1.How to check if Git available on your system ?
ANSWER:-
To check if Git is installed on your system, you can open a terminal or command prompt and type the command git --version.

If Git is installed, this command will display the version of Git that is currently installed on your system. If Git is not installed, you will see an error message indicating that the command is not recognized.

2.How to intialize a new Git repository ?
ANSWER:-
To initialize a new Git repository, you can use the git init command. This command creates a new, empty repository in the current directory.

Here are the steps to initialize a new repository:

1.Open a terminal or command prompt and navigate to the directory where you want to create the repository.

2.Type git init and press enter. This will create a new, empty repository in the current directory.

3.Optional: you can add files to the repository by copying or creating them in the directory and use git add command to stage them for commit.

4.use git commit -m "Initial commit" to commit the changes.

3.How to tell git about your name and email ?
ANSWER:-
To tell Git your name and email, you can use the “git config” command. This command is used to set various configuration options for your Git repository.

Here are the steps to set your name and email:

Open a terminal or command prompt and navigate to the directory of your Git repository.

Type git config --global user.name "Your Name" and press enter. This will set your name in Git.

Type git config --global user.email "your.email@example.com" and press enter. This will set your email in Git.

By using --global option this will set the name and email for all repositories in your system.

4.How to add a file to the staging area ?

ANSWER:-
To add a file to the staging area in Git, you can use the git add command. The staging area is a temporary holding area where changes to files are tracked before they are committed to the repository.

Here are the steps to add a file to the staging area:

1.Open a terminal or command prompt and navigate to the directory of your Git repository.

2.Type git add <file-name> and press enter. This will add the specified file to the staging area.

3.Alternatively, you can use git add . to add all the files in the current directory to the staging area.

4.You can also use git add -A to add all files including the new, modified, and deleted files to the staging area.

5.How to remove a file from the staging area ?
ANSWER:-
To remove a file from the staging area in Git, you can use the git reset command. The git reset command can be used to unstage files and bring them back to their last committed state.

Here are the steps to remove a file from the staging area:

1.Open a terminal or command prompt and navigate to the directory of your Git repository.

2.Type git reset <file-name> and press enter. This will remove the specified file from the staging area.

3.Alternatively, you can use git reset without any file name, this will remove all files from the staging area.

6.How to make a commit ?
ANSWER:-
To make a commit in Git, you first need to add the changes you want to commit to the staging area using the git add command. Once the changes are in the staging area, you can use the git commit command to commit the changes to the repository.

Here are the steps to make a commit:

1.Open a terminal or command prompt and navigate to the directory of your Git repository.

2.Use the git add command to add the changes you want to commit to the staging area. For example, git add <file-name> or git add . to add all the files in the current directory to the staging area.

3.Use the git commit command to commit the changes to the repository. You need to provide a commit message to describe the changes you made. The commit message should be in present tense, and explain what the changes are.

7.How to send your changes to a remote repository ?
ANSWER:-
To send your changes to a remote repository in Git, you first need to use the git push command. The git push command sends the changes you've made locally to a remote repository.

Here are the steps to send changes to a remote repository:

1.Open a terminal or command prompt and navigate to the directory of your Git repository.

2.Use the git push command to send your changes to the remote repository. The command takes the following format: git push <remote> <branch>

3.<remote> is the name of the remote repository where you want to push the changes and <branch> is the name of the branch where you want to push the changes.

4.If you cloned the repository from a remote server, the name of the remote is usually origin, and the default branch is master.

git push origin master

5.If you haven't set up a remote repository, you can use the git remote add command to add a new remote repository.

6.After that, you can use the git push command to send your changes to the remote repository.

8.What is the difference between clone and pull ?
ANSWER:-
Git clone:-
git clone is how you get a local copy of an existing repository to work on. git pull (or git fetch + git merge) is how you update that local copy with new commits from the remote repository.

git clone is used for just downloading exactly what is currently working on the remote server repository and saving it in your machine's folder where that project is placed. Mostly it is used only when we are going to upload the project for the first time. After that pull is the better option.

Git pull:-

The git pull command is used to fetch and download content from a remote repository and immediately update the local repository to match that content. Merging remote upstream changes into your local repository is a common task in Git-based collaboration workflows.

git pull is a (clone(download) + merge) operation and mostly used when you are working as teamwork. In other words, when you want the recent changes in that project, you can pull.