Setup Instructions for Recording Data and Plotting

Installing Python and necessary packages

Note: If you already use Python regularly, we recommend you set up a virtual environment to separate this installation from your other installations. If you use an Anaconda distribution, you can find instructions at this link. If you do not use Anaconda, you can follow the directions here.

1. Learn how to access your terminal/command line prompt.
    a. On Windows: Wikiversity instructions or WikiHow instructions (note: do not use Method 5)
    b. On Mac: WikiHow instructions
    c. On Linux: Ubuntu tutorial
2. In your terminal, check if you have Python (a coding language) pre-installed by typing **`python`** and pressing Enter. If something like the following appears, Python is already installed on your computer and you can skip to step 5!

    ```
    Python 3.9.6 (tags/v3.9.6:db3ff76, Jun 28 2021, 15:26:21) [MSC
    v.1929 64 bit (AMD64)] on win32
    Type "help", "copyright", "credits" or "license" for more
    information.
    ```

3. If you do not have Python on your computer, you should close your terminal by typing **`exit`** and pressing enter, or by closing the window with the red X button. You can install Python by following the directions here at this link (Python main website)
    a. If you're on a Windows computer, make sure to check the box "Add Python 3.11 to PATH" (or whatever version you have) on the first page ("Install Python") of the installer. You can follow the instructions in the YouTube video here
4. Once you have installed Python, repeat step 2 to make sure that it has installed properly.
5. Exit your Python interpreter by typing **`exit()`** in your terminal and pressing Enter. This will exit Python but leave the terminal open.
6. Now, you need to check that you have the Python package installer (pip) working. In your terminal, type **`python -m pip --version`**. If this returns something like `pip X.Y.Z from ... (python 3.N.N)`, pip is already installed and you can skip to step 9!
7. If the previous step returned an error message, install pip with the steps on the Pip Documentation website (link) for the appropriate operating system.
8. Repeat step 6 to make sure that pip has been installed.

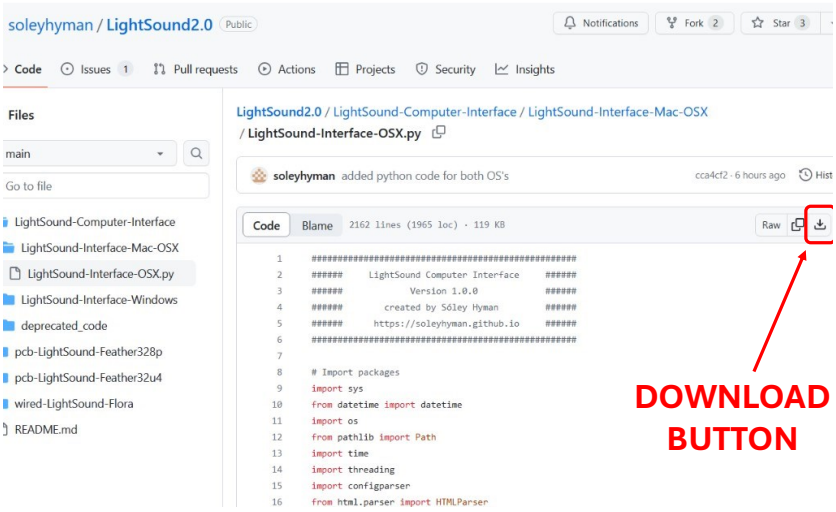
9. Open up your Python interpreter again by typing **`python`** in your terminal again and pressing Enter.
10. Almost there! Now you just need to make sure that you have the appropriate Python packages installed so that you can record LightSound data and plot it. Type the following lines in your Python interpreter, pressing Enter after each one:

    ```
    import numpy
    import matplotlib
    ```

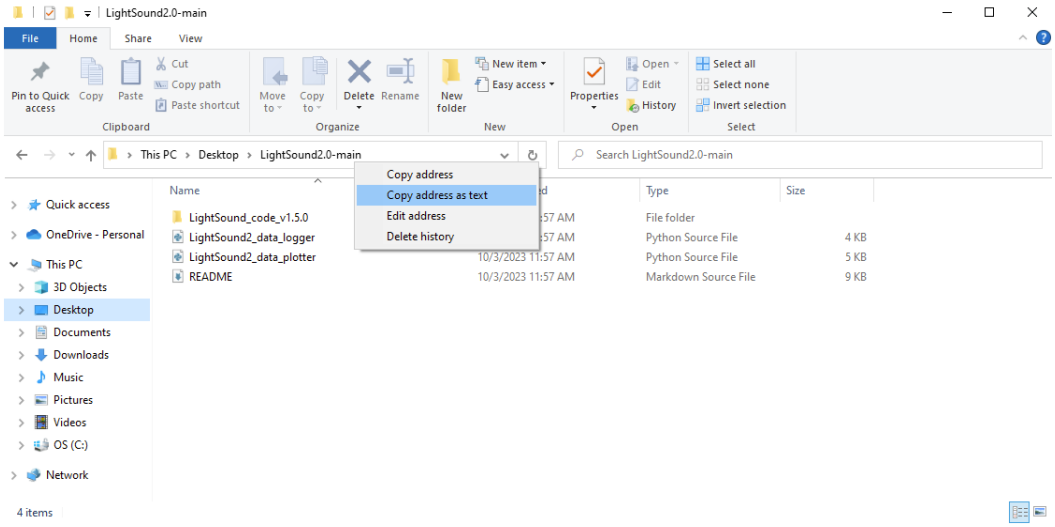
11. If you get an error from either of those (i.e., `ModuleNotFoundError: No module named 'numpy'` or `ModuleNotFoundError: No module named 'matplotlib'`), exit your Python interpreter with the **`exit()`** + Enter command sequence and install whichever packages don't exist with **`pip install numpy`** and/or **`pip install matplotlib`**.
12. Install the pySerial package (for reading in data from a USB port) by typing **`python -m pip install pyserial`** into your terminal and pressing Enter.
13. Install the graphical interface package wx by typing **`pip install wxpython`** and pressing Enter.
14. Congratulations! You have installed Python on your computer.

Setting up a data folder for the LightSound

1. Now that you've got Python all set up, we want to test the LightSound data recording and plotting. Before we do that, we need to download the code from the LightSound GitHub.
    - If you are a **Windows user**, download the code from this link via the Download button (i.e.,).
    - If you are a Mac or Linux user, you can download the code from this link. That should download in your "Downloads" folder.



2. Navigate to your Downloads folder and move the code (named either LightSound-Interface-Windows.py or LightSound-Interface-OSX.py) to the location you want to save your data (e.g., Desktop or Documents – somewhere you can easily find).
3. In the same location, create a folder called **LightSoundData**. This is where your data will save to by default.
4. Determine the "address" of your folder.
    a. Windows: In the path bar of the folder window, right click the option the farthest to the right and click "Copy address as text"



    b. Mac: Control-click the folder in the path bar, then choose Copy *"folder"* as Pathname.
5. Now we need to navigate to the folder your code is in. If you closed your terminal, reopen it again. If you have a Windows computer, go to step 6. If you have an Apple computer, go to step 7.
6. If you are on Windows, type `cd` and then a space and then **either** right-click in the terminal window or use the Ctrl+V paste sequence. This should paste the folder address in the terminal. Press Enter. At the bottom of your terminal, you should see something like `C:\Users\yourname\Desktop\`, depending on where you put the folder. Check that the contents are there by typing **`dir`** and press Enter. You should see a list of files that includes **`LightSound-Interface-Windows.py`** and **`LightSoundData`**.
7. If you are on Mac, type **`cd`** and then a space and then Copy text in another app, and then in Terminal, choose Edit > Paste**.** This should paste the folder address in the terminal. Press Enter. At the bottom of your terminal, you should see something like `/Users/UserName/Desktop/`. Check that the contents are there by typing **`ls`** and press Enter. You should see a list of files that includes **`LightSound-Interface-OSX.py`** and **`LightSoundData`**.

8. Congratulations! To run the program:
    - **Windows:** Type **`python LightSound-Interface-Windows.py`** and press Enter.
    - **Mac/OSX/Linux:** Type **`python LightSound-Interface-OSX.py`** and press Enter.