Basic Linux Commands You Should Know

commends: Definitions:

ls	The ls command is a Linux shell command that is used to list the contents of a directory. When you run the ls command, it will display the names of the files and directories in the current directory.
ls -a	lists hidden files along with other files and directories.
ls -al	lists files and directories with file permissions, ownership, and size information.
ls -ltr	list file names in the last modification time in reverse order.
ls -h	Use human-readable file sizes (e.g. 1K, 2M, 3G) instead of bytes.
ls -r	Reverse the order of the listing.
cd	The cd command is a Linux shell command that is used to change the current working directory. The name "cd" stands for "change directory".
cd **..**	To go back to the previous directory, you can use the cd command with a special directory name .. (two dots), which represents the parent directory. For example:
Mkdir	The mkdir stands for make directory. The mkdir command is used to create a new directory in Linux.
cat	In Linux, the cat command is a shell command that is used to concatenate and display the contents of files. The name "cat" stands for "concatenate". For example, to display the contents of a file named "file.txt" in the terminal window, you can run:

	cat file.txt
cat > file.txt	create a new file named file1.txt, type:
cat file.txt > help.txt	The existing contents of the department.txt file will be overwritten by the contents of users.txt:
cat users.txt | tr a-z A-Z > output.txt	To convert the content of file users.txt from lowercase to uppercase and store it in output.txt, use the following command:
df	The df stands for disk filesystem. It is used to get a complete summary of used and available disk space of the file system in your Linux computer.
df -a	displays all file systems.
df -h	displays output in a human-readable format.
df -T	displays file system type.
df -m	displays output in mebibyte (MiB).
df -g	displays output in gibibyte (GiB).
du	The du command in Linux stands for "disk usage" and is used to obtain disk usage information about files and directories on your computer. By default, the du command displays the disk usage of files in terms of the number of 1024-byte blocks they occupy. However, you can use the -h option with the du command to display the output in a more human-readable format that uses kilobytes (KB), megabytes (MB), gigabytes (GB), and so on.

cp	The cp command is used to copy files and directories from the source directory to another location. Example
	`cp users.txt Documents/records/`
	
	To create a copy of the file file2.txt with the name file2_backup.txt in your current working directory.
	
	`cp file2.txt file2_backup.txt`

mv	Use the mv (stands for move) command to move files or directories from the source to the destination directory. It can be used to rename a file/directory.
	The following mv command moves the users.txt file to the Documents/records directory.
	
	`mv users.txt Documents/records`
	
	To rename the employees.txt file to users.txt:
	
	`mv employees.txt users.txt`

rm	Use the rm (stands for remove) command to remove the given file, multiple files, or a group/type of files. By default, the rm command does not require user confirmation to delete a file. Options can be used with the rm command for the following needs: ● `-i` deletes files in interactive mode. ● `-f` forcefully removes the write-protected file. ● `-r` recursively deletes files, directories, and subdirectories in the specified parent directory. ● `-d` deletes the specified empty directory. Removes the file named documents.txt from the current directory:

	`rm documents.txt`
	
	To remove directories and their contents recursively, type:
	
	`rm -r Documents`
	
	To remove the directory without being prompted, type:
	
	`rm -rf dir1`

man & help	Use the man command to display the user manual of the Linux command. Almost all the Linux commands have man pages, which is a kind of documentation that is displayed on the terminal. The Linux commands manual page explains what a particular command does, syntax, and accepted arguments.
	
	Type man followed by the command name to display the command user manual page.
	
	`man mkdir`
	
	Similarly, you can also use the –-help option to display the help pages of a particular command.
	
	`mkdir --help`

chmod	The chmod command is used to change the mode (permission) of a file. The basic permissions a file can have are r(read), w(write), and x(execute). Using numeric octet's mode you can set read (value is 4), write (value is 2 and

	execute (value is 1)
	
	permissions for owner, group and all others.
	
	For example to give file1.txt owner and group read and write permissions and only read permission to all others, type:
	
	`chmod 664 file1.txt`
	
	You may use -R option to recursively set permissions for all files and directories in a given directory.
	
	To make the file executable, type:
	
	`chmod +x myscript.sh`

pwd	pwd stands for present working directory. The pwd command prints the absolute path of the current directory on your Linux terminal. The present working directory is the directory in which you are currently working.
	
	`pwd`
	
	Output
	
	`/home/linuxopsys`
	
	In the example /home/linuxopsys is the current directory.

uname	Use the uname (stands for UNIX name) command to display fundamental information about your Linux computer. By default, this command prints only the type of the operating system, such as Linux. You can use different options with this command to print other information about your computer such as OS, kernel version, machine name, network, and so on.

	• `-a` display all information about your computer. • `-s` display Linux kernel name. • `-m` displays machine information. • `-i` display hardware platform information. The output of `uname -a` command on my machine: `Linux linux 5.13.0-37-generic #42~20.04.1-Ubuntu SMP Tue Mar 15 15:44:28 UTC 2022 x86_64 x86_64 x86_64 GNU/Linux` All system information including OS type, name, user name, kernel release, and hardware platform is displayed.

who	Use the who command to list currently logged-in users on your Linux computer. This command also supports few options to display specific information, such as:
	
	• `-a` displays all available information about logged-in users.
	• `-b` displays boot time.
	• `-H` displays header names.
	
	The output of who command looks like this
	
	`root pts/0 2022-04-28 01:30 (192.168.1.10)`
	
	`bob pts/1 2022-04-28 02:40 (192.168.1.22)`
	
	`linuxopsys pts/2 2022-04-28 02:05 (192.168.1.22)`
	
	It shows 3 users with their logged in date and time.

hostname	Use hostname command to display the Domain Name System (DNS) name and set the hostname or Network

	Information System (NIS) of your system.
	Display host name and domain name, type:
	
	`hostname`
	
	`linuxopsys`
	
	Some distributions allow you to set hostname for your machine, type:
	
	`sudo hostname NEW_HOSTNAME`
	
	You can replace NEW_HOSTNAME with your actual machine name.

top	Use the top or htop commands to monitor the server's processes or vital system resources in real-time. Both these commands are used for the same purpose, but their output is different. The default output displays information about top CPU-consuming processes along with RAM usage. `top` The command also displays system up time, load average, number of logged in users, memory, total tasks, and running tasks. Htop is an improved version of the top command. It display colorful text and allows for output scrolling. Htop doesnt comes preinstalled on most Linux distros.
touch	Use the touch command to create an empty file on your Linux computer, or to modify the timestamp of a file. The primary function of this command is to update file timestamps, but it is most commonly used to create a file. The touch command can update access and modification timestamps of the specified file, and create the file if it does

	not already exist.
	
	To create a new empty file named install.doc, type:
	
	`touch install.doc`
	
	In this example, the modification time of the file updates.txt is updated to the current time.
	
	`touch -m updates.txt`

zip and unzip	Use the zip command-line utility to create an archive of files, and the unzip command for uncompressing a zip archive. The zip command can also be used to modify a zip archive.
	
	To create a zip archive named users.zip from the specified files:
	
	`zip users.zip output.txt install.doc newfiles.txt updates.txt`
	
	To extract the contents of the users.zip archive to the current directory:
	
	`unzip users.zip`

tar	Use the tar command to create a compressed or uncompressed archive of specified files. You can also use this command to modify, maintain, or extract tar archives. Tar archives can combine multiple files or directories into a single file. The tar command provides multiple options to create a tarball: ● -c creates an archive. ● -x extracts the archive.

	● -f assigns filename to the archive. ● -t displays files in an archive. To create an uncompressed archive file named users.tar of the specified files. `tar cf users.tar updates.txt newfiles.txt`

ln	Use the ln command for creating hard and soft links to existing files or directories. By default, this command creates hard links. Specify the -s option to create symbolic or soft links.
	
	To create a symbolic link from the updates.txt file to the newitems.txt file:
	
	`ln -s updates.txt newitems.txt`
	
	Here we can see the symlink that we created in the previous example:
	
	`ls -l newitems.txt`
	
	`lrwxrwxrwx 1 linuxopsys linuxopsys 11 Apr 14 13:53 newitems.txt -> updates.txt`

sudo	Use the sudo command to execute commands as root or another Linux user, as specified in your security policy. This command allows you to execute commands that otherwise require a root password. Specify the command that requires root permissions after the sudo command. To run the useradd command as sudo to create a new user named bob: `$ sudo useradd bob`

apt	Use the apt (stands for advanced package tool) command to install, update, or delete packages on Ubuntu and Debian Linux distributions.
	
	Some of the most commonly used apt commands are apt update, apt upgrade, apt install, apt remove, apt list, and apt search.
	
	To update the package index by pulling the latest updates from the APT repository:
	
	`sudo apt update`
	
	To install the latest updates to all Linux packages:
	
	`sudo apt upgrade`
	
	Let's use apt command to install a package. To install the htop application, type:
	
	`sudo apt install htop`

dnf	DNF is a software package manager and is the successor to YUM (Yellow-Dog Updater Modified). Use the dnf (stand for dandified yum) command to install, update, or delete packages on Red Hat-based Linux distributions.
	
	To install the latest updates to all Red Hat Linux packages:
	
	`sudo dnf upgrade`
	
	To remove the package epel-release:
	
	`sudo dnf remove epel-release`

useradd	Use the useradd command to add new users to your Linux system. This command enables you to add a new user with specific group preferences, user ID (UID), and login shell.

	When you run this command without any option, it creates a user using the default account settings specified in the /etc/default/useradd file. To create a new user named steve that has GID 1000, and UID 1021, and will expire on 31st May 2022: `sudo useradd -g 1000 -u 1021 -e 2022-05-31 steve`
groupadd	Use the groupadd command to add a new group to your Linux system. This command enables you to add a new group with specific group preferences and override default /etc/login.def default values. When you create a new user group using this command, it adds a new entry in the /etc/group file. To create a new group named docs that has GID 1018: `sudo groupadd -g 1018 docs`
usermod	Use the usermod command to change the properties of existing users in Linux. Using this command, you can modify the password, primary group, login directory, expiry date, and other user attributes. You can also add a user to an existing group using this command. To add the user steve to sudo group, type: `sudo usermod -aG sudo steve`

ps	Use the ps command to check the status of the active processes on your Linux system, and display information about these processes. Administrators can use this information to kill the processes or to set process priorities.
	
	To show all running processes in the default output format:
	
	`ps`
	
	To display all running processes in full format:
	
	`ps -af`

kill	Use the kill command to manually terminate a given process. It is located in /bin/kill and it sends signals to another process that terminates the specified process.
	
	To display all available signals:
	
	`kill -l`
	
	To kill the PID 257:
	
	`kill 257`

ping	Use the ping, also known as the Packet Internet Groper, command to check network connectivity between two computers. You can use this command to troubleshoot connectivity, name resolution, and reachability. It accepts an IP address or an URL as an input. To check network connectivity to 172.168.1.1 IP address and send 5 data packets: `ping -c 5 172.168.1.1`

vi and nano	Use the vi and nano commands to create a new file, read a file, or edit an existing file. Vi is the default text editor for several Linux distributions and it is very simple to use. Nano is a terminal-based text editor that is ideal for making some changes to any files or creating basic plain text files.
	
	To open the existing file or to create a new file:
	
	`vi updates.txt`
	
	or
	
	`nano filename.txt`

history	Use the history command to view the history of all the previously executed terminal commands that you run. The maximum number of entries that BASH should store can be defined in your .bashrc file. Linux commands that you execute are considered events and every event is associated with an event number. You can recall, remove, or change commands using their event numbers.
	
	To list the last 5 commands, including itself:
	
	`history 5`
	
	To remove the history of event number 1525:
	
	`history -d 1525`
	
	Note: history command behavior may change depending on the shell you are being used

passwd	Use the passwd command to change the user account password. A normal user can change only their own password, but the root user or user with sudo privileges can change the password of any user account. This command can also be used to delete or expire an account

	password. To change the password for the user bob: `sudo passwd bob`
which	Use the which command to locate the executable file of the specified command. This command searches for the executable file location in the $PATH environment variable. The which command returns any of the following status: To display executable file locations for useradd command, type: `which useradd /usr/sbin/useradd`
less	Use the less command to view the contents of a file one screen at a time, starting at the beginning. This command provides the capability of both backward and forward navigation. The less command has quick file access because it does not access the entire file if the file is large. To display the contents of the updates.txt file one screen at a time. `less updates.txt`
tail	Use the tail command to view the contents of a file, starting at the bottom. By default, it displays the last 10 lines of the file, but you can change this default behavior. The tail command can also be used to monitor the file. When you add new lines to the file, then the display is updated. To display the last 10 lines of the file updates.txt: `tail updates.txt`

	To display the last 15 lines of the file access.log: `tail -n 15 access.log`
head	Use the head command to display the contents of a file, starting at the beginning. By default, it displays the first 10 lines of the file, but you can change this default behavior. To display the first 10 lines of the file updates.txt: `head updates.txt`
grep	Use the grep, also known as global regular expression print, the command to search for a word or string of characters in a file or directory. The search pattern is termed regular expression. When the grep command finds a matching string in the file, then this command prints the line that contains the string. To search for the string 'welcome' in the file log.txt: `grep welcome log.txt`
diff	Use the diff command to do a line-by-line comparison of two files and display their differences. You can also use this command to compare the contents of Linux directories. To show differences between two files named source.java and newsource.java, type diff /dir1/source.java /dir2/newsource.java To show differences between Documents and Docs directories.

	`diff Documents/ Docs`

find	Use the find command to search and locate the files and directories that match the specified conditions. This command enables you to search using file or folder name, creation date, modification date, permissions, and owner.
	
	To search the file named program.py in the /home/linuxopsys/projects directory, type:
	
	`find /home/linuxopsys/projects -type f -name program.py`
	
	To search for files with given permissions in the current directory:
	
	`find . -perm 664`

echo	Use the echo command to display the specified text or string on the terminal. This command is most commonly used in batch files and scripting languages to display standard text output. You can also use this command to display the values of an environment variable.
	
	To print the string “This is a sample string” on the terminal:
	
	`echo "This is a sample string"`
	
	`This is a sample string`
	
	To print the value of the PATH environment variable:
	
	`echo $PATH`
	
	Output:
	
	`/usr/local/sbin:/usr/local/bin:/usr/sbin:/us`

	`r/bin:/sbin:/bin:/usr/games:/usr/local/games:/snap/bin`

shutdown	Use the shutdown command to gracefully and securely shut down your Linux computer.
	
	Use the reboot command to restart your Linux computer. You can also use the shutdown -r command to restart your computer using the shutdown command.
	
	To restart your system after 5 minutes:
	
	`shutdown -r +5`
	
	To immediately restart your computer:
	
	`reboot`

`alias`	The Alias command allows you to define temporary aliases for current sessions. It allows you to execute one or more commands.
	
	Create a new alias named 'lsall', type:
	
	`alias lsall="ls -a"`
	
	Lists all aliases for the current session:
	
	`alias`
	
	Note: Alias behavior may change depending on the shell you are being used. It is traditionally used in bash shell.

exit	The exit command ends a shell session and closes the terminal you are using exit : `exit`
wget	The wget command is used to retrieve or download content from the internet. To download WordPress using wget, type: `wget https://wordpress.org/latest.tar.gz` The latest.tar.gz file will be downloaded to the current directory.
clear	The clear command is used in Linux to clear the terminal screen. This is similar to cls command in other operating systems. `clear`