Automated Accuracy Testing
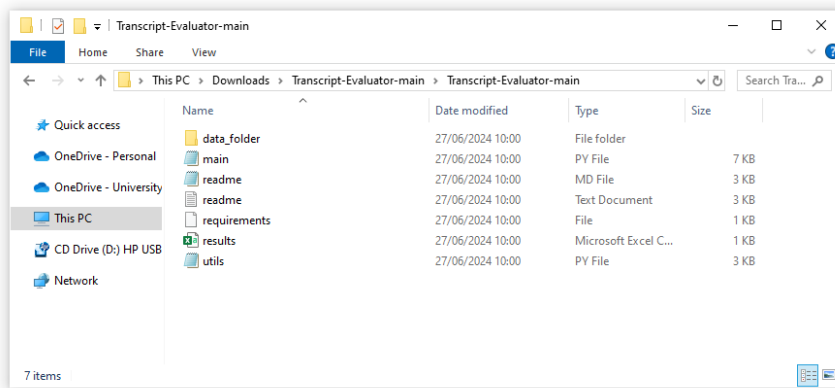
This document outlines how to complete semi-automated accuracy testing using a Python command line tool.

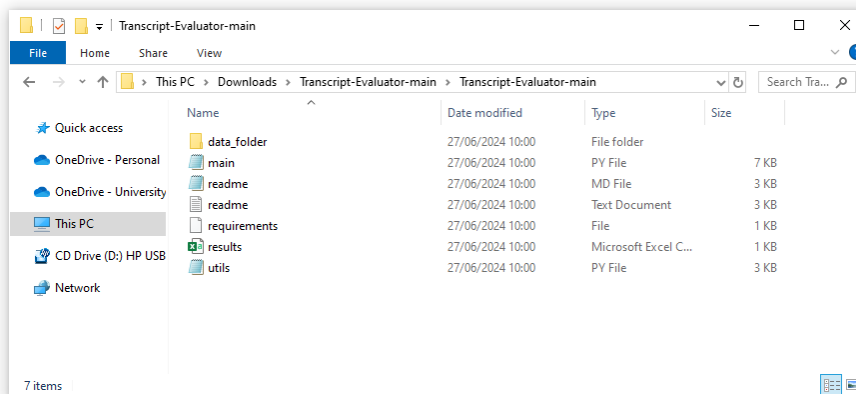## Getting Set Up (installation)

Follow the steps below the first time you run the program.
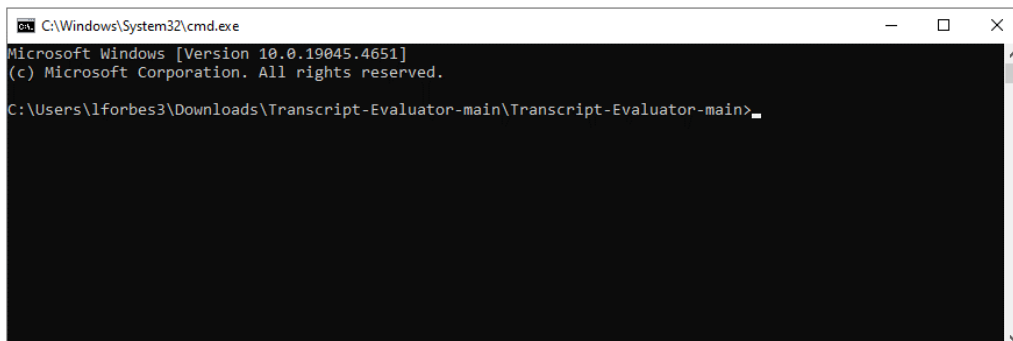
Downloading program and opening terminal

1. Download the Python program from GitHub by clicking Code > Download Zip.
2. Right-click and unzip the folder and open the files:
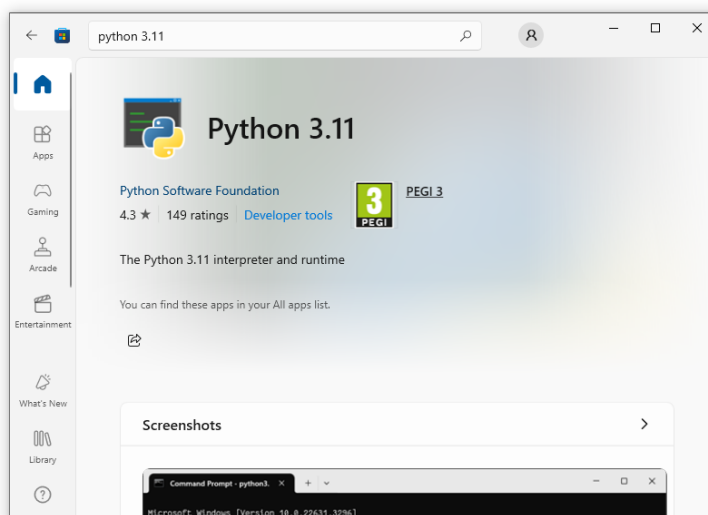

3. Click the bar at the top:

4. Type "cmd" and press enter. Command prompt should open:

Getting correct python version

5. Refer to images at bottom of page for help with this step. In command prompt, type **`python --version`** and press enter.
   a. If you see "'python' is not recognized as an internal or external command", go to step 6.
   b. If you see "Python [version number]" ...
      i. If [version number] is 3.9.X, 3.10.X or 3.11.X (X can be anything), go to step 7.
      ii. If [version number] is anything else, uninstall python completely, then go to step 6.
6. Since Python was not recognised, you need to install it. Open the Microsoft Store, search for 'Python **3.11**', install the app (see image below). Go to step 5.

Scenario Examples

5.a. (go to step 6)

5.b.i. (go to step 7)

5.b.ii. (uninstall Python, go to step 6)

Installing Dependencies (once Python is installed)

7. Uninstall NumPy. Run `pip uninstall numpy` and press enter when prompted. Repeat (press up arrow then enter) until you see "WARNING: Skipping numpy as it is not installed."

8. Type `pip install -r requirements` and press enter. If you see "failed to write executable", repeat this step until it goes away. If it doesn't, change "pip" to "pip3".

9. Type `python main.py` and press enter. If you see the following, you're good to go!

Installation is now complete!

Running Example Command

Assuming you are at step 8 above, check everything's working as follows.

1. Type `del results.csv` and press enter to delete the current results file. This isn't necessary, but ensures the program is actually doing something.
2. Type `python main.py --data data_folder --services service1 service2` to run the program.
3. Open the newly created results.csv in your Transcript-Evaluator-main folder:
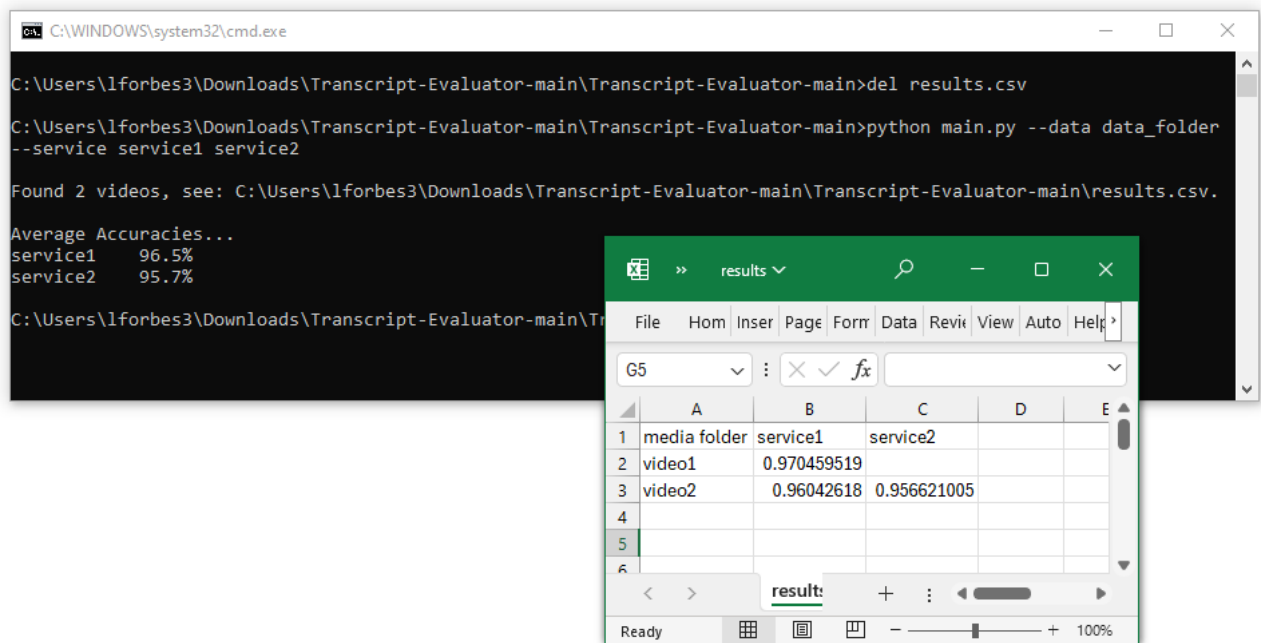
For all usage information, see the Readme file.