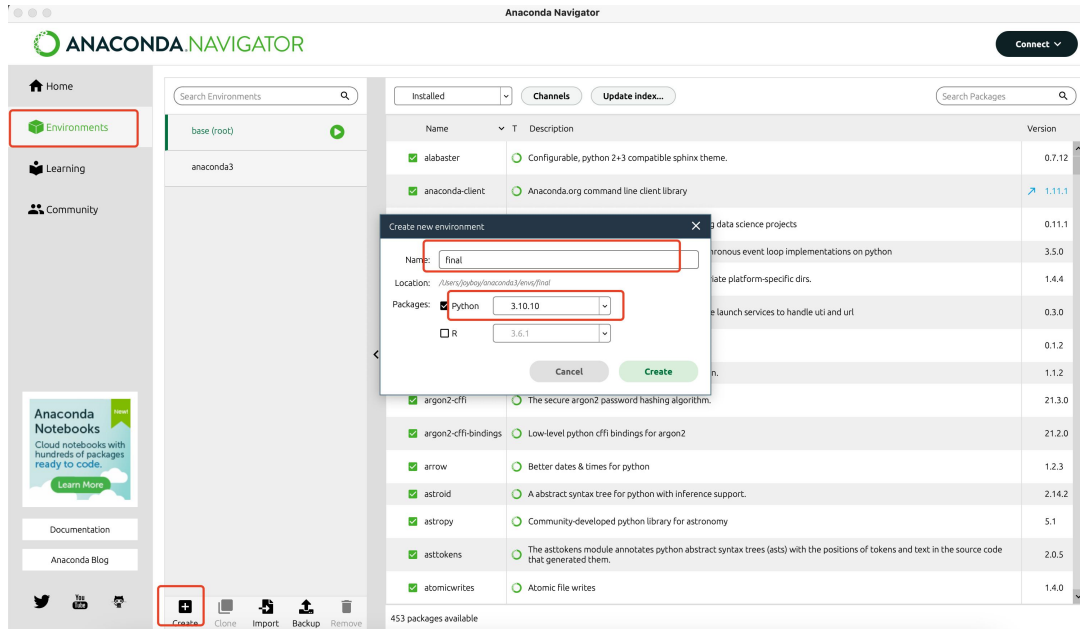


The steps to run the project code in PyCharm or VS Code

Set environment:

1. Open Anaconda Navigator.
2. Click on the "Environments" tab.
3. Click the "Create" button to create a new environment.
4. Name the environment and choose Python 3.9 version.
5. Click the "Create" button to create the environment.



6. Select the new environment you just created and click the "Install" button next to it.
7. In the "Not installed" tab, search for and select the following packages one by one:

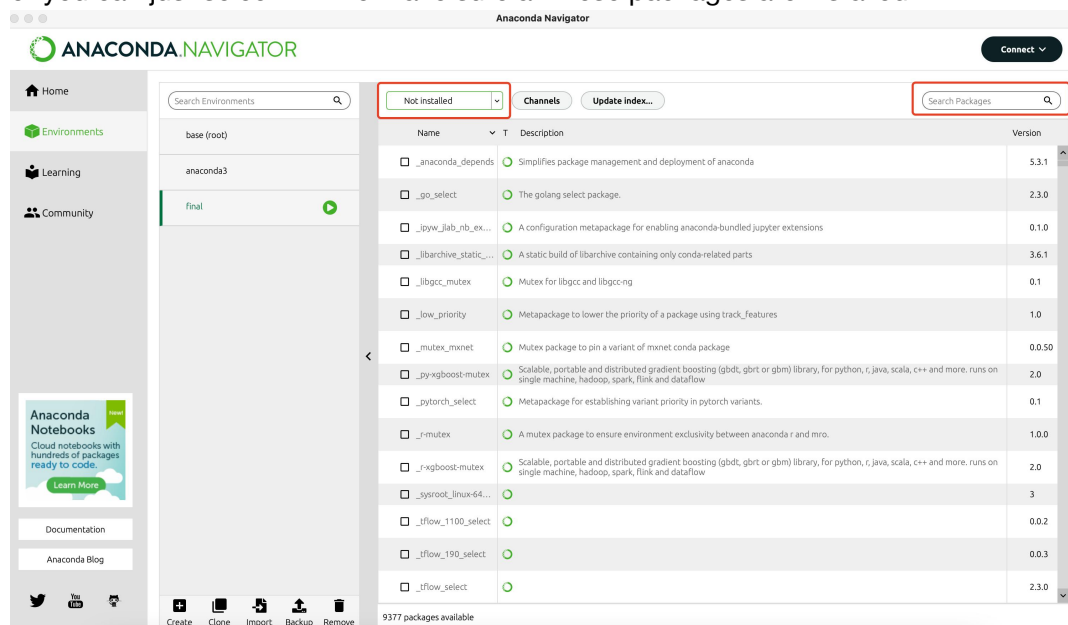
tkinter(or tk)

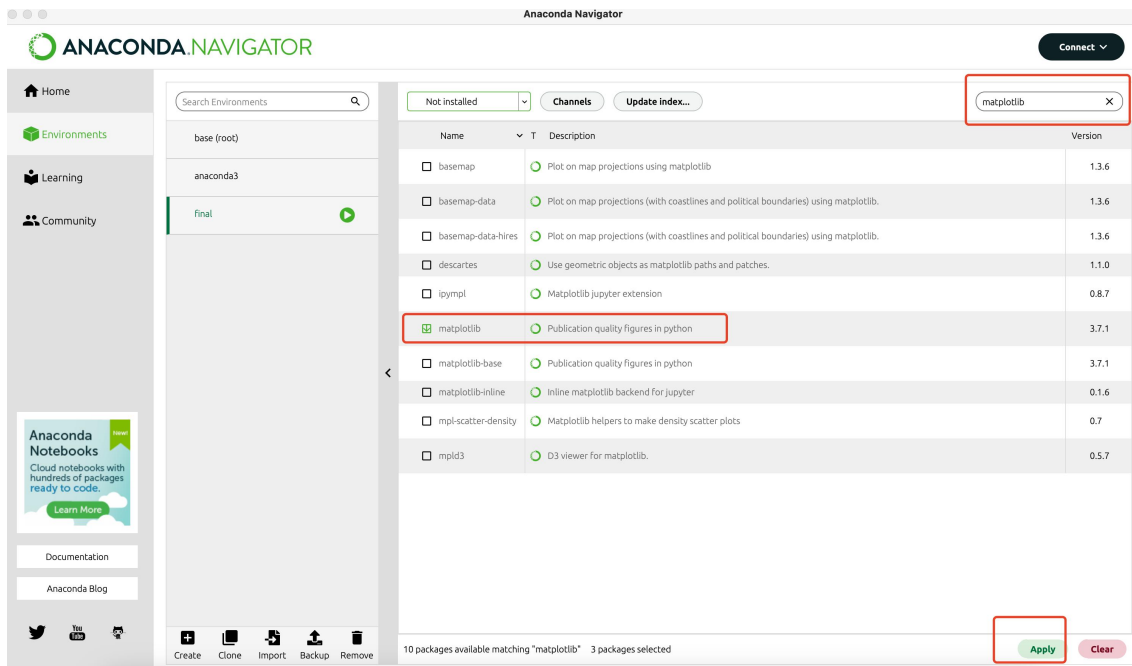
pandas

matplotlib

pillow

or you can just select "All" to make sure all these packages are installed.

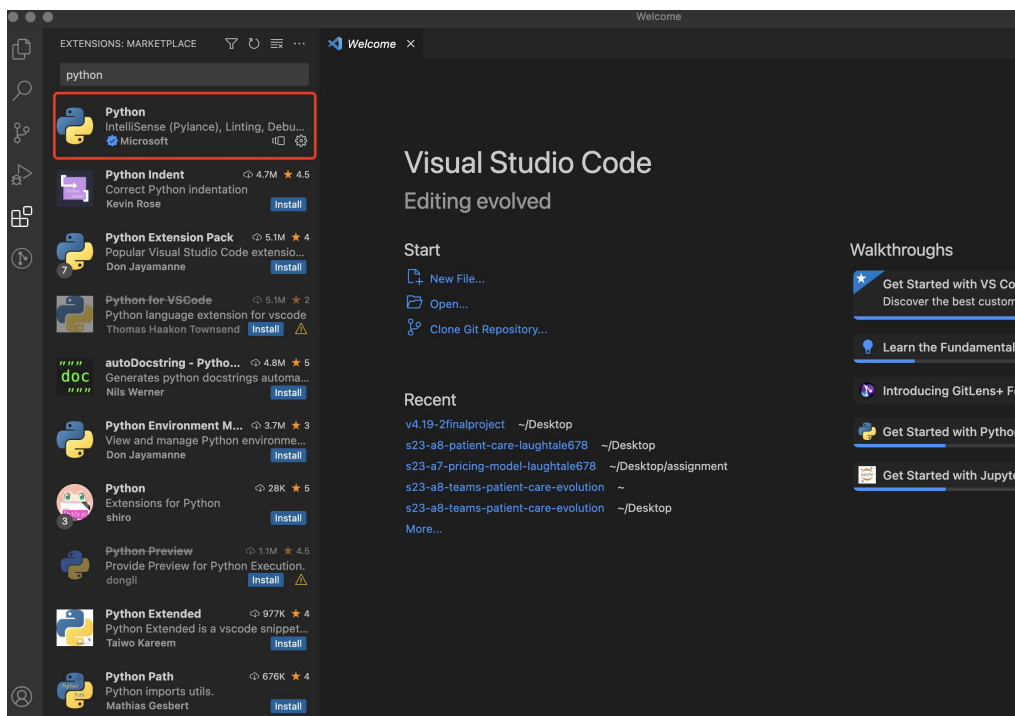




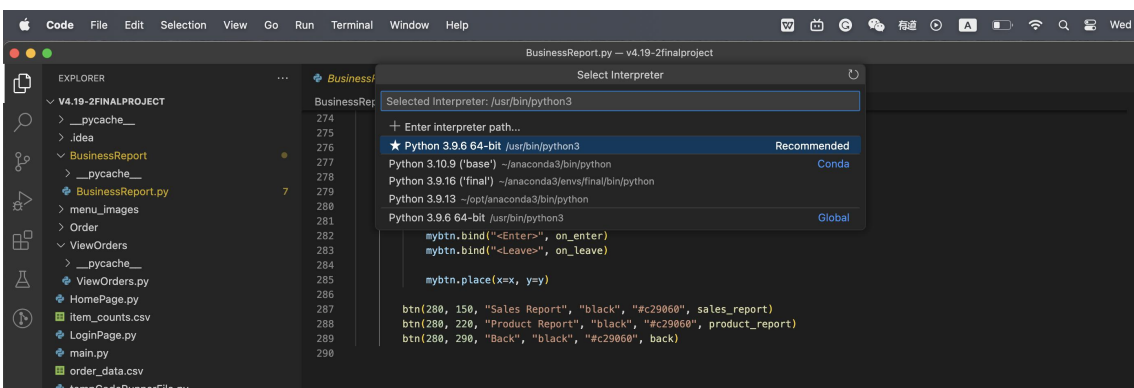
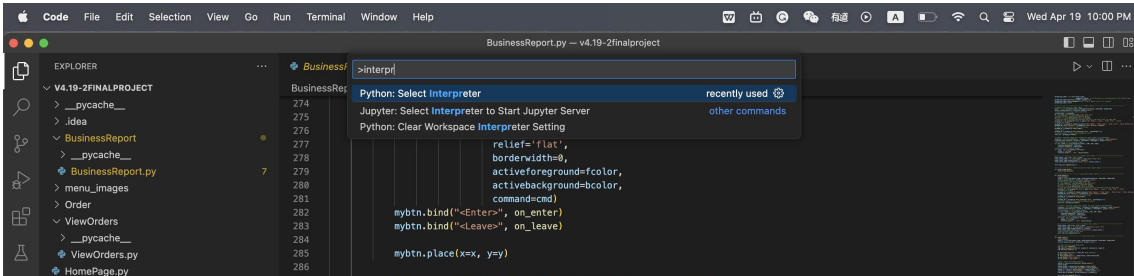
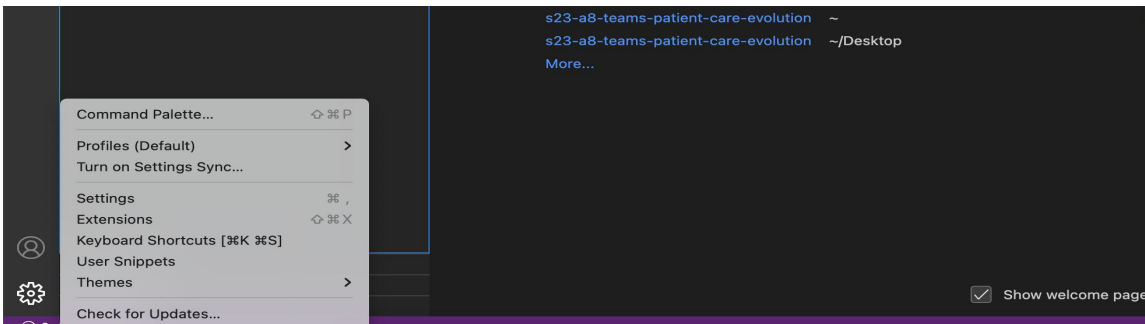
8. Click the "Apply" button to install the packages.

VS Code:

1. Open VS Code and click on "Open Folder" to open your project.
2. Once the project is open, go to the "Extensions" tab on the left sidebar and search for "Python".



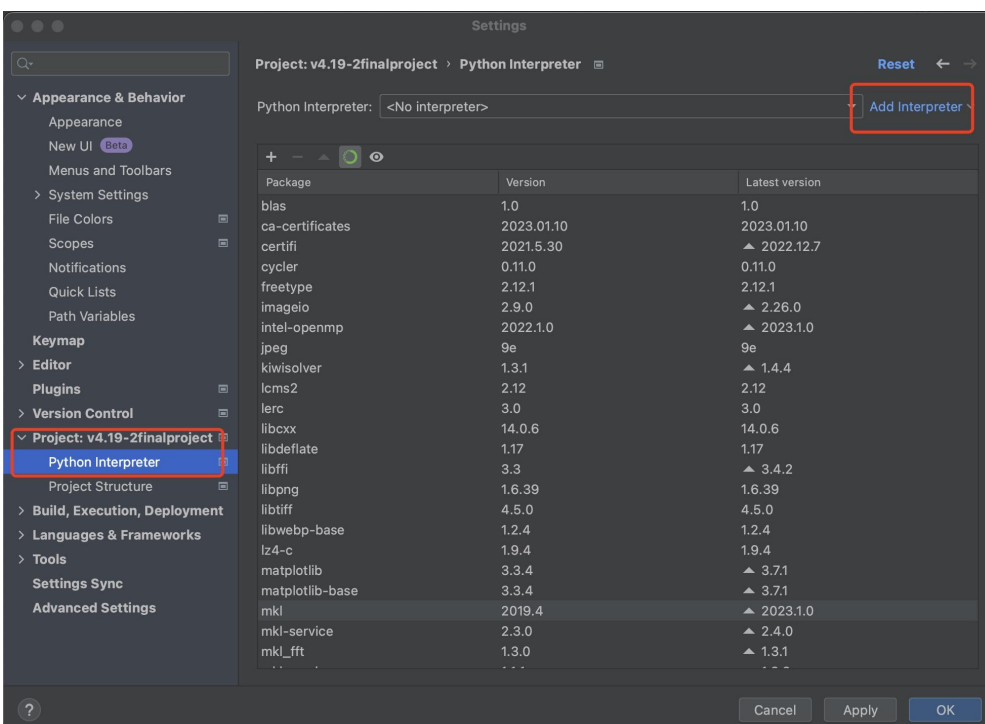
3. Install the "Python" extension and reload VS Code.
4. Press "Command(Ctrl)+Shift+P" or to open the Command Palette, and type "Python: Select Interpreter".



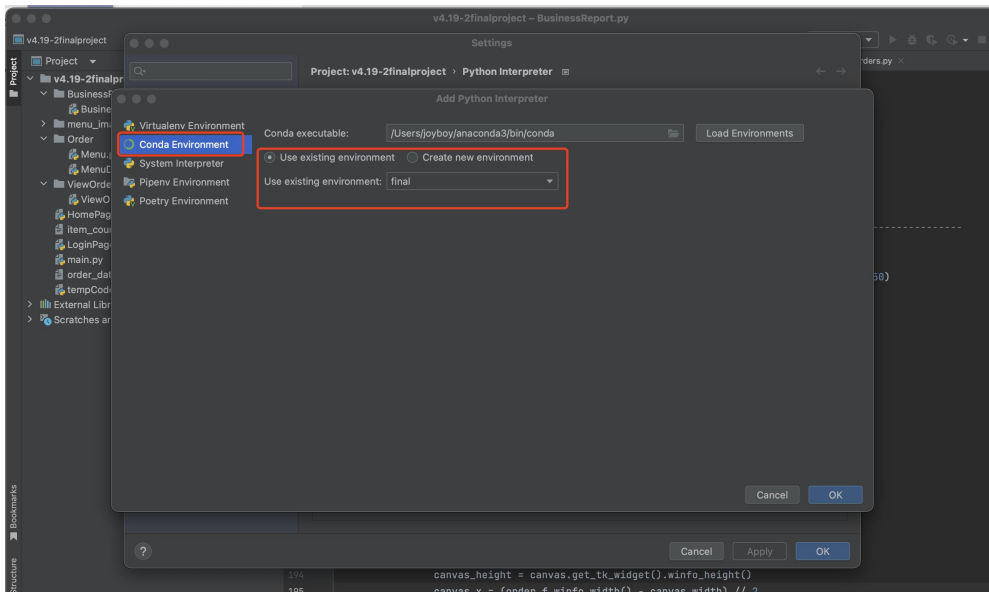
5. Choose the Conda environment you just created.
6. Once the interpreter is selected, you can run the code.

PyCharm:

1. Open PyCharm and click on the "Open" button to open your project.
2. Once the project is open, go to PyCharm > Setting> Project > Project Interpreter.



3. Click on the "Add Interpreter".
4. Select "Conda Environment" and choose the environment you just created.



5. After selecting the interpreter, click on "OK" and wait for PyCharm to configure the environment.
6. Once the environment is configured, you can run the code.