README for the NMR titration practical

version September 2023, USP NMR course

In this practical you will perform an *in silico* quasi-realistic NMR titration experiment to study protein-protein interactions.

The practical runs in a self-guided manner, providing instructions on what to do. There are multiple choice questions along the way, which are automatically graded.

At the end you are asked to send your result by email to me.

This document explains how to get started with the practical. The last page has a FAQ for some common questions.

To do the practical you need to install two programs on your laptop:

1. GNU Octave, a free, open-source version of the commercial MatLAB software
2. the NMR simulator program, which is written in GNU Octave.

1. Installation of GNU Octave (~15 min)

Install GNU Octave on your laptop or PC. GNU Octave is available for Windows, Mac and Linux.

- Windows:
  - go to: https://octave.org/download
  - scroll to the Windows section
  - download either the 32-bit or 64-bit version depending on your computer, i.e. octave-8.3.0-w64-installer.exe
  - click "open file" to run the installer;
    - you may have to allow the program to run
    - select install for all users
    - use all default options
    - The default install location is:
      C:\Program Files\GNU Octave\Octave-8.3.0
      If you use another location, note the path where you install it you will have to edit the startup script of the simulator later.
    - close the installer

- Mac OS:
  - install Octave.app from https://octave-app.org

- Linux:
  - use the package manager of your Linux flavor to install octave.

2. Install the NMR simulator program (~5min)

To install the actual NMR simulator you have to install my Octave based scripts.

- download the file **nmrSim.zip** from:
- Windows:
    - click "open" if prompted; then click "extract all" icon;
      select a suitable location to install the practical in your home folder;
      click "show files after extraction is complete". You now should see the "nmrSim" folder that contains the practical.
    - open the nmrSim folder; here you find the startup script "titrationSimulatorWindows.bat".
    - if you used the default location for GNU Octave in the step above, you're done
    - if you used another install location, you have to edit this script:
        - edit the "titrationSimulatorWindows.bat" script by right-clicking and select edit. if you get a security warning, just run anyway (you can trust me ;))
        - now change "`C:\Program Files\GNU Octave\Octave-8.3.0`" in the 2nd line to point to your install location. Save the file.
- Mac/Linux:
    - click the downloaded **nmrSim.zip** file, its contents will be extracted automatically in the Downloads folder.
    - Move the nmrSim folder to a suitable location in your home folder.

3. Start the NMR titration practical (~90 min)

Open the "nmrSim" folder in your file browser, then

- Windows:
    - double click the "titrationSimulatorWindows.bat" file to start the practical
- Mac:
    - double click the "titrationSimulatorMac" file; you may have to allow the program to run (you can trust me ;))
- Linux:
    - open "titrationSimulatorLinux"

You should see the opening screen:

```
****************************************
***                                  ***
***       titrationSimulator         ***
***                                  ***
****************************************
***      powered by GNU Octave       ***
****************************************

++++++++++++++++++++++++++++++++++++++++
++          DO THE PEAKS MOVE?          ++
++++++++++++++++++++++++++++++++++++++++
```

```
     ++ (c)2023-v15 HvI Utrecht University ++
     ++++++++++++++++++++++++++++++++++++++++


Welcome to the NMR titration simulator.

Whenever you see this symbol: <> press return/enter to
continue!

Whenever you see this symbol: :)] you have to enter a command.
Which command will be clear later on.

Good luck!

<>
```

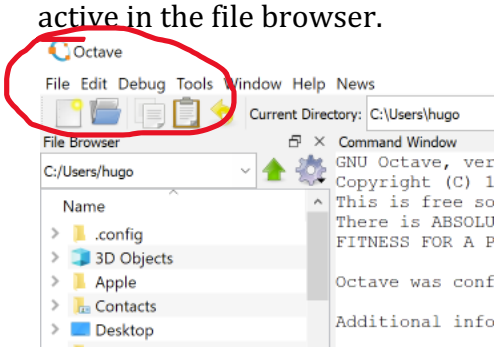
Read it and follow the on-screen instructions.
Next page has a FAQ for some common questions.

4. What to do if the program crashes immediately

The GNU Octave start-up scripts should be able to deal with spaces in file or directory names. Still it could happen that your user name or other folder names contains characters that break them.

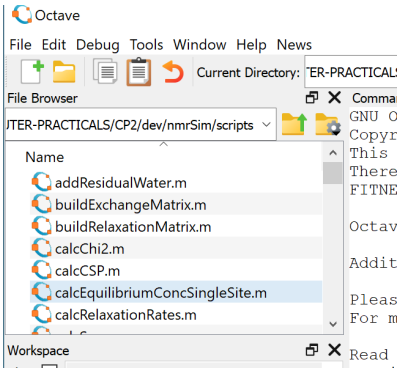
The workaround is to use the GUI app of GNU Octave:

1. Open GNU Octave app (type “octave” in search box)
2. Then find the file browser of GNU Octave, it should be located at the top-left corner of the app window. You can press “Cntrl-2” to have the cursor active in the file browser.



3. Navigate in the file browser to the directory where you saved nmrSim files.

4. Click on the “scripts” dir to enter into that dir so that you see a list of .m files:



5. type “ini” in the command window to start the practical. You can close the other side windows to reduce clutter.
6. The simulator should start now, if not let me know!

5. FAQ

1. help I do not know the answer

Don’t worry, just pick an answer, the program will explain the right answer.

2. help I do not know which command to type

The program will guide you on what to do. Should you get lost, you get a list of all commands available using `listCommand`. If you type a command it will explain how to use it, e.g. when you type `edlev`, you get

```
Usage:
        edlev(number_of_levels, contour_increment, base_level_in_peak_height_fractions)

        edlev(10,1.4,0.05)

        will draw 10 contour lines with 1.4-fold increments starting at 5% of the peak height
```

So the way to use it is to specify the arguments: `edlev(10,1.4, 0.1)`

3. help I do not see my peaks anymore/ I only see one spectrum / my peak labels are gone

type `plotAll` at the command prompt

4. help I made a mistake, I want to go back

you can undo one addition of ligand, before recording the spectrum, by using the `unAdd` command. You can start over by making a new sample with `makeSample`.

5. help I want to stop and continue later

first save your work by `saveState` then type `exit`. When you restart the program later it will reload all your data.

6. help the program says xxx is undefined!

this happens if you make a typo in a command, for instance:

```
:)] zf
error: 'zf' undefined near line 1 column 1
:)] zg
```

7. help, I get some weird error message

in some cases you will get a warning when doing the peak picking:

```
warning: opengl_selector::select: selection buffer
overflow
  warning: called from
```

```
ginput at line 88 column 7
calcCSP at line 137 column 30
```

This is not a problem, you can simply ignore it.

8. help, the chemical shift perturbation analysis does not work

in a few odd cases the clicks in the plot window are not registered properly. First try again, making sure to have the plot window active (you may to first click the title bar), and making sure none of the plot window tools (the zoom , rotate etc buttons) are activated, and click slowly at the peak position.

If it really does not work, save your work using `saveState` then type `exit`. Start GNU Octave main application from the Start Menu or Application folder. Use the GUI version. Contact me with the output of the following commands

```
graphics_toolkit
x=[1:10];
plot(x, sin(x))
[X, Y, BUTTONS] = ginput(1)
```

the last command should give output something like:

```
X =  3.1613
Y = -0.47794
BUTTONS =  1
```