**How to setup stuff on Windows**

**gfortran with NetBeans**

* Go to Cygwin and install 64-bit. You need to make sure that you install *gfortran*, *g++*, *gdb*, *make* and *gcc*. I am not really sure what is needed so I tend to install too many packages rather that too few. Install in **c:\cygwin64**
* Go to your Environment Variables in advanced system settings in windows and include *C:\cygwin64\bin;C:\cygwin64\usr\bin;C:\cygwin64\usr\local\bin;C:\cygwin64\lib;C:\cygwin64\usr\lib* in your Path.
* Install NetBeans from [www.netbeans.org](http://www.netbeans.org). You only need to download the C/C++ version.
* If you don’t have the correct Java, follow the link presented to you and install correct version.
* Copy your code to a folder of your choice.
* Create a new project (C/C++ from Existing Source) and use you folder as the project folder. Keep all other settings.
* You are ready to work.

**NOTE**: Another nice thing to do is to use gfortran from your cluster on your windows PC. Do the following:

* In Netbeans, go to *Tools>Options>C/C++* and click **Edit** next to localhost. Click **Add…** and write **metcl2**. Just keep on clicking until you need to give your username and password for the cluster.
* Now you should be able to run GNU on the cluster from your windows PC.

**Python and PyCharm (Not so good alternative)**

1. Install python 2.7.X, 64 bit from python.org (Windows x86-64 MSI installer). Install with default settings.
2. Visit JetBrain, Pycharm website and obtain a student account (go to buy and renew, <https://www.jetbrains.com/pycharm/buy/>). Click on for *Students and Techers*, go to bottom of the page and click **Apply**. You will get an email where you activate your licence.
3. Create a folder where you can use a project folder. Copy the python code from the suews repository and put it the folder.
4. Download PyCharm professional and install.
5. Start PyCharm and activate license using your JetBrains account.
6. Create a new project (Pure python) and choose the created folder (3) as your project folder and use your python installation as interpreter. Click ok in the next message box.
7. Go to *File>Settings >Project Interpreter*. Add a new package by clicking the green plus sign. Search for **numpy** and install package. If you get errors, you probably need correct version of Visual studio. There is an address of a website where you can download it in the error message when you tried to install **numpy**.
8. Also install matplotlib (used for plotting)
9. Run **mainfileLondon.py** to do stuff.

**Python and PyCharm (good alternative)**

1. Go to qgis.org and click on download. Choose the installation for advanced users (64-bit). Do an express desktop installation and use all other default settings. This give you a python installation with everything you need (pretty much).
2. If you haven’t installed PyCharm, follow set 3 through 5 above.
3. Get the .bat-file from our repository (PyCharmWithQgis.bat) and edit it so that the paths on line 1 and 5 is correct.
4. Double-click on it and Tada, you have the possibility to use scipy etc.

**How to make standalone application using py2exe (this is not used, see below)**

1. In PyCharm, add the **pip** package (if not already there). See bullet point 6. Above.7.
2. Go to <http://www.lfd.uci.edu/~gohlke/pythonlibs/> and download the appropriate py2exe package (.whl).
3. Open a command prompt and go to the folder where you download the py2exe package and write:

pip name\_of\_whl\_file

1. Create a file called setup.py in your working directory with the following code:

from distutils.core import setup

import py2exe

setup(console=['Suews\_wrapper\_v2015a.py'])

1. From a command prompt (can use terminal in PyCharm) write:

python setup.py install

1. Then write:

python setup.py py2exe

1. All files and folders needed are now created in a subfolder call **dist**. You also have to add the SUEWS executable and all files needed to run the model.

**How to make standalone application using Pyinstaller (use this)**

1. Add the pip package (see above)
2. You need to add the path to where pip.exe is located (usually C:\Python27\Scripts\). If you don’t know how to add a path in your environment settings you can temporarily add one in a command prompt by writing:

path %PATH%;C:\Folder\_where\_pipexecutable\_is\_located

1. In the same command prompt, write:

pip install pyinstaller

1. Locate yourself where you have your script and write:

pyinstaller suews\_wrapper\_v3.py