UNIVERSITEIT GENT

Department of Civil Engineering
Technologiepark 904
B-9052 Zwijnaarde

Tim Verbrugghe
+32 9 264 5567
timl.verbrugghe@ugent.be

- openWEC Manual -

Revision	Date	Author
1.0	08/07/2015	Tim Verbrugghe

Contents

1 Introduction

openWEC is an open-source tool to simulate the hydrodynamic behaviour and energy yield from heaving point-absorber type wave energy converters. It is based on the linear wave theory and assumes potential flow. Two software packages are coupled:

- Frequency domain solver Nemoh, developed by Ecole Centrale de Nantes
- Time domain solver, developed in-house

2 Installation & Getting Started

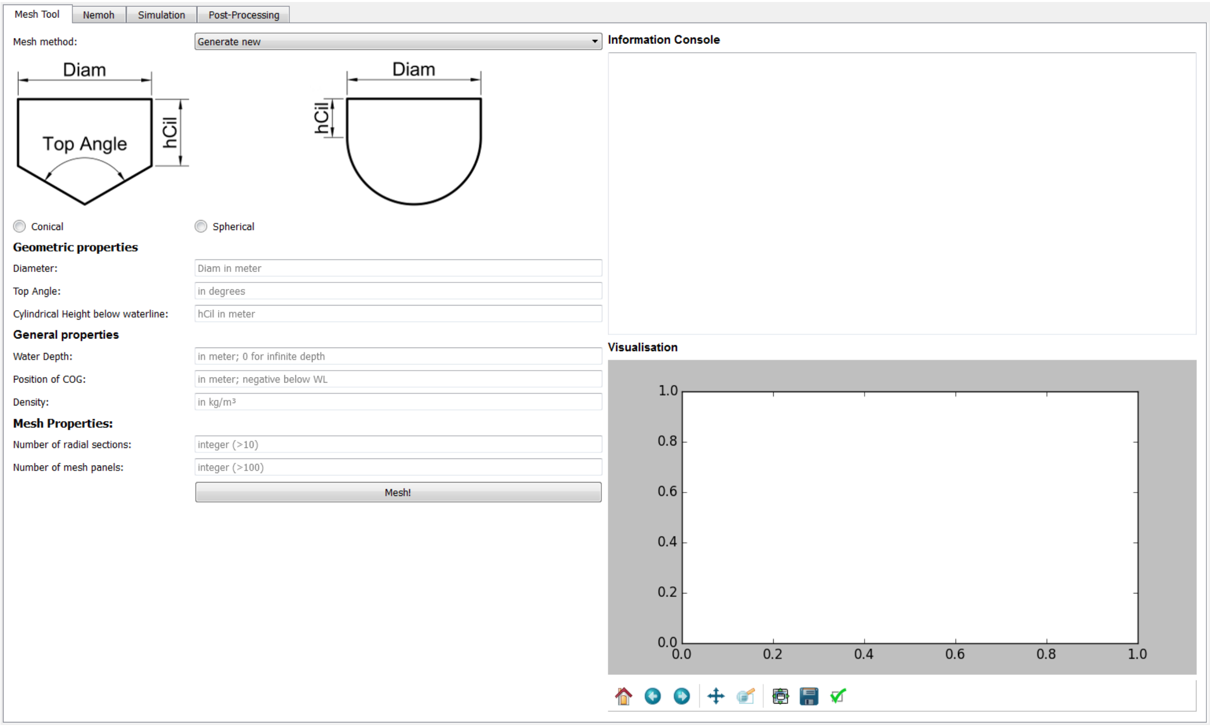
- Double click 'setup_openWEC.exe'
- Follow the steps in the setup

3 Frequency domain modelling

The frequency domain modelling is performed with the Nemoh BEM Solver. It is based on the linear wave theory and thus assumes potential flow. For each panel of a mesh, the hydrodynamic parameters are calculated for a certain frequency range.

3.1 Mesh Tool

First, a mesh needs to be created. This is done with the Mesh Tool:



There are 4 different ways to create a mesh:

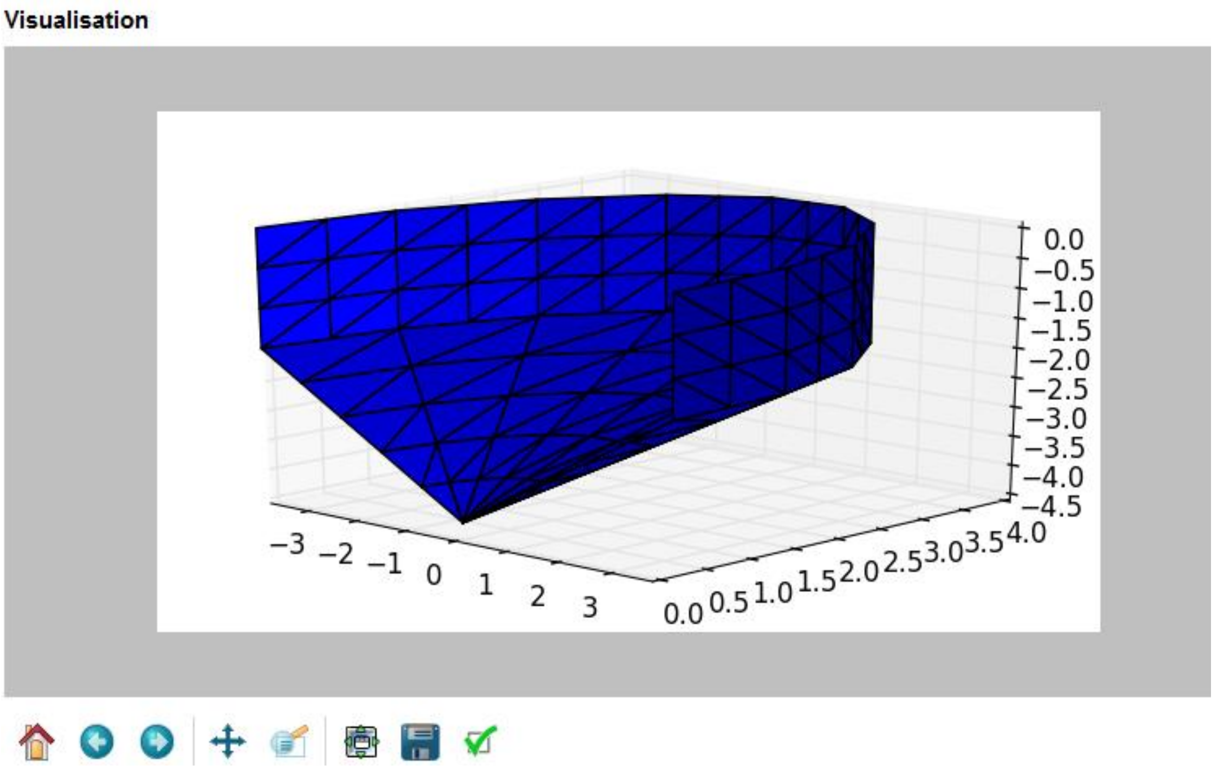
- **Generate a new mesh** based on predescribed shapes: a floating buoy with a conical or spherical bottom, and a cylindrical top.
- **Import a Nemoh mesh** from a previous simulation
- **Convert a .stl mesh** you have created with other software like MeshLab
- **Import results from a previous simulation** and skip to the time-domain solver

Depending on what option you have selected, you must fill in some of the parameters below the WEC shapes. If you have selected anything else than 'Generate new', the information console will display what options need to be filled. When selecting ‘Generate new’, all boxes need to be filled:

- Select a **type of bottom shape**: conical or spherical
- **Diameter**: diameter of the top cylindrical part in meters
- **Top Angle**: angle of the conical bottom shape, fill in any number if you have selected ‘spherical’
- **Cylindrical height below waterline**: all created meshes are fully submerged (negative z coordinates). The part underwater which is still cylindrical is characterised by this parameter
- **Water Depth**: 0 for infinite water depth, otherwise a positive number in meters
- **Position of COG**: The position of the centre of gravity with respect to the waterline: positive is above the waterline, negative is below the waterline. Mostly the COG should be below the waterline to assure good hydrostatic stability
- **Density**: water density in kg/m^3
- **Number of radial section**s: number of mesh discretisations along the circumference of the mesh.
- **Number of mesh panels**: estimate of the desired number of panels. More panels give more accurate results but require more computing time.

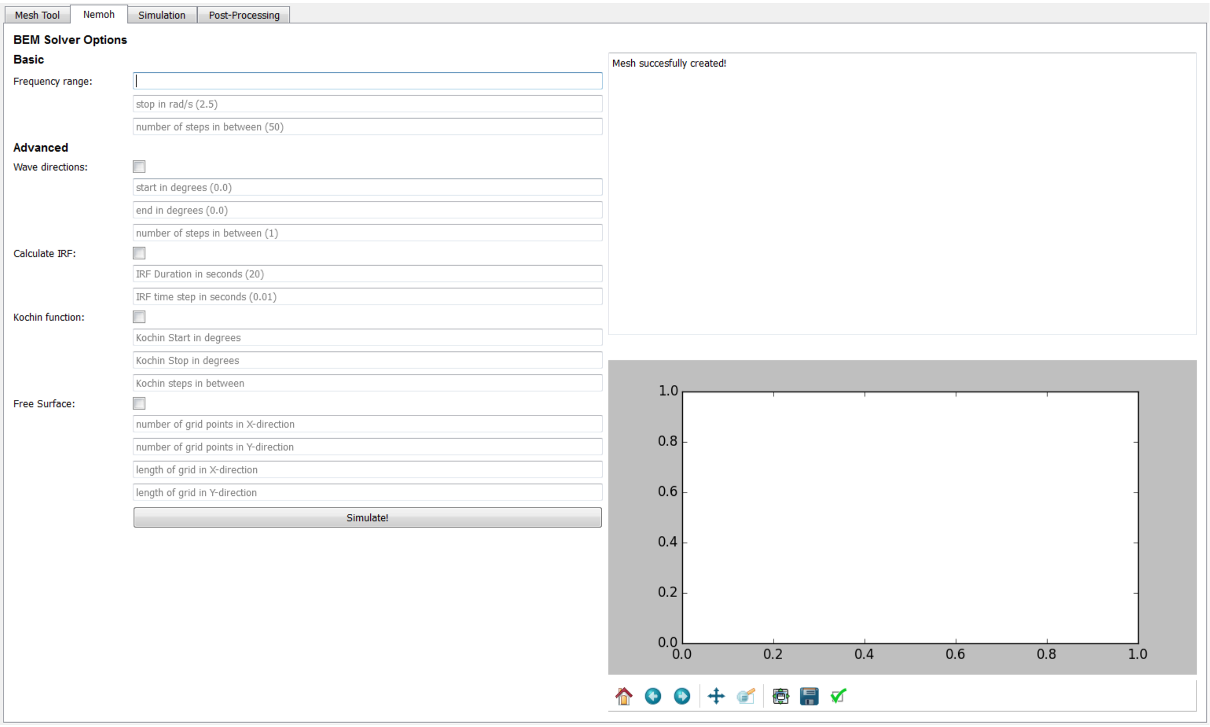
Once every required option is filled, the mesh is created by pressing the ‘Mesh!’ button. In the shell window you will see the code is running. Once finished, the created mesh is displayed in the plotting window. When creating a mesh from a .stl file, make sure you visually check the mesh for inconsistencies. The .stl mesh must be well created to result in a good Nemoh mesh!

The result files of the mesh are found in the ‘./Calculation/mesh’ folder in the main program directory.



3.3 Nemoh

Next, the BEM solver can be run to calculate the hydrodynamic parameters:



The solver can be run with only the basic options, or you can choose to include several advanced options. Within the basic options you are required to give a frequency range for the calculations by entering three values:

- **The starting frequency** in rad/s (e.g. 0.2 rad/s)
- **The ending frequency** in rad/s (e.g. 2.5 rad/s)
- **The number of frequency steps** (preferably >50)

The advance functions can be enabled or disable with the different check boxes:

- **Wave directions**: only applicable for non-axisymmetric shapes, otherwise you will get the same results for every wave direction.
- **Calculate IRF**: When selected, the impulse response function is calculated. This option is needed if you want to use irregular waves in the time-domain.
- **Kochin function**: calculates the kochin function for each solved problem
- **Free Surface**: calculates the free surface elevation around the WEC for all solved problems, for a given grid.

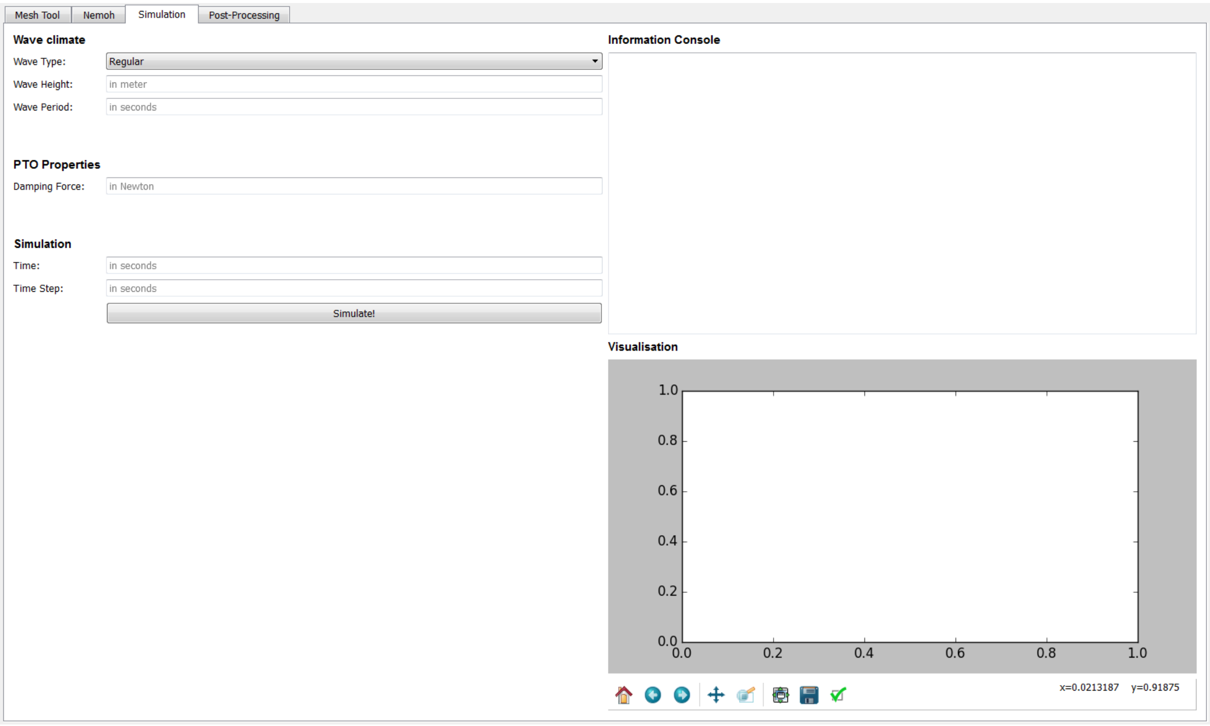
The calculation will start when pressing the 'Simulate!' button. In the shell window you can follow the calculations. Once finished the Added Mass and Hydrodynamic damping will be plotted in the visualisation window.

4 Time domain modelling

Once the frequency domain modelling is finished, the time-domain solver can be used. Here the WEC heaving response and energy absorption is calculated for regular or irregular waves.

4.1 Simulation

The time-domain solver can be accessed through the tab ‘Simulation’:

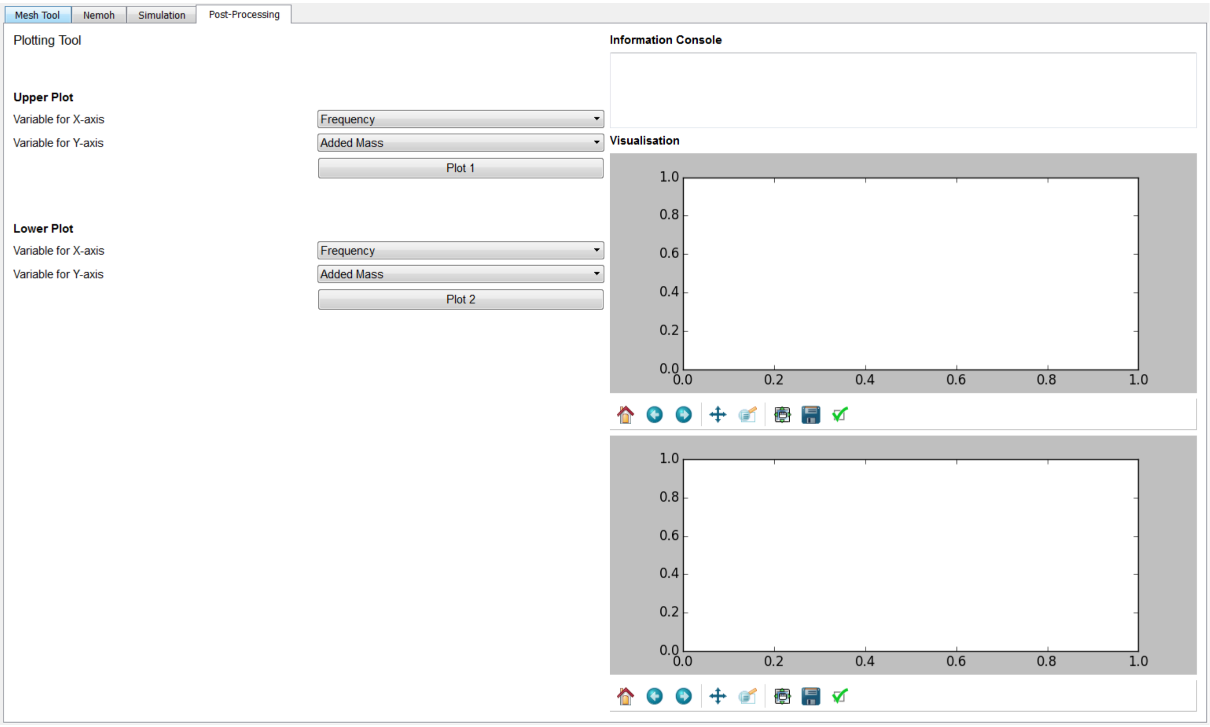


Options are split up into three sections: wave climate, PTO properties and Simulation. All option boxes need to be filled in to be able to simulate.

- **Wave Type**: select between irregular or regular waves
- **Wave Height**: enter the wave height in meters
- **Wave Period**: enter the wave period in seconds
- **Damping force**: enter an optional PTO constant damping force which will be applied during positive velocities (put 0 for a freely floating WEC)
- **Time**: simulation duration
- **Time Step**: time step for each iteration

5 Post-Processing

The final tab allows the user to postprocess the results:



In the plotting tool, you have two visualisation windows at your disposal. For each plot you can select a variable to plot on the x-axis and one on the y-axis. The options for the Y-axis variable will automatically change according to the selected x-axis variable.

6 Theoretical Background