

pymedeas models

Installation instructions for MS Windows

1. Download and install Anaconda 3.5 (Python 3.7) using the default parameters. Write down the installation directory, because you will need it later (we will call this **INSTALL_PATH** from now on). Read *NOTE 1* and *NOTE 2* at the end of this section for particular installation cases (e.g. Anaconda already installed)
2. Add the `INSTALL_PATH` to the Windows path. To do so, open the terminal (click on Windows Start Menu, write `cmd` and type Enter) and run the following command (replacing `INSTALL_PATH` by your actual installation path):

```
SETX PATH "%PATH%;INSTALL_PATH\Scripts;INSTALL_PATH"
```

3. Log out from your user account and log back in to apply the changes.
4. To install `pipenv` package, open the terminal (`cmd`) again and run:

```
pip install pipenv
```

5. Still in the terminal, go to the folder where you downloaded the MEDEAS model using `cd` command (replacing the below path with the actual path to the folder where you downloaded the model):

```
cd C:\Users\UserName\MEDEAS-model
```

6. From there, run the following command to install all Python packages required to run the model in a virtual environment:

```
pipenv install --python=python.exe
```

7. In the output of the previous command you should be able to identify the path where the virtual environment was created. Write it down for later use.
8. Congratulations, you can move to the next step! Go to *Running a simulation from terminal* section to verify that everything went well during the previous steps, and to try to run the

model.

- NOTE 1: By default Anaconda3 installs in your user directory as C:\Users\UserName\Anaconda3 but if you have spaces in the UserName (e.g. C:\Users\Guido van Rossum\Anaconda3) Anaconda might have issues to install further packages. In that case chose to install for All Users (you will need administrative privileges), which will install Anaconda3 in C:\ProgramData\Anaconda3.
- NOTE 2 : If you have an older version of Anaconda installed (Current version 3.5 was released in October 2018) you should install the new one while keeping/removing the old one.

Installation Instructions for Linux

1. Download Anaconda 3.5 for linux. Open the terminal, go to the folder where the package was saved and run (replacing Anaconda3-5.3.0-Linux-x86_64.sh by the name of downloaded file):

```
bash Anaconda3-5.3.0-Linux-x86_64.sh
```

2. Follow the installation instructions, leaving all parameters by default. This will install anaconda in your home directory (e.g. /home/user/anaconda3). When asked: *Do you wish the installer to initialize Anaconda3 in your /home/roger/.bashrc ? [yes|no]*, Answer yes. You can say no to the installation of Microsoft VSCode.

3. The following command should point to your anaconda installation directory:

```
which python
```

4. Install pipenv from terminal with the following command:

```
pip install pipenv
```

5. Still from the console, go to the folder where the model files are, and execute:

```
pipenv install --python 3.7
```

6. In the output of the previous command you should be able to identify the path where the virtual environment was created.

7. You are done. Go to *Running a simulation from terminal* section of this document to verify that everything went well during the previous steps, and to try to run the model.

Installation Instructions for MacOS

1. Download and install Anaconda 3.5 for MacOS (Python 3.7).
2. Open a terminal and install Xcode Command line developer tools with the following command:

```
xcode-select --install
```

3. Confirm that pip executable is inside the anaconda3 folder with the command below (you should see something like `/Users/your_user/anaconda3/bin/pip`):

```
which pip
```

4. Install pipenv with the following command:

```
pip install pipenv
```

5. Still from the console, go to the folder where the model files are, and execute:

```
pipenv install --python 3.7
```

If the previous step did not work, you need to add pipenv path to your system path. To do that, add the following line at the end of your `.bash_profile` (in your home directory, press `Cmd + Shift + .` (dot) to see the `.bash_profile` file) (replace the **ANACONDA_PATH** by the path where you installed anaconda e.g. `/Users/medeas/anaconda3/`):

```
export PATH="ANACONDA_PATH/bin:$PATH"
```

After saving the file, run the following command in the terminal:

```
source ~/.bash_profile
```

Then try again the command of step 4.

6. Congratulations, you can move to the next step! Go to *Running a simulation from terminal* section of this document to verify that everything went well during the previous steps, and to try to run the model.

Running a simulation from terminal (Windows/Linux/MacOS)

1. Open a terminal and go to the project folder (using the `cd` command)
2. Activate the project virtual environment running the following command:

```
pipenv shell
```

3. At this point, you should be able to run a default simulation with the following command:

```
python run.py -x bau -i 1995 -f 2050 -t 0.05 -r 1.0 -p
```

NOTE: to see all user options, run:

```
python run.py -h
```

Using the plot GUI to plot previous simulation results from terminal (in csv format)

1. Open a terminal and go to the project folder (using the `cd` command)
2. If it's not active yet, activate the project virtual environment running the following command:

```
pipenv shell
```

3. Run the following command:

```
python plot_tool.py
```

Python IDE of choice (PyCharm Community Edition)

If you would like to use a graphical IDE instead of the command line, we recommend you to download **Pycharm Community edition**, which is free and open source. You can download it for [Windows](#), [Linux](#) or [macOS](#).

After installation, follow these steps to get the model working:

1. Open the folder where you downloaded the model: go to File\Open and select the model folder
 2. Make Pycharm use the virtual environment that you created for the current project (if it doesn't already):
 - Go to *File\Settings* (PyCharm\Preferences in macOS) and start typing **project interpreter**.
 - If the environment you created appears in the drop-down list you can skip the next step.
 - If the environment does not show up, you need to tell Pycharm where to find it. Click on the sprocket wheel icon, in the top right corner and click on **Add**. This will open a new window, where you should check the option **Existing environment**, and select the path where your environment was stored:
 - You can get the path to your virtualenv by running the following command in a terminal:
- ```
pipenv --venv
```
- on Windows it should be something like  
`C:\Users\user_name\virtualenvs\name_of_virtualenv\Scripts\python.exe`
  - on macOS: `/Users/user_name/.virtualenvs/name_of_virtualenv/bin/python3.7`
  - on Linux: `/home/user_name/.virtualenvs/name_of_virtualenv/bin/python3.7`
- Click *Ok* and *Apply*
3. In the main menu, click on *Run/Edit* configurations
  - Click on the plus sign on the top left corner and choose Python
  - Fill in the boxes as follows:
    - *Name*: Run model
    - *Script path*: select the run.py file from the project folder
    - *Parameters* : -s -t 0.03125 -r 1.0 -x bau -p
    - *Python Interpreter*: select the one you added in step 2 from the drop-down menu
  - Click *Ok*
4. On the top right corner of the screen you should now see a play icon beside the words **Run model**, click on it to run the simulation.

If you would like to be able to run the **plot GUI** to plot results of previous simulations in PyCharm, repeat step 3 changing the parameters to:

- *Name*: Plot

- *Script path*: select the plot\_tool.py file from the project folder
- *Parameters* : (leave blank)
- *Python Interpreter*: select the one you added in step 2 from the drop-down menu