FILE ,FTP, FTPS, GOPHER, HTTP, HTTPS, IMAP, IMAPS, LDAP, LDAPS, POP3, POP3S, RTMP, RTSP, SCP, SFTP, SMB, SMBS, SMTP, SMTPS, TELNET and TFTP).



```
$ curl example.com
```

The command will print the source code of the example.com homepage in the terminal window.

curl

```
$ curl [options...] <url>
```

Options start with one or two dashes. Many of the options require an additional value next to them.

The short "single-dash" form of the options, `-d` for example, may be used with or without a space between it and its value, although a space is a recommended separator. The long "double-dash" form, `-d`, `--data` for example, requires a space between it and its value.

Short version options that don't need any additional values can be used immediately next to each other, like for example you can specify all the options `-O`, `-L` and `-v` at once as `-OLv`.

In general, all boolean options are enabled with `--option` and yet again disabled with `--no-option`. That is, you use the exact same option name but prefix it with `no-`. However, in this list we mostly only list and show the `--option` version of them. (This concept with `--no` options was added in 7.19.0. Previously most options were toggled on/off through repeated use of the same command line option.)

The curl command comes with most of the Linux distributions. But, if the system does not carry the curl by default. You need to install it manually. To install the curl, execute the following commands:

Update the system by executing the following commands:

```
$ sudo apt update  
$ sudo apt upgrade
```

Now, install the curl utility by executing the below command:

```
$ sudo apt install curl
```

Verify the installation by executing the below command:

```
$ curl -version
```

The above command will display the installed version of the curl command.

yes

The **yes** command in linux is used to print a continuous output stream of given *STRING*. If *STRING* is not mentioned then it prints 'y'. It outputs a string repeatedly until killed (using something like ctrl + c).

1. Prints hello world infinitely in the terminal until killed :

```
yes hello world
```

2. A more generalized command:

```
yes [STRING]
```

It accepts the following options:

1. `--help`
display this help and exit
2. `--version`
output version information and exit

last

This command shows you a list of all the users that have logged in and out since the creation of the `var/log/wtmp` file. There are also some parameters you can add which will show you for example when a certain user has logged in and how long he was logged in for.

If you want to see the last 5 logs, just add `-5` to the command like this:

```
last -5
```

And if you want to see the last 10, add `-10`.

Another cool thing you can do is if you add `-F` you can see the login and logout time including the dates.

```
last -F
```

There are quite a lot of stuff you can view with this command. If you need to find out more about this command you can run:

```
last --help
```

locate

The **locate** command searches the file system for files and directories whose name matches a given pattern through a database file that is generated by the **updatedb** command.

1. Running the **locate** command to search for a file named **.bashrc**.

```
locate .bashrc
```

Output

```
/etc/bash.bashrc
/etc/skel/.bashrc
/home/linuxize/.bashrc
/usr/share/base-files/dot.bashrc
/usr/share/doc/adduser/examples/adduser.local.conf.examples/ba
sh.bashrc
/usr/share/doc/adduser/examples/adduser.local.conf.examples/sk
el/dot.bashrc
```

The **/root/.bashrc** file will not be shown because we ran the command as a normal user that doesn't have access permissions to the **/root** directory.

If the result list is long, for better readability, you can pipe the output to the **less** command:

```
locate .bashrc | less
```

2. To search for all **.md** files on the system

```
locate *.md
```

3. To search all `.py` files and display only 10 results

```
locate -n 10 *.py
```

4. To performs case-insensitive search.

```
locate -i readme.md
```

Output

```
/home/linuxize/p1/readme.md  
/home/linuxize/p2/README.md  
/home/linuxize/p3/ReadMe.md
```

5. To return the number of all files containing `.bashrc` in their name.

```
locate -c .bashrc
```

Output

```
6
```

6. The following would return only the existing `.json` files on the file system.

```
locate -e *.json
```

7. To run a more complex search the `-r` (`--regex`) option is used. To search for all `.mp4` and `.avi` files on your system and ignore case.

```
locate --regex -i "(\\.mp4|\\.avi)"
```

```
1. locate [OPTION]... PATTERN...
```

Short Flag	Long Flag	Description
-A	--all	It is used to display only entries that match all PATTERNS instead of requiring only one of them to match.
-b	--basename	It is used to match only the base name against the specified patterns.
-c	--count	It is used for writing the number matching entries instead of writing file names on standard output.
-d	--database DBPATH	It is used to replace the default database with DBPATH.
-e	--existing	It is used to display only entries that refer to existing files during the command is executed.
-L	--follow	If the --existing option is specified, It is used for checking whether files exist and follow trailing symbolic links. It will omit the broken symbolic links to the output. This is the default behavior. The opposite behavior can be specified using the --nofollow option.
-h	--help	It is used to display the help documentation that contains a summary of the available options.
-i	--ignore-case	It is used to ignore case sensitivity of the specified patterns.
-p	--ignore-spaces	It is used to ignore punctuation and spaces when matching patterns.
-t	--transliterate	It is used to ignore accents using iconv transliteration when matching patterns.
-l	--limit, -n LIMIT	If this option is specified, the command exit successfully after finding LIMIT entries.
-m	--mmap	It is used to ignore the compatibility with BSD, and GNU locate.

Short Flag	Long Flag	Description
-0	--null	It is used to separate the entries on output using the ASCII NUL character instead of writing each entry on a separate line.
-S	--statistics	It is used to write statistics about each read database to standard output instead of searching for files.
-r	--regexp REGEXP	It is used for searching a basic regexp REGEXP.
--regex	-	It is used to describe all PATTERNS as extended regular expressions.
-V	--version	It is used to display the version and license information.
-w	--wholename	It is used for matching only the whole path name in specified patterns.

iostat

The **iostat** command in Linux is used for monitoring system input/output statistics for devices and partitions. It monitors system input/output by observing the time the devices are active in relation to their average transfer rates. The iostat produce reports may be used to change the system configuration to raised balance the input/output between the physical disks. iostat is being included in sysstat package. If you don't have it, you need to install first.

```
iostat [ -c ] [ -d ] [ -h ] [ -N ] [ -k | -m ] [ -t ] [ -V ] [
-x ]
      [ -z ] [ [ [ -T ] -g group_name ] { device [...] | ALL
} ]
      [ -p [ device [,...] | ALL ] ] [ interval [ count ] ]
```

1. Display a single history-since-boot report for all CPU and Devices:

```
iostat -d 2
```

2. Display a continuous device report at two-second intervals:

```
iostat -d 2 6
```

3. Display, for all devices, six reports at two-second intervals:

```
iostat -x sda sdb 2 6
```

4. Display, for devices sda and sdb, six extended reports at two-second intervals:


```
iostat -p sda 2 6
```

Short Flag

-x
-c
-d
-xd
-k
-k23
-j ID mmcblk0 sda6 -x -m 2
-p
-N

Description

Show more details statistics information.
Show only the cpu statistic.
Display only the device report
Show extended I/O statistic for device only.
Capture the statistics in kilobytes or megabytes.
Display cpu and device statistics with delay.
Display persistent device name statistics.
Display statistics for block devices.
Display lvm2 statistic information.

sudo

The **sudo** ("substitute user do" or "super user do") command allows a user with proper permissions to execute a command as another user, such as the superuser.

This is the equivalent of "run as administrator" option in Windows. The **sudo** command allows you to elevate your current user account to have root privileges. Also, the root privilege in **sudo** is only valid for a temporary amount of time. Once that time expires, you have to enter your password again to regain root privilege.

WARNING: Be very careful when using the **sudo** command. You can cause irreversible and catastrophic changes while acting as root!

```
sudo [-OPTION] command
```

Flag Description

- V** The -V (version) option causes sudo to print the version number and exit. If the invoking user is already root, the -V option prints out a list of the defaults sudo was compiled with and the machine's local network addresses
- l** The -l (list) option prints out the commands allowed (and forbidden) the user on the current host.
- L** The -L (list defaults) option lists out the parameters set in a Defaults line with a short description for each. This option is useful in conjunction with grep.
- h** The -h (help) option causes sudo to print a usage message and exit.
- v** If given the -v (validate) option, **sudo** updates the user's timestamp, prompting for the user's password if necessary. This extends the sudo timeout for another 5 minutes (or whatever the timeout is set to in sudoers) but does not run a command.
- K** The -K (sure kill) option to sudo removes the user's timestamp entirely. Likewise, this option does not require a password.

Flag Description

- u The -u (user) option causes sudo to run the specified command as a user other than root. To specify a uid instead of a username, use #uid.
- s The -s (shell) option runs the shell specified by the SHELL environment variable if it's set or the shell as specified in the file passwd.
- The -- flag indicates that sudo should stop processing command line arguments. It is most useful in conjunction with the -s flag.

This command switches your command prompt to the BASH shell as a root user:

```
sudo bash
```

Your command line should change to:

```
root@hostname:/home/[username]
```

Adding a string of text to a file is often used to add the name of a software repository to the sources file, without opening the file for editing. Use the following syntax with echo, sudo and tee command:

```
echo 'string-of-text' | sudo tee -a [path_to_file]
```

Example:

```
echo "deb http://nginx.org/packages/debian `lsb_release -cs`  
nginx" \ | sudo tee /etc/apt/sources.list.d/nginx.list
```

apt

apt (Advantage package system) command is used for interacting with **dpkg** (packaging system used by debian). There is already the **dpkg** command to manage **.deb** packages. But **apt** is a more user-friendly and efficient way.

In simple terms **apt** is a command used for installing, deleting and performing other operations on debian based Linux.

You will be using the **apt** command mostly with **sudo** privileges.

install followed by **package_name** is used with **apt** to install a new package.

Syntax:

```
sudo apt install package_name
```

Example:

```
sudo apt install g++
```

This command will install g++ on your system.

remove followed by **package_name** is used with **apt** to remove a specific package.

Syntax:

```
sudo apt remove package_name
```

Example:

```
sudo apt remove g++
```

This command will remove g++ from your system.

search followed by the **package_name** used with apt to search a package across all repositories.

Syntax:

```
apt search package_name
```

note: sudo not required

Example:

```
apt search g++
```

Whenever a new package that depends on other packages is installed on the system, the package dependencies will be installed too. When the package is removed, the dependencies will stay on the system. This leftover packages are no longer used by anything else and can be removed.

Syntax:

```
sudo apt autoremove
```

This command will remove all unused from your system.

apt package index is nothing but a database that stores records of available packages that are enabled on your system.

Syntax:

```
sudo apt update
```

This command will update the package index on your system.

If you want to install the latest updates for your installed packages you may want to run this command.

Syntax:

```
sudo apt upgrade
```

The command doesn't upgrade any packages that require removal of installed packages.

If you want to upgrade a single package, pass the package name:

Syntax:

```
sudo apt upgrade package_name
```

This command will upgrade your packages to the latest version.

yum

The **yum** command is the primary package management tool for installing, updating, removing, and managing software packages in Red Hat Enterprise Linux. It is an acronym for *Yellow Dog Updater, Modified*.

yum performs dependency resolution when installing, updating, and removing software packages. It can manage packages from installed repositories in the system or from .rpm packages.

```
yum -option command
```

1. To see an overview of what happened in past transactions:

```
yum history
```

2. To undo a previous transaction:

```
yum history undo <id>
```

3. To install firefox package with 'yes' as a response to all confirmations

```
yum -y install firefox
```

4. To update the mysql package it to the latest stable version

Command	Description
<code>install</code>	Installs the specified packages
<code>remove</code>	Removes the specified packages
<code>search</code>	Searches package metadata for keywords
<code>info</code>	Lists the description
<code>update</code>	Updates each package to the latest version
<code>repolist</code>	Lists repositories
<code>history</code>	Displays what has happened in past transactions
<code>groupinstall</code>	To install a particular package group
<code>clean</code>	To clean all cached files from enabled repository

Short Flag	Long Flag	Description
<code>-C</code>	<code>--cacheonly</code>	Runs entirely from system cache, doesn't update the cache and use it even in case it is expired.
<code>-</code>	<code>--security</code>	Includes packages that provide a fix for a security issue. Applicable for the upgrade command.
<code>-y</code>	<code>--assumeyes</code>	Automatically answer yes for all questions.
<code>-</code>	<code>--skip-broken</code>	Resolves depsolve problems by removing packages that are causing problems from the transaction. It is an alias for the strict configuration option with value False.
<code>-v</code>	<code>--verbose</code>	Verbose operation, show debug messages.

zip

The **zip** command is used to compress files and reduce their size. It outputs an archive containing one or more compressed files or directories.

In order to compress a single file with the **zip** command the syntax would be the following:

```
zip myZipFile.zip filename.txt
```

This also works with multiple files as well:

```
zip multipleFiles.zip file1.txt file2.txt
```

If you are compressing a whole directory, don't forget to add the **-r** flag:

```
zip -r zipFolder.zip myFolder/
```

```
zip [OPTION] zipFileName filesList
```

Flag Description

- d** Removes the file from the zip archive. After creating a zip file, you can remove a file from the archive using the **-d** option

Flag Description

- u Updates the file in the zip archive. This option can be used to update the specified list of files or add new files to the existing zip file. Update an existing entry in the zip archive only if it has been modified more recently than the version already in the zip archive.
- m Deletes the original files after zipping.
- r To zip a directory recursively, it will recursively zip the files in a directory. This option helps to zip all the files present in the specified directory.
- x Exclude the files in creating the zip
- v Verbose mode or print diagnostic version info. Normally, when applied to real operations, this option enables the display of a progress indicator during compression and requests verbose diagnostic info about zip file structure oddities

unzip

The **unzip** command extracts all files from the specified ZIP archive to the current directory.

In order to extract the files the syntax would be the following:

```
unzip myZipFile.zip
```

To unzip a ZIP file to a different directory than the current one, don't forget to add the **-d** flag:

```
unzip myZipFile.zip -d /path/to/directory
```

To unzip a ZIP file and exclude specific file or files or directories from being extracted, don't forget to add the **-x** flag:

```
unzip myZipFile.zip -x file1.txt file2.txt
```

```
unzip zipFileName [OPTION] [PARAMS]
```

Flag Description	Params
-d Unzip an archive to a different directory.	/path/to/directory

Flag	Description	Params
-x	Extract the archive but do not extract the specified files.	filename(s)
-j	Unzip without creating new folders, if the zipped archive contains a folder structure.	-
-l	Lists the contents of an archive file without extracting it.	-
-n	Do not overwrite existing files; supply an alternative filename instead.	-
-o	Overwrite files.	-
-P	Supplies a password to unzip a protected archive file.	password
-q	Unzips without writing status messages to the standard output.	-
-t	Tests whether an archive file is valid.	-
-v	Displays detailed (verbose) information about the archive without extracting it.	-

shutdown

The **shutdown** command lets you bring your system down in a secure way. When **shutdown** is executed the system will notify all logged-in users and disallow further logins. You have the option to shut down your system immediately or after a specific time.

Only users with root (or sudo) privileges can use the **shutdown** command.

1. Shut down your system immediately:

```
sudo shutdown now
```

2. Shut down your system after 10 minutes:

```
sudo shutdown +10
```

3. Shut down your system with a message after 5 minutes:

```
sudo shutdown +5 "System will shutdown in 5 minutes"
```

```
shutdown [OPTIONS] [TIME] [MESSAGE]
```

Short Flag Long Flag Description

- | | | |
|------------|---|-------------------------------|
| - r | - | Reboot the system |
| - C | - | Cancel an scheduled shut down |

dir

The **dir** command lists the contents of a directory(*the current directory by default*). **It differs from ls command in the format of listing the content.** By default, the dir command lists the files and folders in columns, sorted vertically and special characters are represented by backslash escape sequences.

```
dir [OPTIONS] [FILE]
```

1. To list files in the current directory:

```
dir
```

2. To list even the hidden files in the current directory:

```
dir -a
```

3. To list the content with detailed information for each entry

```
dir -l
```


Short Flag	Long Flag	Description
-a	--all	It displays all the hidden files(starting with <code>.</code>) along with two files denoted by <code>.</code> and <code>..</code>
-A	--almost-all	It is similar to -a option except that it <i>does not display files that signals the current directory and previous directory.</i>
-l	-	Display detailed information for each entry
-s	--size	Print the allocated size of each file, in blocks File
-h	--human-readable	Used with with -l and -s, to print sizes like in human readable format like 1K, 2M and so on
-F	-	Classifies entries into their type based on appended symbol (<code>/</code> , <code>*</code> , <code>@</code> , <code>%</code> , <code>=</code>)
-v	--verbose	Print source and destination files
-	--group-directories-first	To group directories before files
-R	--recursive	To List subdirectories recursively.
-S	-	sort by file size, display largest first

reboot

The **reboot** command is used to restart a linux system. However, it requires elevated permission using the **sudo** command. Necessity to use this command usually arises after significant system or network updates have been made to the system.

```
reboot [OPTIONS...]
```

- **-help** : This option prints a short help text and exit.
- **-halt** : This command will stop the machine.
- **-w, -wtmp-only** : This option only writes wtmp shutdown entry, it do not actually halt, power-off, reboot.

1. Basic Usage. Mainly used to restart without any further details

```
$ sudo reboot
```

However, alternatively the shutdown command with the **-r** option

```
$ sudo shutdown -r now
```

Note that the usage of the reboot, halt and power off is almost similar in syntax and effect. Run each of these commands with **-help** to see the details.

2. The **reboot** command has limited usage, and the **shutdown** command is being used instead of reboot command to fulfill much more advance reboot and shutdown requirements. One of those situations is a scheduled restart. Syntax is as follows

```
$ sudo shutdown -r [TIME] [MESSAGE]
```

Here the TIME has various formats. The simplest one is **now**, already been listed in the previous section, and tells the system to restart immediately. Other valid formats we have are **+m**, where m is the number of minutes we need to wait until restart and

HH:MM which specifies the TIME in a 24hr clock.

Example to reboot the system in 2 minutes

```
$ sudo shutdown -r +2
```

Example of a scheduled restart at 03:00 A.M

```
$ sudo shutdown -r 03:00
```

3. Cancelling a Reboot. Usually happens in case one wants to cancel a scheduled restart

Syntax

```
$ sudo shutdown -c [MESSAGE]
```

Usage

```
$sudo shutdown -c "Scheduled reboot cancelled because the  
chicken crossed the road"
```

4. Checking your reboot logs

```
$ last reboot
```

sort

the **sort** command is used to sort a file, arranging the records in a particular order. By default, the sort command sorts a file assuming the contents are ASCII. Using options in the sort command can also be used to sort numerically.

Suppose you create a data file with name file.txt:

```
Command :  
$ cat > file.txt  
abhishek  
chitransh  
satish  
rajan  
naveen  
divyam  
harsh
```

Sorting a file: Now use the sort command

Syntax :

```
sort filename.txt
```

```
Command:  
$ sort file.txt
```

```
Output :  
abhishek  
chitransh  
divyam  
harsh  
naveen  
rajan  
satish
```

Note: This command does not actually change the input file, i.e. file.txt.

i.e. uppercase and lower case: When we have a mix file with both uppercase and lowercase letters then first the upper case letters would be sorted following with the lower case letters.

Example:

Create a file mix.txt

```
Command :  
$ cat > mix.txt  
abc  
apple  
BALL  
Abc  
bat
```

Now use the sort command



```
Command :  
$ sort mix.txt  
Output :  
Abc  
BALL  
abc  
apple  
bat
```

paste

The **paste** command writes lines of two or more files, sequentially and separated by TABs, to the standard output

```
paste [OPTIONS]... [FILE]...
```

1. To paste two files

```
paste file1 file2
```

2. To paste two files using new line as delimiter

```
paste -d '\n' file1 file2
```

Short Flag	Long Flag	Description
-d	--delimiter	use character of TAB
-s	--serial	paste one file at a time instead of in parallel
-z	--zero-terminated	set line delimiter to NUL, not newline
	--help	print command help
	--version	print version information

exit

The `exit` command is used to terminate (close) an active shell session



```
exit
```

Shortcut: Instead of typing `exit`, press `ctrl + D`, it will do the same Functionality.

diff/sdiff

This command is used to display the differences in the files by comparing the files line by line.

```
diff [options] File1 File2
```

1. Lets say we have two files with names a.txt and b.txt containing 5 Indian states as follows-:

```
$ cat a.txt
Gujarat
Uttar Pradesh
Kolkata
Bihar
Jammu and Kashmir
```

```
$ cat b.txt
Tamil Nadu
Gujarat
Andhra Pradesh
Bihar
Uttar pradesh
```

On typing the diff command we will get below output.

```
$ diff a.txt b.txt
0a1
> Tamil Nadu
2,3c3
< Uttar Pradesh
  Andhra Pradesh
5c5
  Uttar pradesh
```

Short Flag

Description

- c** To view differences in context mode, use the -c option.
- u** To view differences in unified mode, use the -u option. It is similar to context mode
- i** By default this command is case sensitive. To make this command case insensitive use -i option with diff.
- version** This option is used to display the version of diff which is currently running on your system.

tar

The **tar** command stands for tape archive, is used to create Archive and extract the Archive files. This command provides archiving functionality in Linux. We can use tar command to create compressed or uncompressed Archive files and also maintain and modify them.

1. To create a tar file in abel directory:

```
tar -cvf file-14-09-12.tar /home/abel/
```

2. To un-tar a file in the current directory:

```
tar -xvf file-14-09-12.tar
```

```
tar [options] [archive-file] [file or directory to be archived]
```

Use Flag	Description
-c	Creates Archive
-x	Extract the archive
-f	Creates archive with given filename
-t	Displays or lists files in archived file
-u	Archives and adds to an existing archive file
-v	Displays Verbose Information

Use Flag	Description
-A	Concatenates the archive files
-Z	zip, tells tar command that creates tar file using gzip
-j	Filter archive tar file using tbzip
w	Verify a archive file
r	update or add file or directory in already existed .tar file
-?	Displays a short summary of the project
-d	Find the difference between an archive and file system
--usage	shows available tar options
--version	Displays the installed tar version
--show-defaults	Shows default enabled options
Option Flag	Description
--check-device	Check device numbers during incremental archive
-g	Used to allow compatibility with GNU-format incremental ackups
--hole-detection	Used to detect holes in the sparse files
-G	Used to allow compatibility with old GNU-format incremental backups
--ignore-failed-read	Don't exit the program on file read errors
--level	Set the dump level for created archives
-n	Assume the archive is seekable
--no-check-device	Do not check device numbers when creating archives
--no-seek	Assume the archive is not seekable
--occurrence=N	`Process only the Nth occurrence of each file
--restrict	`Disable use of potentially harmful options
--sparse-version=MAJOR,MINOR	Set version of the sparce format to use
-S	Handle sparse files efficiently.
Overwright control Flag	Description
-k	Don't replace existing files
--keep-newer-files	Don't replace existing files that are newer than the archives version
--keep-directory-symlink	Don't replace existing symlinks
--no-overwrite-dir	Preserve metadata of existing directories
--one-top-level=DIR	Extract all files into a DIR
--overwrite	Overwrite existing files
--overwrite-dir	Overwrite metadata of directories
--recursive-unlink	Recursively remove all files in the directory before extracting

Overwright control Flag

--remove-files

--skip-old-files

-u

-w

Description

Remove files after adding them to a directory

Don't replace existing files when extracting

Remove each file before extracting over it

Verify the archive after writing it

gunzip

The **gunzip** command is an antonym command of [gzip command](#). In other words, it decompresses files deflated by the **gzip** command.

gunzip takes a list of files on its command line and replaces each file whose name ends with *.gz*, *-gz*, *.z*, *-z*, or *_z* (ignoring case) and which begins with the correct magic number with an uncompressed file without the original extension. **gunzip** also recognizes the special extensions *.tgz* and *.taz* as shorthands for *.tar.gz* and *.tar.Z* respectively.

1. Uncompress a file

```
gunzip filename.gz
```

2. Recursively uncompress content inside a directory, that match extension (suffix) compressed formats accepted by **gunzip**:

```
gunzip -r directory_name/
```

3. Uncompress all files in the current/working directory whose suffix match *.tgz*:

```
gunzip -S .tgz *
```

4. List compressed and uncompressed sizes, compression ratio and uncompressed name of input compressed file/s:

```
gunzip -l file_1 file_2
```

```
gunzip [ -acfhkLLnNrtvV ] [-S suffix] [ name ... ]
```

[This video](#) shows how to compress and decompress in a Unix shell. It uses **gunzip** as decompression command.

Short Flag	Long Flag	Description
-c	--stdout	write on standard output, keep original files unchanged
-h	--help	give help information
-k	--keep	keep (don't delete) input files
-l	--list	list compressed file contents
-q	--quiet	suppress all warnings
-r	--recursive	operate recursively on directories
-S	--suffix=SUF	use suffix SUF on compressed files
	--synchronous	synchronous output (safer if system crashes, but slower)
-t	--test	test compressed file integrity
-v	--verbose	verbose mode
-V	--version	display version number

hostnamectl

The **hostnamectl** command provides a proper API used to control Linux system hostname and change its related settings. The command also helps to change the hostname without actually locating and editing the **/etc/hostname** file on a given system.



```
$ hostnamectl [OPTIONS...] COMMAND ...
```

where **COMMAND** can be any of the following

status: Used to check the current hostname settings

set-hostname NAME: Used to set system hostname

set-icon-name NAME: Used to set icon name for host

1. Basic usage to view the current hostnames

```
$ hostnamectl
```

or

```
$ hostnamectl status
```

2. To change the static host name to *myhostname*. It may or may not require root access

```
$ hostnamectl set-hostname myhostname --static
```

3. To set or change a transient hostname

```
$ hostnamectl set-hostname myotherhostname --transient
```

4. To set the pretty hostname. The name that is to be set needs to be in the double quote(" ").

```
$ hostname set-hostname "prettyname" --pretty
```

iptables

The **iptables** command is used to set up and maintain tables for the Netfilter firewall for IPv4, included in the Linux kernel. The firewall matches packets with rules defined in these tables and then takes the specified action on a possible match.

```
iptables --table TABLE -A/-C/-D... CHAIN rule --jump Target
```

This command will append to the chain provided in parameters:

```
iptables [-t table] --append [chain] [parameters]
```

This command drops all the traffic coming on any port:

```
iptables -t filter --append INPUT -j DROP
```

Flag Description

- C Check if a rule is present in the chain or not. It returns 0 if the rule exists and returns 1 if it does not.
- A Append to the chain provided in parameters.

netstat

The term **netstat** stands for Network Statistics. In layman's terms, netstat command displays the current network connections, networking protocol statistics, and a variety of other interfaces.

Check if you have **netstat** on your PC:

```
netstat -v
```

If you don't have **netstat** installed on your PC, you can install it with the following command:

```
sudo apt install net-tools
```

netstat

- **Netstat** command with **-nr** flag shows the routing table detail on the terminal.

Example:

```
netstat -nr
```

- **Netstat** command with **-i** flag shows statistics for the currently configured network interfaces. This command will display the first 10 lines of file **foo.txt**.

Example:

```
netstat -i
```

- **Netstat** command with **-tunlp** will gives a list of networks, their current states, and their associated ports.

Example:

```
netstat -tunlp
```

- You can get the list of all TCP port connection by using **-at** with **netstat**.

```
netstat -at
```

- You can get the list of all UDP port connection by using **-au** with **netstat**.

```
netstat -au
```

- You can get the list of all active connection by using **-l** with **netstat**.

```
netstat -l
```

lsof

The **lsof** command shows **file information** of all the files opened by a running process. Its name is also derived from the fact that, list open files > **lsof**

An open file may be a regular file, a directory, a block special file, a character special file, an executing text reference, a library, a stream or a network file (Internet socket, NFS file or UNIX domain socket). A specific file or all the files in a file system may be selected by path.

```
lsof [-OPTION] [USER_NAME]
```

1. To show all the files opened by all active processes:

```
lsof
```

2. To show the files opened by a particular user:

```
lsof -u [USER_NAME]
```

3. To list the processes with opened files under a specified directory:

```
lsof +d [PATH_TO_DIR]
```

Option Additional Options Description

-i	tcp/udp/:port	List all network connections running, Additionally, on udp/tcp or on specified port.
-i4	-	List all processes with ipv4 connections.
-i6	-	List all processes with ipv6 connections.
-c	[PROCESS_NAME]	List all the files of a particular process with given name.
-p	[PROCESS_ID]	List all the files opened by a specified process id.
-p	^[PROCESS_ID]	List all the files that are not opened by a specified process id.
+d	[PATH]	List the processes with opened files under a specified directory
+R	-	List the files opened by parent process Id.

Run below command to view the complete guide to **lsof** command.

```
man lsof
```


bzip2

The **bzip2** command lets you compress and decompress the files i.e. it helps in binding the files into a single file which takes less storage space as the original file use to take.

```
bzip2 [OPTIONS] filenames ...
```

Note : Each file is replaced by a compressed version of itself, with the name original name of the file followed by extension bz2.

Option	Alias	Description
-d	--decompress	to decompress compressed file
-f	--force	to force overwrite an existing output file
-h	--help	to display the help message and exit
-k	--keep	to enable file compression, doesn't deletes the original input file
-L	--license	to display the license terms and conditions
-q	--quiet	to suppress non-essential warning messages
-t	--test	to check integrity of the specified .bz2 file, but don't want to decompress them
-v	--verbose	to display details for each compression operation
-V	--version	to display the software version
-z	--compress	to enable file compression, but deletes the original input file

By default, when bzip2 compresses a file, it deletes the original (or input) file. However, if you don't want that to happen, use the -k command line option.

1. To force compression:

```
bzip2 -z input.txt
```

Note: This option deletes the original file also

2. To force compression and also retain original input file:

```
bzip2 -k input.txt
```

3. To force decompression:

```
bzip2 -d input.txt.bz2
```

4. To test integrity of compressed file:

```
bzip2 -t input.txt.bz2
```

5. To show the compression ratio for each file processed:

```
bzip2 -v input.txt
```

service

Service runs a System V init script in as predictable environment as possible, removing most environment variables and with current working directory set to /.

The SCRIPT parameter specifies a System V init script, located in /etc/init.d/SCRIPT. The supported values of COMMAND depend on the invoked script, service passes COMMAND and OPTIONS it to the init script unmodified. All scripts should support at least the start and stop commands. As a special case, if COMMAND is --full-restart, the script is run twice, first with the stop command, then with the start command.

The COMMAND can be at least start, stop, status, and restart.

service --status-all runs all init scripts, in alphabetical order, with the **status** command

Examples :

1. To check the status of all the running services:

```
service --status-all
```

2. To run a script

```
service SCRIPT-Name start
```

3. A more generalized command:

```
service [SCRIPT] [COMMAND] [OPTIONS]
```

vmstat

The **vmstat** command lets you monitor the performance of your system. It shows you information about your memory, disk, processes, CPU scheduling, paging, and block IO. This command is also referred to as **virtual memory statistic report**.

The very first report that is produced shows you the average details since the last reboot and after that, other reports are made which report over time.

vmstat



As you can see it is a pretty useful little command. The most important things that we see above are the **free**, which shows us the free space that is not being used, **si** shows us how much memory is swapped in every second in kB, and **so** shows how much memory is swapped out each second in kB as well.

vmstat -a

If we run **vmstat -a**, it will show us the active and inactive memory of the system running.



vmstat -d

The **vmstat -d** command shows us all the disk statistics.



As you can see this is a pretty useful little command that shows you different statistics about your virtual memory

mpstat

The **mpstat** command is used to report processor related statistics. It accurately displays the statistics of the CPU usage of the system and information about CPU utilization and performance.

```
mpstat [options] [<interval> [<count>]]
```

Note : It initializes the first processor with CPU 0, the second one with CPU 1, and so on.

Option	Description
-A	to display all the detailed statistics
-h	to display mpstat help
-I	to display detailed interrupts statistics
-n	to report summary CPU statistics based on NUMA node placement
-N	to indicate the NUMA nodes for which statistics are to be reported
-P	to indicate the processors for which statistics are to be reported
-o	to display the statistics in JSON (Javascript Object Notation) format
-T	to display topology elements in the CPU report
-u	to report CPU utilization
-v	to display utilization statistics at the virtual processor level
-V	to display mpstat version
-ALL	to display detailed statistics about all CPUs

1. To display processor and CPU statistics:

```
mpstat
```

2. To display processor number of all CPUs:

```
mpstat -P ALL
```

3. To get all the information which the tool may collect:

```
mpstat -A
```

4. To display CPU utilization by a specific processor:

```
mpstat -P 0
```

5. To display CPU usage with a time interval:

```
mpstat 1 5
```

Note: This command will print 5 reports with 1 second time interval

ncdu

ncdu (NCurses Disk Usage) is a curses-based version of the well-known **du** command. It provides a fast way to see what directories are using your disk space.

1. Quiet Mode



```
ncdu -q
```

2. Omit mounted directories



```
ncdu -q -x
```




```
ncdu [-hqvX] [--exclude PATTERN] [-X FILE] dir
```

Short Flag	Long Flag	Description
-h	-	Print a small help message
-q	-	Quiet mode. While calculating disk space, ncdu will update the screen 10 times a second by default, this will be decreased to once every 2 seconds in quiet mode. Use this feature to save bandwidth over remote connections.
-v	-	Print version.
-x	-	Only count files and directories on the same filesystem as the specified dir.
-	--exclude PATTERN	Exclude files that match PATTERN. This argument can be added multiple times to add more patterns.
-X FILE	--exclude-from FILE	Exclude files that match any pattern in FILE. Patterns should be separated by a newline.

uniq

The **uniq** command in Linux is a command line utility that reports or filters out the repeated lines in a file. In simple words, **uniq** is the tool that helps you to detect the adjacent duplicate lines and also deletes the duplicate lines. It filters out the adjacent matching lines from the input file(that is required as an argument) and writes the filtered data to the output file .

In order to omit the repeated lines from a file, the syntax would be the following:

```
uniq kt.txt
```

In order to tell the number of times a line was repeated, the syntax would be the following:

```
uniq -c kt.txt
```

In order to print repeated lines, the syntax would be the following:

```
uniq -d kt.txt
```

In order to print unique lines, the syntax would be the following:

```
uniq -u kt.txt
```

In order to allows the N fields to be skipped while comparing uniqueness of the lines, the syntax would be the following:

```
uniq -f 2 kt.txt
```

In order to allow the N characters to be skipped while comparing uniqueness of the lines, the syntax would be the following:

```
uniq -s 5 kt.txt
```

In order to make the comparison case-insensitive, the syntax would be the following:

```
uniq -i kt.txt
```

```
uniq [OPTION] [INPUT[OUTPUT]]
```

Flag	Description	Params
-c	It tells how many times a line was repeated by displaying a number as a prefix with the line.	-
-d	It only prints the repeated lines and not the lines which aren't repeated.	-
-i	By default, comparisons done are case sensitive but with this option case insensitive comparisons can be made.	-
-f	It allows you to skip N fields(a field is a group of characters, delimited by whitespace) of a line before determining uniqueness of a line.	N
-s	It doesn't compare the first N characters of each line while determining uniqueness. This is like the -f option, but it skips individual characters rather than fields.	N
-u	It allows you to print only unique lines.	-
-z	It will make a line end with 0 byte(NULL), instead of a newline.	-
-w	It only compares N characters in a line.	N
--help	It displays a help message and exit.	-
--version	It displays version information and exit.	-

RPM

rpm - RPM Package Manager

rpm is a powerful **Package Manager**, which can be used to build, install, query, verify, update, and erase individual software packages. A **package** consists of an archive of files and meta-data used to install and erase the archive files. The meta-data includes helper scripts, file attributes, and descriptive information about the package. Packages come in two varieties: binary packages, used to encapsulate software to be installed, and source packages, containing the source code and recipe necessary to produce binary packages.

One of the following basic modes must be selected: **Query, Verify, Signature Check, Install/Upgrade/Freshen, Uninstall, Initialize Database, Rebuild Database, Resign, Add Signature, Set Owners/Groups, Show Querytags, and Show Configuration.**

General Options

These options can be used in all the different modes.

Short Flag	Long Flag	Description
-?	--help	Print a longer usage message then normal.
-	--version	Print a single line containing the version number of rpm being used.
-	--quiet	Print as little as possible - normally only error messages will be displayed.
-v	-	Print verbose information - normally routine progress messages will be displayed.
-vv	-	Print lots of ugly debugging information.
-	--rcfile FILELIST	Each of the files in the colon separated FILELIST is read sequentially by rpm for configuration information. Only the first file in the list must exist, and tildes will be expanded to the value of \$HOME. The default FILELIST is /usr/lib/rpm/rpmrc:/usr/lib/rpm/redhat/rpmrc:/etc/rpmrc:~/rpmrc.
-	--pipe CMD	Pipes the output of rpm to the command CMD.
-	--dbpath DIRECTORY	Use the database in DIRECTORY rather than the default path /var/lib/rpm

Short Flag	Long Flag	Description
-	--root DIRECTORY	Use the file system tree rooted at DIRECTORY for all operations. Note that this means the database within DIRECTORY will be used for dependency checks and any scriptlet(s) (e.g. %post if installing, or %prep if building, a package) will be run after a chroot(2) to DIRECTORY.
-D	--define='MACRO EXPR'	Defines MACRO with value EXPR.
-E	--eval='EXPR'	Prints macro expansion of EXPR.


```
rpm {-q|--query} [select-options] [query-options]

rpm {-V|--verify} [select-options] [verify-options]

rpm --import PUBKEY ...

rpm {-K|--checksig} [--nosignature] [--nodigest] PACKAGE_FILE
...
```



```
rpm {-i|--install} [install-options] PACKAGE_FILE ...  
rpm {-U|--upgrade} [install-options] PACKAGE_FILE ...  
rpm {-F|--freshen} [install-options] PACKAGE_FILE ...  
rpm {-e|--erase} [--allmatches] [--nodeps] [--noscripts] [--  
notriggers] [--test] PACKAGE_NAME ...
```

```
rpm {--initdb|--rebuilddb}
```

```
rpm {--addsign|--resign} PACKAGE_FILE...
```

```
rpm {--querytags|--showrc}
```

```
rpm {--setperms|--setugids} PACKAGE_NAME .
```

```
[--changelog] [-c,--configfiles] [-d,--docfiles] [--dump]  
[--filesbypkg] [-i,--info] [--last] [-l,--list]  
[--provides] [--qf,--queryformat QUERYFMT]  
[-R,--requires] [--scripts] [-s,--state]  
[--triggers,--triggerscripts]
```

```
[--nodeps] [--nofiles] [--noscripts]  
[--nodigest] [--nosignature]  
[--nolinkto] [--nofiledigest] [--nosize] [--nouser]  
[--nogroup] [--nomtime] [--nomode] [--nordev]  
[--nocaps]
```

```
[--aid] [--allfiles] [--badreloc] [--excludepath OLDPATH]
[--excludedocs] [--force] [-h,--hash]
[--ignoresize] [--ignorearch] [--ignoreeos]
[--includedocs] [--justdb] [--nodeps]
[--nodigest] [--nosignature] [--nosuggest]
[--noorder] [--noscripts] [--notriggers]
[--oldpackage] [--percent] [--prefix NEWPATH]
[--relocate OLDPATH=NEWPATH]
[--replacefiles] [--replacepkgs]
[--test]
```

scp

SCP (secure copy) is a command-line utility that allows you to securely copy files and directories between two locations.

Both the files and passwords are encrypted so that anyone snooping on the traffic doesn't get anything sensitive.

- From local system to a remote system.
- From a remote system to a local system.
- Between two remote systems from the local system.

1. To copy the files from a local system to a remote system:

```
scp /home/documents/local-file root@{remote-ip-address}:/home/
```

2. To copy the files from a remote system to the local system:

```
scp root@{remote-ip-address}:/home/remote-file  
/home/documents/
```

3. To copy the files between two remote systems from the local system.

```
scp root@{remote1-ip-address}:/home/remote-file root@{remote2-  
ip-address}/home/
```

```
scp [OPTION] [user@]SRC_HOST:]file1 [user@]DEST_HOST:]file2
```

- **OPTION** - scp options such as cipher, ssh configuration, ssh port, limit, recursive copy ...etc.
- **[user@]SRC_HOST:]file1** - Source file
- **[user@]DEST_HOST:]file2** - Destination file

Local files should be specified using an absolute or relative path, while remote file names should include a user and host specification.

scp provides several that control every aspect of its behaviour. The most widely used options are:

Short Flag	Long Flag	Description
-P	-	Specifies the remote host ssh port.
-p	-	Preserves files modification and access times.
-q	-	Use this option if you want to suppress the progress meter and non-error messages.
-C	-	This option forces scp to compresses the data as it is sent to the destination machine.
-r	-	This option tells scp to copy directories recursively.

The **scp** command relies on **ssh** for data transfer, so it requires an **ssh key** or **password** to authenticate on the remote systems.

The **colon (:)** is how scp distinguish between local and remote locations.

To be able to copy files, you must have at least read permissions on the source file and write permission on the target system.

Be careful when copying files that share the same name and location on both systems, **scp** will overwrite files without warning.

When transferring large files, it is recommended to run the scp command inside a **screen** or **tmux** session.

sleep

The **sleep** command is used to create a dummy job. A dummy job helps in delaying the execution. It takes time in seconds by default but a small suffix(s, m, h, d) can be added at the end to convert it into any other format. This command pauses the execution for an amount of time which is defined by NUMBER.

Note: If you will define more than one NUMBER with sleep command then this command will delay for the sum of the values.

1. To sleep for 10s

```
sleep 10s
```

2. A more generalized command:

```
sleep NUMBER[SUFFIX]...
```

It accepts the following options:

1. --help
display this help and exit
 2. --version
output version information and exit
-

split

The `split` command in Linux is used to split a file into smaller files.

1. Split a file into a smaller file using file name

```
split filename.txt
```

2. Split a file named filename into segments of 200 lines beginning with prefix file

```
split -l 200 filename file
```

This will create files of the name fileaa, fileab, fileac, filead, etc. of 200 lines.

3. Split a file named filename into segments of 40 bytes with prefix file

```
split -b 40 filename file
```

This will create files of the name fileaa, fileab, fileac, filead, etc. of 40 bytes.

4. Split a file using `--verbose` to see the files being created.

```
split filename.txt --verbose
```


`split [options] filename [prefix]`

Short Flag	Long Flag	Description
-a	--suffix-length=N	Generate suffixes of length N (default 2)
	--additional-suffix=SUFFIX	Append an additional SUFFIX to file names
-b	--bytes=SIZE	Put SIZE bytes per output file
-C	--line-bytes=SIZE	Put at most SIZE bytes of records per output file
-d		Use numeric suffixes starting at 0, not alphabetic
	--numeric-suffixes[=FROM]	Same as -d, but allow setting the start value
-x		Use hex suffixes starting at 0, not alphabetic
	--hex-suffixes[=FROM]	Same as -x, but allow setting the start value
-e	--elide-empty-files	Do not generate empty output files with '-n'
	--filter=COMMAND	Write to shell COMMAND; file name is \$FILE
-l	--lines=NUMBER	Put NUMBER lines/records per output file
-n	--number=CHUNKS	Generate CHUNKS output files; see explanation below
-t	--separator=SEP	Use SEP instead of newline as the record separator; '\0' (zero) specifies the NUL character
-u	--unbuffered	Immediately copy input to output with '-n r/...'
	--verbose	Print a diagnostic just before each output file is opened
	--help	Display this help and exit
	--version	Output version information and exit

The SIZE argument is an integer and optional unit (example: 10K is 10*1024). Units are K,M,G,T,P,E,Z,Y (powers of 1024) or KB,MB,... (powers of 1000).

CHUNKS may be:

CHUNKS Description

N	Split into N files based on size of input
K/N	Output Kth of N to stdout
l/N	Split into N files without splitting lines/records
l/K/N	Output Kth of N to stdout without splitting lines/records
r/N	Like 'l' but use round robin distribution
r/K/N	Likewise but only output Kth of N to stdout

stat

The **stat** command lets you display file or file system status. It gives you useful information about the file (or directory) on which you use it.

1. Basic command usage

```
stat file.txt
```

2. Use the **-c** (or **--format**) argument to only display information you want to see (here, the total size, in bytes)

```
stat file.txt -c %s
```

```
stat [OPTION] [FILE]
```

Short Flag	Long Flag	Description
-L	--dereference	Follow links
-f	--file-system	Display file system status instead of file status
-c	--format=FORMAT	Specify the format (see below)
-t	--terse	Print the information in terse form
-	--cached=MODE	Specify how to use cached attributes. Can be: always , never , or default

Short Flag	Long Flag	Description
-	<code>--printf=FORMAT</code>	Like <code>--format</code> , but interpret backslash escapes (<code>\n</code> , <code>\t</code> , ...)
-	<code>--help</code>	Display the help and exit
-	<code>--version</code>	Output version information and exit

Format	Description
<code>%a</code>	Permission bits in octal
<code>%A</code>	Permission bits and file type in human readable form
<code>%d</code>	Device number in decimal
<code>%D</code>	Device number in hex
<code>%F</code>	File type
<code>%g</code>	Group ID of owner
<code>%G</code>	Group name of owner
<code>%h</code>	Number of hard links
<code>%i</code>	Inode number
<code>%m</code>	Mount point
<code>%n</code>	File name
<code>%N</code>	Quoted file name with dereference if symbolic link
<code>%s</code>	Total size, in bytes
<code>%u</code>	User ID of owner
<code>%U</code>	User name of owner
<code>%w</code>	Time of file birth, human-readable; - if unknown
<code>%x</code>	Time of last access, human-readable
<code>%y</code>	Time of last data modification, human-readable
<code>%z</code>	Time of last status change, human-readable

useradd

The **useradd** command is used to add or update user accounts to the system.

To add a new user with the **useradd** command the syntax would be the following:

```
useradd NewUser
```

To add a new user with the **useradd** command and give a home directory path for this new user the syntax would be the following:

```
useradd -d /home/NewUser NewUser
```

To add a new user with the **useradd** command and give it a specific id the syntax would be the following:

```
useradd -u 1234 NewUser
```

```
useradd [OPTIONS] NameOfUser
```

Flag	Description	Params
-d	The new user will be created using /path/to/directory as the value for the user's login directory	/path/to/directory
-u	The numerical value of the user's ID	ID
-g	Create a user with specific group id	GroupID
-M	Create a user without home directory	-
-e	Create a user with expiry date	DATE (format: YYYY-MM-DD)
-c	Create a user with a comment	COMMENT
-s	Create a user with changed login shell	/path/to/shell
-p	Set an unencrypted password for the user	PASSWORD

userdel

The **userdel** command is used to delete a user account and related files

To delete a user with the **userdel** command the syntax would be the following:

```
userdel userName
```

To force the removal of a user account even if the user is still logged in, using the **userdel** command the syntax would be the following:

```
userdel -f userName
```

To delete a user along with the files in the user's home directory using the **userdel** command the syntax would be the following:

```
userdel -r userName
```

```
userdel [OPTIONS] userName
```

Flag Description

-f Force the removal of the specified user account even if the user is logged in

Flag Description

- r Remove the files in the user's home directory along with the home directory itself and the user's mail spool
- Z Remove any SELinux(Security-Enhanced Linux) user mapping for the user's login.

usermod

The **usermod** command lets you change the properties of a user in Linux through the command line. After creating a user we sometimes have to change their attributes, like their password or login directory etc. So in order to do that we use the **usermod** command.

```
usermod [options] USER
```

Note : Only superuser (root) is allowed to execute **usermod command**

Option Description

- a to add anyone of the group to a secondary group
- c to add comment field for the useraccount
- d to modify the directory for any existing user account
- g change the primary group for a User
- G to add supplementary groups
- l to change existing user login name
- L to lock system user account
- m to move the contents of the home directory from existing home dir to new dir
- p to create an un-encrypted password
- s to create a specified shell for new accounts
- u to assigned UID for the user account
- U to unlock any locked user

1. To add a comment/description for a user:

```
sudo usermod -c "This is test user" test_user
```

2. To change the home directory of a user:

```
sudo usermod -d /home/sam test_user
```

3. To change the expiry date of a user:

```
sudo usermod -e 2021-10-05 test_user
```

4. To change the group of a user:

```
sudo usermod -g sam test_user
```

5. To change user login name:

```
sudo usermod -l test_account test_user
```

6. To lock a user:

```
sudo usermod -L test_user
```

7. To unlock a user:

```
sudo usermod -U test_user
```

8. To set an unencrypted password for the user:

```
sudo usermod -p test_password test_user
```

9. To create a shell for the user:

```
sudo usermod -s /bin/sh test_user
```

10. To change the user id of a user:

```
sudo usermod -u 1234 test_user
```

ionice

The **ionice** command is used to set or get process I/O scheduling class and priority.

If no arguments are given , **ionice** will query the current I/O scheduling class and priority for that process.



```
ionice [options] -p <pid>
```



```
ionice [options] -P <pgid>
```



```
ionice [options] -u <uid>
```



```
ionice [options] <command>
```

-

A program with idle I/O priority will only get disk time when **no other program has asked for disk I/O for a defined grace period**.

The impact of idle processes on normal system actively should be **zero**.

This scheduling class **doesn't take priority** argument.

Presently this scheduling class is permitted for an **ordinary user (since kernel 2.6.25)**.

-

This is **effective** scheduling class for any process that has **not asked for a specific I/O priority**.

This class **takes priority argument from 0-7**, with **lower** number being **higher priority**.

Programs running at the same best effort priority are served in **round-robbin fashion**.

Note that before kernel 2.6.26 a process that has not asked for an I/O priority formally uses "None" as scheduling class , but the io scheduler will treat such processes as if it were in the best effort class.

The priority within best effort class will be dynamically derived from the CPU nice level of the process : $io_priority = (cpu_nice + 20) / 5$ for kernels after 2.6.26 with CFQ I/O scheduler a process that has not asked for an io priority inherits CPU scheduling class.

The I/O priority is derived from the CPU nice level of the process (since before kernel 2.6.26).

-

The real time scheduler class is given first access to disk, regardless of what else is going on in the system.

Thus the real time class needs to be used with some care, as it can starve other processes .

As with the best effort class, 8 priority levels are defined denoting how big a time slice a given process will receive on each scheduling window.

This scheduling class is not permitted for an ordinary user(non-root).

Options	Description
-c, --class	name or number of scheduling class, 0: none, 1: realtime, 2: best-effort, 3: idle
-n, --classdata	priority (0..7) in the specified scheduling class, only for the realtime and best-effort classes
-p, --pid ...	act on these already running processes
-P, --pgid ...	act on already running processes in these groups
-t, --ignore	ignore failures
-u, --uid ...	act on already running processes owned by these users
-h, --help	display this help
-V, --version	display version

For more details see `ionice(1)`.

Command	O/P	Explanation
\$ ionice	<i>none: prio 4</i>	Running alone ionice will give the class and priority of current process
\$ ionice -p 101	<i>none : prio 4</i>	Give the details(class : priority) of the process specified by given process id
\$ ionice -p 2	<i>none: prio 4</i>	Check the class and priority of process with pid 2 it is none and 4 resp.
\$ ionice -c2 -n0 -p2	2 (best-effort) priority 0 process 2	Now lets set process(pid) 2 as a best-effort program with highest priority
\$ ionice -p 2	best-effort : prio 0	Now if I check details of Process 2 you can see the updated one
\$ ionice /bin/ls		get priority and class info of bin/ls
\$ ionice -n4 -p2		set priority 4 of process with pid 2
\$ ionice -p 2	best-effort: prio 4	Now observe the difference between the command ran above and this one we have changed priority from 0 to 4
\$ ionice -c0 -n4 -p2	ionice: ignoring given class data for none class	(Note that before kernel 2.6.26 a process that has not asked for an I/O priority formally uses "None" as scheduling class , but the io scheduler will treat such processes as if it were in the best effort class.) -t option : ignore failure
\$ ionice -c0 -n4 -p2 -t		For ignoring the warning shown above we can use -t option so it will ignore failure

Thus we have successfully learnt about `ionice` command.

du

The **du** command, which is short for **disk usage** lets you retrieve information about disk space usage information in a specified directory. In order to customize the output according to the information you need, this command can be paired with the appropriate options or flags.

1. To show the estimated size of sub-directories in the current directory:

```
du
```

2. To show the estimated size of sub-directories inside a specified directory:

```
du {PATH_TO_DIRECTORY}
```

```
du [OPTION]... [FILE]...  
du [OPTION]... --files0-from=F
```

Note: This does not include an exhaustive list of options.

Short Flag	Long Flag	Description
-a	--all	Includes information for both files and directories
-c	--total	Provides a grand total at the end of the list of files/directories

Short Flag	Long Flag	Description
-d	--max-depth=N	Provides information up to N levels from the directory where the command was executed
-h	--human-readable	Displays file size in human-readable units, not in bytes
-s	--summarize	Display only the total filesize instead of a list of files/directories

ping

The **ping** (Packet Internet Groper) command is used to check the network connectivity between host and server/host. This command takes as input the IP address or the URL and sends a data packet to the specified address with the message "PING" and get a response from the server/host this time is recorded which is called latency. Ping uses ICMP(Internet Control Message Protocol) to send an ICMP echo message to the specified host if that host is available then it sends ICMP reply message. Ping is generally measured in millisecond every modern operating system has this ping pre-installed.

The basic ping syntax includes ping followed by a hostname, a name of a website, or the exact IP address.

```
ping [option] [hostname] or [IP address]
```

1. To get ping version installed on your system.

```
sudo ping -v
```

2. To check whether a remote host is up, in this case, google.com, type in your terminal:

```
ping google.com
```

3. Controlling the number of pings: Earlier we did not define the number of packets to send to the server/host by using -c option we can do so.

```
ping -c 5 google.com
```

4. Controlling the number of pings: Earlier a default sized packets were sent to a host but we can send light and heavy packet by using -s option.

```
ping -s 40 -c 5 google.com
```

5. Changing the time interval: By default ping wait for 1 sec to send next packet we can change this time by using -i option.

```
ping -i 2 google.com
```

rsync

The **rsync** command is probably one of the most used commands out there. It is used to securely copy files from one server to another over SSH.

Compared to the **scp** command, which does a similar thing, **rsync** makes the transfer a lot faster, and in case of an interruption, you could restore/resume the transfer process.

In this tutorial, I will show you how to use the **rsync** command and copy files from one server to another and also share a few useful tips!

Before you get started, you would need to have 2 Linux servers. I will be using DigitalOcean for the demo and deploy 2 Ubuntu servers.

You can use my referral link to get a free \$100 credit that you could use to deploy your virtual machines and test the guide yourself on a few DigitalOcean servers:

[DigitalOcean \\$100 Free Credit](#)

This is one of the most common causes. Essentially this is how you would copy the files from the server that you are currently on (the source server) to remote/destination server.

What you need to do is SSH to the server that is holding your files, cd to the directory that you would like to transfer over:

```
cd /var/www/html
```

And then run:

```
rsync -avz user@your-remote-server.com:/home/user/dir/
```

The above command would copy all the files and directories from the current folder on your server to your remote server.

Rundown of the command:

- **-a**: is used to specify that you want recursion and want to preserve the file permissions and etc.
- **-v**: is verbose mode, it increases the amount of information you are given during the transfer.
- **-z**: this option, rsync compresses the file data as it is sent to the destination machine, which reduces the amount of data being transmitted -- something that is useful over a slow connection.

I recommend having a look at the following website which explains the commands and the arguments very nicely:

<https://explainshell.com/explain?cmd=rsync+-avz>

In case that the SSH service on the remote server is not running on the standard 22 port, you could use **rsync** with a special SSH port:



```
rsync -avz -e 'ssh -p 1234' user@your-remote-  
server.com:/home/user/dir/
```

In some cases you might want to transfer files from your remote server to your local server, in this case, you would need to use the following syntax:

```
rsync -avz your-user@your-remote-server.com:/home/user/dir/  
/home/user/local-dir/
```

Again, in case that you have a non-standard SSH port, you can use the following command:

```
rsync -avz -e 'ssh -p 2510' your-user@your-remote-  
server.com:/home/user/dir/ /home/user/local-dir/
```

If you would like to transfer only the missing files you could use the `--ignore-existing` flag.

This is very useful for final sync in order to ensure that there are no missing files after a website or a server migration.

Basically the commands would be the same apart from the appended `--ignore-existing` flag:

```
rsync -avz --ignore-existing user@your-remote-  
server.com:/home/user/dir/
```

Using **rsync** is a great way to quickly transfer some files from one machine over to another in a secure way over SSH.

For more cool Linux networking tools, I would recommend checking out this tutorial here:

[Top 15 Linux Networking tools that you should know!](#)

Hope that this helps!

Initially posted here: [How to Transfer Files from One Linux Server to Another Using rsync](#)

dig

dig - DNS lookup utility

The **dig** is a flexible tool for interrogating DNS name servers. It performs DNS lookups and displays the answers that are returned from the name server(s) that were queried.

1. Dig is a network administration command-line tool for querying the Domain Name System.

```
dig google.com
```

2. The system will list all google.com DNS records that it finds, along with the IP addresses.

```
dig google.com ANY
```

```
dig [server] [name] [type] [q-type] [q-class] {q-opt}  
    {global-d-opt} host [@local-server] {local-d-opt}  
    [ host [@local-server] {local-d-opt} [...]]
```

```
domain      is in the Domain Name System  
q-class     is one of (in,hs,ch,...) [default: in]
```

```

        q-type    is one of
(a,any,mx,ns,soa,hinfo,axfr,txt,...) [default:a]
                (Use ixfr=version for type ixfr)
        q-opt     is one of:
-4                                (use IPv4 query transport
only)
-6                                (use IPv6 query transport
only)
-b address[#port]    (bind to source
address/port)
-c class              (specify query class)
-f filename          (batch mode)
-k keyfile            (specify tsig key file)
-m                  (enable memory usage
debugging)
-p port              (specify port number)
-q name              (specify query name)
-r                  (do not read ~/.digrc)
-t type              (specify query type)
-u                  (display times in usec
instead of msec)
-x dot-notation      (shortcut for reverse
lookups)
-y [hmac:]name:key   (specify named base64
tsig key)
        d-opt     is of the form +keyword[=value], where
keyword is:
+[no]aaflag          (Set AA flag in query
+[no]aaflag))
+[no]aaonly          (Set AA flag in query
+[no]aaflag))
+[no]additional      (Control display of
additional section)
+[no]adflag          (Set AD flag in query
(default on))
+[no]all              (Set or clear all display
flags)
+[no]answer          (Control display of
answer section)
+[no]authority       (Control display of
authority section)
+[no]badcookie       (Retry BADCOOKIE
responses)
+[no]besteffort      (Try to parse even
illegal messages)
+bufsize[=###]      (Set EDNS0 Max UDP packet

```

size)		
flag in query)	+ <i>[no]</i> cdflag	(Set checking disabled
in records)	+ <i>[no]</i> class	(Control display of class
command line -	+ <i>[no]</i> cmd	(Control display of
		global option)
packet header	+ <i>[no]</i> comments	(Control display of
comments)		and section name
the request)	+ <i>[no]</i> cookie	(Add a COOKIE option to
cryptographic	+ <i>[no]</i> crypto	(Control display of
		fields in records)
(+ <i>[no]</i> search))	+ <i>[no]</i> defname	(Use search list
	+ <i>[no]</i> dnssec	(Request DNSSEC records)
	+domain=###	(Set default domainname)
### [0..63])	+ <i>[no]</i> dscp=###]	(Set the DSCP value to
	+ <i>[no]</i> edns=###]	(Set EDNS version) [0]
	+ednsflags=###	(Set EDNS flag bits)
negotiation)	+ <i>[no]</i> ednsnegotiation	(Set EDNS version
option)	+ednsopt=###[:value]	(Send specified EDNS
options)	+noednsopt	(Clear list of +ednsopt
	+ <i>[no]</i> expandaaaa	(Expand AAAA records)
	+ <i>[no]</i> expire	(Request time to expire)
SERVFAIL)	+ <i>[no]</i> fail	(Don't try next server on
question section)	+ <i>[no]</i> header-only	(Send query without a
answers)	+ <i>[no]</i> identify	(ID responders in short
[default=on on tty])	+ <i>[no]</i> idnin	(Parse IDN names
[default=on on tty])	+ <i>[no]</i> idnout	(Convert IDN response
TC responses.)	+ <i>[no]</i> ignore	(Don't revert to TCP for
	+ <i>[no]</i> keepalive	(Request EDNS TCP

keepalive)		
	<code>+[no]keepopen</code>	(Keep the TCP socket open
between queries)		
	<code>+[no]mapped</code>	(Allow mapped IPv4 over
IPv6)		
	<code>+[no]multiline</code>	(Print records in an
expanded format)		
	<code>+ndots=###</code>	(Set search <i>NDOTS</i> value)
	<code>+[no]nsid</code>	(Request Name Server ID)
	<code>+[no]nssearch</code>	(Search all authoritative
nameservers)		
	<code>+[no]onesoa</code>	(AXFR prints only one soa
record)		
	<code>+[no]opcode=###</code>	(Set the opcode of the
request)		
	<code>+padding=###</code>	(Set padding block size
[0])		
	<code>+[no]qr</code>	(Print question before
sending)		
	<code>+[no]question</code>	(Control display of
question section)		
	<code>+[no]raflag</code>	(Set RA flag in query
(+[no]raflag))		
	<code>+[no]rdflag</code>	(Recursive mode
(+[no]recurse))		
	<code>+[no]recurse</code>	(Recursive mode
(+[no]rdflag))		
	<code>+retry=###</code>	(Set number of UDP
retries) [2]		
	<code>+[no]rrcomments</code>	(Control display of per-
record comments)		
	<code>+[no]search</code>	(Set whether to use
searchlist)		
	<code>+[no]short</code>	(Display nothing except
short		
		form of answers - global
option)		
	<code>+[no]showsearch</code>	(Search with intermediate
results)		
	<code>+[no]split=##</code>	(Split hex/base64 fields
into chunks)		
	<code>+[no]stats</code>	(Control display of
statistics)		
	<code>+subnet=addr</code>	(Set edns-client-subnet
option)		
	<code>+[no]tcflag</code>	(Set TC flag in query


```

(+[no]tcflag))
    +[no]tcp                (TCP mode (+[no]vc))
    +timeout=###           (Set query timeout) [5]
    +[no]trace              (Trace delegation down
from root [+dnssec])
    +tries=###             (Set number of UDP
attempts) [3]
    +[no]ttlid              (Control display of ttls
in records)
    +[no]ttlunits           (Display TTLs in human-
readable units)
    +[no]unexpected         (Print replies from
unexpected sources
                        default=off)
    +[no]unknownformat      (Print RDATA in RFC 3597
"unknown" format)
    +[no]vc                 (TCP mode (+[no]tcp))
    +[no]yaml               (Present the results as
YAML)
    +[no]zflag              (Set Z flag in query)
    global d-opts and servers (before host name) affect
all queries.
    local d-opts and servers (after host name) affect only
that lookup.
    -h                      (print help and exit)
    -v                      (print version and exit)

```

whois

The **whois** command in Linux to find out information about a domain, such as the owner of the domain, the owner's contact information, and the nameservers that the domain is using.

1. Performs a whois query for the domain name:

```
whois {Domain_name}
```

2. -H option omits the lengthy legal disclaimers that many domain registries deliver along with the domain information.

```
whois -H {Domain_name}
```

```
whois [ -h HOST ] [ -p PORT ] [ -aCFH\LMmrRSVx ] [ -g  
SOURCE:FIRST-LAST ]  
      [ -i ATTR ] [ -S SOURCE ] [ -T TYPE ] object
```

```
whois -t TYPE
```

```
whois -v TYPE
```

Flag	Description
-h HOST, --host HOST	Connect to HOST.
-H	Do not display the legal disclaimers some registries like to show you.
-p, --port PORT	Connect to PORT.
--verbose	Be verbose.
--help	Display online help.
--version	Display client version information. Other options are flags understood by whois.ripe.net and some other RIPE-like servers.
-a	Also search all the mirrored databases.
-b	Return brief IP address ranges with abuse contact.
-B	Disable object filtering (<i>show the e-mail addresses</i>)
-c	Return the smallest IP address range with a reference to an irt object.
-d	Return the reverse DNS delegation object too.
-g SOURCE:FIRST-LAST	Search updates from SOURCE database between FIRST and LAST update serial number. It's useful to obtain Near Real Time Mirroring stream.
-G	Disable grouping of associated objects.
-i ATTR[,ATTR]...	Search objects having associated attributes. ATTR is attribute name. Attribute value is positional OBJECT argument.
-K	Return primary key attributes only. Exception is members attribute of set object which is always returned. Another exceptions are all attributes of objects organisation, person, and role that are never returned.
-l	Return the one level less specific object.
-L	Return all levels of less specific objects.
-m	Return all one level more specific objects.
-M	Return all levels of more specific objects.
-q KEYWORD	Return list of keywords supported by server. KEYWORD can be version for server version, sources for list of source databases, or types for object types.
-r	Disable recursive look-up for contact information.

Flag	Description
-R	Disable following referrals and force showing the object from the local copy in the server.
-s SOURCE[,SOURCE]...	Request the server to search for objects mirrored from SOURCES. Sources are delimited by comma and the order is significant. Use -q sources option to obtain list of valid sources.
-t TYPE	Return the template for a object of TYPE.
-T TYPE[,TYPE]...	Restrict the search to objects of TYPE. Multiple types are separated by a comma.
-v TYPE	Return the verbose template for a object of TYPE.
-x	Search for only exact match on network address prefix.

ssh

The **ssh** command in Linux stands for "Secure Shell". It is a protocol used to securely connect to a remote server/system. ssh is more secure in the sense that it transfers the data in encrypted form between the host and the client. ssh runs at TCP/IP port 22.

1. Use a Different Port Number for SSH Connection:

```
ssh test.server.com -p 3322
```

2. -i ssh to remote server using a private key?

```
ssh -i private.key user_name@host
```

3. -l ssh specifying a different username

```
ssh -l alternative-username sample.ssh.com
```

```
ssh user_name@host(IP/Domain_Name)
```

```
ssh -i private.key user_name@host
```

```
ssh sample.ssh.com ls /tmp/doc
```

Flag	Description
-1	Forces ssh to use protocol SSH-1 only.
-2	Forces ssh to use protocol SSH-2 only.
-4	Allows IPv4 addresses only.
-A	Authentication agent connection forwarding is enabled..
-a	Authentication agent connection forwarding is disabled.
-B bind_interface	Bind to the address of bind_interface before attempting to connect to the destination host. This is only useful on systems with more than one address.
-b bind_address	Use bind_address on the local machine as the source address of the connection. Only useful on systems with more than one address.
-C	Compresses all data (including stdin, stdout, stderr, and data for forwarded X11 and TCP connections) for a faster transfer of data.
-c cipher_spec	Selects the cipher specification for encrypting the session.
-D [bind_address:]port	Dynamic application-level port forwarding. This allocates a socket to listen to port on the local side. When a connection is made to this port, the connection is forwarded over the secure channel, and the application protocol is then used to determine where to connect to from the remote machine.
-E log_file	Append debug logs instead of standard error.

Flag	Description
<code>-e escape_char</code>	Sets the escape character for sessions with a pty (default: '~'). The escape character is only recognized at the beginning of a line. The escape character followed by a dot ('.') closes the connection; followed by control-Z suspends the connection; and followed by itself sends the escape character once. Setting the character to "none" disables any escapes and makes the session fully transparent.
<code>-F configfile</code>	Specifies a per-user configuration file. The default for the per-user configuration file is ~/.ssh/config.
<code>-f</code>	Requests ssh to go to background just before command execution.
<code>-G</code>	Causes ssh to print its configuration after evaluating Host and Match blocks and exit.
<code>-g</code>	Allows remote hosts to connect to local forwarded ports.
<code>-I pkcs11</code>	Specify the PKCS#11 shared library ssh should use to communicate with a PKCS#11 token providing keys.
<code>-i identity_file</code>	A file from which the identity key (private key) for public key authentication is read.
<code>-J [user@]host[:port]</code>	Connect to the target host by first making a ssh connection to the pjump host[(/iam/jump-host) and then establishing a TCP forwarding to the ultimate destination from there.
<code>-K</code>	Enables GSSAPI-based authentication and forwarding (delegation) of GSSAPI credentials to the server.
<code>-k</code>	Disables forwarding (delegation) of GSSAPI credentials to the server.

Flag

`-L`
`[bind_address:]port:host:hostport, -`
`L`
`[bind_address:]port:remote_socket, -`
`L local_socket:host:hostport, -L`
`local_socket:remote_socket`

`-l login_name`

`-M`

`-m mac_spec`

`-N`

`-n`

Description

Specifies that connections to the given TCP port or Unix socket on the local (client) host are to be forwarded to the given host and port, or Unix socket, on the remote side. This works by allocating a socket to listen to either a TCP port on the local side, optionally bound to the specified `bind_address`, or to a Unix socket. Whenever a connection is made to the local port or socket, the connection is forwarded over the secure channel, and a connection is made to either host port `hostport`, or the Unix socket `remote_socket`, from the remote machine.

Specifies the user to log in as on the remote machine.

Places the ssh client into “master” mode for connection sharing. Multiple `-M` options places ssh into “master” mode but with confirmation required using `ssh-askpass` before each operation that changes the multiplexing state (e.g. opening a new session).

A comma-separated list of MAC (message authentication code) algorithms, specified in order of preference.

Do not execute a remote command. This is useful for just forwarding ports.

Prevents reading from stdin.

Flag	Description
<code>-O ctl_cmd</code>	Control an active connection multiplexing master process. When the <code>-O</code> option is specified, the <code>ctl_cmd</code> argument is interpreted and passed to the master process. Valid commands are: "check" (check that the master process is running), "forward" (request forwardings without command execution), "cancel" (cancel forwardings), "exit" (request the master to exit), and "stop" (request the master to stop accepting further multiplexing requests).
<code>-o</code>	Can be used to give options in the format used in the configuration file. This is useful for specifying options for which there is no separate command-line flag.
<code>-p, --port PORT</code>	Port to connect to on the remote host.
<code>-Q query_option</code>	Queries ssh for the algorithms supported for the specified version 2. The available features are: cipher (supported symmetric ciphers), cipher-auth (supported symmetric ciphers that support authenticated encryption), help (supported query terms for use with the <code>-Q</code> flag), mac (supported message integrity codes), kex (key exchange algorithms), kex-gss (GSSAPI key exchange algorithms), key (keytypes), key-cert (certificate key types), key-plain (non-certificate key types), key-sig (all keytypes and signature algorithms), protocol-version (supported SSH protocol versions), and sig (supported signature algorithms). Alternatively, any keyword from <code>ssh_config(5)</code> or <code>sshd_config(5)</code> that takes an algorithm list may be used as an alias for the corresponding query_option.
<code>-q</code>	Quiet mode. Causes most warning and diagnostic messages to be suppressed.

Flag	Description
-R [bind_address:]port:host:hostport, -R [bind_address:]port:local_socket, - R remote_socket:host:hostport, -R remote_socket:local_socket, -R [bind_address:]port	Specifies that connections to the given TCP port or Unix socket on the remote (server) host are to be forwarded to the local side.
-S ctl_path	Specifies the location of a control socket for connection sharing, or the string "none" to disable connection sharing.
-S	May be used to request invocation of a subsystem on the remote system. Subsystems facilitate the use of SSH as a secure transport for other applications (e.g. sftp(1)). The subsystem is specified as the remote command.
-T	Disable pseudo-terminal allocation.
-t	Force pseudo-terminal allocation. This can be used to execute arbitrary screen-based programs on a remote machine, which can be very useful, e.g. when implementing menu services. Multiple -t options force tty allocation, even if ssh has no local tty.
-V	Display the version number.
-v	Verbose mode. It echoes everything it is doing while establishing a connection. It is very useful in the debugging of connection failures.
-W host:port	Requests that standard input and output on the client be forwarded to host on port over the secure channel. Implies -N, -T, ExitOnForwardFailure and ClearAllForwardings, though these can be overridden in the configuration file or using -o command line options.

Flag**Description**

-w `local_tun[remote_tun]`

Requests tunnel device forwarding with the specified tun devices between the client (`local_tun`) and the server (`remote_tun`). The devices may be specified by numerical ID or the keyword “any”, which uses the next available tunnel device. If `remote_tun` is not specified, it defaults to “any”. If the Tunnel directive is unset, it will be set to the default tunnel mode, which is “point-to-point”. If a different Tunnel forwarding mode is desired, then it should be specified before -w.

-X

Enables X11 forwarding (GUI Forwarding).

-x

Disables X11 forwarding (GUI Forwarding).

-Y

Enables trusted X11 Forwarding.

-y

Send log information using the syslog system module. By default this information is sent to stderr.

awk

Awk is a general-purpose scripting language designed for advanced text processing. It is mostly used as a reporting and analysis tool.

WHAT CAN WE DO WITH AWK?

1. AWK Operations: (a) Scans a file line by line (b) Splits each input line into fields (c) Compares input line/fields to pattern (d) Performs action(s) on matched lines
2. Useful For: (a) Transform data files (b) Produce formatted reports
3. Programming Constructs: (a) Format output lines (b) Arithmetic and string operations (c) Conditionals and loops

Syntax

```
awk options 'selection _criteria {action }' input-file >  
output-file
```

Example

Consider the following text file as the input file for below example:

```
```\n$cat > employee.txt\n```\n```\n\najay manager account 45000\nsunil clerk account 25000\nvarun manager sales 50000\namit manager account 47000\ntarun peon sales 15000\n```\n
```

1. Default behavior of Awk: By default Awk prints every line of data from the specified file.

```
$ awk '{print}' employee.txt
```

```
ajay manager account 45000\nsunil clerk account 25000\nvarun manager sales 50000\namit manager account 47000\ntarun peon sales 15000
```

In the above example, no pattern is given. So the actions are applicable to all the lines. Action print without any argument prints the whole line by default, so it prints all the lines of the file without failure.

2. Print the lines which match the given pattern.

```
awk '/manager/ {print}' employee.txt
```

```
ajay manager account 45000\nvarun manager sales 50000\namit manager account 47000
```

In the above example, the awk command prints all the line which matches with the 'manager'.

3. Splitting a Line Into Fields : For each record i.e line, the awk command splits the record delimited by whitespace character by default and stores it in the \$n variables. If the line has 4 words, it will be stored in \$1, \$2, \$3 and \$4 respectively. Also, \$0 represents the whole line.

```
$ awk '{print $1,$4}' employee.txt
```

```
ajay 45000
sunil 25000
varun 50000
amit 47000
tarun 15000
```

### Built-In Variables In Awk

Awk's built-in variables include the field variables—\$1, \$2, \$3, and so on (\$0 is the entire line) — that break a line of text into individual words or pieces called fields.

NR: NR command keeps a current count of the number of input records. Remember that records are usually lines. Awk command performs the pattern/action statements once for each record in a file. NF: NF command keeps a count of the number of fields within the current input record. FS: FS command contains the field separator character which is used to divide fields on the input line. The default is “white space”, meaning space and tab characters. FS can be reassigned to another character (typically in BEGIN) to change the field separator. RS: RS command stores the current record separator character. Since, by default, an input line is the input record, the default record separator character is a newline. OFS: OFS command stores the output field separator, which separates the fields when Awk prints them. The default is a blank space. Whenever print has several parameters separated with commas, it will print the value of OFS in between each parameter. ORS: ORS command stores the output record separator, which separates the output lines when Awk prints them. The default is a newline character. print automatically outputs the contents of ORS at the end of whatever it is given to print.

## crontab

**crontab** is used to maintain crontab files for individual users (Vixie Cron)

crontab is the program used to install, uninstall or list the tables used to drive the cron(8) daemon in Vixie Cron. Each user can have their own crontab, and though these are files in `/var/spool/cron/crontabs`, they are not intended to be edited directly.

```
crontab [-u user] file
crontab [-u user] [-i] { -e | -l | -r }
```

1. The `-l` option causes the current crontab to be displayed on standard output.

```
crontab -l
```

2. The `-r` option causes the current crontab to be removed.

```
crontab -r
```

3. The `-e` option is used to edit the current crontab using the editor specified by the `VISUAL` or `EDITOR` environment variables. After you exit from the editor, the modified crontab will be installed automatically. If neither of the environment variables is defined, then the default editor `/usr/bin/editor` is used.

```
crontab -e
```

4. You can specify the user you want to edit the crontab for. Every user has its own

crontab. Assume you have a **www-data** user, which is in fact the user Apache is default running as. If you want to edit the crontab for this user you can run the following command

```
crontab -u www-data -e
```

Run below command to view the complete guide to **crontab** command.

```
man crontab
```



## xargs

**xargs** is used to build and execute command lines from standard input

Some commands like `grep` can accept input as parameters, but some commands accept arguments, this is place where `xargs` came into picture.

```
xargs [options] [command [initial-arguments]]
```

```
-0, --null
```

Input items are terminated by a null character instead of by whitespace, and the quotes and backslash are not special (every character is taken literal-ly). Disables the end of file string, which is treated like any other argument. Useful when input items might contain white space, quote marks, or back-slashes.

```
-a file, --arg-file=file
```

Read items from file instead of standard input. If you use this option, `stdin` remains unchanged when commands are run. Otherwise, `stdin` is redirected from `/dev/null`.

```
-o, --open-tty
```

Reopen `stdin` as `/dev/tty` in the child process before executing the command. This is useful if you want `xargs` to run an interactive application.

```
--delimiter=delim, -d delim
```

Input items are terminated by the specified character. The specified delimiter may be a single character, a C-style character escape such as `\n`, or an octal or hexadecimal escape code. Octal and hexadecimal escape codes are understood as for the `printf` command. Multibyte characters are not supported. When processing the input, quotes and backslash are not special; every character in the input is taken literally. The `-d` option disables any end-of-file string, which is treated like any other argument. You can use this option when the input consists of simply newline-separated items, although it is almost always better to design your program to use `--null` where this is possible.

```
-p, --interactive
```

Prompt the user about whether to run each command line and read a line from the terminal. Only run the command line if the response starts with `y'` or `Y'`. Implies `-t`.

```
find /tmp -name core -type f -print | xargs /bin/rm -f
```

Find files named `core` in or below the directory `/tmp` and delete them. Note that this will work incorrectly if there are any filenames containing newlines or spaces.

```
find /tmp -name core -type f -print0 | xargs -0 /bin/rm -f
```

Find files named `core` in or below the directory `/tmp` and delete them, processing filenames in such a way that file or directory names containing spaces or newlines are correctly handled.

```
find /tmp -depth -name core -type f -delete
```

Find files named `core` in or below the directory `/tmp` and delete them, but more efficiently than in the previous example (because we avoid the need to use `fork(2)` and `exec(2)` to launch `rm` and we don't need the extra `xargs` process).

```
cut -d: -f1 < /etc/passwd | sort | xargs echo
```

Generates a compact listing of all the users on the system.

Run below command to view the complete guide to **xargs** command.

```
man xargs
```

## nohup

When a shell exits (maybe while logging out of an SSH session), the HUP ('hang up') signal is sent to all of its child processes, causing them to terminate. If you require a long-running process to continue after exiting shell, you'll need the **nohup** command. Prefixing any command with **nohup** causes the command to become *immune* to HUP signals. Additionally, STDIN is being ignored and all output gets redirected to local file **./nohup.out**.

1. Applying nohup to a long-running debian upgrade:

```
nohup apt-get -y upgrade
```

```
nohup COMMAND [ARG]...
nohup OPTION
```

## pstree

The **pstree** command is similar to **ps**, but instead of listing the running processes, it shows them as a tree. The tree-like format is sometimes more suitable way to display the processes hierarchy which is a much simpler way to visualize running processes. The root of the tree is either init or the process with the given pid.

1. To display a hierarchical tree structure of all running processes:

```
pstree
```

2. To display a tree with the given process as the root of the tree:

```
pstree [pid]
```

3. To show only those processes that have been started by a user:

```
pstree [USER]
```

4. To show the parent processes of the given process:

```
pstree -s [PID]
```

5. To view the output one page at a time, pipe it to the **less** command:

```
pstree | less
```

ps [OPTIONS] [USER or PID]

Short Flag	Long Flag	Description
-a	--arguments	Show command line arguments
-A	--ascii	use ASCII line drawing characters
-c	--compact	Don't compact identical subtrees
-h	--highlight-all	Highlight current process and its ancestors
-H PID	--highlight-pid=PID	highlight this process and its ancestors
-g	--show-pgids	show process group ids; implies -C
-G	--vt100	use VT100 line drawing characters
-l	--long	Don't truncate long lines
-n	--numeric-sort	Sort output by PID
-N type	--ns-sort=type	Sort by namespace type (cgroup, ipc, mnt, net, pid, user, uts)
-p	--show-pids	show PIDs; implies -c
-s	--show-parents	Show parents of the selected process
-S	--ns-changes	show namespace transitions
-t	--thread-names	Show full thread names
-T	--hide-threads	Hide threads, show only processes
-u	--uid-changes	Show uid transitions
-U	--unicode	Use UTF-8 (Unicode) line drawing characters
-V	--version	Display version information
-Z	--security-context	Show SELinux security contexts

## tree

The **tree** command in Linux recursively lists directories as tree structures. Each listing is indented according to its depth relative to root of the tree.

1. Show a tree representation of the current directory.

```
tree
```

2. -L NUMBER limits the depth of recursion to avoid display very deep trees.

```
tree -L 2 /
```

```
tree [-acdfghilnpqrstuvxACDFQNSUX] [-L level [-R]] [-H
baseHREF] [-T title]
 [-o filename] [--nolinks] [-P pattern] [-I pattern] [-
-inodes]
 [--device] [--noreport] [--dirsfirst] [--version] [--
help] [--filelimit #]
 [--si] [--prune] [--du] [--timefmt format] [--
matchdirs] [--from-file]
 [--] [directory ...]
```

Flag	Description
-a	Print all files, including hidden ones.

Flag	Description
<b>-d</b>	Only list directories.
<b>-l</b>	Follow symbolic links into directories.
<b>-f</b>	Print the full path to each listing, not just its basename.
<b>-x</b>	Do not move across file-systems.
<b>-L #</b>	Limit recursion depth to #.
<b>-P REGEX</b>	Recurse, but only list files that match the REGEX.
<b>-I REGEX</b>	Recurse, but do not list files that match the REGEX.
<b>--ignore-case</b>	Ignore case while pattern-matching.
<b>--prune</b>	Prune empty directories from output.
<b>--filelimit #</b>	Omit directories that contain more than # files.
<b>-o FILE</b>	Redirect STDOUT output to FILE.
<b>-i</b>	Do not output indentation.



## whereis

The **whereis** command is used to find the location of source/binary file of a command and manuals sections for a specified file in Linux system. If we compare **whereis** command with find command they will appear similar to each other as both can be used for the same purposes but **whereis** command produces the result more accurately by consuming less time comparatively.

### Points to be kept on mind while using the whereis command:

Since the **whereis** command uses chdir(change directory 2V) to give you the result in the fastest possible way, the pathnames given with the -M, -S, or -B must be full and well-defined i.e. they must begin with a / and should be a valid path that exist in the system's directories, else it exits without any valid result. **whereis** command has a hard-coded(code which is not dynamic and changes with specification) path, so you may not always find what you're looking for.

```
whereis [options] [filename]
```

-b : This option is used when we only want to search for binaries. -m : This option is used when we only want to search for manual sections. -s : This option is used when we only want to search for source files. -u: This option search for unusual entries. A source file or a binary file is said to be unusual if it does not have any existence in system as per [-bmsu] described along with "-u". Thus `whereis -m -u \*` asks for those files in the current directory which have unusual entries.

-B : This option is used to change or otherwise limit the places where whereis searches for binaries. -M : This option is used to change or otherwise limit the places where whereis searches for manual sections. -S : This option is used to change or otherwise limit the places where whereis searches for source files.

-f : This option simply terminate the last directory list and signals the start of file names. This must be used when any of the -B, -M, or -S options are used. -V: Displays

version information and exit. -h: Displays the help and exit.

## printf

This command lets you print the value of a variable by formatting it using rules. It is pretty similar to the printf in C language.

```
$printf [-v variable_name] format [arguments]
```

OPTION	Description
FORMAT	FORMAT controls the output, and defines the way that the ARGUMENTs will be expressed in the output
ARGUMENT	An ARGUMENT will be inserted into the formatted output according to the definition of FORMAT
--help	Display help and exit
--version	Output version information and exit

The anatomy of the FORMAT string can be extracted into three different parts,

- *ordinary characters*, which are copied exactly the same characters as were used originally to the output.
- *interpreted character sequences*, which are escaped with a backslash ("").
- *conversion specifications*, this one will define the way the ARGUMENTs will be expressed as part of the output.

You can see those parts in this example,

```
printf " %s is where over %d million developers shape \"the
future of software.\" " " Github 65
```

The output:

```
Github is where over 65 million developers shape "the future
of software."
```

There are two conversion specifications `%s` and `%d`, and there are two escaped characters which are the opening and closing double-quotes wrapping the words of *the future of software*. Other than that are the ordinary characters.

Each conversion specification begins with a `%` and ends with a **conversion character**. Between the `%` and the **conversion character** there may be, in order:

- A minus sign. This tells `printf` to left-adjust the conversion of the argument
- An integer that specifies field width; `printf` prints a conversion of ARGUMENT *number* in a field at least number characters wide. If necessary it will be padded on the left (or right, if left-adjustment is called for) to make up the field width
- . A period, which separates the field width from the precision
- An integer, the precision, which specifies the maximum number of characters *number* to be printed from a string, or the number of digits after the decimal point of a floating-point value, or the minimum number of digits for an integer
- h** or **l** These differentiate between a short and a long integer, respectively, and are generally only needed for computer programming

The conversion characters tell `printf` what kind of argument to print out, are as follows:

Conversion char	Argument type
<b>s</b>	A string
<b>c</b>	An integer, expressed as a character corresponds ASCII code
<b>d, i</b>	An integer as a decimal number
<b>o</b>	An integer as an unsigned octal number
<b>x, X</b>	An integer as an unsigned hexadecimal number
<b>u</b>	An integer as an unsigned decimal number

Conversion char	Argument type
<code>f</code>	A floating-point number with a default precision of 6
<code>e, E</code>	A floating-point number in scientific notation
<code>p</code>	A memory address pointer
<code>%</code>	No conversion

Here is the list of some examples of the `printf` output the ARGUMENT. we can put any word but in this one we put a 'linuxcommand' word and enclosed it with quotes so we can see easier the position related to the whitespaces.

FORMAT string	ARGUMENT string	Output string
<code>"%s"</code>	<code>"linuxcommand"</code>	<code>"linuxcommand"</code>
<code>"%5s"</code>	<code>"linuxcommand"</code>	<code>"linuxcommand"</code>
<code>"%.5s"</code>	<code>"linuxcommand"</code>	<code>"linux"</code>
<code>"%-8s"</code>	<code>"linuxcommand"</code>	<code>"linuxcommand"</code>
<code>"%-15s"</code>	<code>"linuxcommand"</code>	<code>"linuxcommand "</code>
<code>"%12.5s"</code>	<code>"linuxcommand"</code>	<code>" linux"</code>
<code>"%-12.5"</code>	<code>"linuxcommand"</code>	<code>"linux "</code>
<code>"%-12.4"</code>	<code>"linuxcommand"</code>	<code>"linu "</code>

Notes:

- `printf` requires the number of conversion strings to match the number of ARGUMENTs
- `printf` maps the conversion strings one-to-one, and expects to find exactly one ARGUMENT for each conversion string
- Conversion strings are always interpreted from left to right.

Here's the example:

The input

```
printf "We know %f is %s %d" 12.07 "larger than" 12
```

The output:

```
We know 12.070000 is larger than 12
```

The example above shows 3 arguments, *12.07*, *larger than*, and *12*. Each of them

interpreted from left to right one-to-one with the given 3 conversion strings (%f, %d, %s).

Character sequences which are interpreted as special characters by `printf`:

Escaped char	Description
<code>\a</code>	issues an alert (plays a bell). Usually ASCII BEL characters
<code>\b</code>	prints a backspace
<code>\c</code>	instructs <code>printf</code> to produce no further output
<code>\e</code>	prints an escape character (ASCII code 27)
<code>\f</code>	prints a form feed
<code>\n</code>	prints a newline
<code>\r</code>	prints a carriage return
<code>\t</code>	prints a horizontal tab
<code>\v</code>	prints a vertical tab
<code>\"</code>	prints a double-quote (")
<code>\\</code>	prints a backslash ( )
<code>\NNN</code>	prints a byte with octal value <code>NNN</code> (1 to 3 digits)
<code>\xHH</code>	prints a byte with hexadecimal value <code>HH</code> (1 to 2 digits)
<code>\uHHHH</code>	prints the unicode character with hexadecimal value <code>HHHH</code> (4 digits)
<code>\UHHHHHHHH</code>	prints the unicode character with hexadecimal value <code>HHHHHHHH</code> (8 digits)
<code>%b</code>	prints ARGUMENT as a string with "\" escapes interpreted as listed above, with the exception that octal escapes take the form <code>\0</code> or <code>\0NN</code>

The format specifiers usually used with `printf` are stated in the examples below:

- `%s`

```
$printf "%s\n" "Printf command documentation!"
```

This will print `Printf command documentation!` in the shell.

- `%b` - Prints arguments by expanding backslash escape sequences.

- **%q** - Prints arguments in a shell-quoted format which is reusable as input.
- **%d** , **%i** - Prints arguments in the format of signed decimal integers.
- **%u** - Prints arguments in the format of unsigned decimal integers.
- **%o** - Prints arguments in the format of unsigned octal(base 8) integers.
- **%x** , **%X** - Prints arguments in the format of unsigned hexadecimal(base 16) integers. %x prints lower-case letters and %X prints upper-case letters.
- **%e** , **%E** - Prints arguments in the format of floating-point numbers in exponential notation. %e prints lower-case letters and %E prints upper-case.
- **%a** , **%A** - Prints arguments in the format of floating-point numbers in hexadecimal(base 16) fractional notation. %a prints lower-case letters and %A prints upper-case.
- **%g** , **%G** - Prints arguments in the format of floating-point numbers in normal or exponential notation, whichever is more appropriate for the given value and precision. %g prints lower-case letters and %G prints upper-case.
- **%c** - Prints arguments as single characters.
- **%f** - Prints arguments as floating-point numbers.
- **%s** - Prints arguments as strings.
- **%%** - Prints a "%" symbol.

### More Examples:

The input:

```
printf 'Hello\nyoung\nman!'
```

The output:


```
hello
young
man!
```

The two **\n** break the sentence into 3 parts of words.

The input:

```
printf "%f\n" 2.5 5.75
```

The output



```
2.500000
5.750000
```

The `%f` specifier combined with the `\n` interpreted the two arguments in the form of floating point in the separated new lines.



## cut

The **cut** command lets you remove sections from each line of files. Print selected parts of lines from each FILE to standard output. With no FILE, or when FILE is -, read standard input.

1. Selecting specific fields in a file

```
cut -d "delimiter" -f (field number) file.txt
```

2. Selecting specific characters:

```
cut -c [(k)-(n)/(k),(n)/(n)] filename
```

Here, **k** denotes the starting position of the character and **n** denotes the ending position of the character in each line, if *k* and *n* are separated by "-" otherwise they are only the position of character in each line from the file taken as an input.

3. Selecting specific bytes:

```
cut -b 1,2,3 filename //select bytes 1,2
and 3
cut -b 1-4 filename //select bytes 1
through 4
cut -b 1- filename //select bytes
1 through the end of file
cut -b -4 filename //select bytes
from the beginning till the 4th byte
```

**Tabs and backspaces** are treated like as a character of 1 byte.

cut OPTION... [FILE]...

Short Flag	Long Flag	Description
-b	--bytes=LIST	select only these bytes
-c	--characters=LIST	select only these characters
-d	--delimiter=DELIM	use DELIM instead of TAB for field delimiter
-f	--fields	select only these fields; also print any line that contains no delimiter character, unless the -s option is specified
-s	--only-delimited	do not print lines not containing delimiters
-z	--zero-terminated	line delimiter is NUL, not newline

## sed

**sed** command stands for stream editor. A stream editor is used to perform basic text transformations on an input stream (a file or input from a pipeline). For instance, it can perform lot's of functions on files like searching, find and replace, insertion or deletion. While in some ways it is similar to an editor which permits scripted edits (such as **ed**), **sed** works by making only one pass over the input(s), and is consequently more efficient. But it is sed's ability to filter text in a pipeline that particularly distinguishes it from other types of editors.

The most common use of **sed** command is for a substitution or for find and replace. By using sed you can edit files even without opening it, which is a much quicker way to find and replace something in the file. It supports basic and extended regular expressions that allow you to match complex patterns. Most Linux distributions come with GNU and **sed** is pre-installed by default.

### 1. To Find and Replace String with **sed**

```
sed -i 's/{search_regex}/{replace_value}/g' input-file
```

### 2. For Recursive Find and Replace (*along with **find***)

Sometimes you may want to recursively search directories for files containing a string and replace the string in all files. This can be done using commands such as find to recursively find files in the directory and piping the file names to **sed**. The following command will recursively search for files in the current working directory and pass the file names to **sed**. It will recursively search for files in the current working directory and pass the file names to **sed**.

```
find . -type f -exec sed -i
's/{search_regex}/{replace_value}/g' {} +
```

```
sed [OPTION]... {script-only-if-no-other-script} [INPUT-FILE]...
```

- **OPTION** - sed options in-place, silent, follow-symlinks, line-length, null-data ...etc.
- **{script-only-if-no-other-script}** - Add the script to command if available.
- **INPUT-FILE** - Input Stream, A file or input from a pipeline.

If no option is given, then the first non-option argument is taken as the sed script to interpret. All remaining arguments are names of input files; if no input files are specified, then the standard input is read.

GNU sed home page: <http://www.gnu.org/software/sed/>

Short Flag	Long Flag	Description
<b>-i[SUFFIX]</b>	<b>--in-place[=SUFFIX]</b>	Edit files in place (makes backup if SUFFIX supplied).
<b>-n</b>	<b>--quiet, --silent</b>	Suppress automatic printing of pattern space.
<b>-e script</b>	<b>--expression=script</b>	Add the script to the commands to be executed.
<b>-f script-file</b>	<b>--file=script-file</b>	Add the contents of script-file to the commands to be executed.
<b>-l N</b>	<b>--line-length=N</b>	Specify the desired line-wrap length for the <b>l</b> command.
<b>-r</b>	<b>--regexp-extended</b>	Use extended regular expressions in the script.
<b>-s</b>	<b>--separate</b>	Consider files as separate rather than as a single continuous long stream.
<b>-u</b>	<b>--unbuffered</b>	Load minimal amounts of data from the input files and flush the output buffers more often.
<b>-z</b>	<b>--null-data</b>	Separate lines by NULL characters.

It may seem complicated and complex at first, but searching and replacing text in files with sed is very simple.

To find out more: <https://www.gnu.org/software/sed/manual/sed.html>

## vim

The [vim](#) is a text editor for Unix that comes with Linux, BSD, and macOS. It is known to be fast and powerful, partly because it is a small program that can run in a terminal (although it has a graphical interface). Vim text editor is developed by [Bram Moolenaar](#). It supports most file types and the vim editor is also known as a programmer's editor. It is mainly because it can be managed entirely without menus or a mouse with a keyboard.

**Note:** Do not confuse [vim](#) with [vi](#). [vi](#), which stands for "Visual", is a text editor that was developed by [Bill Joy](#) in 1976. [vim](#) stands for "Vi Improved", and is an improved clone of the [vi](#) editor.

## vim

### How to exit vim editor?

The most searched question about vim editor looks very funny but it's true that the new user gets stuck at the very beginning when using vim editor.

The command to save the file and exit vim editor: `:wq`

The command to exit vim editor without saving the file: `:q!`

### Fun reading:

Here's a [survey](#) for the same question, look at this and do not think to quit the vim editor.

First check if vim is already installed or not, enter the following command:

```
vim --version
```

If it is already installed it will show its version, else we can run the below commands for the installations:

On Ubuntu/Debian:

```
sudo apt-get install vim
```

On CentOS/Fedora:

```
sudo yum install vim
```

If you want to use advanced features on CentOS/Fedora, you'll need to install enhanced vim editor, to do this run the following command:

```
sudo yum install -y vim-enhanced
```

```
vim [FILE_PATH/FILE_NAME]
```

1. To open the file named "demo.txt" from your current directory:

```
vim demo.txt
```

2. To open the file in a specific directory:

```
vim {File_Path/filename}
```

3. To open the file starting on a specific line in the file:

```
vim {File_Path/filename} +LINE_NUMBER
```

There are some arguments as to how many modes that vim has, but the modes you're most likely to use are **command mode** and **insert mode**. These [modes](#) will allow you to do just about anything you need, including creating your document, saving your document, and doing advanced editing, including taking advantage of search and replace functions.

## vim

1. Open a new or existing file with **vim filename**.
2. Type **i** to switch into insert mode so that you can start editing the file.
3. Enter or modify the text of your file.
4. When you're done, press the **Esc** key to exit insert mode and back to command mode.
5. Type **:w** or **:wq** to save the file or save and exit from the file respectively.

In this interactive tutorial, you will learn the different ways to use the **vim** command:

[The Open vim Tutorial](#)

Flags/Options	Description
<b>-e</b>	Start in Ex mode (see <a href="#">Ex-mode</a> )
<b>-R</b>	Start in read-only mode
<b>-R</b>	Start in read-only mode
<b>-g</b>	Start the <a href="#">GUI</a>
<b>-eg</b>	Start the GUI in Ex mode
<b>-Z</b>	Like "vim", but in restricted mode
<b>-d</b>	Start in diff mode <a href="#">diff-mode</a>
<b>-h</b>	Give usage (help) message and exit



**Flags/Options****Description****+NUMBER**

Open a file and place the cursor on the line number specified by NUMBER

vim can not be learned in a single day, use in day-to-day tasks to get hands-on in vim editor.

To learn more about **vim** follow the given article:

[Article By Daniel Miessler](#)

## chown

The **chown** command makes it possible to change the ownership of a file or directory. Users and groups are fundamental in Linux, with **chown** you can change the owner of a file or directory. It's also possible to change ownership on folders recursively

1. Change the owner of a file

```
chown user file.txt
```

2. Change the group of a file

```
chown :group file.txt
```

3. Change the user and group in one line

```
chown user:group file.txt
```

4. Change to ownership on a folder recursively

```
chown -R user:group folder
```

```
chown [-OPTION] [DIRECTORY_PATH]
```

## find

The **find** command lets you **search for files in a directory hierarchy**

- Search a file with specific name.
- Search a file with pattern
- Search for empty files and directories.

1. Search a file with specific name:

```
find ./directory1 -name sample.txt
```

2. Search a file with pattern:

```
find ./directory1 -name '*.txt'
```

3. To find all directories whose name is test in / directory.

```
find / -type d -name test
```

4. Searching empty files in current directory

```
find . -size 0k
```

```
find [options] [paths] [expression]
```

## In Simple words

```
find [where to start searching from]
 [expression determines what to find] [-options] [what to
 find]
```

Commonly-used primaries include:

- **name** pattern - tests whether the file name matches the shell-glob pattern given.
- **type** type - tests whether the file is a given type. Unix file types accepted include:

options	Description
<b>b</b>	block device (buffered)
<b>d</b>	directory
<b>f</b>	regular file
<b>l</b>	Symbolic link
<b>-print</b>	always returns true; prints the name of the current file plus a newline to the stdout.
<b>-mtime n</b>	find's all the files which are modified n days back.
<b>-atime n</b>	find's all the files which are accessed 50 days back.
<b>-cmin n</b>	find's all the files which are modified in the last 1 hour.
<b>-newer file</b>	find's file was modified more recently than file.
<b>-size n</b>	File uses n units of space, rounding up.

Run below command to view the complete guide to **find** command or [click here](#).

```
man find
```

## rmkdir

The **rmkdir** command is used to remove empty directories from the filesystem in Linux. The rmkdir command removes each and every directory specified in the command line only if these directories are empty.

### 1. remove directory and its ancestors

```
rmkdir -p a/b/c // is similar to 'rmkdir
a/b/c a/b a'
```

### 2. remove multiple directories

```
rmkdir a b c // removes empty
directories a,b and c
```

```
rmkdir [OPTION]... DIRECTORY...
```

Short Flag	Long Flag	Description
-	--ignore-fail-on-non-empty	ignore each failure that is solely because a directory is non-empty
-p	--parents	remove DIRECTORY and its ancestors
-d	--delimiter=DELIM	use DELIM instead of TAB for field delimiter

Short Flag	Long Flag	Description
<b>-v</b>	<b>--verbose</b>	output a diagnostic for every directory processed

lsblk

The `lsblk` command displays the block and loop devices on the system. It is especially useful when you want to format disks, write filesystems, check the filesystem and know the mount point of a device.



1. Basic usage is fairly simple - just execute 'lsblk' sans any option.

```
lsblk
```

2. Make lsblk display empty devices as well

```
lsblk -a
```

3. Make lsblk print size info in bytes

```
lsblk -b
```

4. Make lsblk print zone model for each device

```
lsblk -z
```

5. Make lsblk skip entries for slaves

```
lsblk -d
```

6. Make lsblk use ascii characters for tree formatting

```
lsblk -i
```

7. Make lsblk display info about device owner, group, and mode

```
lsblk -m
```

8. Make lsblk output select columns

```
lsblk -o NAME,SIZE
```



```
lsblk [options] [<device> ...]
```

## lsblk

On running `lsblk` with no flags or command-line arguments, it writes general disk information to the STDOUT. Here is a table that interpretes that information:

Column Name	Meaning	Interpretation
NAME	Name of the device.	Shows name of the device.
RM	Removable.	Shows 1 if the device is removable, 0 if not.
SIZE	Size of the device.	Shows size of the device.
RO	Read-Only.	Shows 1 if read-only, 0 if not.
TYPE	The type of block or loop device.	Shows <code>disk</code> for entire disk and <code>part</code> for partitions.
MOUNTPOINTS	Where the device is mounted.	Shows where the device is mounted. Empty if not mounted.

`lsblk` can display information of a specific device when the device's absolute path is passed to it. For example, `lsblk` command for displaying the information of the `sda` disk is:

```
lsblk /dev/sda
```

## lsblk

Here is a table that show some of the useful flags that can be used with lsblk

Short Flag	Long Flag	Description
-a	--all	<b>lsblk</b> does not list empty devices by default. This option disables this restriction.
-b	--bytes	Print the SIZE column in bytes rather than in human-readable format.
-d	--nodeps	Don't print device holders or slaves.
-D	--discard	Print information about the discard (TRIM, UNMAP) capabilities for each device.
-E	--dedup column	Use column as a de-duplication key to de-duplicate output tree. If the key is not available for the device, or the device is a partition and parental whole-disk device provides the same key than the device is always printed.
-e	--exclude list	Exclude the devices specified by a comma-separated list of major device numbers. Note that RAM disks (major=1) are excluded by default. The filter is applied to the top-level devices only.
-f	--fs	Displays information about filesystem.
-h	--help	Print a help text and exit.
-l	--include list	Displays all the information in List Format.
-J	--json	Displays all the information in JSON Format.
-l	--list	Displays all the information in List Format.
-m	--perms	Displays info about device owner, group and mode.
-M	--merge	Group parents of sub-trees to provide more readable output for RAIDs and Multi-path devices. The tree-like output is required.
-n	--noheadings	Do not print a header line.
-o	--output list	Specify which output columns to print. Use --help to get a list of all supported columns.
-O	--output-all	Displays all available columns.
-p	--paths	Displays absolute device paths.
-P	--pairs	Use key="value" output format. All potentially unsafe characters are hex-escaped (\x)
-r	--raw	Use the raw output format. All potentially unsafe characters are hex-escaped (\x) in NAME, KNAME, LABEL, PARTLABEL and MOUNTPOINT columns.
-S	--scsi	Output info about SCSI devices only. All partitions, slaves and holder devices are ignored.
-s	--inverse	Print dependencies in inverse order.
-t	--topology	Output info about block device topology. This option is equivalent to "-o NAME,ALIGNMENT,MIN-IO,OPT-IO,PHY-SEC,LOG-SEC,ROTA,SCHED,RQ-SIZE".
-T	--tree[=column]	Displays all the information in Tree Format.
-V	--version	Output version information and exit.
-w	--width	Specifies output width as a number of characters. The default is the number of the terminal columns, and if not executed on a terminal, then output width is not restricted at all by default.
-x	--sort [column]	Sort output lines by column. This option enables --list output format by default. It is possible to use the option --tree to force tree-like output and then the tree branches are sorted by the column.
-z	--zoned	Print the zone model for each device.
-	--sysroot directory	Gather data for a Linux instance other than the instance from which the lsblk command is issued. The specified directory is the system root of the Linux instance to be inspected.

Like every Unix / Linux Program, `lsblk` returns an exit code to the environment. Here is a table of all the exit codes.

**Exit Code Meaning**

0	Exit with success.
1	Exit with failure.
32	Specified device(s) not found.
64	Some of the specified devices were found while some not.

## 05 `cmatrix`

This command doesn't come by default in Linux. It has to be installed, and as seen in chapter [052](#) we need to run the following command:

```
sudo apt-get install cmatrix
```

And after everything is installed, you have become a 'legit hacker'. In order to use this command, just type in `cmatrix` and press enter:

```
cmatrix
```

And this is what you should see:



As you can see you have access to the matrix now. Well, not really.

What this actually is just a fun little command to goof around with. There are actually a few options you can use. For example you can change the text colour. You can choose from **green, red, blue, white, yellow, cyan, magenta and black**.

```
cmatrix -C red
```



And the falling characters will be red. This command isn't really something that will help you with your job or anything, but it is fun to know that you can have some fun in Linux.



## chmod

The **chmod** command allows you to change the permissions on a file using either a symbolic or numeric mode or a reference file.

1. Change the permission of a file using symbolic mode:

```
chmod u=rwx,g=rx,o=r myfile
```

The command above means :

- user can read, write, execute **myfile**
- group can read, execute **myfile**
- other can read **myfile**

2. Change the permission of a file using numeric mode

```
chmod 754 myfile user:group file.txt
```

The command above means :

- user can read, write, execute **myfile**
- group can read, execute **myfile**
- other can read **myfile**

3. Change the permission of a folder recursively

```
chmod -R 754 folder
```



```
chmod [OPTIONS] MODE FILE(s)
```

- **[OPTIONS]** : **-R**: recursive, mean all file inside directory
- **MODE**: different way to set permissions:
- **Symbolic mode explained**
  - u: user
  - g: group
  - o: other
  - =: set the permission
  - r: read
  - w: write
  - x: execute
  - example **u=rwx** means user can read write and execute
- **Numeric mode explained:**

The **numeric mode** is based off of a binary representation of the permissions for user, group, and others, for more information please look at this [explanation](#) from Digital Ocean's community section:

- 4 stands for "read",
- 2 stands for "write",
- 1 stands for "execute", and
- 0 stands for "no permission."
- example 7 mean read + write + execute

## grep

The **grep** filter searches a file for a particular pattern of characters, and displays all lines that contain that pattern. **grep** stands for globally search for regular expression and print out. The pattern that is searched in the file is referred to as the regular expression.

1. To search the contents of the destination.txt file for a string("KeY") case insensitively.

```
grep -i "KeY" destination.txt
```

2. Displaying the count of number of matches

```
grep -c "key" destination.txt
```

3. We can search multiple files and only display the files that contains the given string/pattern.

```
grep -l "key" destination1.txt destination2.txt
destination3.txt destination4.txt
```

4. To show the line number of file with the line matched.

```
grep -n "key" destination.txt
```

5. If you want to grep the monitored log files, you can add the **--line-buffered** to search them in real time.

```
tail -f destination.txt | grep --line-buffered "key"
```

The general syntax for the grep command is as follows:

```
grep [options] pattern [files]
```

Short Flag	Long Flag	Description
-c	--count	print a count of matching lines for each input file
-h	--no-filename	Display the matched lines, but do not display the filenames
-i	--ignore-case	Ignores, case for matching
-l	--files-with-matches	Displays list of a filenames only.
-n	--line-number	Display the matched lines and their line numbers.
-v	--invert-match	This prints out all the lines that do not matches the pattern
-e	--regexp=	Specifies expression with this option. Can use multiple times
-f	--file=	Takes patterns from file, one per line.
-E	--extended-regexp	Treats pattern as an extended regular expression (ERE)
-w	--word-regexp	Match whole word
-o	--only-matching	Print only the matched parts of a matching line, with each such part on a separate output line.
	--line-buffered	Force output to be line buffered.

## screen

**screen** - With screen you can start a screen session and then open any number of windows (virtual terminals) inside that session. Processes running in Screen will continue to run when their window is not visible even if you get disconnected. This is very handy for running long during session such as bash scripts that run very long.

To start a screen session you type **screen**, this will open a new screen session with a virtual terminal open.

Below are some most common commands for managing Linux Screen Windows:

Command	Description
<b>Ctrl+a+ c</b>	Create a new window (with shell).
<b>Ctrl+a+ "</b>	List all windows.
<b>Ctrl+a+ 0</b>	Switch to window 0 (by number).
<b>Ctrl+a+ A</b>	Rename the current window.
<b>Ctrl+a+ S</b>	Split current region horizontally into two regions.
<b>Ctrl+a+ '</b>	Split current region vertically into two regions.
<b>Ctrl+a+ tab</b>	Switch the input focus to the next region.
<b>Ctrl+a+ Ctrl+a</b>	Toggle between the current and previous windows
<b>Ctrl+a+ Q</b>	Close all regions but the current one.
<b>Ctrl+a+ X</b>	Close the current region.

To restore to a screen session you type `screen -r`, if you have more than one open screen session you have to add the session id to the command to connect to the right session.

To find the session ID you can list the current running screen sessions with:

```
screen -ls
```

There are screens on:

```
18787.pts-0.your-server (Detached)
15454.pts-0.your-server (Detached)
2 Sockets in /run/screens/S-yourserver.
```

If you want to restore screen 18787.pts-0, then type the following command:

```
screen -r 18787
```

## nc

The **nc** (or netcat) command is used to perform any operation involving TCP (Transmission Control Protocol, connection oriented), UDP (User Datagram Protocol, connection-less, no guarantee of data delivery) or UNIX-domain sockets. It can be thought of as swiss-army knife for communication protocol utilities.

```
nc [options] [ip] [port]
```

**1. Open a TCP connection to port 80 of host, using port 1337 as source port with timeout of 5s:**

```
$ nc -p 1337 -w 5 host.ip 80
```

**2. Open a UDP connection to port 80 on host:**

```
$ nc -u host.ip 80
```

**3. Create and listen on UNIX-domain stream socket:**

```
$ nc -lU /var/tmp/dsocket
```

**4. Create a basic server/client model:**

This creates a connection, with no specific server/client sides with respect to nc, once the connection is established.



```
$ nc -l 1234 # in one console
$ nc 127.0.0.1 1234 # in another console
```

## 5. Build a basic data transfer model:

After the file has been transferred, sequentially, the connection closes automatically

```
$ nc -l 1234 > filename.out # to start listening in one
console and collect data
$ nc host.ip 1234 < filename.in
```

## 6. Talk to servers:

Basic example of retrieving the homepage of the host, along with headers.

```
$ printf "GET / HTTP/1.0\r\n\r\n" | nc host.ip 80
```

## 7. Port scanning:

Checking which ports are open and running services on target machines. **-z** flag commands to inform about those rather than initiate a connection.

```
$ nc -zv host.ip 20-2000 # range of ports to check for
```

Short Flag	Description
<b>-4</b>	Forces nc to use IPv4 addresses
<b>-6</b>	Forces nc to use IPv6 addresses
<b>-b</b>	Allow broadcast
<b>-D</b>	Enable debugging on the socket
<b>-i</b>	Specify time interval delay between lines sent and received
<b>-k</b>	Stay listening for another connection after current is over
<b>-l</b>	Listen for incoming connection instead of initiate one to remote

Short Flag	Description
-T	Specify length of TCP
-p	Specify source port to be used
-r	Specify source and/or destination ports randomly
-s	Specify IP of interface which is used to send the packets
-U	Use UNIX-domain sockets
-u	Use UDP instead of TCP as protocol
-w	Declare a timeout threshold for idle or unestablished connections
-x	Should use specified protocol when talking to proxy server
-z	Specify to scan for listening daemons, without sending any data

## make

The **make** command is used to automate the reuse of multiple commands in certain directory structure.

An example for that would be the use of **terraform init**, **terraform plan**, and **terraform validate** while having to change different subscriptions in Azure. This is usually done in the following steps:

```
az account set --subscription "Subscription - Name"
terraform init
```

How the **make** command can help us is it can automate all of that in just one go: **make tf-init**

```
make [-f makefile] [options] ... [targets] ...
```

1. Create **Makefile** in your guide directory
2. Include the following in your **Makefile** :

```
hello-world:
 echo "Hello, World!"

hello-bobby:
 echo "Hello, Bobby!"

touch-letter:
 echo "This is a text that is being inputted into our
letter!" > letter.txt

clean-letter:
 rm letter.txt
```

3. Execute **make hello-world** - this echoes "Hello, World" in our terminal.
4. Execute **make hello-bobby** - this echoes "Hello, Bobby!" in our terminal.
5. Execute **make touch-letter** - This creates a text file named **letter.txt** and populates a line in it.
6. Execute **make clean-letter**

(linoxide - linux make command examples)[<https://linoxide.com/linux-make-command-examples/>] (makefiletutorial.com - the name itself gives it out)[<https://makefiletutorial.com/>]

## basename

The **basename** is a command-line utility that strips directory from given file names. Optionally, it can also remove any trailing suffix. It is a simple command that accepts only a few options.

The most basic example is to print the file name with the leading directories removed:

```
basename /etc/bar/foo.txt
```

The output will include the file name:

```
foo.txt
```

If you run **basename** on a path string that points to a directory, you will get the last segment of the path. In this example, `/etc/bar` is a directory.

```
basename /etc/bar
```

Output

```
bar
```

The **basename** command removes any trailing `/` characters:

```
basename /etc/bar/foo.txt/
```

Output

```
foo.txt
```

1. By default, each output line ends in a newline character. To end the lines with NUL, use the `-z` (`--zero`) option.

```
$ basename -z /etc/bar/foo.txt
foo.txt$
```

2. The `basename` command can accept multiple names as arguments. To do so, invoke the command with the `-a` (`--multiple`) option, followed by the list of files separated by space. For example, to get the file names of `/etc/bar/foo.txt` and `/etc/spam/eggs.docx` you would run:

```
basename -a /etc/bar/foo.txt /etc/spam/eggs.docx
```

```
foo.txt
eggs.docx
```

The `basename` command supports two syntax formats:

```
basename NAME [SUFFIX]
basename OPTION... NAME...
```

**Removing a Trailing Suffix:** To remove any trailing suffix from the file name, pass the suffix as a second argument:



```
basename /etc/hostname name
host
```

Generally, this feature is used to strip file extensions

Run the following command to view the complete guide to **basename** command.



```
man basename
```

## banner

The **banner** command writes ASCII character Strings to standard output in large letters. Each line in the output can be up to 10 uppercase or lowercase characters in length. On output, all characters appear in uppercase, with the lowercase input characters appearing smaller than the uppercase input characters.

Note: If you will define more than one NUMBER with sleep command then this command will delay for the sum of the values.

1. To display a banner at the workstation, enter:

```
banner LINUX!
```

2. To display more than one word on a line, enclose the text in quotation marks, as follows:

```
banner "Intro to" Linux
```

This displays Intro to on one line and Linux on the next

3. Printing "101LinuxCommands" in large letters.

```
banner 101LinuxCommands
```

It will print only 101LinuxCo as banner has a default capacity of 10



## alias

The **alias** command lets you create shortcuts for commands or define your own commands.

This is mostly used to avoid typing long commands.

1. To show the list of all defined aliases in the reusable form **alias NAME=VALUE** :

```
alias -p
```

2. To make **ls -A** shortcut:

```
alias la='ls -A'
```

```
alias [-p] [name[=value]]
```

As with most Linux custom settings for the terminal, any alias you defined is only applied to the current opening terminal session.

For any alias to be active for all new sessions you need to add that command to your rc file to be executed in the startup of every new terminal. this file can be as follows:

- **Bash:** ~/.bashrc
- **ZSH:** ~/.zshrc

- **Fish** - ~/.config/fish/config.fish

you can open that file with your favorite editor as follows:

```
vim ~/.bashrc
```

type your commands one per line, then save the file and exit. the commands will be automatically applied in the next session.

If you want to apply it in the current session, run the following command:

```
source ~/.bashrc
```

To remove predefined alias you can use **unalias** command as follows:

```
unalias alias_name
```

to remove all aliases

```
unalias -a
```

## which

**which** command identifies the executable binary that launches when you issue a command to the shell. If you have different versions of the same program on your computer, you can use which to find out which one the shell will use.

It has 3 return status as follows:

```
0 : If all specified commands are found and executable.
1 : If one or more specified commands is nonexistent or not
executable.
2 : If an invalid option is specified.
```

1. To find the full path of the ls command, type the following:

```
which ls
```

2. We can provide more than one arguments to the which command:

```
which netcat uptime ping
```

The which command searches from left to right, and if more than one matches are found in the directories listed in the PATH path variable, which will print only the first one.

3. To display all the paths for the specified command:

```
which [filename] -a
```

4. To display the path of node executable files, execute the command:

```
which node
```

5. To display the path of Java executable files, execute:

```
which java
```

```
which [filename1] [filename2] ...
```

You can pass multiple programs and commands to which, and it will check them in order.

For example:

```
which ping cat uptime date head
```

-a : List all instances of executables found (instead of just the first one of each).

-s : No output, just return 0 if all the executables are found, or 1 if some were not found

## date

The **date** command is used to print the system current date and time.

**date** command is also used to set the date and time of the system, but you need to be the super-user (*root*) to do it.

1. To show the current date and time:

```
date
```

2. You can use -u option to show the date and time in UTC (*Coordinated Universal Time*) time zone

```
date -u
```

1. To display any given date string in formatted date:

```
date --date="2/02/2010"
date --date="2 years ago"
```

```
date [OPTION]... [+FORMAT]
date [-u|--utc|--universal] [MMDDhhmm[[CC]YY][.ss]]
```

Short Flag	Long Flag	Description
-d	--date=STRING	convert the provided string into formatted date
-f	--file=DATEFILE	like --date but for files
-I[FMT]	--iso-8601[=FMT]	Display date and time in ISO 8601 format
-r	--reference=FILE	Display the last modification time of FILE
-s	--set=STRING	sets the time to the one described by STRING
-u	--universal	show the date and time in UTC ( <i>Coordinated Universal Time</i> ) time zone
-R	--rfc-email	Display date and time in ISO 8601 format Example: (Fri, 22 Oct 2021 05:18:42 +0200)
-	rfc-3339=FMT	Display date and time in RFC 3339 format
-	--debug	Usually used with --date to annotate the parsed date and warn about questionable usage to stderr

You can use Format specifiers to control the output date and time.

Command	Output
\$ date "+%D"	10/22/21
\$ date "+%D %T"	10/22/21 05:33:51
\$ date "+%A %B %d %T %y"	Friday October 22 05:34:47 21

```
date "+%[format-options ...]"
```

### Specifiers Description

%a abbreviated weekday name (*e.g.*, *Sun*)

## Specifiers Description

%A	full weekday name ( <i>e.g., Sunday</i> )
%b	abbreviated month name ( <i>e.g., Jan</i> )
%B	full month name ( <i>e.g., January</i> )
%c	date and time ( <i>e.g., Thu Mar 3 23:05:25 2005</i> )
%C	century; like %Y, except omit last two digits ( <i>e.g., 20</i> )
%d	day of month ( <i>e.g., 01</i> )
%D	date; same as %m/%d/%y
%e	day of month, space padded; same as %_d
%F	full date; same as %Y-%m-%d
%g	last two digits of year of ISO week number (see %G)
%G	year of ISO week number (see %V); normally useful only with %V
%h	same as %b
%H	hour (00..23)
%I	hour (01..12)
%j	day of year (001..366)
%k	hour, space padded ( 0..23); same as %_H
%l	hour, space padded ( 1..12); same as %_I
%m	month (01..12)
%M	minute (00..59)
%n	a newline
%N	nanoseconds (000000000..999999999)
%p	locale's equivalent of either AM or PM; blank if not known
%P	like %p, but lower case
%q	quarter of year (1..4)
%r	locale's 12-hour clock time ( <i>e.g., 11:11:04 PM</i> )
%R	24-hour hour and minute; same as %H:%M
%s	seconds since 1970-01-01 00:00:00 UTC
%S	second (00..60)
%t	a tab
%T	time; same as %H:%M:%S
%u	day of week (1..7); 1 is Monday
%U	week number of year, with Sunday as first day of week (00..53)
%V	ISO week number, with Monday as first day of week (01..53)
%w	day of week (0..6); 0 is Sunday
%W	week number of year, with Monday as first day of week (00..53)
%x	locale's date representation ( <i>e.g., 12/31/99</i> )
%X	locale's time representation ( <i>e.g., 23:13:48</i> )

**Specifiers Description**

<b>%y</b>	last two digits of year (00..99)
<b>%Y</b>	year
<b>%Z</b>	+hhmm numeric time zone (e.g., -0400)
<b>%%:Z</b>	+hh:mm numeric time zone (e.g., -04:00)
<b>%%::Z</b>	+hh:mm:ss numeric time zone (e.g., -04:00:00)
<b>%%:::Z</b>	numeric time zone with : to necessary precision (e.g., -04, +05:30)
<b>%Z</b>	alphabetic time zone abbreviation (e.g., EDT)



## mount

The **mount** command is used to mount 'attach' a filesystem and make it accessible by an existing directory structure tree.

1. Displays version information:

```
mount -V
```

2. Attaching filesystem found on device and of type type at the directory dir:

```
mount -t type device dir
```

```
mount [-lhV]
```

```
mount -a [-fFnrsvw] [-t vfstype] [-O optlist]
```

```
mount [-fnrsvw] [-t fstype] [-o options] device dir
```

Short Flag	Long Flag	Description
-h	--help	Display a help message and exists
-n	--no-mtab	Mount without writing in /etc/mtab
-a	--all	Mount all filesystems (of the given types) mentioned in fstab
-r	--read-only	Mount the filesystem read-only
-w	--rw	Mount the filesystem as read/write.
-M	--move	Move a subtree to some other place.
-B	--bind	Remount a subtree somewhere else ( <i>so that its contents are available in both places</i> ).

## nice/renice

The **nice/renice** commands is used to modify the priority of the program to be executed. The priority range is between -20 and 19 where 19 is the lowest priority.

1. Running cc command in the background with a lower priority than default (slower):

```
nice -n 15 cc -c *.c &
```

2. Increase the priority to all processes belonging to group "test":

```
renice --20 -g test
```

```
nice [-Increment| -n Increment] Command [Argument ...]
```

Short Flag	Long Flag	Description
-Increment	-	Increment is the value of priority you want to assign.
-n Increment	-	Same as -Increment

## WC

the **WC** command stands for word count. It's used to count the number of lines, words, and bytes (*characters*) in a file or standard input then prints the result to the standard output.

1. To count the number of lines, words and characters in a file in order:

```
wc file.txt
```

2. To count the number of directories in a directory:

```
ls -F | grep / | wc -l
```

```
wc [OPTION]... [FILE]...
```

Short Flag	Long Flag	Description
-c	--bytes	print the byte counts
-m	--chars	print the character counts
-l	--lines	print the newline counts
-	--files0-from=F	read input from the files specified by NUL-terminated names in file F. If F is - then read names from standard input

Short Flag	Long Flag	Description
-L	--max-line-length	print the maximum display width
-w	--words	print the word counts

- Passing more than one file to **wc** command prints the counts for each file and the total counts of them.
- you can combine more than one flag to print the result as you want.

## tr

The `tr` command in UNIX is a command line utility for translating or deleting characters. It supports a range of transformations including uppercase to lowercase, squeezing repeating characters, deleting specific characters and basic find and replace. It can be used with UNIX pipes to support more complex translation. `tr` stands for translate.

1. Convert all lowercase letters in `file1` to uppercase.

```
$ cat file1
foo
bar
baz
tr a-z A-Z < file1
FOO
BAR
BAZ
```

2. Make consecutive line breaks into one.

```
$ cat file1
foo

bar

baz
$ tr -s "\n" < file1
foo
bar
baz
```

3. Remove the newline code.

```
$ cat file1
foo
bar
baz
$ tr -d "\n" < file1
foobarbaz%
```

The general syntax for the tr command is as follows:

```
tr [options] string1 [string2]
```

Short Flag	Long Flag	Description
-C		Complement the set of characters in string1, that is -C ab includes every character except for a and b.
-c		Same as -C.
-d		Delete characters in string1 from the input.
-s		If there is a sequence of characters in string1, combine them into one.

## fdisk

The **fdisk** command is used for controlling the disk partition table and making changes to it and this is a list of some of options provided by it :

- Organize space for new drives.
- Modify old drives.
- Create space for new partitions.
- Move data to new partitions.

1. To view basic details about all available partitions on the system:

```
fdisk -l
```

2. To show the size of the partition:

```
fdisk -s /dev/sda
```

3. To view the help message and all options of the command:

```
fdisk -h
```

```
fdisk [options] device
```



On writing the following command

```
fdisk /dev/sdb
```

the following window appears :

```
linux@ubuntu:~$ sudo fdisk /dev/sda

Welcome to fdisk (util-linux 2.31.1).
Changes will remain in memory only, until you decide to write them.
Be careful before using the write command.

Command (m for help):
```

and then you type m which will show you all options you need such as creating new partition and deleting a partition as in the following picture :

```
Command (m for help): m
Help:

DOS (MBR)
a toggle a bootable flag
b edit nested BSD disklabel
c toggle the dos compatibility flag

Generic
d delete a partition
F list free unpartitioned space
l list known partition types
n add a new partition
p print the partition table
t change a partition type
v verify the partition table
i print information about a partition

Misc
m print this menu
u change display/entry units
x extra functionality (experts only)

Script
I load disk layout from sfdisk script file
O dump disk layout to sfdisk script file

Save & Exit
w write table to disk and exit
q quit without saving changes

Create a new label
g create a new empty GPT partition table
G create a new empty SGI (IRIX) partition table
o create a new empty DOS partition table
s create a new empty Sun partition table

Command (m for help):
```

## Wait

It is a command that waits for completing any running process of given id. if the process id is not given then it waits for all current child processes to complete.

This example shows how the `wait` command works :

**Step-1:**

Create a file named "wait\_example.sh" and add the following script to it.

```
#!/bin/bash
echo "Wait command" &
process_id=$!
wait $process_id
echo "Exited with status $?"
```

**Step-2:**

Run the file with bash command.

```
$ bash wait_example.sh
```

## zcat

The **zcat** allows you to look at a compressed file.

1. To view the content of a compressed file:

```
~$ zcat test.txt.gz
Hello World
```

2. It can also Works with multiple files:

```
~$ zcat test2.txt.gz test.txt.gz
hello
Hello world
```

The general syntax for the **zcat** command is as follows:

```
zcat [-n] [-V] [File ...]
```

## fold

The **fold** command in Linux wraps each line in an input file to fit a specified width and prints it to the standard output.

By default, it wraps lines at a maximum width of 80 columns but this is configurable.

To fold input using the fold command pass a file or standard input to the command.

```
fold [OPTION]... [FILE]...
```

**-w** : By using this option in fold command, we can limit the width by number of columns.

By using this command we change the column width from default width of 80. Syntax:

```
fold -w[n] [FILE]
```

Example: wrap the lines of file1.txt to a width of 60 columns

```
fold -w60 file1.txt
```

**-b** : This option of fold command is used to limit the width of the output by the number of bytes rather than the number of columns.

By using this we can enforce the width of the output to the number of bytes.

```
fold -b[n] [FILE]
```

Example: limit the output width of the file to 40 bytes and the command breaks the output at 40 bytes.

```
fold -b40 file1.txt
```

**-s** : This option is used to break the lines on spaces so that words are not broken.

If a segment of the line contains a blank character within the first width column positions, break the line after the last such blank character meeting the width constraints.

```
fold -w[n] -s [FILE]
```

## quota

The **quota** display disk usage and limits.

You can simply go ahead and install quota on ubuntu systems by running:

```
sudo apt-get install quota
```

for Debian use the install command without sudo:

```
apt-get install quota
```

The general syntax for the **quota** command is as follows:

```
quota [-u [User]] [-g [Group]] [-v | -q]
```

## **aplay**

**aplay** is a command-line audio player for ALSA(Advanced Linux Sound Architecture) sound card drivers. It supports several file formats and multiple soundcards with multiple devices. It is basically used to play audio on command-line interface. aplay is much the same as arecord only it plays instead of recording. For supported soundfile formats, the sampling rate, bit depth, and so forth can be automatically determined from the soundfile header.





```
$ aplay [flags] [filename [filename]] ...
```

```
-h, -help : Show the help information.
-d, -duration=# : Interrupt after # seconds.
-r, -rate=# : Sampling rate in Hertz. The default rate is 8000
Hertz.
-version : Print current version.
-l, -list-devices : List all soundcards and digital audio
devices.
-L, -list-pcms : List all PCMs(Pulse Code Modulation) defined.
-D, -device=NAME : Select PCM by name.
```

Note: This command contain various other options that we normally don't need. If you want to know more about you can simply run following command on your terminal.

```
aplay --help
```

1. To play audio for only 10 secs at 2500hz frequency.

```
$ aplay -d 10 -r 2500hz sample.mp3
```

Plays sample.mp3 file for only 10 secs at 2500hz frequency.

2. To play full audio clip at 2500hz frequency.

```
$ aplay -r 2500hz sample.mp3
```

Plays sample.mp3 file at 2500hz frequency.

3. To Display version information.

```
$ aplay --version
```

Displays version information. For me it shows aplay: version 1.1.0

## **spd-say**

**spd-say** sends text-to-speech output request to speech-dispatcher process which handles it and ideally outputs the result to the audio system.



```
$ spd-say [options] "some text"
```

-r, --rate  
Set the rate of the speech (between -100 and +100, default: 0)

-p, --pitch  
Set the pitch of the speech (between -100 and +100, default: 0)

-i, --volume  
Set the volume (intensity) of the speech (between -100 and +100, default: 0)

-o, --output-module  
Set the output module

-l, --language  
Set the language (iso code)

-t, --voice-type  
Set the preferred voice type (male1, male2, male3, female1, female2, female3, child\_male, child\_female)

-m, --punctuation-mode  
Set the punctuation mode (none, some, all)

-s, --spelling  
Spell the message

-x, --ssml  
Set SSML mode on (default: off)

-e, --pipe-mode  
Pipe from stdin to stdout plus Speech Dispatcher

-P, --priority  
Set priority of the message (important, message, text, notification, progress; default: text)

-N, --application-name

```
 Set the application name used to establish the
connection to specified string value
 (default: spd-say)

-n, --connection-name
 Set the connection name used to establish the
connection to specified string value
 (default: main)

-w, --wait
 Wait till the message is spoken or discarded

-S, --stop
 Stop speaking the message being spoken in Speech
Dispatcher

-C, --cancel
 Cancel all messages in Speech Dispatcher

-v, --version
 Print version and copyright info

-h, --help
 Print this info
```

1. To Play the given text as the sound.



```
$ spd-say "Hello"
```

Plays "Hello" in sound.



## xeyes

Xeyes is a graphical user interface program that creates a set of eyes on the desktop that follow the movement of the mouse cursor. It seems much of a funny command, than of any useful use. Being funny is as much useful, is another aspect.



### xeyes

**xeyes** is not for fun, at least not only. The purpose of this program is to let you follow the mouse pointer which is sometimes hard to see. It is very useful on multi-headed computers, where monitors are separated by some distance, and if someone (say teacher at school) wants to present something on the screen, the others on their monitors can easily follow the mouse with **xeyes**.

## parted

The **parted** command is used to manage hard disk partitions on Linux. It can be used to add, delete, shrink and extend disk partitions along with the file systems located on them. You will need root access to the system to run **parted** commands.

**NOTE:** Parted writes the changes immediately to your disk, be careful when you are modifying the disk partitions.

1. Displays partition layout of all block devices:

```
sudo parted -l
```

2. Display partition table of a specific **disk**

```
sudo parted disk print
```

Examples of **disk** are /dev/sda, /dev/sdb

3. Create a new disk label of **label-type** for a specific disk

```
sudo parted mklabel disk label-type
```

**label-type** can take values "aix", "amiga", "bsd", "dvh", "gpt", "loop", "mac", "msdos", "pc98", or "sun"

4. Create a new partition in a specific **disk** of type **part-time**, file system is **fs-type** and of size **size** Mb.

```
sudo parted disk mkpart part-time fs-type 1 size
```

**part-time** can take values "primary", "logical", "extended".

**fs-type** is optional. It can take values "btrfs", "ext2", "ext3", "ext4", "fat16", "fat32", "hfs", "hfs+", "linux-swaps", "ntfs", "reiserfs", "udf", or "xfs"

**size** has to be less than the total size of the specified disk. To create a partition of size 50Mb, will take the value of 50

5. **parted** can also be run in an interactive format. Operations to manage the disk partitions can be performed by entering appropriate commands in the interactive session. **help** command in the interactive session shows a list of all possible disk management operations which can be performed.

```
$ sudo parted
GNU Parted 3.3
Using /dev/sda
Welcome to GNU Parted! Type 'help' to view a list of
commands.
(parted) print # prints the partition table of the default
selected disk - /dev/sda
Model: ATA VBOX HARDDISK (scsi)
Disk /dev/sda: 53.7GB
Sector size (logical/physical): 512B/512B
Partition Table: msdos
Disk Flags:

Number Start End Size Type File system Flags
 1 1049kB 53.7GB 53.7GB primary ext4 boot

(parted) select /dev/sdb # change the current disk on which
operations have to be performed
Using /dev/sdb
(parted) quit # exit the interactive session
```

```
parted [options] [device [command [options...]]...]]
```

Short Flag	Long Flag	Description
-h	--help	displays a help message listing all possible <b>commands</b> <b>[options]</b>
-l	--list	lists partition layout on all block devices
-m	--machine	displays machine parseable output
-v	--version	displays the version
		set alignment type for newly created partition. It can take the following values:
		<b>none</b> : Use the minimum alignment allowed by the disk type
		<b>cylinder</b> : Align partitions to cylinders
-a	--align	<b>minimal</b> : Use minimum alignment as given by the disk topology information
		<b>optimal</b> : Use optimum alignment as given by the disk topology information

999-wrap-up.md