

Linux Basics

Most Commonly Used Linux Commands

Before proceeding to the list of basic commands, you need to open Terminal first. If you are still unsure, check out our [CLI tutorial](#).

Although the steps may differ depending on your Linux distribution, the Terminal application is usually found in the **Utilities** section.

Here is the list of basic Linux commands:

1. sudo command

Short for superuser do, **sudo** is one of the most popular basic Linux commands that lets you perform tasks that require administrative or root permissions.

When using sudo, the system will prompt users to authenticate themselves with a password. Then, the Linux system will log a timestamp as a tracker. By default, every root user can run sudo commands for **15 minutes/session**.

If you try to run sudo in the command line without authenticating yourself, the system will log the activity as a security event.

Here's the general syntax:

sudo (command)

You can also add an option, such as:

- **-k** or **--reset-timestamp** invalidates the timestamp file.
- **-g** or **--group=group** runs commands as a specified group name or ID.
- **-h** or **--host=host** runs commands on the host.

2. pwd command

Use the **pwd** command to find the path of your current working directory. Simply entering **pwd** will return the full current path – a path of all the directories that starts with a forward slash (/). For example, **/home/username**.

The **pwd** command uses the following syntax:

pwd [option]

It has two acceptable options:

- **-L** or **--logical** prints environment variable content, including symbolic links.
- **-P** or **--physical** prints the actual path of the current directory.

3. cd command

To navigate through the Linux files and directories, use the **cd** command. Depending on your current working directory, it requires either the full path or the directory name.

Running this command without an option will take you to the home folder. Keep in mind that only users with **sudo** privileges can execute it.

Let's say you're in **/home/username/Documents** and want to go to **Photos**, a subdirectory of **Documents**. To do so, enter the following command:

cd Photos.

If you want to switch to a completely new directory, for example, **/home/username/Movies**, you have to enter **cd** followed by the directory's absolute path:

cd /home/username/Movies

Here are some shortcuts to help you navigate:

- **cd ~[username]** goes to another user's home directory.

Linux Basics

- **cd ..** moves one directory up.
- **cd-** moves to your previous directory.

4. ls command

The **ls** command lists files and directories within a system. Running it without a flag or parameter will show the current working directory's content.

To see other directories' content, type **ls** followed by the desired path. For example, to view files in the **Documents** folder, enter:

ls /home/username/Documents

Here are some options you can use with the **ls** command:

- **ls -R** lists all the files in the subdirectories.
- **ls -a** shows hidden files in addition to the visible ones.
- **ls -lh** shows the file sizes in easily readable formats, such as MB, GB, and TB.

5. cat command

Concatenate, or **cat**, is one of the most frequently used Linux commands. It lists, combines, and writes file content to the standard output. To run the cat command, type **cat** followed by the file name and its extension. For instance:

cat filename.txt

Here are other ways to use the [cat command](#):

- **cat > filename.txt** creates a new file.
- **cat filename1.txt filename2.txt > filename3.txt** merges **filename1.txt** and **filename2.txt** and stores the output in **filename3.txt**.
- **tac filename.txt** displays content in reverse order.

6. cp command

Use the **cp** command to copy files or directories and their content. Take a look at the following use cases.

To copy one file from the current directory to another, enter **cp** followed by the file name and the destination directory. For example:

cp filename.txt /home/username/Documents

To copy files to a directory, enter the file names followed by the destination directory:

cp filename1.txt filename2.txt filename3.txt /home/username/Documents

To copy the content of a file to a new file in the same directory, enter **cp** followed by the source file and the destination file:

cp filename1.txt filename2.txt

To copy an entire directory, pass the **-R** flag before typing the source directory, followed by the destination directory:

cp -R /home/username/Documents /home/username/Documents_backup

7. mv command

The primary use of the [mv command](#) is to move and rename files and directories. Additionally, it doesn't produce an output upon execution.

Simply type **mv** followed by the filename and the destination directory. For example, you want to move **filename.txt** to the **/home/username/Documents** directory:

mv filename.txt /home/username/Documents

You can also use the **mv** command to rename a file:

mv old_filename.txt new_filename.txt

Linux Basics

8. mkdir command

Use the **mkdir** command to create one or multiple directories at once and set permissions for each of them. The user executing this command must have the privilege to make a new folder in the parent directory, or they may receive a permission denied error.

Here's the basic syntax:

mkdir [option] directory_name

For example, you want to create a directory called **Music**:

mkdir Music

To make a new directory called **Songs** inside **Music**, use this command:

mkdir Music/Songs

The **mkdir** command accepts many options, such as:

- **-p** or **-parents** create a directory between two existing folders. For example, **mkdir -p Music/2020/Songs** will make the new "2020" directory.
- **-m** sets the file permissions. For instance, to create a directory with full read, write, and execute permissions for all users, enter **mkdir -m777 directory_name**.
- **-v** prints a message for each created directory.

9. rmdir command

To permanently delete an empty directory, use the [rmdir command](#). Remember that the user running this command should have **sudo** privileges in the parent directory.

For example, you want to remove an empty subdirectory named **personal1** and its main folder **mydir**:

rmdir -p mydir/personal1

10. rm command

The **rm** command is used to delete files within a directory. Make sure that the user performing this command has write permissions.

Remember the directory's location as this will remove the file(s) and you can't undo it.

Here's the general syntax:

rm filename

To remove multiple files, enter the following command:

rm filename1 filename2 filename3

Here are some acceptable options you can add:

- **-i** prompts system confirmation before deleting a file.
- **-f** allows the system to remove without a confirmation.
- **-r** deletes files and directories recursively.

11. touch command

The [touch command](#) allows you to create an empty file or generate and modify a timestamp in the Linux command line.

For example, enter the following command to create an HTML file named **Web** in the **Documents** directory:

touch /home/username/Documents/Web.html

12. locate command

The [locate command](#) can find a file in the database system.

Moreover, adding the **-i** argument will turn off case sensitivity, so you can search for a file even if you don't remember its exact name.

To look for content that contains two or more words, use an asterisk (*). For example:

Linux Basics

locate -i school*not

The command will search for files that contain the words **school** and **note**, whether they use uppercase or lowercase letters.

13. find command

Use the **find** command to search for files within a specific directory and perform subsequent operations. Here's the general syntax:

find [option] [path] [expression]

For example, you want to look for a file called **notes.txt** within the **home** directory and its subfolders:

find /home -name notes.txt

Here are other variations when using **find**:

- **find -name filename.txt** to find files in the current directory.
- **find ./ -type d -name directoryname** to look for directories.

14. grep command

Another basic Linux command on the list is **grep** or global regular expression print. It lets you find a word by searching through all the texts in a specific file.

Once the [grep command](#) finds a match, it prints all lines that contain the specific pattern. This command helps filter through large log files.

For example, you want to search for the word **blue** in the **notepad.txt** file:

grep blue notepad.txt

The command's output will display lines that contain **blue**.

15. df command

Use the [df command](#) to report the system's disk space usage, shown in percentage and kilobyte (KB). Here's the general syntax:

df [options] [file]

For example, enter the following command if you want to see the current directory's system disk space usage in a human-readable format:

df -h

These are some acceptable options to use:

- **df -m** displays information on the file system usage in **MBs**.
- **df -k** displays file system usage in **KBs**.
- **df -T** shows the file system **type** in a new column.

16. du command

If you want to check how much space a file or a directory takes up, use the **du** command. You can run this command to identify which part of the system uses the storage excessively.

Remember, you must specify the directory path when using the **du** command. For example, to check **/home/user/Documents** enter:

du /home/user/Documents

Adding a flag to the **du** command will modify the operation, such as:

- **-s** offers the total size of a specified folder.
- **-m** provides folder and file information in **MB**
- **k** displays information in **KB**.

Linux Basics

- **-h** informs the last modification date of the displayed folders and files.

17. head command

The **head** command allows you to view the first ten lines of a text. Adding an option lets you change the number of lines shown. The **head** command is also used to output piped data to the CLI.

Here's the general syntax:

head [option] [file]

For instance, you want to view the first ten lines of **note.txt**, located in the current directory:

head note.txt

Below are some options you can add:

- **-n** or **-lines** prints the first customized number of lines. For example, enter **head -n 5 filename.txt** to show the first five lines of **filename.txt**.
- **-c** or **-bytes** prints the first customized number of bytes of each file.
- **-q** or **-quiet** will not print headers specifying the file name.

18. tail command

The [tail command](#) displays the last ten lines of a file. It allows users to check whether a file has new data or to read error messages.

Here's the general format:

tail [option] [file]

For example, you want to show the last ten lines of the **colors.txt** file:

tail -n colors.txt

19. diff command

Short for difference, the **diff** command compares two contents of a file line by line. After analyzing them, it will display the parts that do not match.

Programmers often use the **diff** command to alter a program instead of rewriting the entire source code.

Here's the general format:

diff [option] file1 file2

For example, you want to compare two text files – **note.txt** and **note_update.txt**:

diff note.txt note_update.txt

Here are some acceptable options to add:

- **-c** displays the difference between two files in a context form.
- **-u** displays the output without redundant information.
- **-i** makes the **diff** command case insensitive.

20. tar command

The [tar command](#) archives multiple files into a **TAR** file – a common Linux format similar to **ZIP**, with optional compression.

Here's the basic syntax:

tar [options] [archive_file] [file or directory to be archived]

For instance, you want to create a new **TAR** archive named **newarchive.tar** in the **/home/user/Documents** directory:

tar -cvf newarchive.tar /home/user/Documents

The **tar** command accepts many options, such as:

- **-x** extracts a file.
- **-t** lists the content of a file.

Linux Basics

- **-u** archives and adds to an existing archive file.

Check out the more [practical examples](#) to know more about the other functions.

21. chmod command

[chmod](#) is a common command that modifies a file or directory's read, write, and execute permissions. In Linux, each file is associated with three user classes – **owner**, **group member**, and **others**.

Here's the basic syntax:

chmod [option] [permission] [file_name]

For example, the owner is currently the only one with full permissions to change **note.txt**. To allow group members and others to read, write, and execute the file, change it to the **-rwxrwxrwx** permission type, whose numeric value is **777**:

chmod 777 note.txt

This command supports many options, including:

- **-c** or **–changes** displays information when a change is made.
- **-f** or **–silent** suppresses the error messages.
- **-v** or **–verbose** displays a diagnostic for each processed file.

22. chown command

The [chown command](#) lets you change the ownership of a file, directory, or symbolic link to a specified username.

Here's the basic format:

chown [option] owner[:group] file(s)

For example, you want to make **linuxuser2** the owner of **filename.txt**:

chown linuxuser2 filename.txt

23. jobs command

A job is a process that the shell starts. The **jobs** command will display all the running processes along with their statuses. Remember that this command is only available in **cs**h, **bash**, **tc**sh, and **k**sh shells.

This is the basic syntax:

jobs [options] jobID

To check the status of jobs in the current shell, simply enter **jobs** to the CLI.

Here are some options you can use:

- **-l** lists process IDs along with their information.
- **-n** lists jobs whose statuses have changed since the last notification.
- **-p** lists process IDs only.

24. kill command

Use the [kill command](#) to terminate an unresponsive program manually. It will signal misbehaving applications and instruct them to close their processes.

To kill a program, you must know its process identification number (PID). If you don't know the PID, run the following command:

ps ux

After knowing what signal to use and the program's PID, enter the following syntax:

kill [signal_option] pid

There are [64 signals](#) that you can use, but these two are among the most commonly used:

- **SIGTERM** requests a program to stop running and gives it some time to save all of its progress. The system will use this by default if you don't specify the signal when entering the kill command.

Linux Basics

- **SIGKILL** forces programs to stop, and you will lose unsaved progress.

For example, the program's PID is **63773**, and you want to force it to stop:
kill SIGKILL 63773

25. ping command

The [ping command](#) is one of the most used basic Linux commands for checking whether a network or a server is reachable. In addition, it is used to troubleshoot various connectivity issues.

Here's the general format:

ping [option] [hostname_or_IP_address]

For example, you want to know whether you can connect to **Google** and measure its response time:

ping google.com

26. wget command

The Linux command line lets you download files from the internet using the [wget command](#). It works in the background without hindering other running processes.

The **wget** command retrieves files using HTTP, HTTPS, and FTP protocols. It can perform recursive downloads, which transfer website parts by following directory structures and links, creating local versions of the web pages.

To use it, enter the following command:

wget [option] [url]

For example, enter the following command to download the latest version of [WordPress](#):

wget https://wordpress.org/latest.zip

27. uname command

The **uname** or unix name command will print detailed information about your Linux system and hardware. This includes the machine name, operating system, and kernel. To run this command, simply enter **uname** into your CLI.

Here's the basic syntax:

uname [option]

These are the acceptable options to use:

- **-a** prints all the system information.
- **-s** prints the kernel name.
- **-n** prints the system's node hostname.

28. top command

The [top command](#) in Linux Terminal will display all the running processes and a dynamic real-time view of the current system. It sums up the resource utilization, from CPU to memory usage.

The **top** command can also help you identify and terminate a process that may use too many system resources.

To run the command, simply enter **top** into the CLI.

29. history command

With **history**, the system will list up to 500 previously executed commands, allowing you to reuse them without re-entering. Keep in mind that only users with **sudo** privileges can execute this command. How this utility runs also depends on which Linux shell you use.

To run it, enter the command below:

history [option]

This command supports many options, such as:

- **-c** clears the complete history list.

Linux Basics

- **-d offset** deletes the history entry at the **OFFSET** position.
- **-a** appends history lines.

30. man command

The **man** command provides a user manual of any commands or utilities you can run in Terminal, including the name, description, and options.

It consists of nine sections:

- Executable programs or shell commands
- System calls
- Library calls
- Games
- Special files
- File formats and conventions
- System administration commands
- Kernel routines
- Miscellaneous

To display the complete manual, enter:

man [command_name]

For example, you want to access the manual for the **ls** command:

man ls

Enter this command if you want to specify the displayed section:

man [option] [section_number] [command_name]

For instance, you want to see section 2 of the **ls** command manual:

man 2 ls

31. echo command

The **echo** command is a built-in utility that displays a line of text or string using the standard output. Here's the basic syntax:

echo [option] [string]

For example, you can display the text **Hostinger Tutorials** by entering:

echo "Hostinger Tutorials"

This command supports many options, such as:

- **-n** displays the output without the trailing newline.
- **-e** enables the interpretation of the following backslash escapes:
- **\a** plays sound alert.
- **\b** removes spaces in between a text.
- **\c** produces no further output.
- **-E** displays the default option and disables the interpretation of backslash escapes.

32. zip, unzip commands

Linux Basics

Use the **zip** command to compress your files into a **ZIP** file, a universal format commonly used on Linux. It can automatically choose the best compression ratio.

The **zip** command is also useful for archiving files and directories and reducing disk usage.

To use it, enter the following syntax:

zip [options] zipfile file1 file2....

For example, you have a file named **note.txt** that you want to compress into **archive.zip** in the current directory:

zip archive.zip note.txt

On the other hand, the [unzip command](#) extracts the zipped files from an archive. Here's the general format:

unzip [option] file_name.zip

So, to unzip a file called **archive.zip** in the current directory, enter:

unzip archive.zip

33. hostname command

Run the **hostname** command to know the system's hostname. You can execute it with or without an option. Here's the general syntax:

hostname [option]

There are many optional flags to use, including:

- **-a** or **-alias** displays the hostname's alias.
- **-A** or **-all-fqdns** displays the machine's Fully Qualified Domain Name (FQDN).
- **-i** or **-ip-address** displays the machine's IP address.

For example, enter the following command to know your computer's IP address:

hostname -i

34. useradd, userdel commands

Linux is a multi-user system, meaning more than one person can use it simultaneously. **useradd** is used to create a new account, while the **passwd** command allows you to add a password. Only those with root privileges or **sudo** can run the **useradd** command.

When you use the **useradd** command, it performs some major changes:

- Edits the **/etc/passwd**, **/etc/shadow**, **/etc/group**, and **/etc/gshadow** files for the newly created accounts.
- Creates and populates a home directory for the user.
- Sets file permissions and ownerships to the home directory.

Here's the basic syntax:

useradd [option] username

To set the password:

passwd the_password_combination

For example, to add a new person named John, enter the following command simultaneously:

useradd John

passwd 123456789

To delete a user account, use the **userdel** command:

userdel username

35. apt-get command

apt-get is a command line tool for handling Advanced Package Tool (APT) libraries in Linux. It lets you retrieve information and bundles from authenticated sources to manage, update, remove, and install software and its dependencies.

Running the **apt-get** command requires you to use **sudo** or root privileges.

Here's the main syntax:

apt-get [options] (command)

These are the most common commands you can add to **apt-get**:

Linux Basics

- **update** synchronizes the package files from their sources.
- **upgrade** installs the latest version of all installed packages.
- **check** updates the package cache and checks broken dependencies.

36. nano, vi, jed commands

Linux allows users to edit and manage files via a text editor, such as **nano**, **vi**, or **jed**. **nano** and **vi** come with the operating system, while **jed** has to be installed.

The **nano command** denotes keywords and can work with most languages. To use it, enter the following command:

nano [filename]

vi uses two operating modes to work – **insert** and **command**. **insert** is used to edit and create a text file. On the other hand, the **command** performs operations, such as saving, opening, copying, and pasting a file.

To use **vi** on a file, enter:

vi [filename]

jed has a drop-down menu interface that allows users to perform actions without entering keyboard combinations or commands. Like **vi**, it has modes to load modules or plugins to write specific texts.

To open the program, simply enter **jed** to the command line.

37. alias, unalias commands

alias allows you to create a shortcut with the same functionality as a command, file name, or text. When executed, it instructs the shell to replace one string with another.

To use the **alias** command, enter this syntax:

alias Name=String

For example, you want to make **k** the alias for the **kill** command:

alias k='kill'

On the other hand, the **unalias** command deletes an existing alias.

Here's what the general syntax looks like:

unalias [alias_name]

38. su command

The switch user or **su** command allows you to run a program as a different user. It changes the administrative account in the current log-in session. This command is especially beneficial for accessing the system through **SSH** or using the GUI display manager when the root user is unavailable.

Here's the general syntax of the command:

su [options] [username [argument]]

When executed without any option or argument, the **su** command runs through root privileges. It will prompt you to authenticate and use the **sudo** privileges temporarily.

Here are some acceptable options to use:

- **-p** or **–preserve-environment** keeps the same shell environment, consisting HOME, SHELL, USER, and LOGNAME.
- **-s** or **–shell** lets you specify a different shell environment to run.
- **-l** or **–login** runs a login script to switch to a different username. Executing it requires you to enter the user's password.

39. htop command

The **htop** command is an interactive program that monitors system resources and server processes in real time. It is available on most Linux distributions, and you can install it using the default package manager.

Compared to the **top** command, **htop** has many improvements and additional features, such as mouse operation and visual indicators.

To use it, run the following command:

Linux Basics

htop [options]

You can also add options, such as:

- **-d** or **-delay** shows the delay between updates in tenths of seconds.
- **-C** or **-no-color** enables the monochrome mode.
- **-h** or **-help** displays the help message and exit.

40. ps command

The process status or **ps** command produces a snapshot of all running processes in your system. The static results are taken from the virtual files in the **/proc** file system.

Executing the **ps** command without an option or argument will list the running processes in the shell along with:

- The unique process ID (**PID**)
- The type of the terminal (**TTY**)
- The running time (**TIME**)
- The command that launches the process (**CMD**)

Here are some acceptable options you can use:

- **-T** displays all processes associated with the current shell session.
- **-u username** lists processes associated with a specific user.
- **-A** or **-e** shows all the running processes.

Bonus Tips and Tricks

Here are some tips and tricks you can use to manage the Linux system:

- Enter the **clear** command to clean the Terminal screen.
 - Press the **Tab** button to autofill after entering a command with an argument.
 - Use **Ctrl + C** to terminate a running command.
 - Press **Ctrl + Z** to pause a working command.
 - Use **Ctrl + S** to freeze your Terminal temporarily.
 - Press **Ctrl + Q** to undo the Terminal freeze.
 - Use **Ctrl + A** to move to the beginning of the line.
 - Press **Ctrl + E** to bring you to the end of the line.
 - When executing multiple commands in a single line, use **(;)** to separate them. Alternatively, use **&&** to only allow the next command to run if the previous one is successful.
-

41. How to replace text in a file with a sed command

Linux Basics

The Linux **sed command** is most commonly used for substituting text. It searches for the specified pattern in a file and replaces it with the wanted string.

How to Replace a String Using the sed Command

The sed command is commonly used for replacing text. It will search for a specified pattern in a file and change it with the desired string.

To do it, use the substitution command **s** and delimiters to separate each field. Replace the “**old_string**” value with the original name and “**new_string**” with the preferred text:

```
sed 's/old_string/new_string/' samplefile.txt
```

For example, to substitute the word “**images**” with “**photos**” in the **scenery.txt** file, enter the following:

```
sed 's/images/photos/' scenery.txt
```

How to Replace the nth Occurrence of a Pattern in a Line Using a sed Command

Insert a corresponding character, either a slash (/) or vertical bar (|), and substitute the hash (#) symbol with the desired number to replace the nth occurrence of a pattern in a line. Here's the basic syntax:

```
sed 's/old_string/new_string/#' samplefile.txt
```

For example, to replace the first occurrence of the word “**music**” with “**song**” in a line inside the **playlist.txt** file, enter:

```
sed 's/music/song/1' playlist.txt
```

How to Replace All the Occurrences of the Pattern in a Line Using the sed Command

By default, sed only substitutes the first occurrence of a specified string in every line. It looks for the initial instance, replaces it, and moves on to the following input line.

To replace all the pattern occurrences in one line, add a substitute flag **/g** for global replacement. Here's the sed script:

```
sed 's/old_string/new_string/g' samplefile.txt
```

For example, to replace all occurrences that contain “**eagle**” with “**falcon**” in a line inside **animals.txt**, run:

```
sed 's/eagle/falcon/g' animals.txt
```

42. What is a cURL Command and How to Use It?

cURL commands can download files from a remote location. You can do it in two different ways:

- **-O** will save the file in the current working directory with the same file name as remote.
- **-o** lets you specify a different file name or location

An example of this is as shown below:

```
curl -O http://testdomain.com/testfile.tar.gz
```

The above command will save this as **testfile.tar.gz**.

```
curl -o newtestfile.tar.gz http://testdomain.com/testfile.tar.gz
```

The above command will save this as **newtestfile.tar.gz**.

Linux Basics

If for some reason, the download gets interrupted, you can resume it using cURL. You can do this with the following command:

```
curl -C - -O http://testdomain.com/testfile.tar.gz
```

Using cURL, we can also download multiple files, like shown below:

```
curl -O http://testdomain.com/testfile.tar.gz -O http://mydomain.com/myfile.tar.gz
```

43. ifconfig

Moving on to the networking section in Linux, we come across the ifconfig and traceroute commands which will be frequently used if you manage a network.

The ifconfig command will give you the list of all the network interfaces along with the IP addresses, MAC addresses and other information about the interface.

```
root@ubuntu:~ -->> ifconfig
```

There are multiple parameters that can be used but we'll work with the basic command here.

```
root@ubuntu:~ -->> ifconfig
docker0: flags=4099<UP,BROADCAST,MULTICAST> mtu 1500
    inet 172.17.0.1 netmask 255.255.0.0 broadcast 172.17.255.255
    ether 02:42:3b:09:02:00 txqueuelen 0 (Ethernet)
    RX packets 0 bytes 0 (0.0 B)
    RX errors 0 dropped 0 overruns 0 frame 0
    TX packets 0 bytes 0 (0.0 B)
    TX errors 0 dropped 0 overruns 0 carrier 0 collisions 0
```