

Preparations for online course simulations

To do this online course on Gravitational wave laser interferometry you need to have Python, FINESSE and PyKat installed. If you do not have these installed already, follow the installation instructions provided below. If you have questions, need help, or have some comments on these instructions, please contact us:

Daniel Töyrä: dtoyra@star.sr.bham.ac.uk,

Daniel Brown: ddb@star.sr.bham.ac.uk

Andreas Freise: adf@star.sr.bham.ac.uk

Python installation

Before starting the school the Python ecosystem has to be installed for the optical simulation sessions. One of the easiest way to install Python and the required dependencies is through the miniconda suite: <http://conda.pydata.org/miniconda.html>



	 Windows	 Mac OS X	 Linux
Python 2.7	64-bit (exe installer) 32-bit (exe installer)	64-bit (bash installer)	64-bit (bash installer) 32-bit (bash installer)
Python 3.5	64-bit (exe installer) 32-bit (exe installer)	64-bit (bash installer)	64-bit (bash installer) 32-bit (bash installer)

(If you have a preferred method for installing Python on your computer, feel free to use that. The instructions provided here are meant for students who do not have experience with installing Python, which can be significantly more difficult than installing other software packages.)

The notebooks are written for Python 3, so we recommend that you use Python 3 as well. Select the installer according to your operating system on the last line of the above table.

For **Windows**, simply double click on the *.exe and follow the instruction on screen. For **Linux** and **OS X**, you first have to make the file executable:

```
chmod +x Miniconda3-latest-Linux-x86_64.sh
```

Then execute the script:

```
sudo ./Miniconda3-latest-Linux-x86_64.sh
```

And answer positively to the questions.

Open a command windows or a new terminal (this will not work in the window used for the installation, close that and open a new one!) and type: `conda info`

If the installation has been successful, you should see the information related to conda on the screen terminal. For Linux, it looks like

```
Current conda install:
```

```
platform : linux-64
```

```
conda version : 3.9.1
conda-build version : not installed
python version : 3.4.2.final.0
requests version : 2.5.3
...
```

Now when we have conda installed, we will use conda to install the Python packages that we need. For **Linux** and **OS X**, set your miniconda folder with all the permission first (this is not necessary in **Windows**):

```
sudo chmod -R 777 miniconda3
```

Now, to install the package in the terminal window use the following command:

```
conda install numpy scipy matplotlib ipython ipython-notebook pip
```

And that's all! One can see the list of the installed packages with the command:

```
conda list
```

OS X only: miniconda has now created the file `.bash_profile` if you did not already have it. This is a problem if `.bash_profile` was not already your primary login shell (the file executed to configure your shell before the initial command prompt), because **OS X** will now execute `.bash_profile` instead. To check and solve this:

1. Open `.bash_profile` and check if the only line in there is from miniconda. If not, everything is fine, `.bash_profile` was already your login shell. In case the only line is from miniconda, then proceed to the next step below.
2. Copy the text line in `.bash_profile`. Check if you have a file called `.bash_login` in your home directory. If you do, paste your copied text line into this file, save it, and then delete `.bash_profile`. Otherwise, you should find a file called `.profile` in your home directory. Paste your copied text line into this file, save it, and then delete `.bash_profile`.

Finesse installation

FINESSE is an interferometer simulation software. There are binaries available for several operating systems. After downloading the appropriate package for your operating system from <http://www.gwoptics.org/finesse/> you can install FINESSE simply by unpacking the zip (or tar.gz) file. This will create a directory 'Finesse2.1' with all the necessary files.

Linux and OS X:

Once you have move the downloaded folder to the location you want to store FINESSE you must setup some environment variables to allow you to call FINESSE.


To do this you must edit the file `.bashrc` (**linux**) or the file `.bash_profile/.bash_login/.profile` (**OSX**). These files can be found in your home directory "cd ~". If they are not there, just create the relevant file. Using a text editor add the following lines:

```
export FINESSE_DIR = [FINESSE_FOLDER_LOCATION]
export KATINI="${FINESSE_DIR}kat.ini"
export PATH=${FINESSE_DIR}:$PATH
```

Where you must replace [FINESSE_FOLDER_LOCATION] with the path to where you have saved the Finesse2.1 folder. For example, in my .profile file I have:

```
export FINESSE_DIR= /Users/toyradaw/Finesse2.1/
export KATINI="${FINESSE_DIR}kat.ini"
export PATH=${FINESSE_DIR}:$PATH
```

Save this file, close the terminal and then reopen it. Type 'kat' and press enter. You should now see the following output:



```
-----
      o_._=._      FINESSE pre-2.1.0      (build pre-2.1.0-101-g41626c1)
     \'.\".\\      Frequency domain INTERferomEter Simulation Software
      .>' C_--_      10.04.2015      http://www.gwoptics.org/finesse/
    _=/d' ,^\\      Input file *,
   ~ \\-' '      Output file *,
    / |      Gnuplot file *
   ' '
                                     Thu Apr 16 02:25:07 2015
-----

*** Error:
usage: kat [options] infile [outfile [gnufile]]
      or usage: kat [options] basename
      (kat -h displays help message)
```

Windows:

Firstly you must move the FINESSE folder to a location on your computer where you want to store it. Unless you are an expert we recommend put it **not** in a system wide directory such as 'Program Files' but for example use a new folder on the Desktop. Once you have moved FINESSE to the desired location you should double click the 'install.bat' file. Running this will bring up a command window to update your PATH variables. After this has been run you must **not** move this Finesse folder, as the system PATH variable has been set for this directory only. If you do move the folder then please re-run install.bat.

Pykat installation

Pykat is a selection of Python-based tools to run optical simulations and to call FINESSE from Python. The easiest way to install pykat is to use pip which was installed with miniconda. Type the command "pip install pykat". Once this is complete, open a new python session (type 'python' in the terminal) and type the command "import pykat", you should then see the following output:



If you are running a different variant of python than Anaconda or Miniconda you will need to search the web on how to install pip for your particular setup. Email us if you have any issues.

iPython notebooks

We will work exclusively with and in 'ipython notebooks'. To start a notebook, start a terminal window and go to your preferred working folder. Type `ipython notebook`. The notebook server should open as a new window in your browser. Press new and chose Python3. Copy and paste the below code to test if everything is working as expected. If everything works, a .ascii drawing of a cat eating a pie is shown, and no error messages should be shown. If you have any problems, please let us know.

```
from pykat import finesse
import matplotlib
import matplotlib.pyplot as plt
from IPython.utils import io
with io.capture_output() as captured:
    from IPython import get_ipython
    ipython = get_ipython()
    ipython.magic("matplotlib inline");
```

```
kat = finesse.kat()
code = ""
l laser 1 0 n0
s s1 1 n0 n1
m mirror 0.9 0.1 0 n1 n2
pd pout n1
xaxis laser P lin 0 10 200
yaxis abs
""
kat.parseCommands(code)
out = kat.run()
fig = out.plot()
```