HOW TO INSTALL AND USE PYCHARM IDE

PyCharm is developed by JetBrains and it is world no. 1 IDE for Python programmers. At present, this Python IDE is being used by large enterprises like Twitter, Pinterest, HP, Symantec and Groupon.

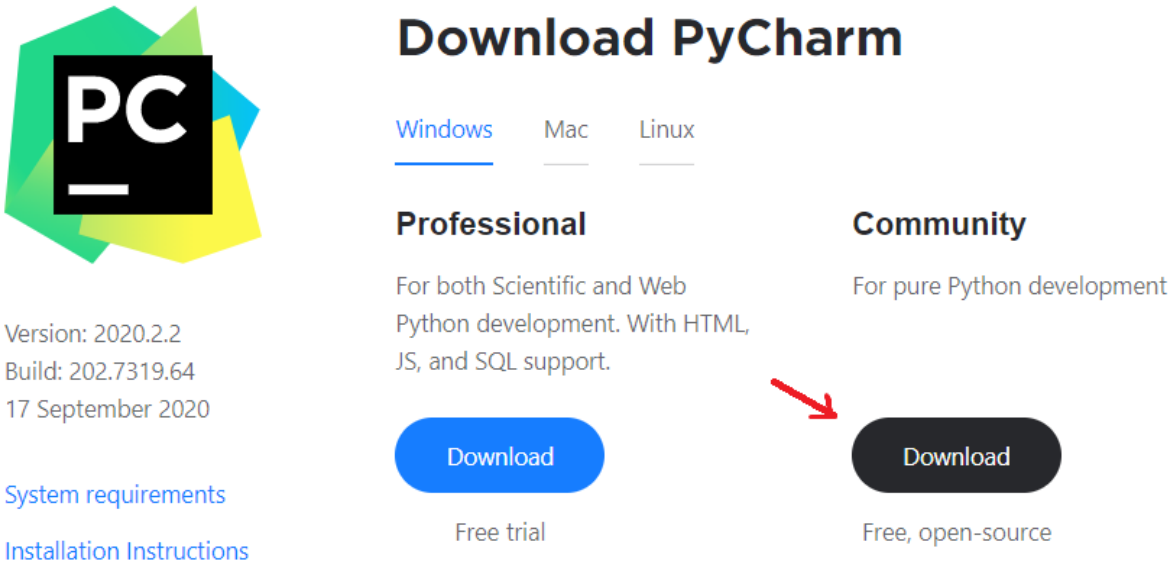
PyCharm is also compatible with Windows, Linux, and macOS. Pycharm provides all the tools we need for writing and executing Python programs. Also, Pycharm can be used to create Django projects.

PyCharm does not come with its own Python software. Hence, we should install Python in our system before we install PyCharm. Now, let us see how to install Community edition of PyCharm, since it is freely available.

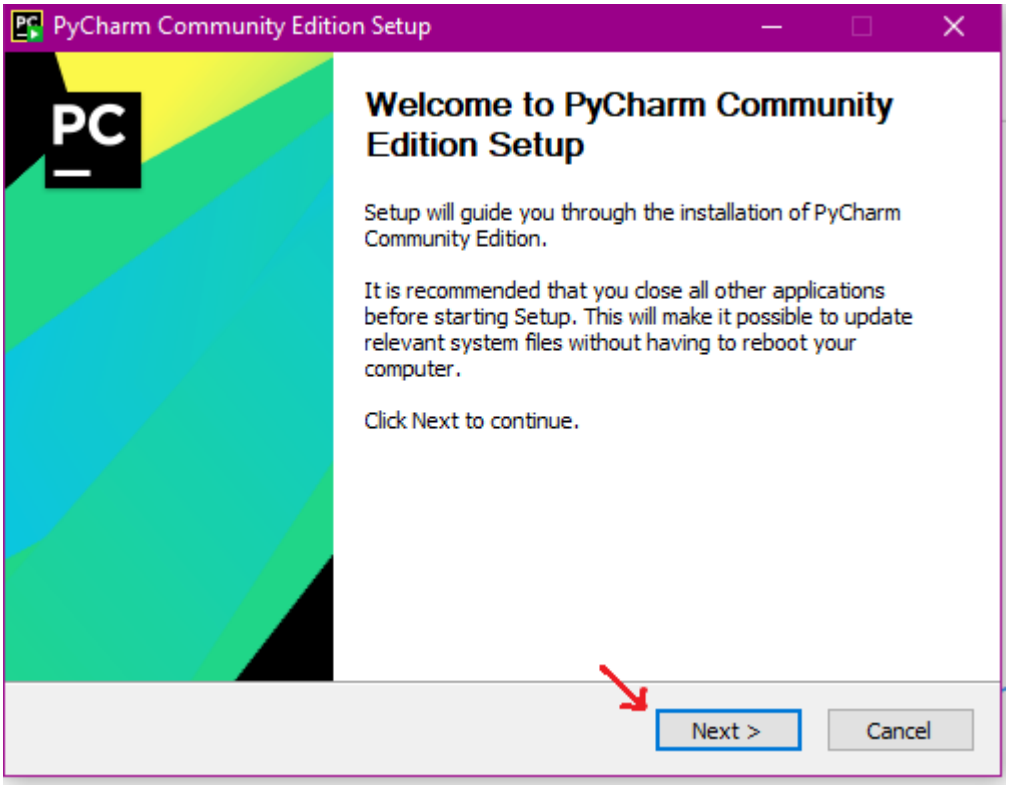
Step 1) To download PyCharm visit the website url:

https://www.jetbrains.com/pycharm/download/

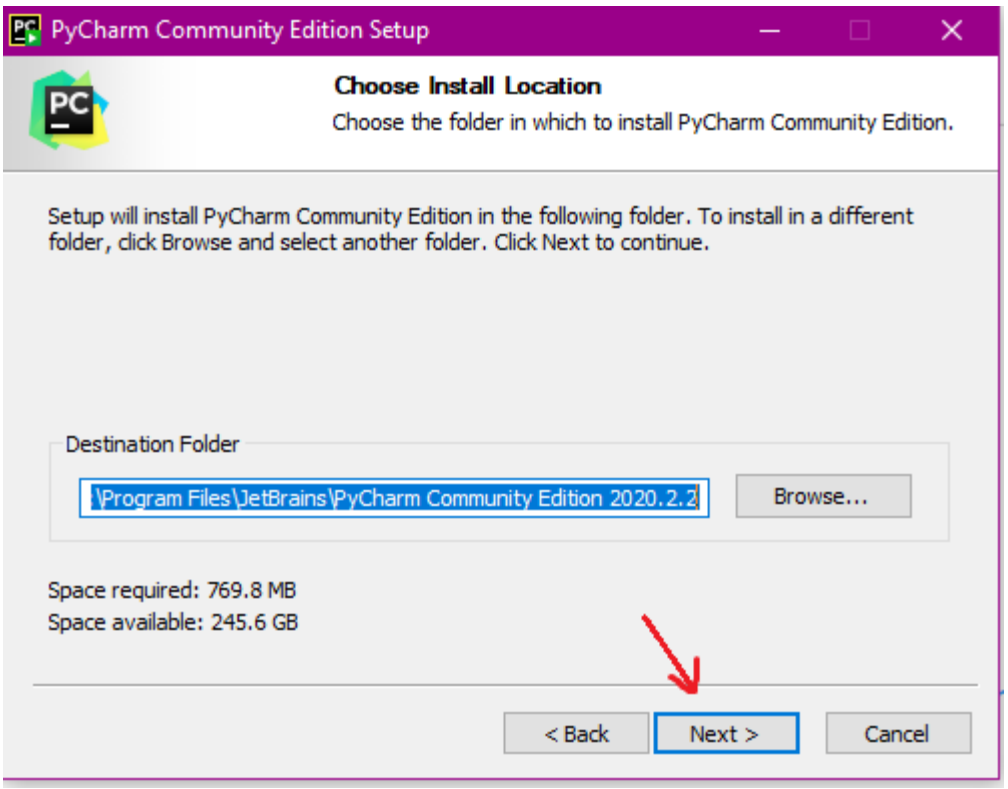
and click the 'Download' link under the Community Section. A file by the name "pycharm-community-2020.2.2.exe" will be downloaded into our system.



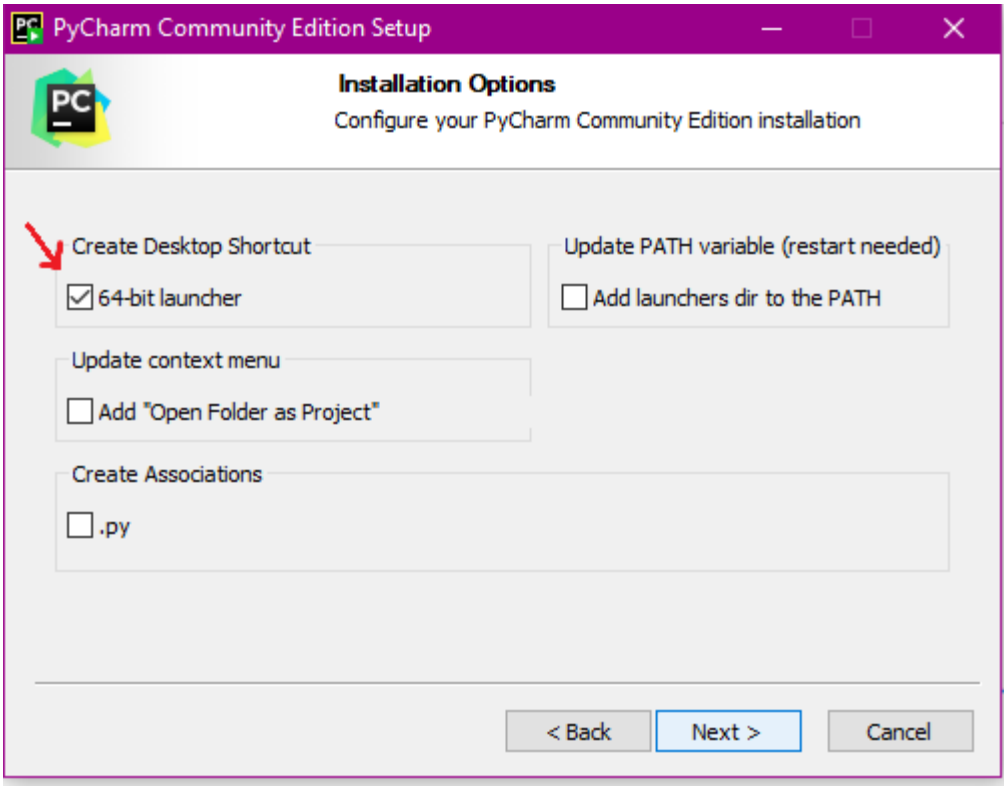
Step 2) Once the download is complete, run the .exe file to install PyCharm. The setup wizard should have started. Click "Next".



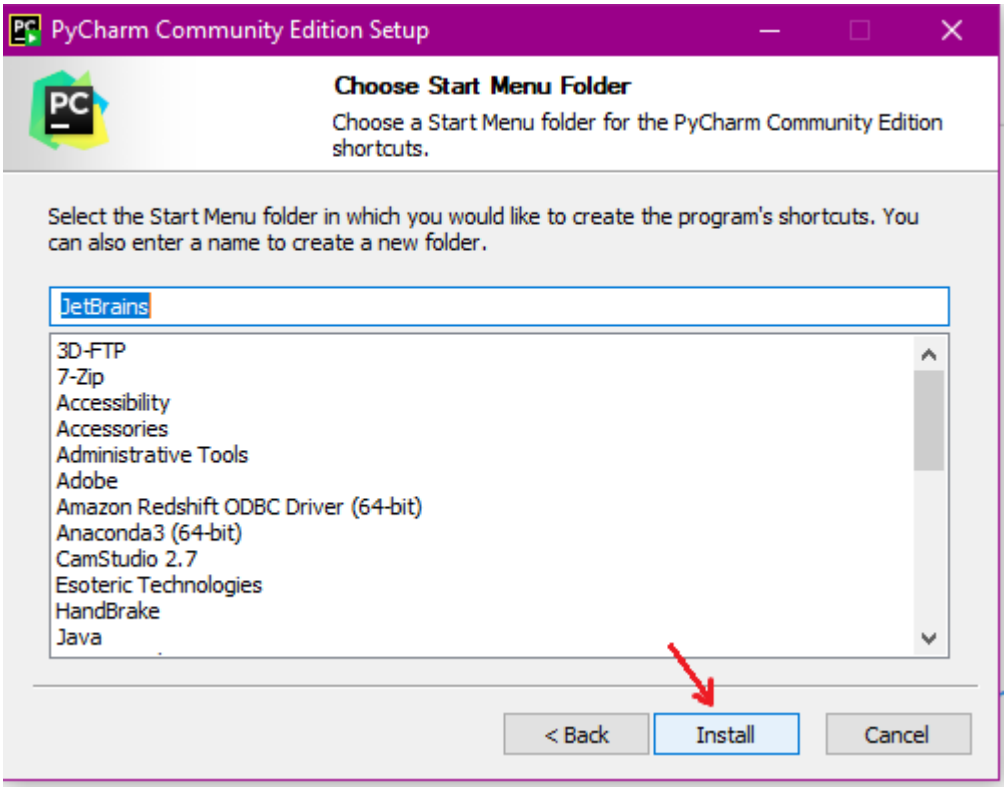
Step 3) On the next screen, change the installation path if required. Click “Next”.



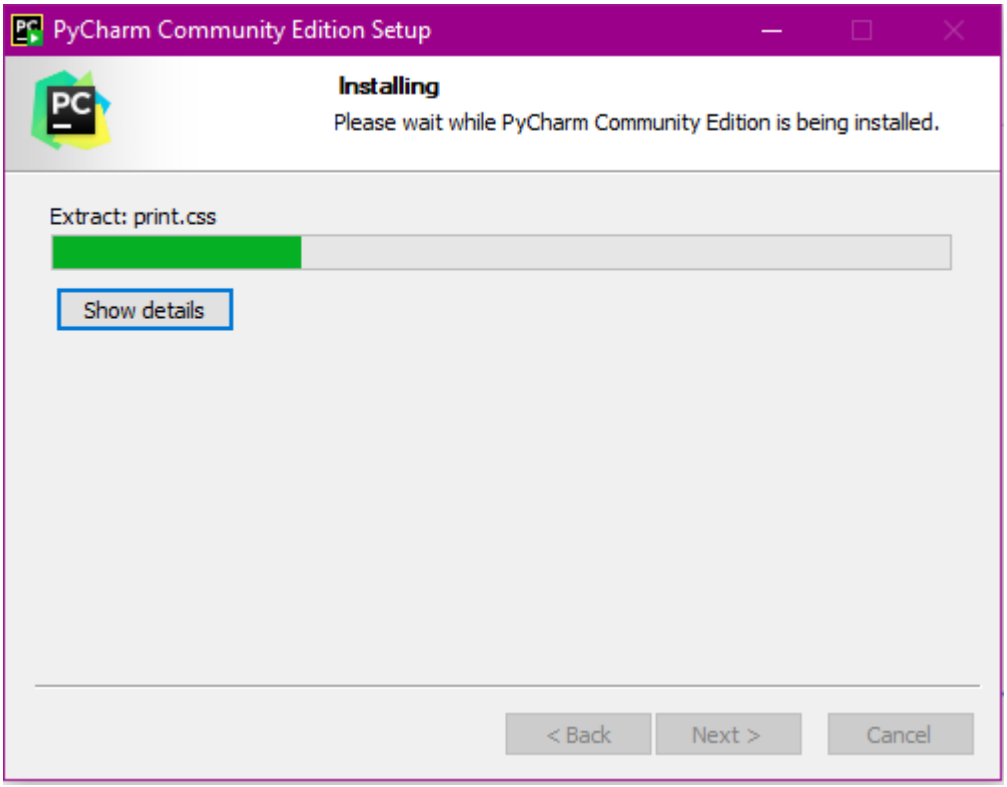
Step 4) On the next screen, we can create a desktop shortcut if we want. Select “64-bit launcher”. We can also select “Create Associations” checkbox to make Python programs open in PyCharm by default. After that, click on “Next”.



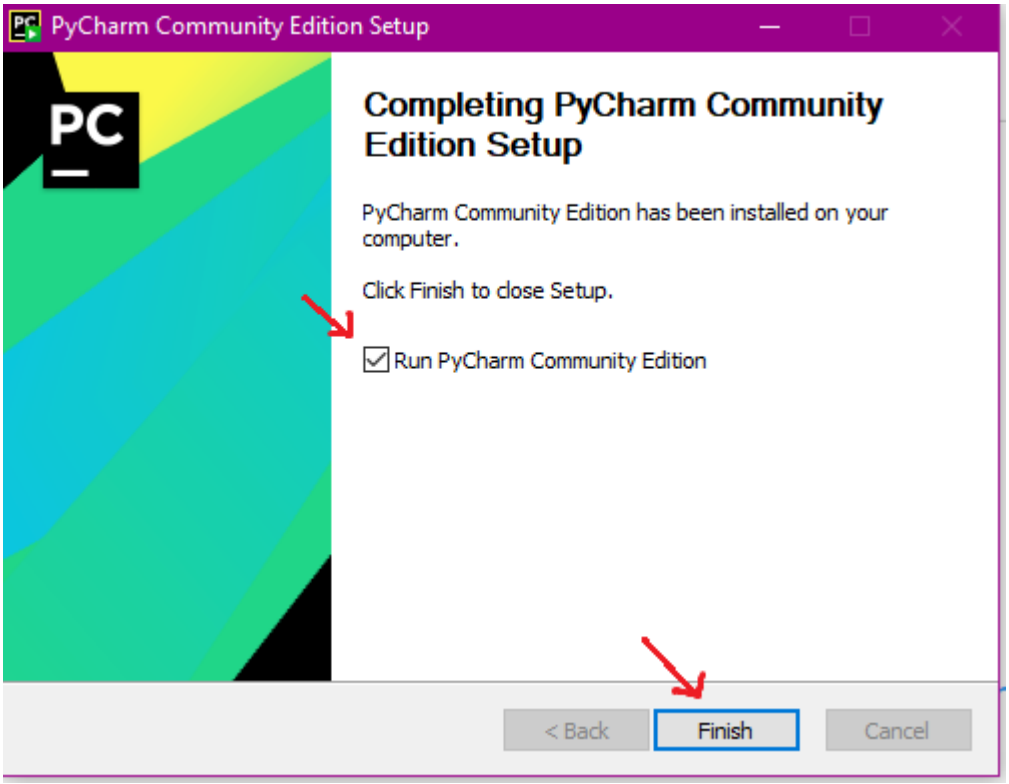
Step 5) Choose the start menu folder. Keep selected JetBrains and click on “Install”.



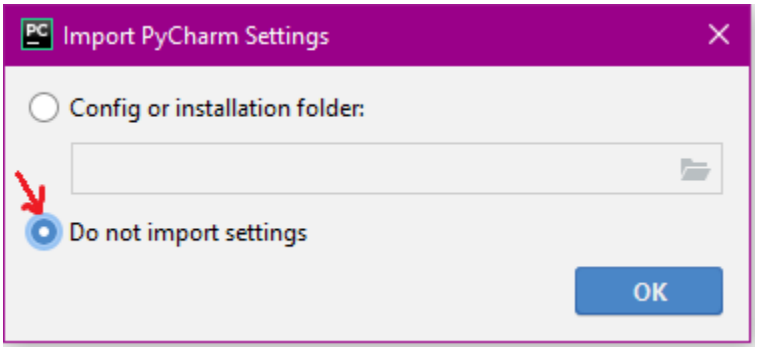
Step 6) The installation will start and we have to wait for its completion.



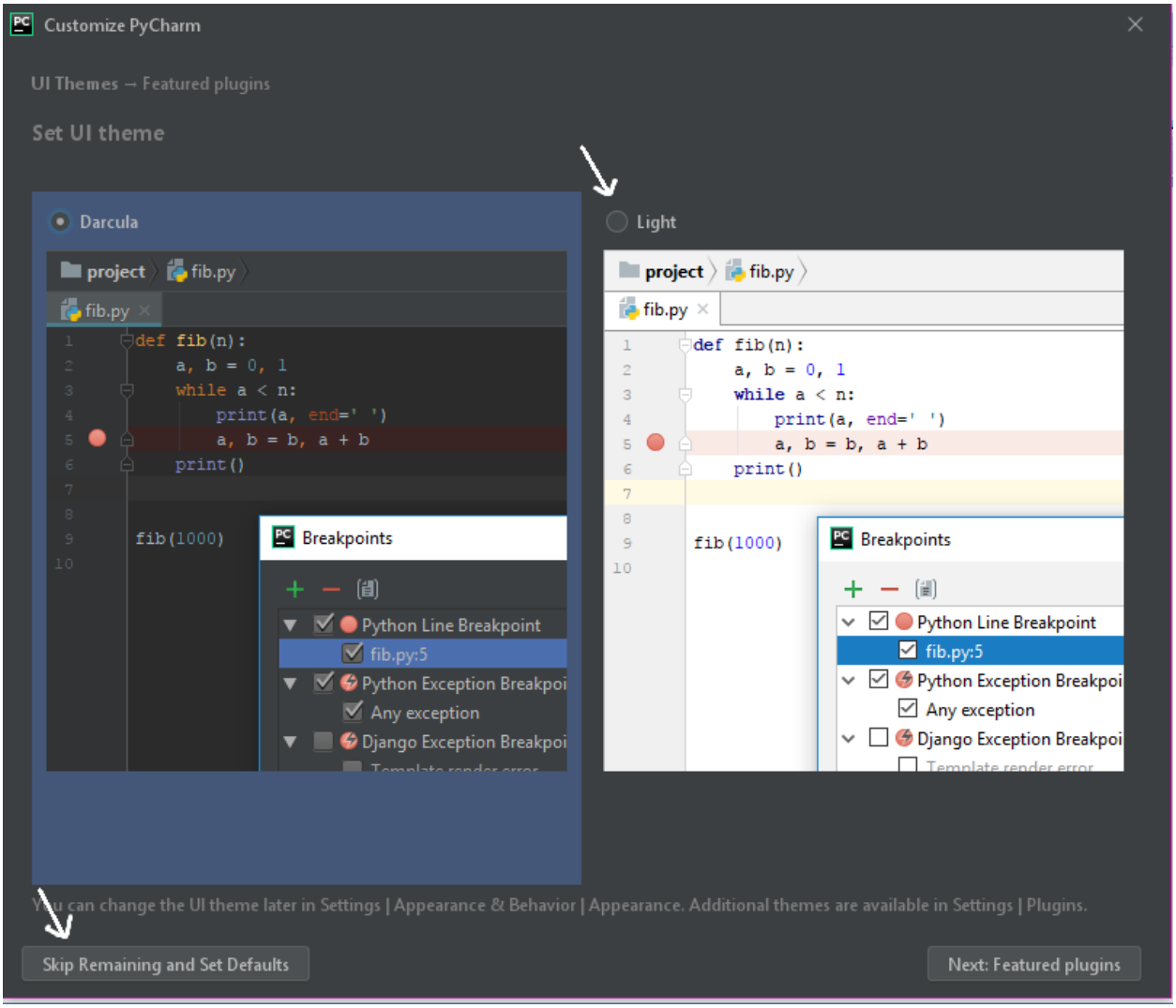
Step 7) Once installation is finished, we can click the “Run PyCharm Community Edition” box first and then click “Finish”.



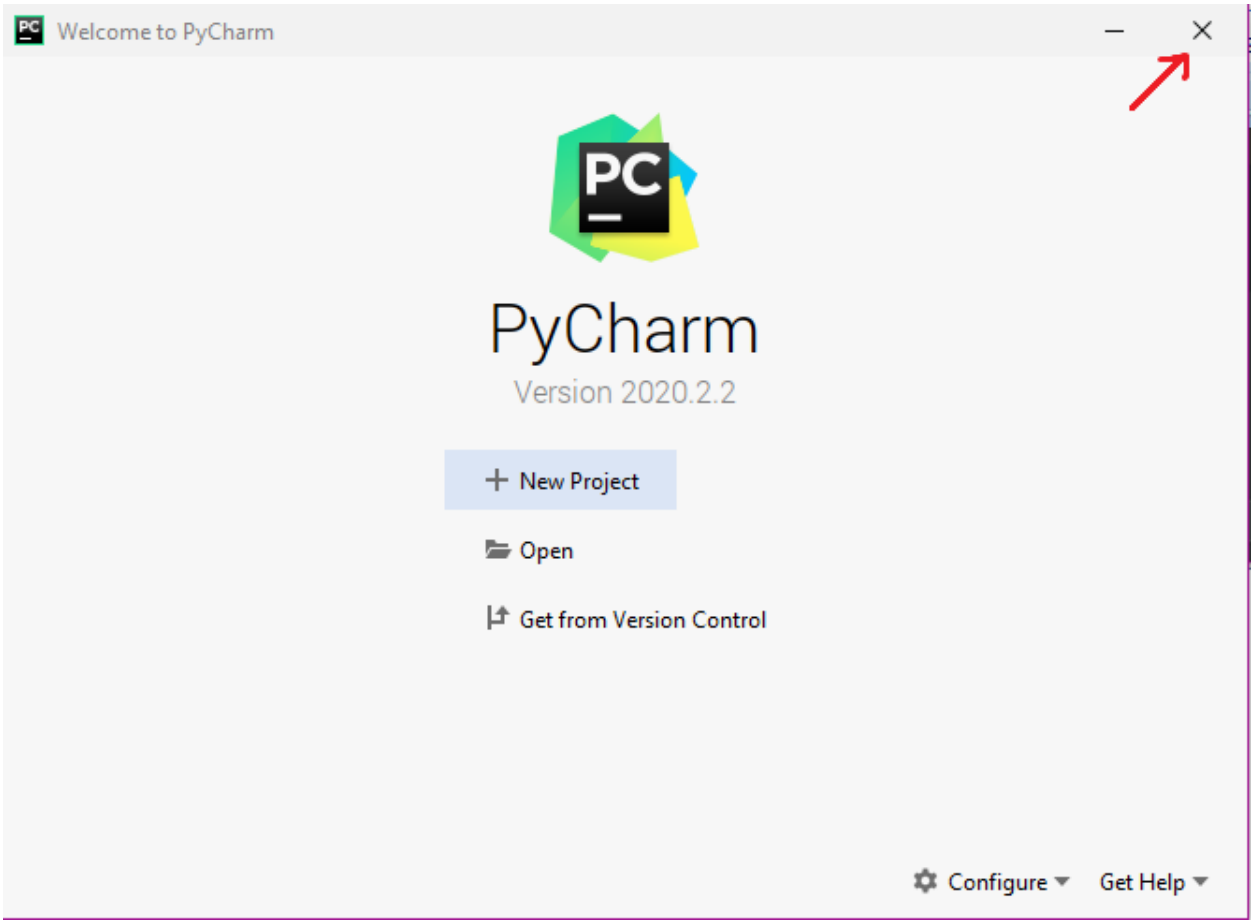
Step 8) After we click on "Finish", the Following screen will appear. Select “Do not import settings” and click on “OK”.



Step 9) In the next step, we can select a theme either “Darcula” or “Light” . Then click on “Skip Remaining and Set Defaults” button.



Step 10) Installation completes with the following screen. Click on ‘X’ button to close this screen.



CREATING FIRST PROGRAM

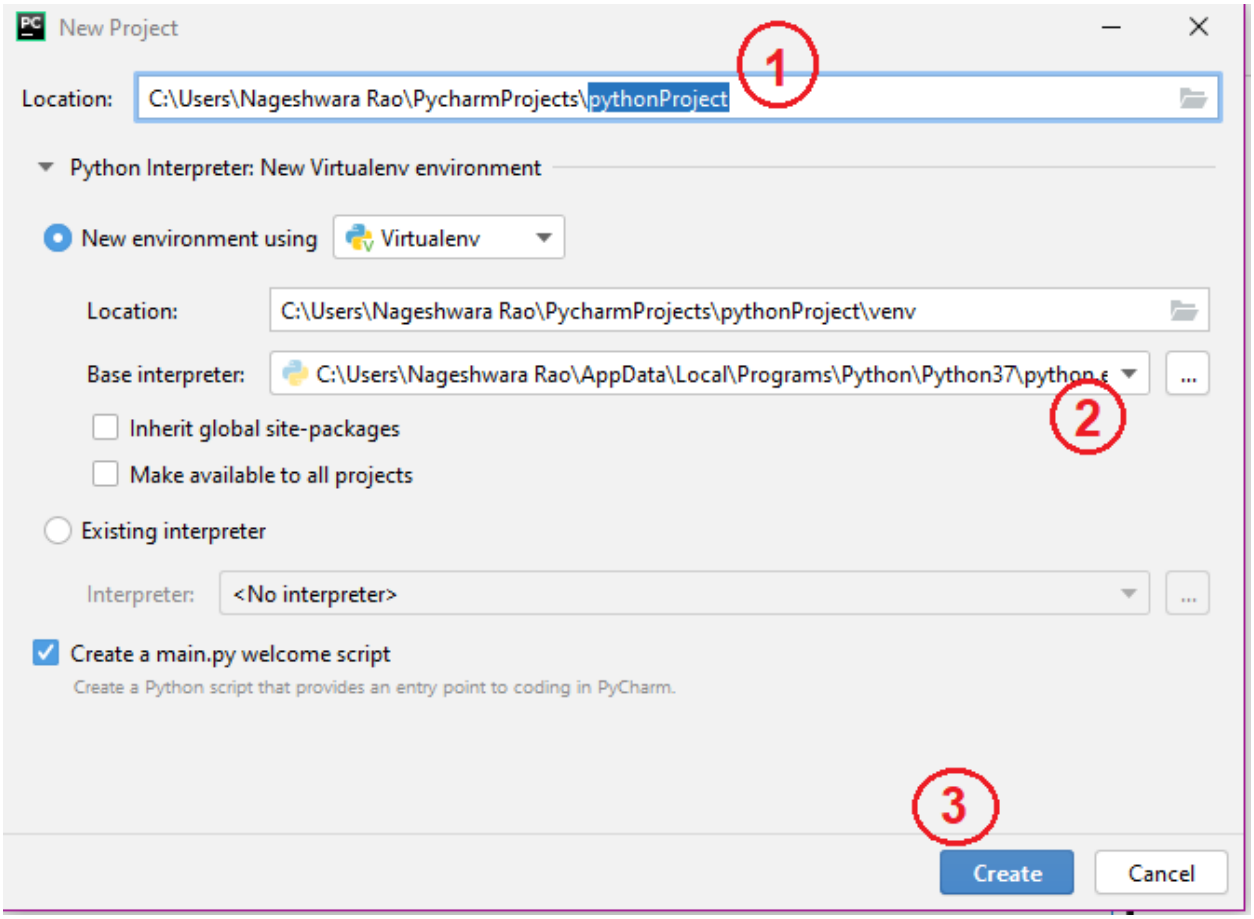
Step 1) Open PyCharm Editor by double clicking the PyCharm icon created on your desktop. You can see the introductory screen for PyCharm. To create a new project, click on “Create New Project”.

Step 2) You will need to select a location.

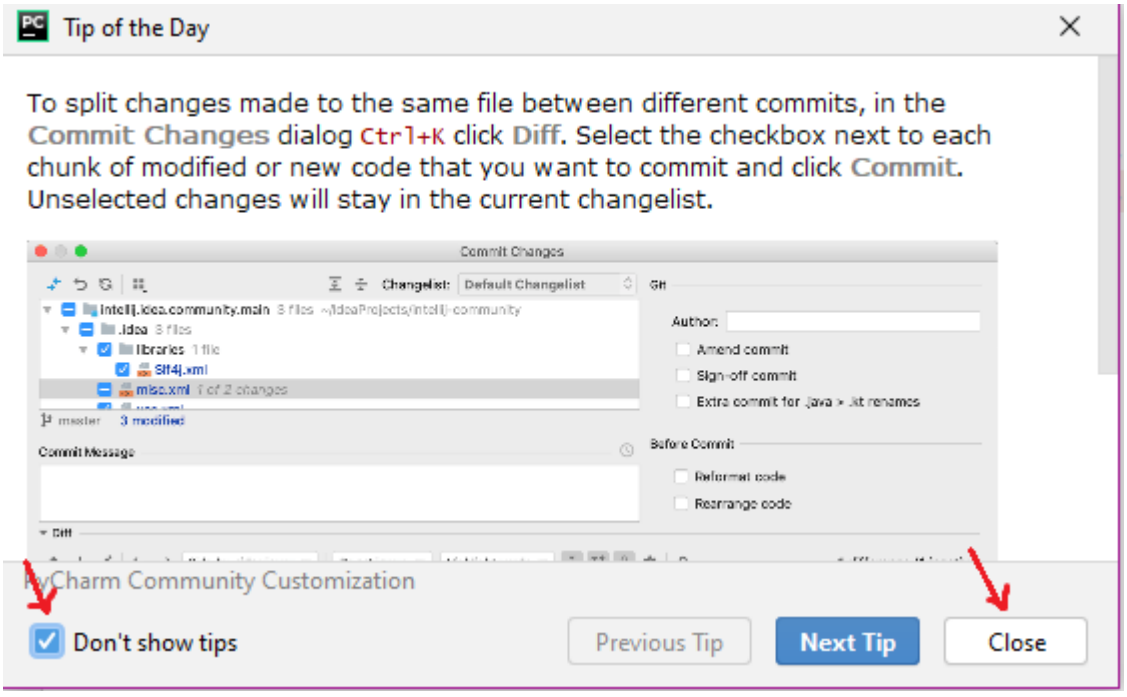
1. You can select the location where you want the project to be created. If you don’t want to change location then keep it as it is but at least change the name from “untitled” to something more meaningful, like “FirstProject”.

2. PyCharm should have found the Python interpreter you installed earlier.

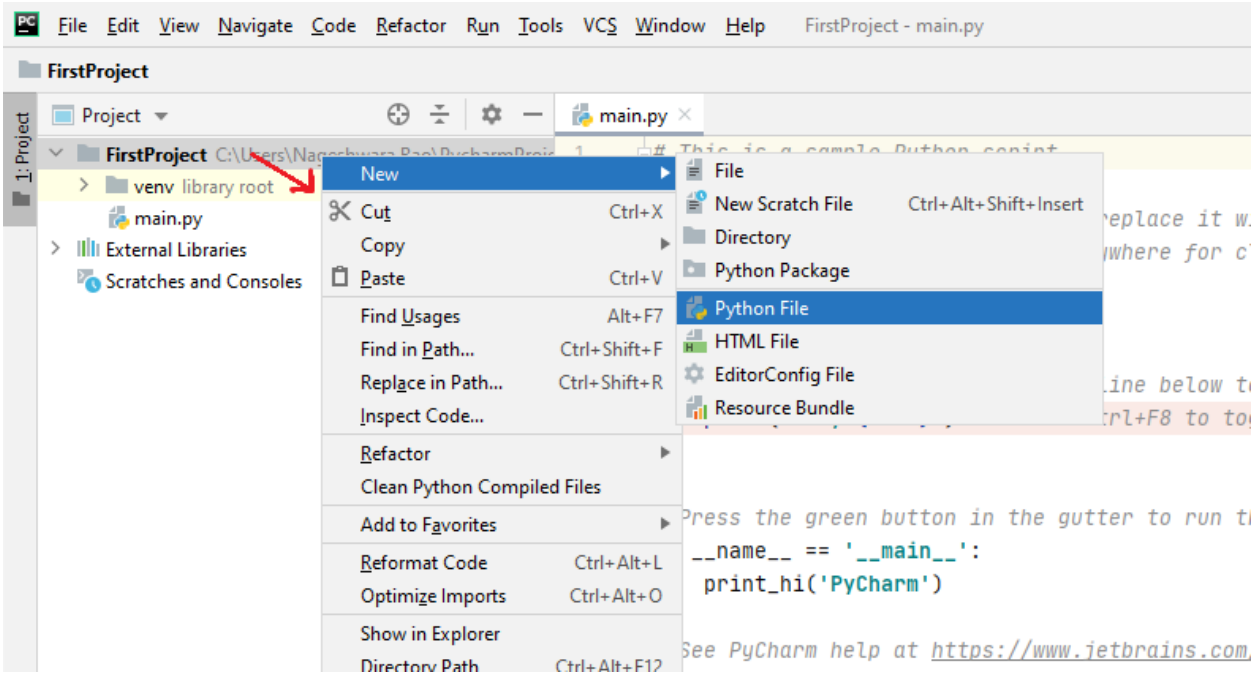
3. Next Click the “Create” Button.



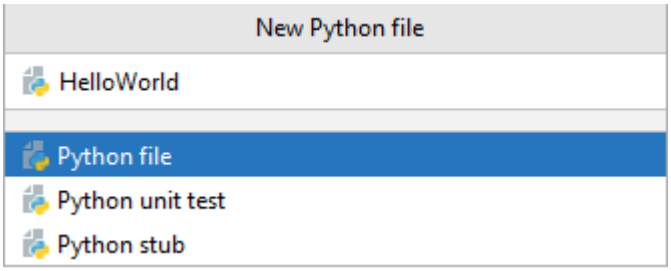
Step 3) If it shows “Tip of the Day” window, select “Don’t show tips” checkbox and then click on “Close” button.



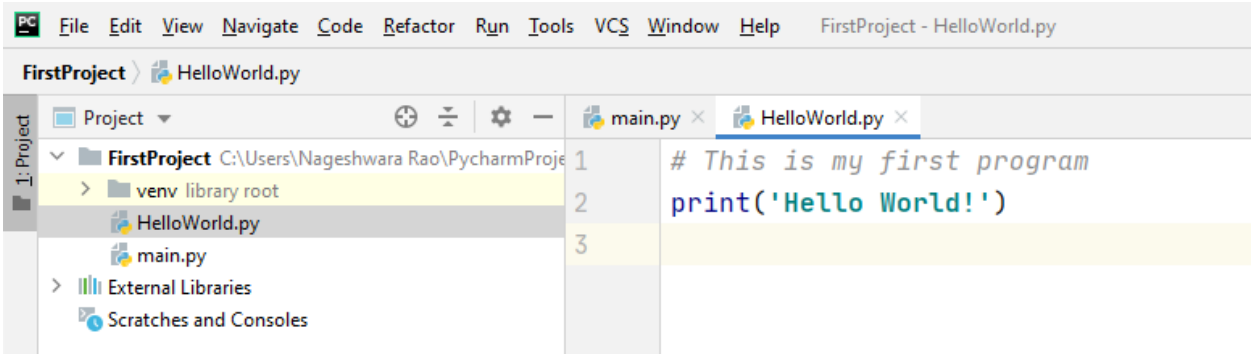
Step 4) Right click on the project name and then select “New”. Next, select “Python File”.



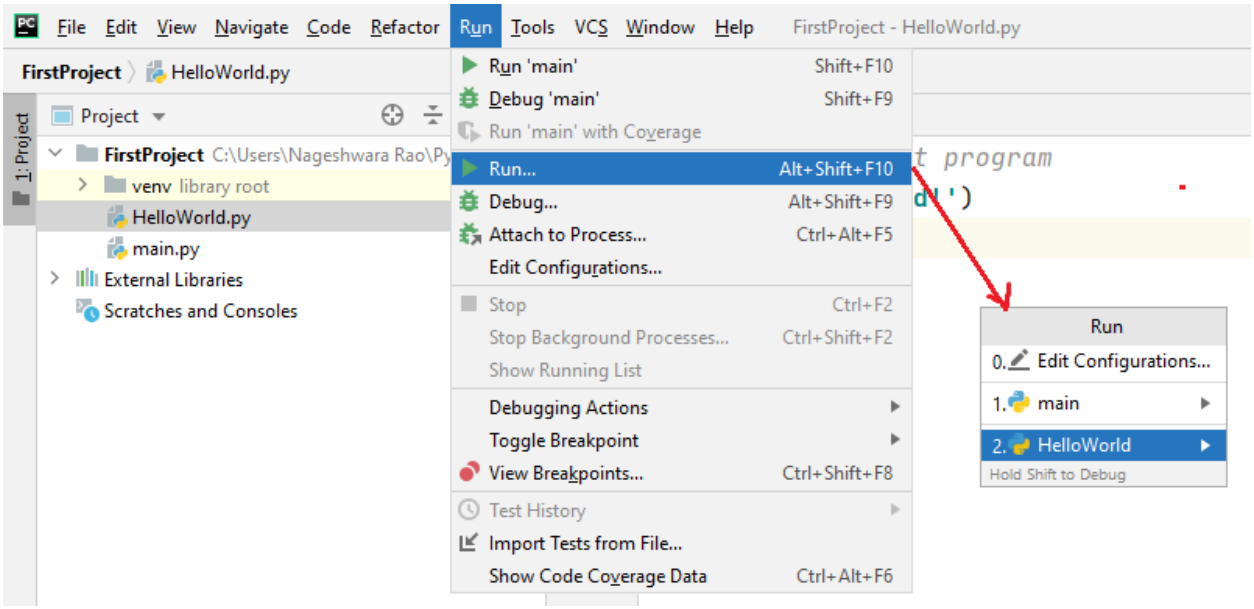
Step 5) A new pop up will appear. Now type the name of the file you want (Here we give "HelloWorld") and press <Enter> button.



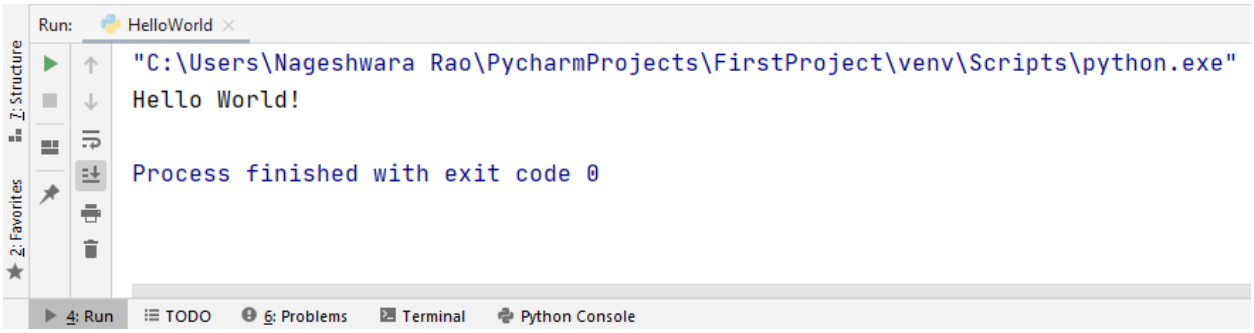
Step 6) It opens a blank page where we can type our program. Now type a simple program as shown in the screenshot.



Step 7) Now go up to the "Run" menu and select "Run" to run your program. Then you can see the program name that is to run. Select it.



Step 8) You can see the output of your program at the bottom of the screen.



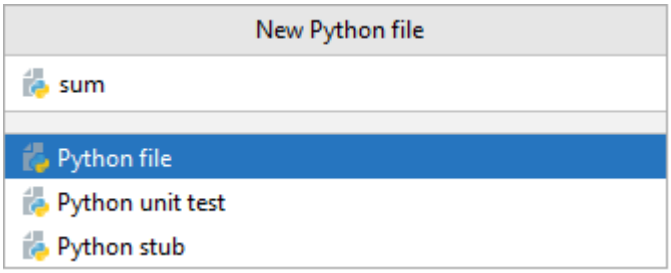
Note: To save all of your work, click on File and then Save All.

Step 9) Select “File” and then “Exit” to terminate PyCharm.

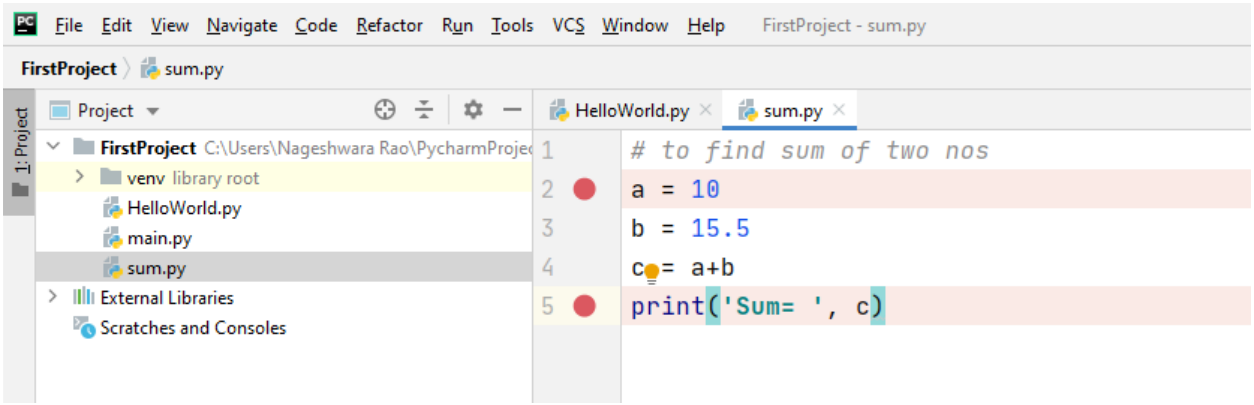
DEBUGGING AND TRACING

It is possible to execute every line of the program and see the output generated at that line. This is called program tracing and is useful in debugging.

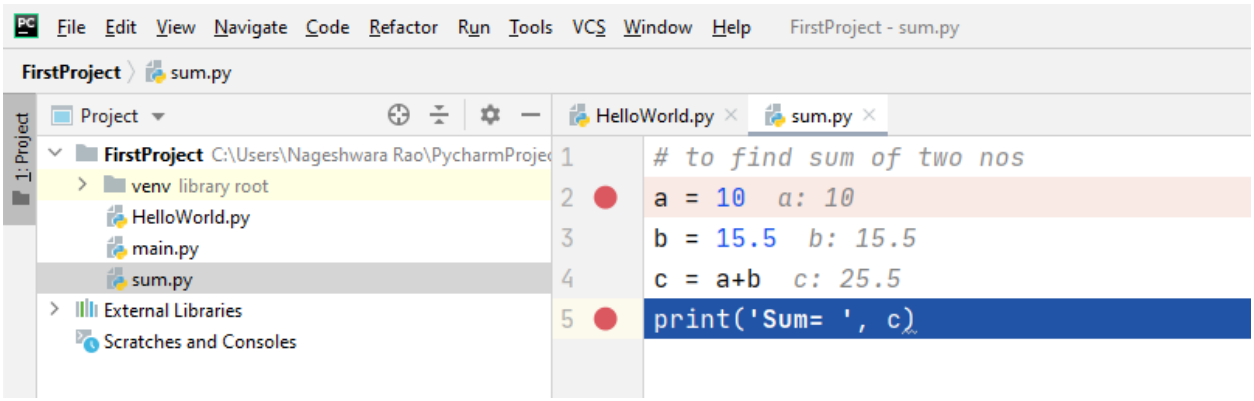
Step 1) Double click to open “PyCharm Community Edition” shortcut. It will open the recently stored project by default. In the File menu, select “New” and then “Python File”. Then type a name for the program as “sum” and press Enter key.



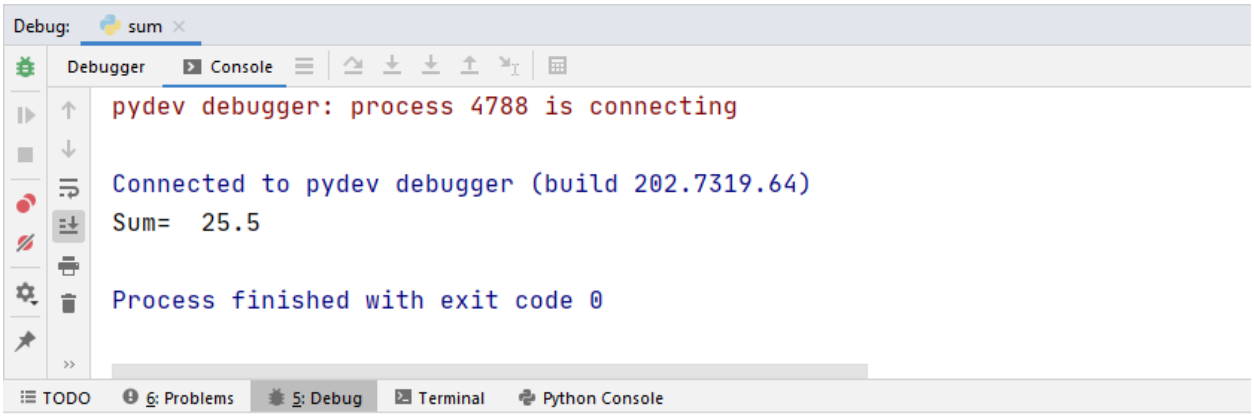
Step 2) Set breakpoints at a line (beginning) and another line (ending) in the program. For this purpose, click on the gray area at the left side of the program. When clicked, it displays a red dot.



Step 3) Click on Run and then Debug and then select program name. When you press “F8” button, you can see the result at the end of each line. Keep on pressing F8 button to go to the next line.



Step 4) Open 'Console' window at the bottom to see the result of the program.

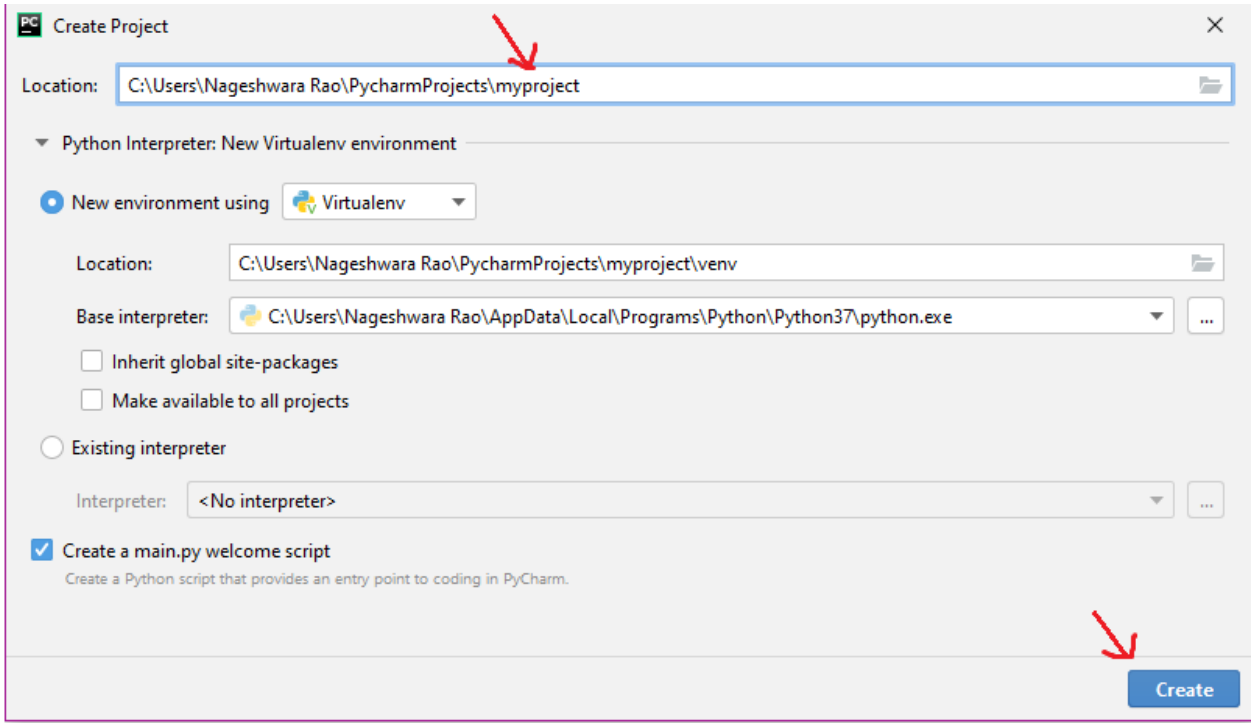


Note: To remove the breakpoints, click on them once again.

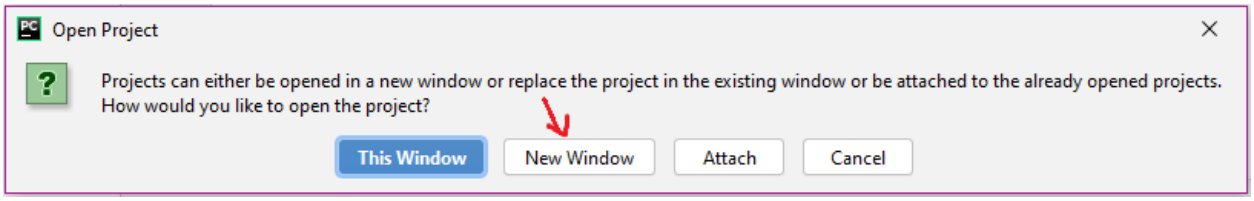
WORKING WITH PACKAGES

A package is a folder that contains several modules. A module is a python program that contains classes, functions, or any Python objects. Once we create a package, we can import and use it in another program using "import" statement.

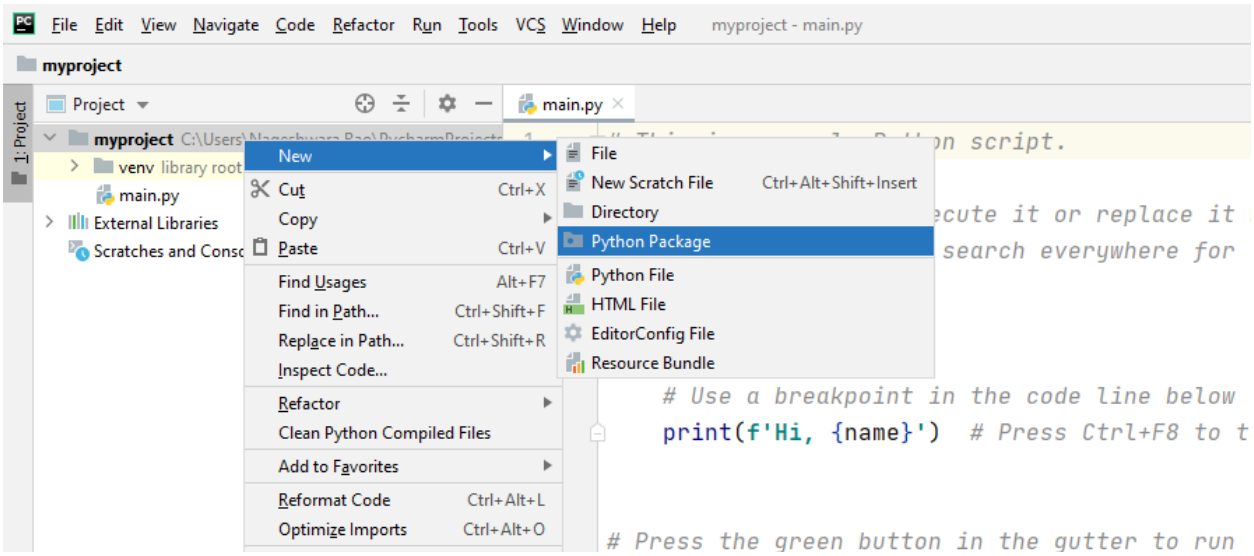
Step 1) Open the PyCharm IDE. Click on "File" and then "New Project". Then type a name "myproject". Click on "Create" button.



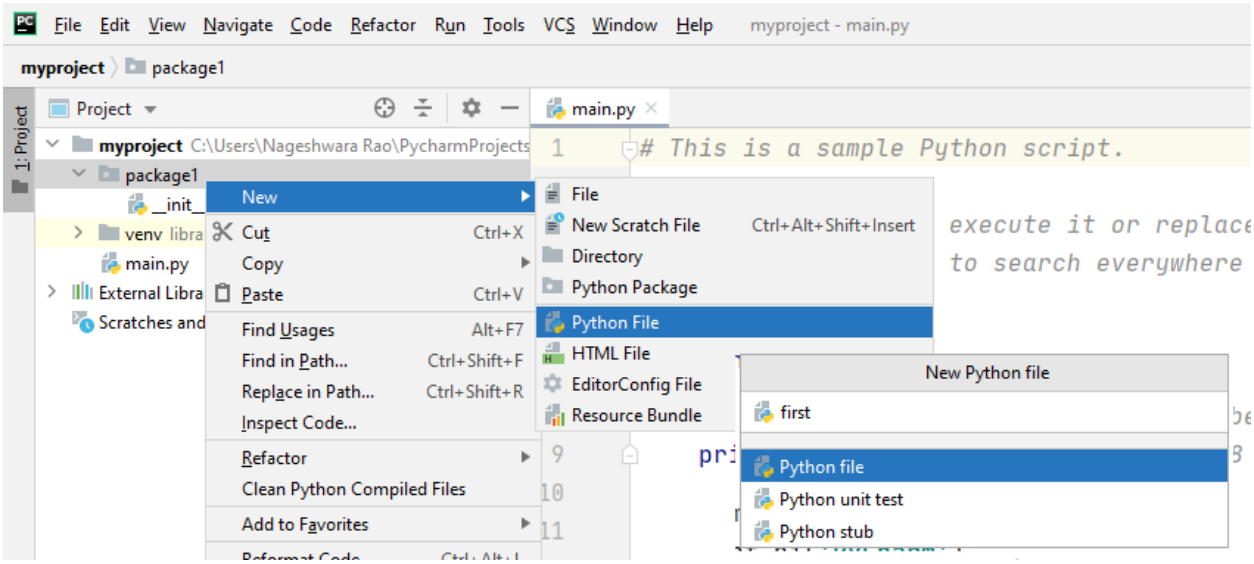
Step 2) Click on "Open in New window".



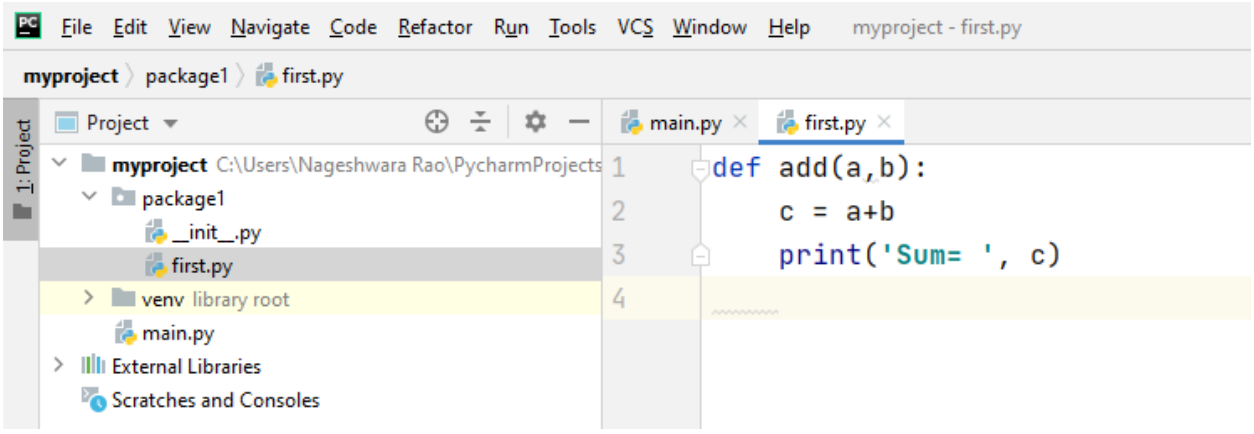
Step 3) First, select "myproject' by clicking on it. Then right click on "myproject" and click on "New" and "Python Package". Type the package name as "package1". Then press Enter key.



Step 4) Click on “myproject” and you will see the “package1” as a folder. Right click on “package1” and then click on “New” and “Python File”. Type the file name as “first”.

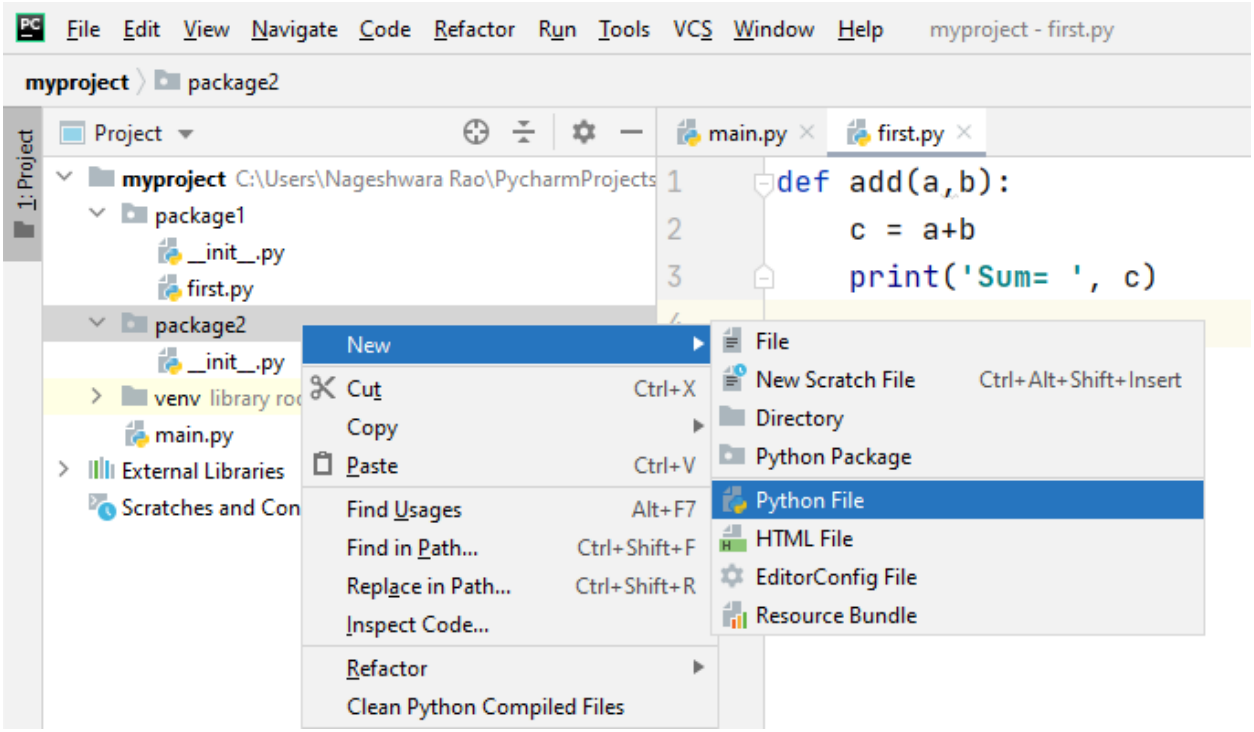


Step 5) A new tab sheet opens by the name “first.py” . Type a function or class etc. there. Now we created a package by the name “package1” and in that a module “first.py” that contains add() function.

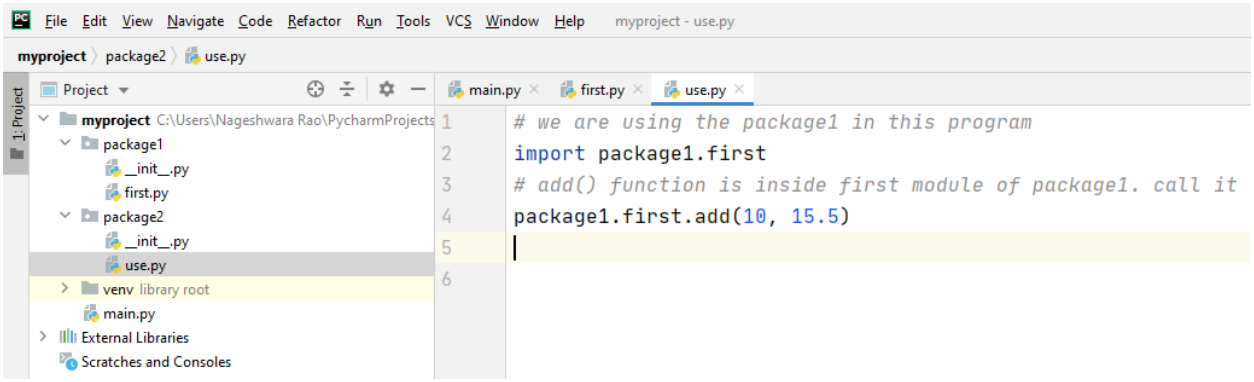


Step 6) We can reuse “package1” in another package or program. Let us create another package by the name “package2”. For this purpose, click on “myproject”. Then right click on it. Then click on “New” and then “Python Package”. Enter the package name as “package2”.

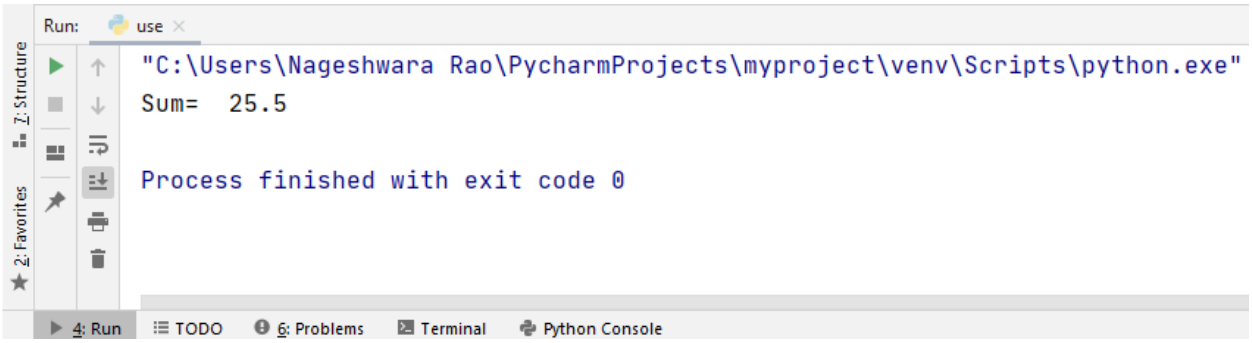
Step 7) Now click on “package2” under the project folder. Then right click on it. Then select “New” and “Python File”. Then type the file name as “use”.



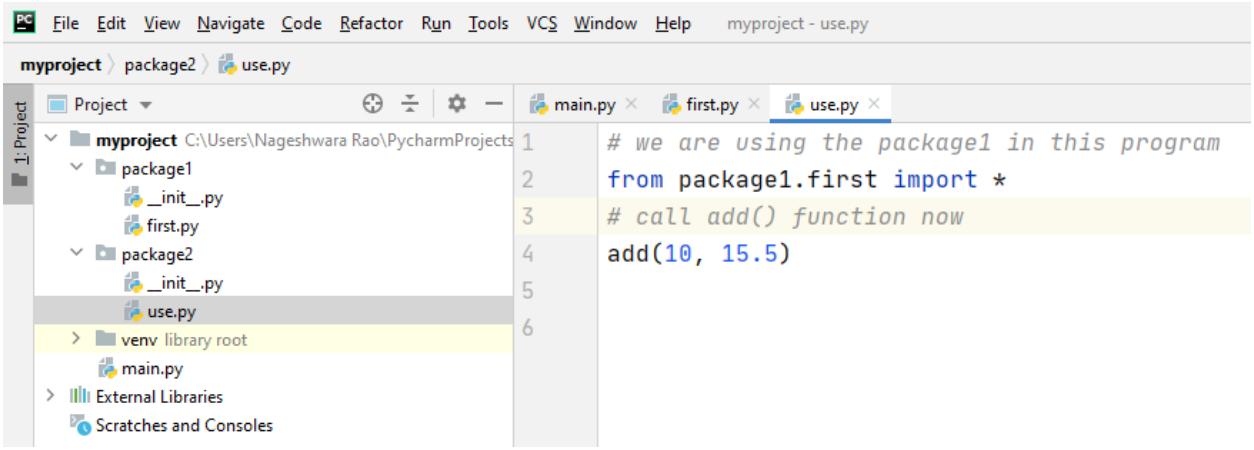
Step 8) You can see use.py appearing in the new tab sheet. Type a program where you want to use “package1”. See the following screen.



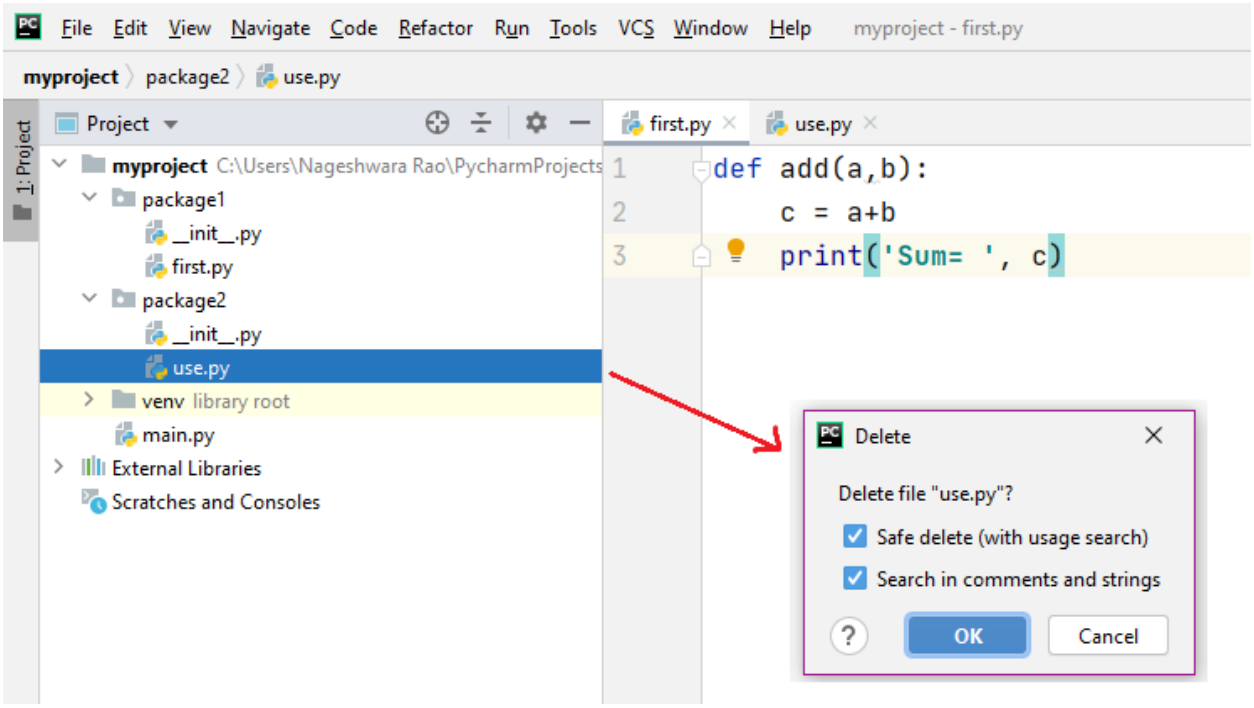
Step 9) Click on “run” and “run use” to see the result at the bottom.



Note: Another way of importing the package is shown in the following screen shot.



Note: To delete any file from the project, we can select that file and then press Delete key on the key board. Then click on “OK”.



Note: To save the project along with all the programs, we can select “File” in the menu and then “Save All”.