Setting up Pycharm and running the code

Installations
1) Download the Pycharm community version here:
   https://www.jetbrains.com/pycharm/download/#section=windows
2) Select these options for an easier user experience
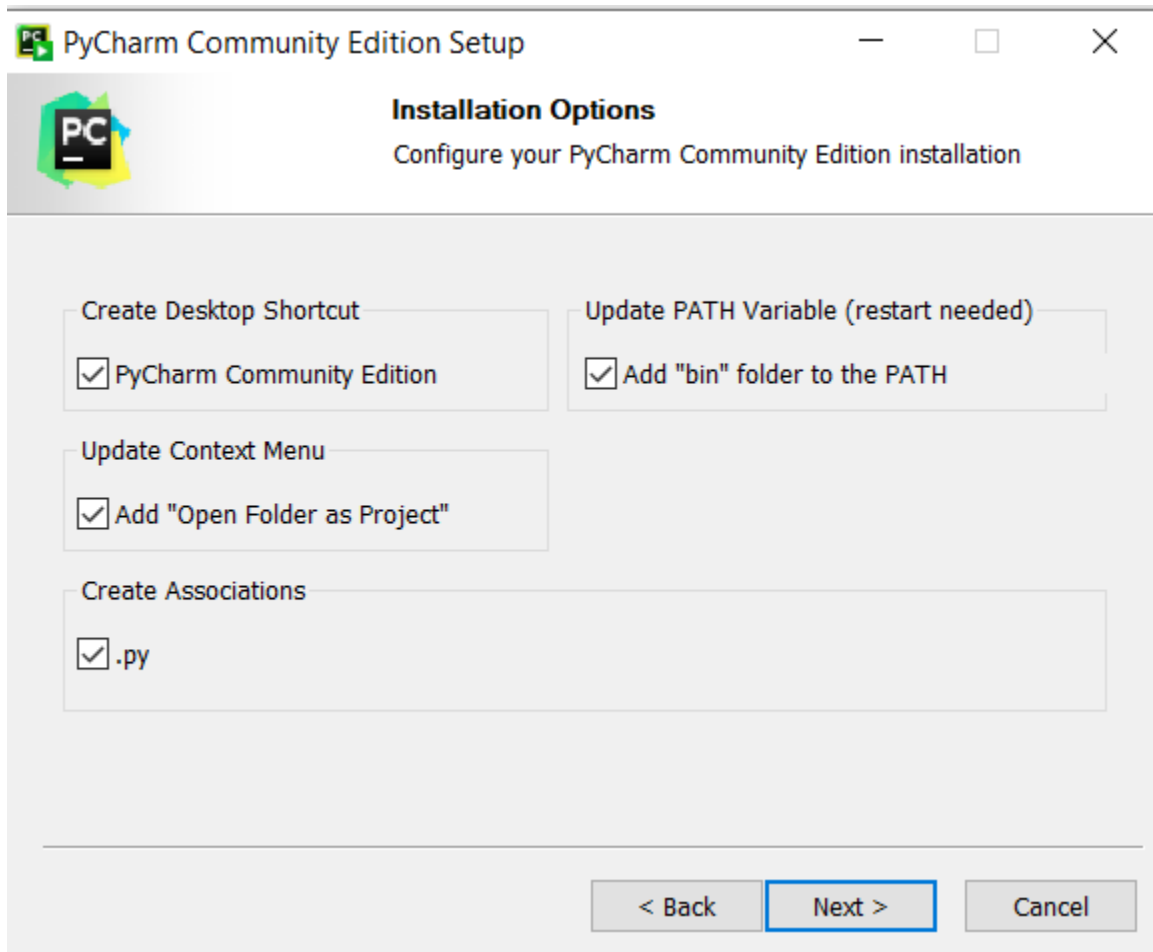
3) Click "Reboot now" when finished.

The code repository
1) The code is hosted on github at this url: https://github.com/benmbrew/tmsp
2) Click the "Code" button and download as a .zip file to destination of your choosing (ie Desktop or Documents). Note: the "defunct" folder can be removed (it was just scripts I used for testing, but aren't relevant now that we have this code).
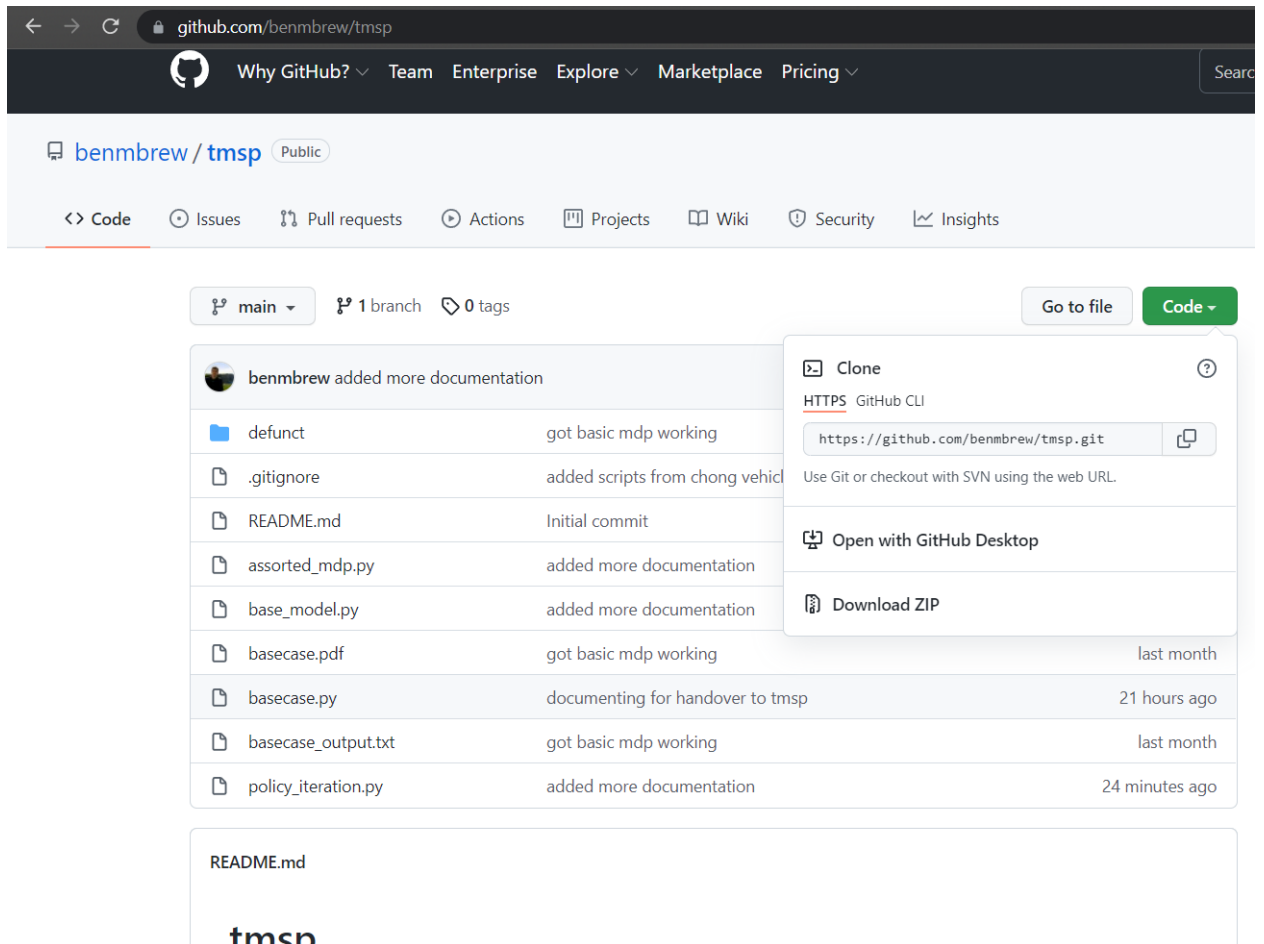
3) "Extract" the folder from the .zip file.
4) Double click on one of the python file "basecase" and it should automatically open the file in Pycharm (you will have to confirm the user agreement).
5) When Pycharm opens, click on the "Open in Project" button. This will make Pycharm interpret the folder "tmsp-main" as a "project" and therefore easily allow you to move from script to script and call functions from other scripts.

Navigating pycharm
1) Create a virtual environment for version control (to download packages you may need). File -> Settings -> Project: tmsp-main -> Python Interpreter
2) Click on the gear in the upper-right corner and then click "Add"
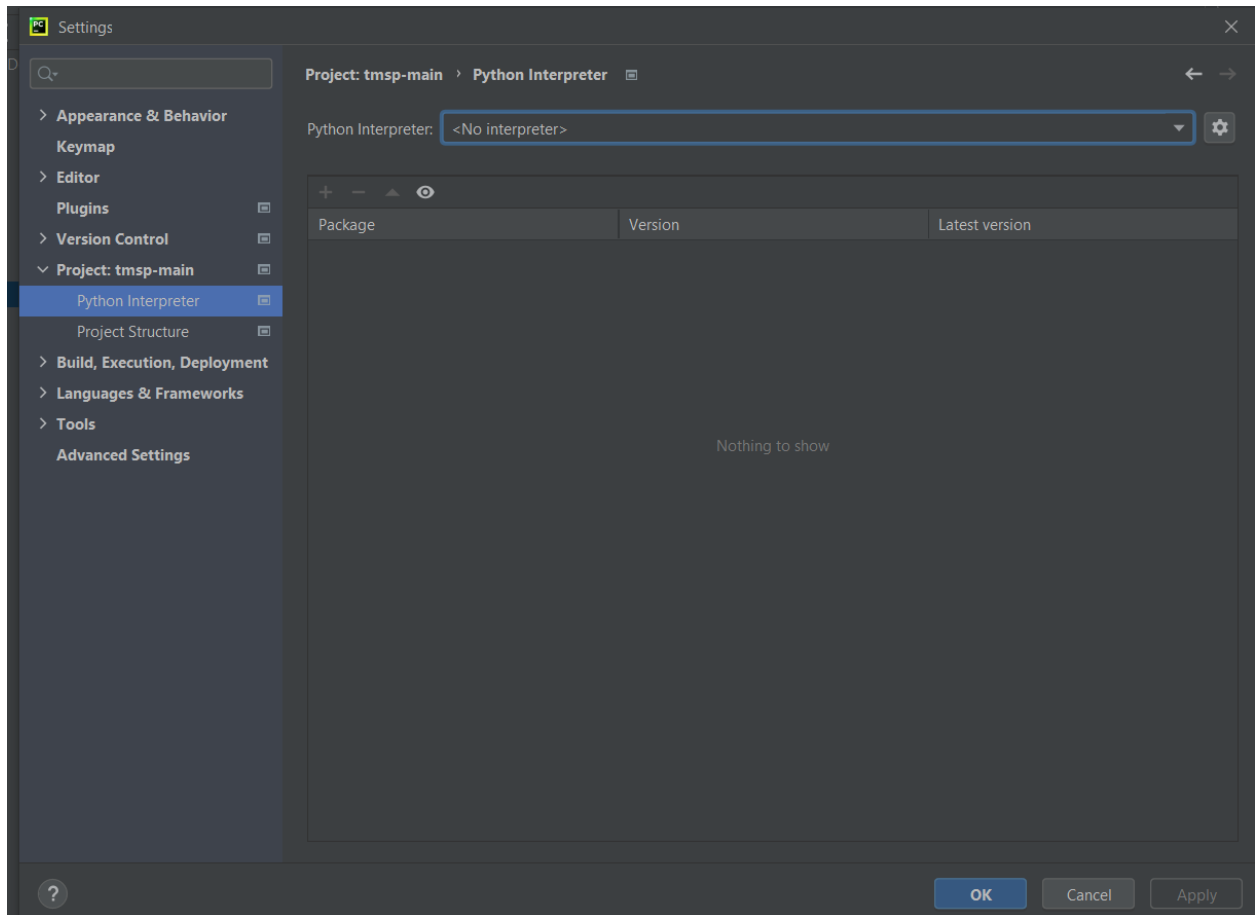
3) Click "New Environment". Then click the "Inherit global site-packages" and "Make available to all projects". Finally, press "OK" and the latest version of python should install. When you add packages, they will only be added to this virtual environment. This is just best practice for version control. You will likely be prompted by the Windows taskbar to approve the installation.
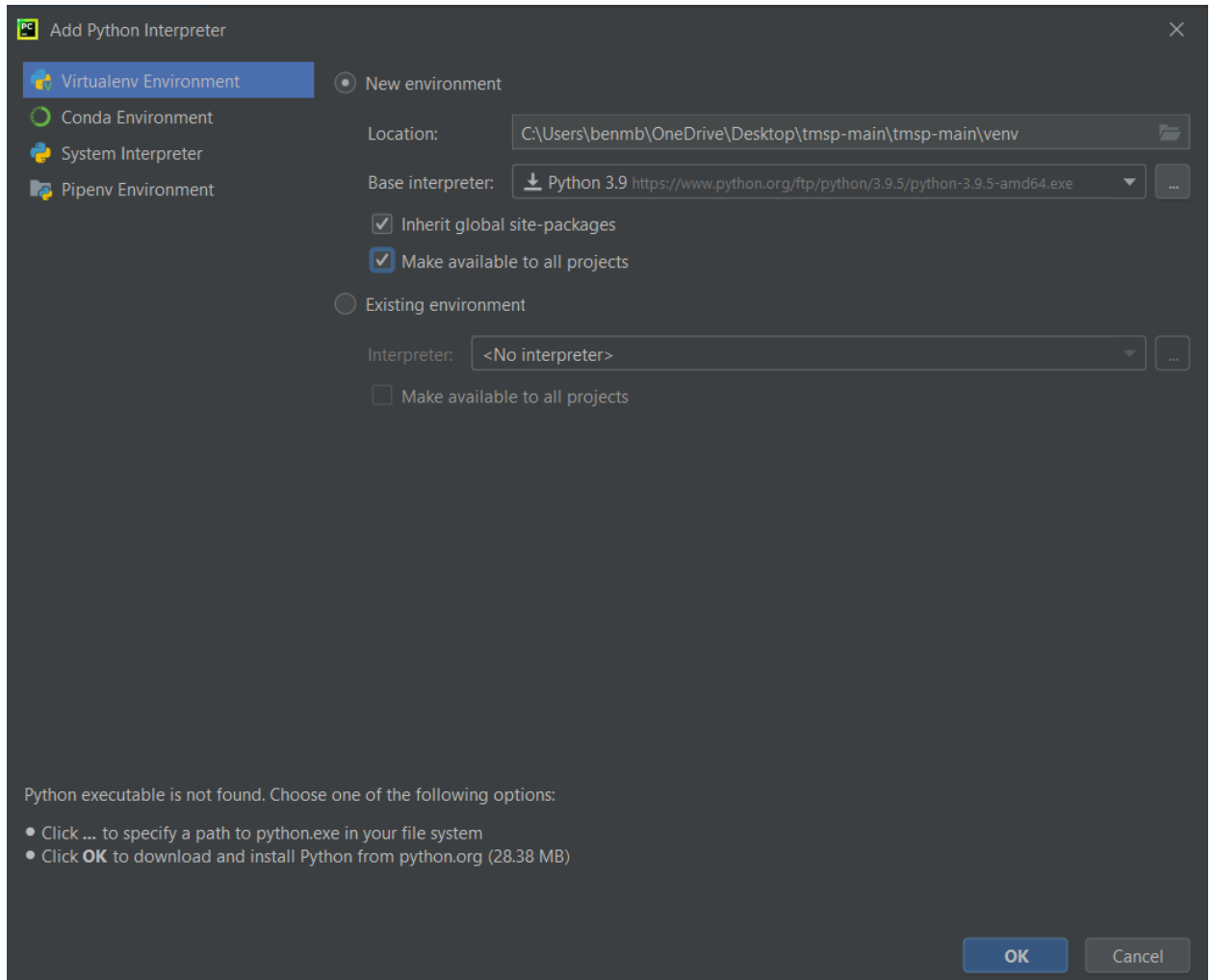
5) The only external packages being used are Numpy and matplotlib. We need to download them to our virtual environment. File -> Settings -> Project: tmsp-main -> Python Interpreter. Click the "+" sign (above "Package" in the picture below) to open up the option for downloading a package
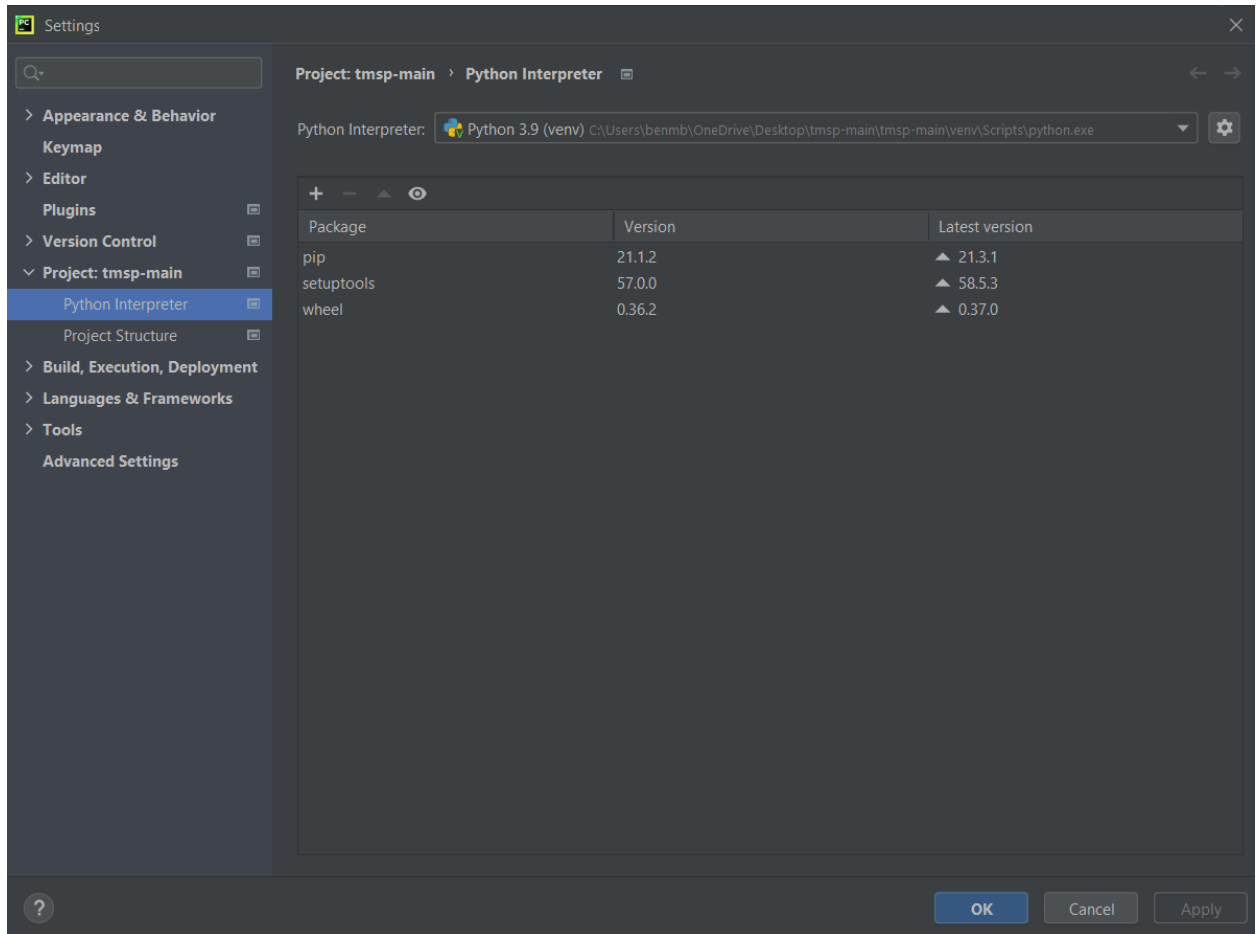
6) Once opened, search for numpy and click on it. Then click the button "Install package" (bottom left corner in the picture below).
7) We also need "matplotlib" package, so do the same thing as above.
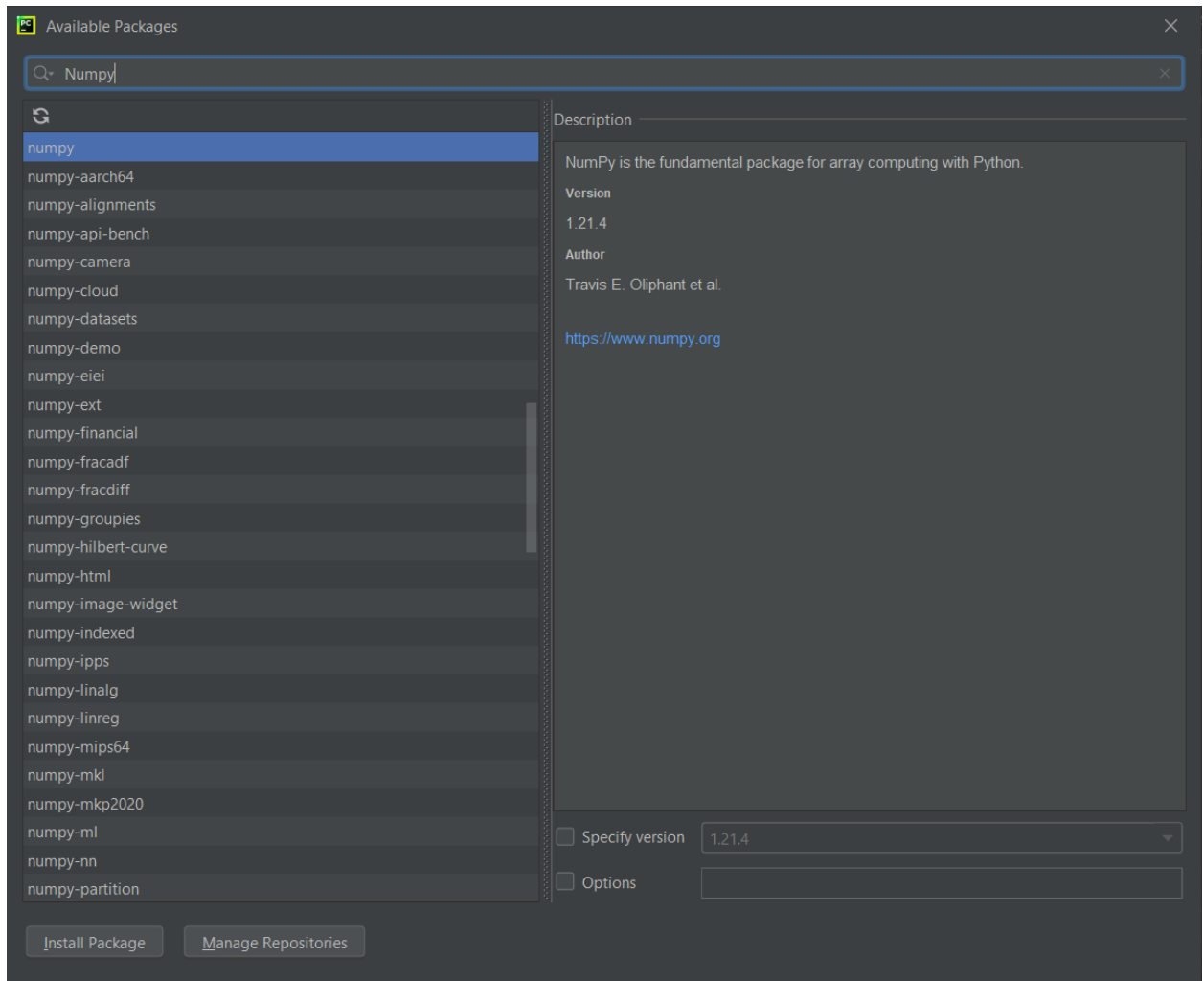
Running code

1) Basecase.py is the script you need to run to produce the first plot. It sources the other 3 scripts at the top, as well as the numpy and matplotlib packages.

2) To run the code line by line: click on a line and the push Shift+Alt+E.

3) To run the entire script you can highlight the whole script and hit Shift+Alt+E. Or you can click on "Run" at the top and click the "Run" button. Alternatively you can click Alt+Shift+f10. If you run basecase.py, the plot will be produced.

4) I would recommend running line by line or highlighting and running with Alt+Shift+E, as that will keep you in the "python console", where you can see the previous code and the objects currently in the environment (the picture below shows the python console at the bottom. The objects (dataframes, arrays, lists, etc) that are used in the script are stored in the bottom right panel.
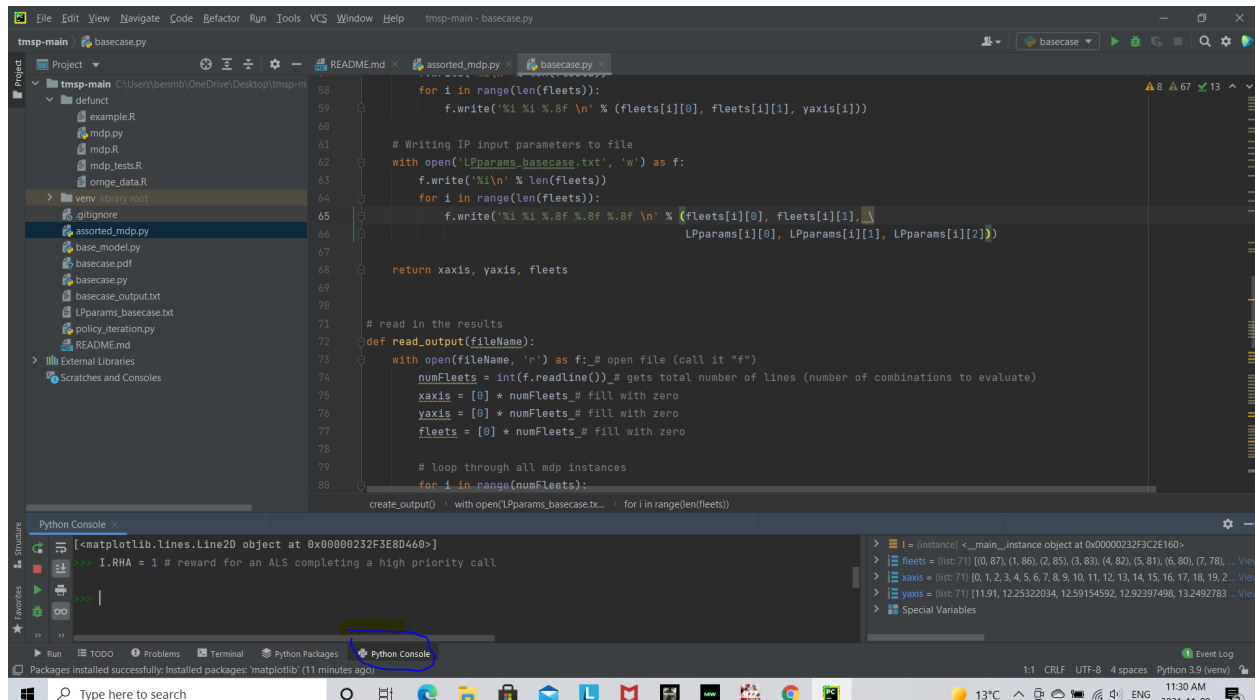
5) To see other keyboard shortcuts press "Ctrl+Alt+S". Though the main ones I use are Shift+Alt+E to run the code and then "#" to comment on the code.

6) When typing the name of an existing object or function, you can use "tab completion".