

SeaFEM tutorials manual

<http://www.compassis.com>

info@compassis.com

November 2018

Version: 15.1.0

1. Bottom mounted cylinder

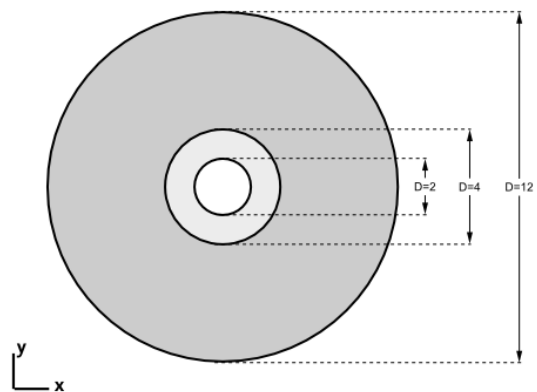
This tutorial shows how to create the CAD/CAE model corresponding to the validation case Bottom mounted cylinder of SeaFEM that analyzes the diffraction problem of a bottom mounted cylinder of radius $R=1m$ and height $H=1m$. The analysis is carried out for a monochromatic wave of amplitude $A=0.1m$ and wave length $\lambda=2m$ propagating in a domain of 1 meter depth.

Since water depth (d) is exactly one half of the wave length (λ) considered here, the corresponding wave period can be evaluated from a deep water approximation, this being generally valid for $d/\lambda > 0.5$. Under this assumption, wave length and wave period are correlated by the following expression:

$$\lambda = \frac{g \cdot T^2}{2\pi} = 1.56 \cdot T^2$$

The following image shows a top view of the computational domain. The intermediate zone centered at the origin and with diameter $D=4m$ has a relatively small element size assigned so that a more accurate solution can be obtained within the region close to the body.


It shall be usually necessary to introduce some artificial dissipation away from the body, specially when dealing with irregular sea state conditions. Such a dissipation is introduced in order to damp those waves refracted and radiated by the body and to avoid them to be reflected in the outer boundary of the computational domain. The region where artificial dissipation is introduced is called the absorption area, while the region close to the body where no dissipation exists is referred as the analysis area. It is a good practice to make the dissipation start sufficiently away from the body but inside of a finner mesh region to avoid internal wave reflections due to large element sizes and large mesh size transitions. Because of this, it is recommended to avoid abrupt changes in the mesh element sizes like those arising when intermediate auxiliary volumes with small element sizes assigned are combined with large values of the unstructured mesh size transition. Hence, in the present case a unique volume encompassing the entire computational domain is used. And the beginning of the absorption area (beach) is placed at a distance of 2 meters from the center of the scattering cylinder. This way, the free surface analysis area exactly coincides in this case with the finner mesh region centered at the origin.



The analytical solution for this analysis is taken from the work "R. McCamy and R. Fuchs, Wave forces on piles: a diffraction theory. Tech. Memo No. 69, U.S. Army Corps of Engrs, 1954". For further information on this validation case, please review the "Validation Case 1: Bottom mounted cylinder", at the following link:

<http://www.compassis.com/compass/en/Soporte>

To automatically load the model, ready to be meshed and run, simply left-click on the button below.

 [Load model](#)

On the contrary, to build the model from scratch proceed as indicated in the following sections of this tutorial.

1.1. Geometry definition

In this section, we will describe in detail the geometry generation process. SeaFEM has plenty of tools for geometry generation. Hence, the way to construct the geometry that is shown in this tutorial may constitute just one of many possibilities. For a more detailed description of geometry generation tools and other pre-processing capabilities of SeaFEM, we strongly recommend the user to take a look at the GiD pre-processing manual (see <http://www.gidhome.com/support>).

For the present case, both the body and the computational domain are cylindrical, and therefore they can be easily created by means of the geometry generation tools available in the GUI of SeaFEM. Before starting to construct the geometry, it is important to remark that the entire computational domain in

SeaFEM must be located below the undisturbed water surface (i.e $z \leq 0$). In the present case, the domain will be also centered at the origin (0,0,0).

Before starting to construct the geometry, few general advertisements become pertinent:

1. For the cases where no currents are imposed, the most indicated shape for the computational domain is a cylinder. The computational depth should be no larger than the physical depth. Moreover, computational depth might be smaller than physical and/or recommended depth if the bottom boundary condition is used.

2. A region with higher mesh resolution is usually needed close to the floating object in order to correctly capture the scattered waves solution (waves radiated and diffracted by the floating object). In the present case, the border of this area will coincide with the beginning of the damping/absorption area (beach), although this is not mandatory in general. Actually, the absorption area should be at least as long as the maximum wave length, although the recommended length is twice the maximum wave length. Nevertheless, if monochromatic wave is used along with Sommerfeld radiation condition, the absorption area might be reduced to half the wave length.

3. To further facilitate boundary conditions assignment and many other pre-processing operations, it is advisable to organize the geometry by layers.

Now, one of the several possible ways for generating the geometry of the present tutorial will be explained. Simply follow the steps in the given order.

First we proceed to create the bottom surface of the entire computational domain.

1. Open the layers window and rename the existing layer to *domain_contour*.

Utilities ► Layers

- Right click on the layer named *Layer0*

- Select the *Rename* option

- Enter the name of the new layer (i.e. *domain_contour*).

2. Create the bottom boundary of the computational domain. It consists of a circle located at $z = -1.0$ m. and of radius $R = 6.0$ m.

Geometry ► Create ► Object ► Circle

- Enter center: 0 0 -1

- Enter normal: Positive Z

- Enter radius: 6.0

3. Create a new layer called *Body*.

Utilities ► Layers

- Right click on the layers window

- Select the New layer option

- Enter the name of the new layer (i.e. *Body*).

4. Create a circle representing the base of the body.

Geometry ► Create ► Object ► Circle

- Enter center: 0 0 -1

- Enter normal: Positive Z

- Enter radius: 1.0

5. Delete the surface of the bottom mounted cylinder to retain just the lines that define the perimeter of the body.

Geometry ► Delete ► Surfaces

- Select the surface of the bottom mounted cylinder

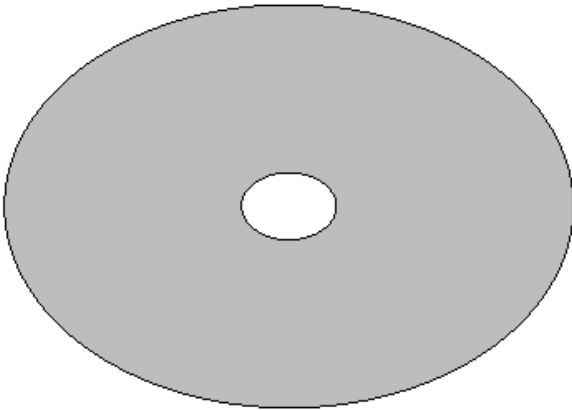
7. Generate a hole within the NURBS surface corresponding to the bottom boundary of the computational domain.

Geometry ► Edit ► Hole NURBS surface

- Enter surface: Select the domain bottom boundary surface.

- Enter lines to define the hole: Select the lines that define the base of the bottom mounted cylinder.

Up to this point, the geometry of the model should look like the figure below.



Isometric view of the bottom boundary of the computational domain

8. Create the lateral boundary of the computational domain. It can be simply created by extrusion of the lines delimiting the outer bottom boundary.

Utilities ► Copy

- Entity type: Lines
- Transformation: Translation
- First point: 6.0, 0.0, -1.0
- Second point: 6.0, 0.0, 0.0
- Do extrude: Surfaces
- Select the lines that define the outer perimeter of the domain's bottom surface

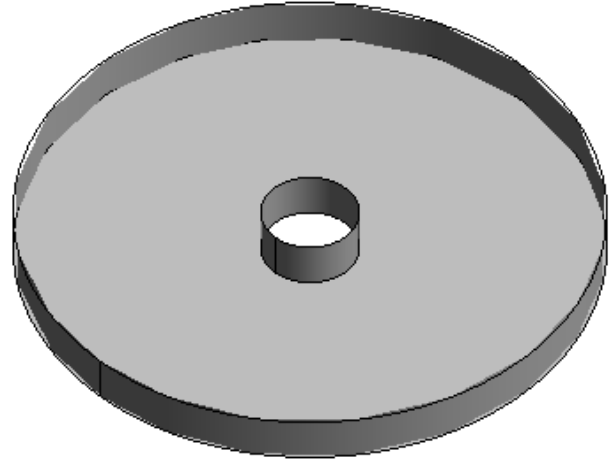
9. Create the inner lateral boundary of the computational domain, which coincides with the wall of the bottom mounted cylinder. As in the previous step, this can be done by copying and extruding the inner and outer lines of the domain's bottom surface.

Utilities ► Copy

- Entity type: lines
- Transformation: Translation
- First point: 6.0, 0.0, -1.0
- Second point: 6.0, 0.0, 0.0
- Do extrude: Surfaces

- Select the lines that define the perimeter of the cylinder's base

After this, the geometry of the computational domain should look similar to what is shown in the following figure.



Isometric view of the computational domain after extrusion of the domain's bottom lines, and before generation of the free surface

10. Create a new layer called *free_surface*.

Utilities ► Layers

- Right click on the layers window
- Select the New layer option
- Enter the name of the new layer (i.e. *free_surface*)

11. Create the perimeter of the analysis area.

Geometry ► Create ► Object ► Circle

- Enter center: 0 0 0
- Enter normal: Positive Z
- Enter radius: 2.0
- Delete the created surface to retain just the lines

12. Create the free surface analysis area.

Geometry ► Create ► NURBS Surface ► By contour

- Select the lines that define both, the inner and outer boundaries of the free

surface analysis area.

13. Create the free surface absorption area.

Geometry ► Create ► NURBS Surface ► By contour

- Select the lines that define both, the inner and outer boundaries of the absorption area.

14. Create a new layer named *volumes*

Utilities ► Layers

- Right click on the layers window

- Select the New layer option

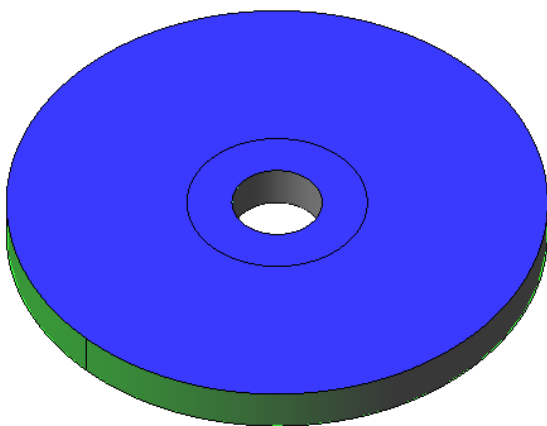
- Enter the name of the new layer (i.e. *volumes*).

15. Create a volume encompassing the entire computational domain

Geometry ► Create ► Volume ► By contour

- Select all the surfaces of the model.

The final geometry should look similar to what is shown in the next image:

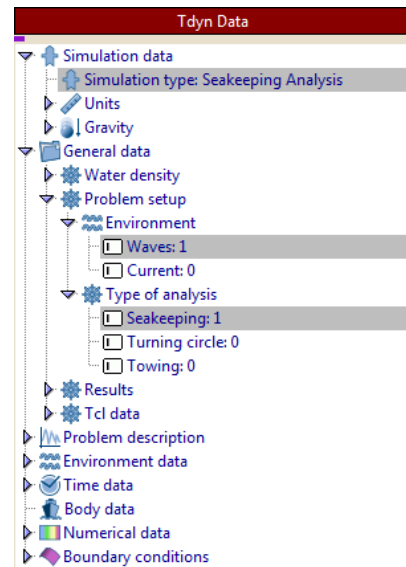


Isometric view of the entire computational domain for the bottom mounted cylinder analysis

1.2. General data

Once the geometry is ready, it is necessary to set up the problem. First of all, select to perform a seakeeping analysis as

shown in the following image caption of the SeaFEM data tree.



Environment and analysis type options highlighted within the SeaFEM data tree

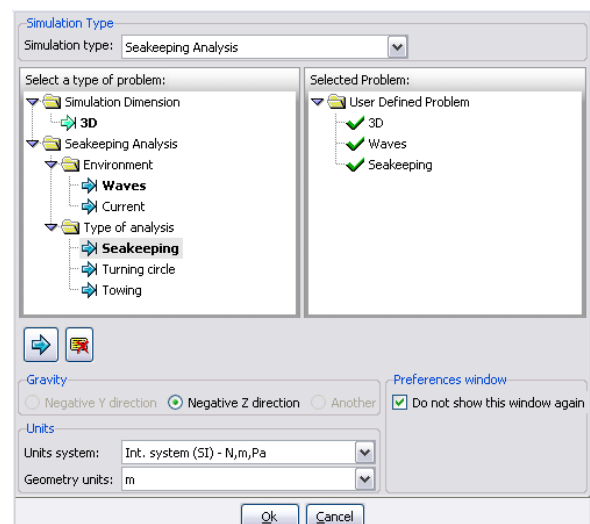
Seakeeping analyses undertaken using SeaFEM always consider that the fluid medium is sea water. Nevertheless, water density can be adjusted to match the actual fluid properties variation. For the present analysis, water density is taken as $\rho = 1025 \text{ kg/m}^3$.

General data ► Water density ► Water density: 1025 kg/m3

Note that even though the cylinder is fixed to the bottom and, therefore, no seakeeping analysis is needed to be performed in this case, the *Seakeeping* option is activated in order to allow the *Body* condition to be assigned to the geometrical entities representing the cylinder.

Also note that these general simulation parameters can also be set from the *Start Data* window which prompts when the program is started, or when clicking the following option within the menu bar.

Data ► Start data



SeaFEM start data window

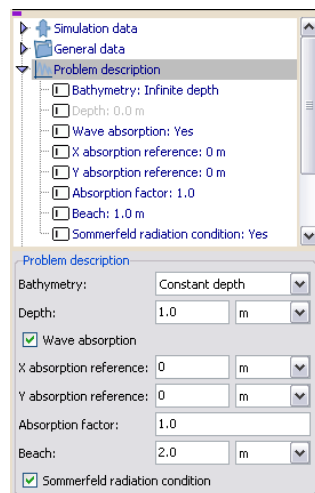
1.3. Problem description

As mentioned previously, a zone where no artificial dissipation exists must be defined for numerical convenience. Such a none-dissipative region is usually centered at the body under analysis, and its outer boundary is specified by the *beach* parameter that must be inserted in the data tree. In this particular example, the beach parameter is set up so that the none-dissipative region coincides with the region with finer mesh resolution created around the body. Nevertheless, such a coincidence is in general not mandatory.

The beach parameter and other related data must be specified as indicated in what follows:

Problem description

- *Bathymetry*: Constant depth
- *Depth*: 1.0 m
- *Wave absorption*: Yes
- *X absorption reference*: 0 m
- *Y absorption reference*: 0 m
- *Absorption factor*: 1.0
- *Beach*: 2.0 m
- *Sommerfeld radiation condition*: Yes



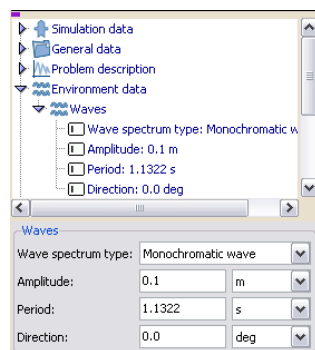
Problem description data shown within the SeaFEM data tree and on the corresponding data window.

1.4. Environment data

Next, environment data basically related to the waves spectra of the analysis must be specified. In the present case, the wave environment simply consist on a monochromatic wave whose properties must be specified as indicated here:

Environment data ► Waves

- *Wave spectrum type*: Monochromatic wave
- *Amplitude*: 0.1 m
- *Period*: 1.1322 s
- *Direction*: 0.0 deg



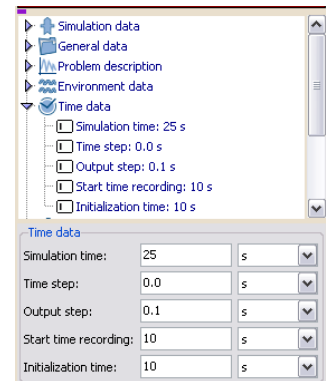
Wave spectra data shown within the SeaFEM data tree and on the corresponding data window.

1.5. Time data

Apart from wave environment, analysis data concerning the time step and total simulation time must be introduced. For the present simulation these data reads as follows:

Time data

- *Simulation time*: 25 s
- *Time step*: 0.0 s
- *Output step*: 0.1
- *Start time recording*: 10 s
- *Initialization time*: 10 s

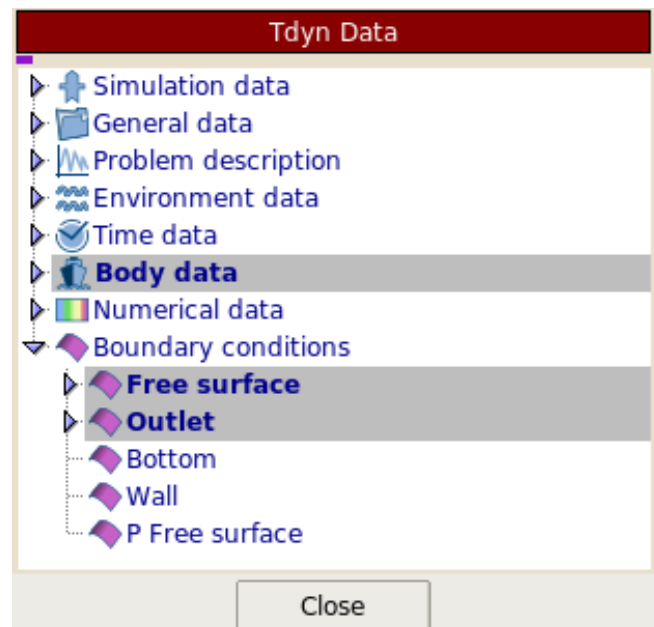


Time data section within the SeaFEM data tree and on the corresponding data introduction window.

Time step is set to 0.0, so that it will be internally evaluated by SeaFEM based on a stability criteria related to the *Free surface stability factor*.

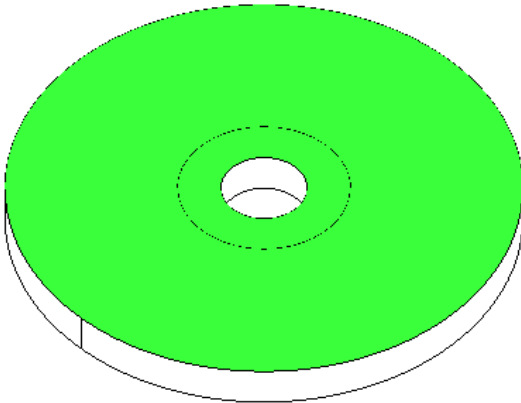
1.6. Boundary conditions

To completely define the numerical problem to be solved, some boundary conditions must be assigned to the geometry. In this section we will see a number of conditions that must be defined for every SeaFEM problem. Some other conditions exist within SeaFEM that can be used to define more specific types of problems. Such advanced conditions will be further presented in more advanced tutorials. In the present case, it will suffice to define *free surface*, *outlet* and *body* conditions as shown in the figure below.



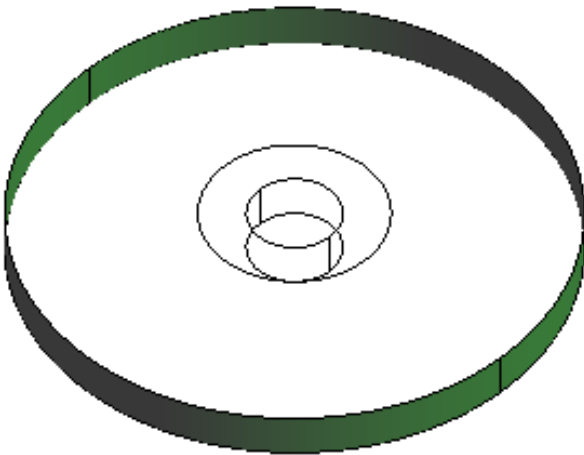
Boundary conditions used in the present tutorial are shown highlighted here within the data tree of SeaFEM.

First, a *free surface* boundary condition must be applied. As the name indicates, such a condition must be applied to all surface entities of the geometry that are located at the free surface of the model (i.e. at $z=0$).



Isometric view showing the free surface boundary condition applied to the model.

Next, an *outlet* boundary condition must be applied. As the name also indicates, the outlet condition is applied to the outmost boundary of the computational domain.



Isometric view showing the outlet boundary condition applied to the model

Finally, a *body* boundary condition must be applied to all surfaces defining the boundary of the body structure under analysis. In the present case, it simply consist on the boundary surface of the bottom mounted cylinder.

Tdyn Data

- Simulation data
- General data
- Problem description
- Environment data
- Time data
- Body data**
- Numerical data
- Boundary conditions

body prop | degrees of free | external | initial po | initial ve

Body properties

Mass: 0.0 kg

XG: 0 m

YG: 0 m

ZG: 0 m

Hydrostatics Type: Linear

Radii of gyration

R: 1.0 m

Group: [Select]

OK Cancel

Tdyn Data

- Simulation data
- General data
- Problem description
- Environment data
- Time data
- Body data**
- Numerical data
- Boundary conditions

body prop | degrees of free | external | initial po | initial ve

Degrees of freedom

☐ Surge

☐ Sway

☐ Heave

☐ Roll

☐ Pitch

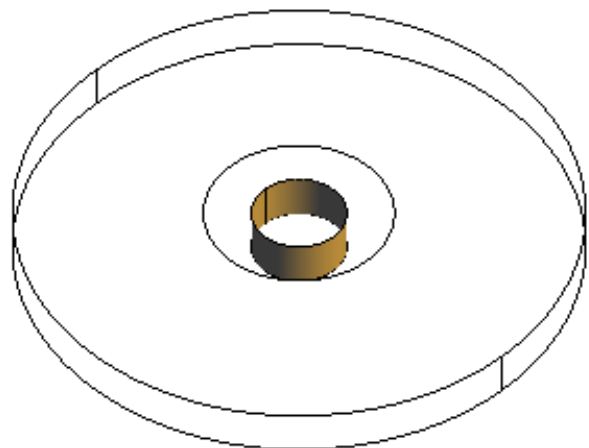
☐ Yaw

Group: [Select]

OK Cancel

Body degrees of freedom data window

Body properties data window



Isometric view showing the body boundary condition applied to the wall of the bottom mounted cylinder

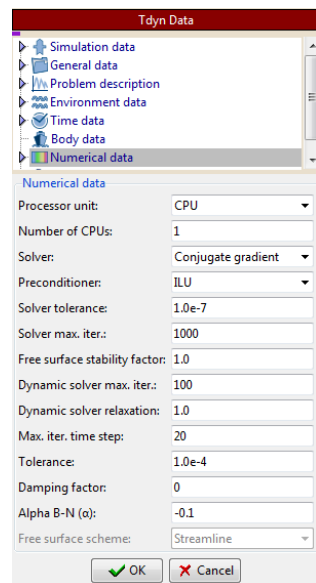
In this basic tutorial, the body structure under analysis is anchored to the bottom surface of the domain so that seakeeping analysis is not necessary. Nevertheless, body boundary condition must be applied to identify the solid structure responsible for the reflection/diffraction of the incident monochromatic wave. As shown in the figures above, body properties such as mass and center of gravity can be all set to zero since they do not play any roll in the simulation of a completely anchored body. All degrees of freedom remain also deactivated.

1.7. Numerical data

Finally, several parameters are available within the SeaFEM GUI concerning numerical aspects of the solver. For the present simulation only the following parameters are relevant:

Numerical data

- *Solver*: Stabilized bi-conjugate gradient (tolerance = $1.0E-7$)
- *Preconditioner*: ILU
- *Free surface stability factor*: 0.5



Numerical data section of the SeaFEM data tree, and the corresponding data introduction window.

1.8. Mesh

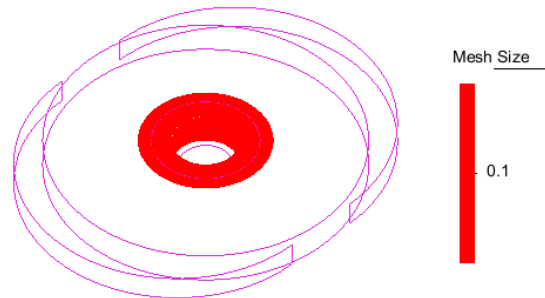
The last step before calculation is mesh generation. Mesh properties for the present analysis are summarized in what follows.

A minimum element size of 0.1 is assigned to the boundary surface of the bottom mounted cylinder and to the free surface portion contained within the analysis area. The following main menu options must be used for the assignment of unstructured mesh sizes to surfaces and volumes respectively:

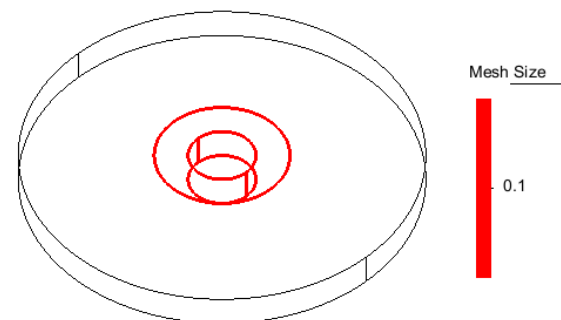
Mesh ► Unstructured ► Assign sizes on surfaces

Mesh ► Unstructured ► Assign sizes on volumes

The following pictures show the result of these mesh size assignments:



Mesh sizes assigned to the surface of the bottom mounted cylinder and to the free surface analysis area

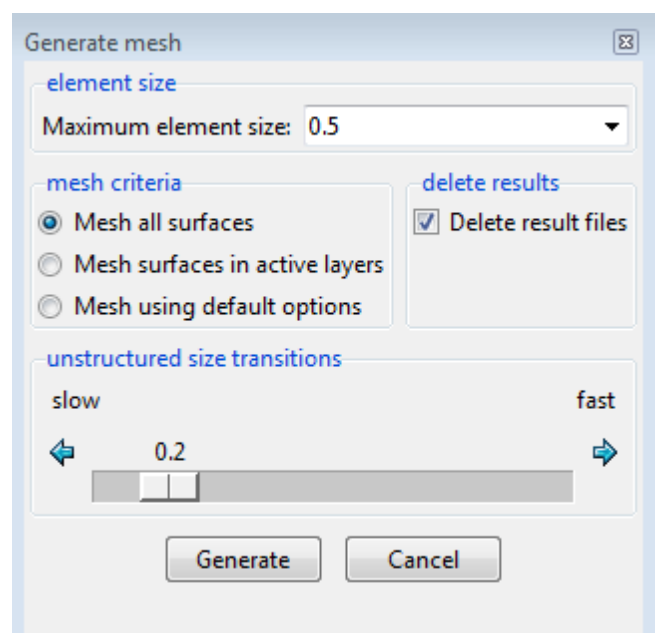


Mesh sizes assigned to the lines that defined the boundaries of the cylinder and of the free surface analysis area

Once mesh element sizes are assigned, we can proceed to mesh generation by using the following main menu option:

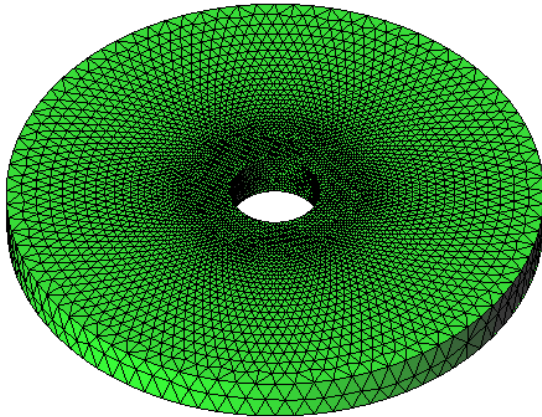
Mesh ► Generate mesh

The following window pops up, where maximum element size and mesh transition can be specified. By using a maximum element size of about 0.5 and an unstructured size transition of 0.2 a mesh is obtained consisting of 35.921 nodes and 208.107 tetrahedra elements.



Mesh generation window showing maximum element size and unstructured size transition.

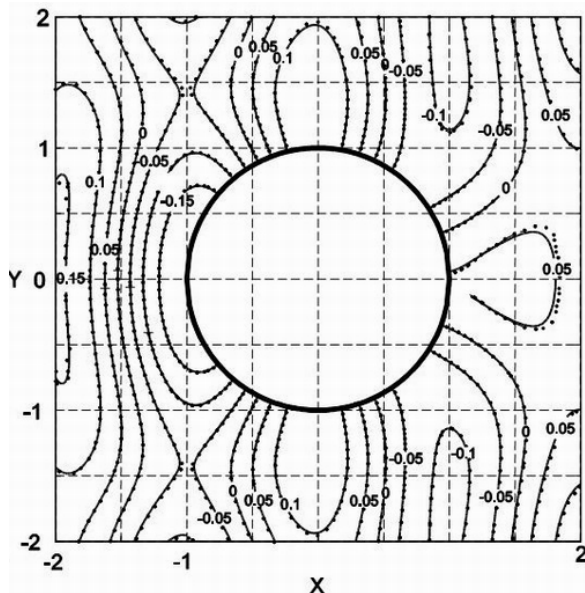
The next picture shows an isometric view of the resulting mesh. It can be appreciated how the mesh is gradually refined when approaching the surface of the bottom mounted cylinder.



Computational domain mesh used for the bottom mounted cylinder analysis.

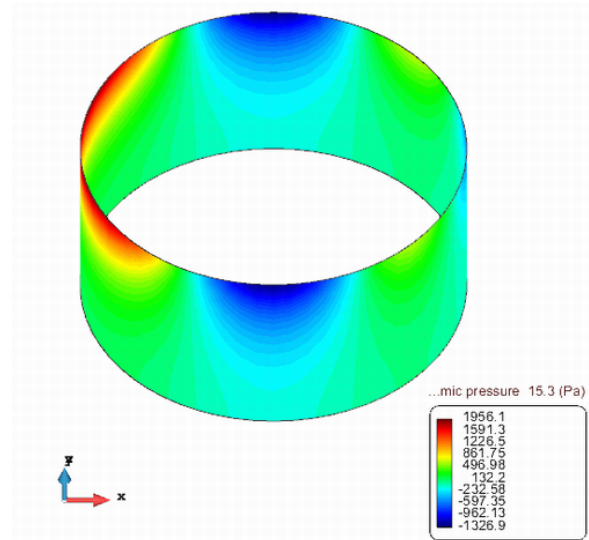
1.9. Results

Wave elevation results obtained using SeaFEM can be compared against the analytical solution. In the figure below wave elevation iscontours are shown in the free surface region close to the mounted cylinder. Solid lines correspond to the analytical solution given by the reference, while dashed lines correspond to the solution computed using SeaFEM. Note that both solutions are in very good agreement.



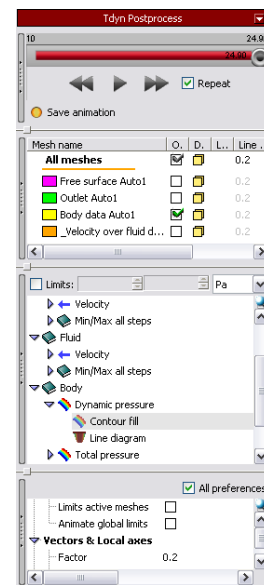
Reference versus SeaFEM solutions concerning wave elevation results.

The following figure shows the pressure distribution at the cylinder surface.



Contour fill of the dynamic pressure over the body.

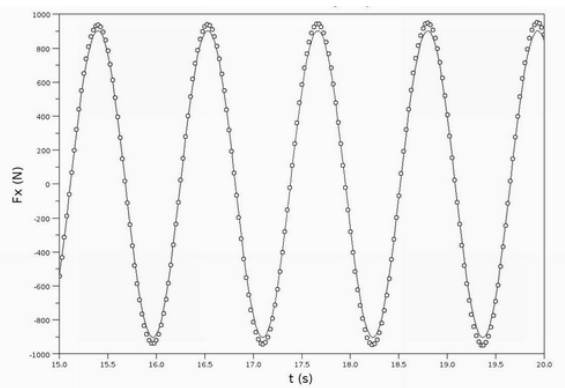
For visualizing pressure contours, just select the corresponding result in the post-process data tree:



Postprocess panel of SeaFEM

Finally, the force exerted by the incident waves over the body was recorded during a significant time period. The resulting oscillating force in the X direction (incidence direction) is compared herein with the corresponding analytical solution in the reference.

As in the case of previous verification data, the two signals are in good agreement in terms of booth, amplitude and oscillation period.

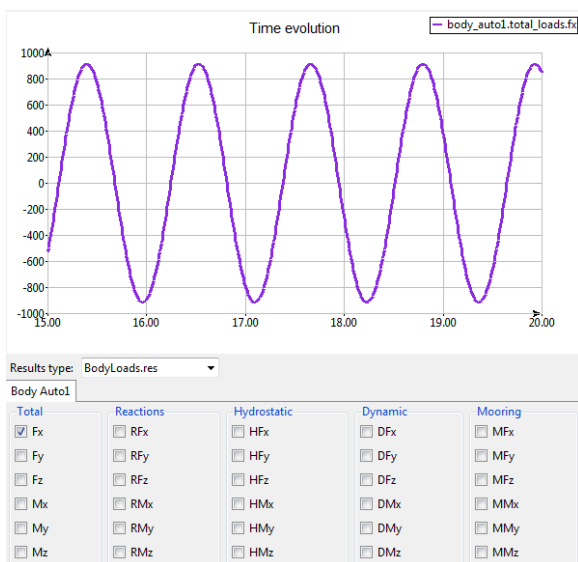


Fx force exerted by the monochromatic wave over the bottom mounted cylinder.

In order to generate plots and graphs of the computed results, you can access the SeaFEM graphs window by going to the menu option

Postprocess ► SeaFEM graphs

and selecting the desired result:

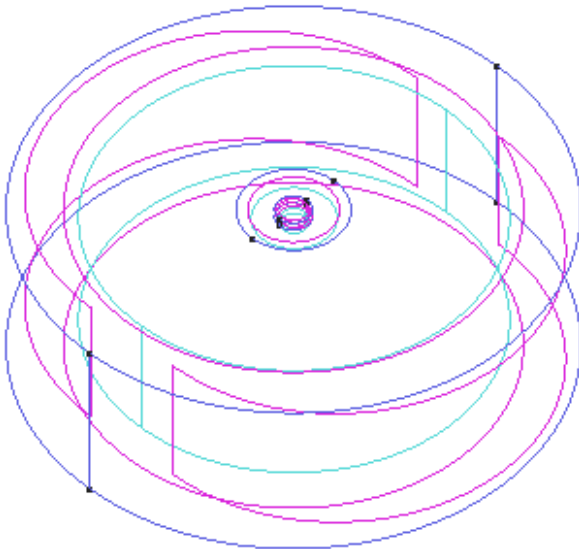


Graph utils window within the SeaFEM postprocess.

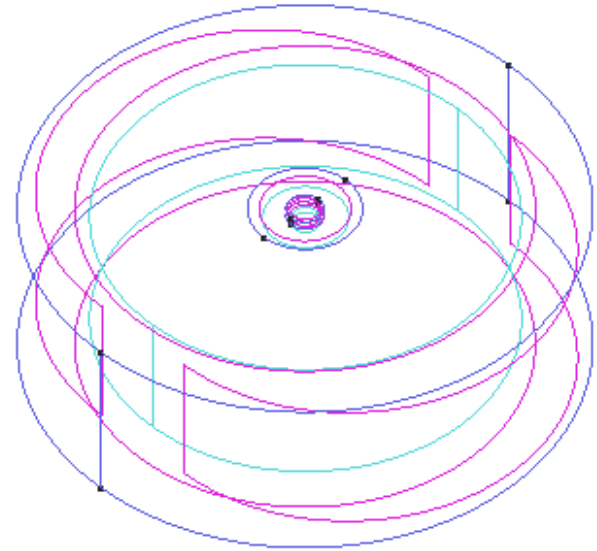
2. Floating cylinder

The present tutorial analyzes the seakeeping behaviour of a freely floating cylinder subject to the action of a monochromatic wave. In addition to the usual preprocessor steps necessary to define a SeaFEM model, in this tutorial postprocess capabilities will be described in detail.

The following image shows an isometric view of the computational domain.



Isometric view of the computational domain.



Geometry of the whole computational domain.

First, proceed to create the bottom surface of the computational domain.

1. Open the layers window and create a layer called *Bottom*:

Utilities ► Layers

- Right click on the layers window

- Select the option *New layer*

- Rename the layer to *Bottom*

2. Create a circle for the base of the computational domain:

Geometry ► Create ► Object ► Circle

- Enter center: 0.0 0.0 -10.0

- Enter normal: Positive Z

- Enter radius: 15.0

3. To facilitate the meshing process, it is sometimes a good idea to generate a more symmetric geometry. To this aim, simply divide the created circle in two segments:

Geometry ► Edit ► Divide ► Lines ► Num divisions ► 2

Next, let's create the lateral surfaces of the computational domain.

4. Create lateral domain surfaces by extrusion of the bottom

2.1. Geometry definition

In this section, we will describe in detail the geometry generation process. Both, the floating body and the computational domain analyzed in this tutorial are cylindrical and can be easily created using the CAD capabilities of the SeaFEM GUI in a similar way as that described in the previous tutorial [Tutorial 1](#).

The geometry of the present model consists on a cylindrical body of radius $R=1.0$ m and draft $D=0.5$ m embedded within a cylindrical computational domain of radius $R=15.0$ m and depth $H=10.0$ m. For the sake of convenience, an intermediate circular region at the free surface and around the floating cylinder is created so that the mesh size can be controlled more easily. We refer to this intermediate area as the analysis region since we will force this area to coincide with the region of the model where no wave dissipation is introduced.

The final geometry of the model will look similar to that shown in the following picture.

boundary surface.

Utilities ► Copy

- Entities type: surfaces

- Transformation: Translation

- First point: 0.0, 0.0, 0.0

- Second point: 0.0, 0.0, 10.0

- Duplicate entities: Off

- Do extrude: surfaces

- Select Bottom layer surfaces and proceed with the copy/extrusion process.

5. Create the boundary surfaces of the floating body. To this aim, first create the bottom surface of the floating cylinder:

Geometry ► Create ► Object ► Circle

- Enter center: 0.0 0.0 -0.5

- Enter normal: Positive Z

- Enter radius: 1.0

6. Divide the created circle in two segments:

Geometry ► Edit ► Divide ► Lines ► Num divisions ► 2

7. Extrude the bottom circular surface of the floating cylinder up to the free surface

Utilities ► Copy

- Entities type = surfaces

- Transformation = Translation

- First point = 0.0, 0.0, 0.0

- Second point = 0.0, 0.0, 0.5

- Duplicate entities = Off

- Do extrude = surfaces

- Select Bottom surface of the floating cylinder

8. Create the free surface analysis area

Geometry ► Create ► Object ► Circle

- Enter center: 0.0 0.0 0.0

- Enter normal: Positive Z

- Enter radius: 3.0

9. Divide the created circle in two segments:

Geometry ► Edit ► Divide ► Lines ► Num divisions ► 2

10. Delete the created surface and regenerate the free surface analysis area containing the hole corresponding to the waterline of the floating cylinder.

Geometry ► Delete ► Surfaces

- Select the inner free surface and proceed to delete.

Geometry ► Create ► NURBS surface ► By contour

- Select the edges of both, the perimeter of the floating cylinder and the boundary of the analysis region.

- Proceed to create the analysis area free surface.

11. Create the outer free surface.

Geometry ► Create ► NURBS surface ► By contour

- Select the edges of both, the perimeter of the analysis region and the outer perimeter of the computational domain.

- Proceed to create the outer free surface.

12. Finally, we must simply generate a volume encompassing the entire computational domain.

Geometry ► Create ► Volume ► By contour

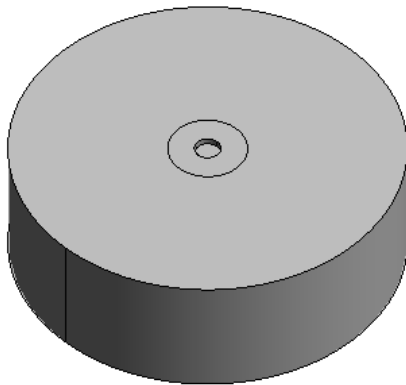
- Select all the surfaces in the model.
- Proceed to create the computational domain volume.

13. Organize geometry by layers.

Utilities ► Layers

- Right click on the layers window
- Select the option *New layer*
- Rename the layer to *Auxiliar*
- Repeat the process to create *Body*, *Free_surface*, *Outlet* and *Volumes* layers.
- Move geometrical entities to their corresponding layers to further facilitate boundary conditions assignment.

The final appearance of the generated computational domain is shown in the following figure.



2.2. General data

Once the geometry is ready, it is necessary to set up the problem. First of all, select to perform a seakeeping analysis. For this, open the Start Data window and select the following options:

Data ► Start data

- Simulation dimension = 3D

- Seakeeping analysis environment = Waves

- Seakeeping analysis type = Seakeeping

Seakeeping analyses undertaken using SeaFEM always consider that the fluid medium is sea water. Nevertheless, water density can be adjusted to match the actual fluid properties variation. For the present analysis, water density is taken as $\rho = 1025 \text{ kg/m}^3$.

General data ► Water density ► Water density: 1025 kg/m3

2.3. Problem description

A zone where no artificial dissipation is introduced must be defined for numerical convenience. Such a none-dissipative region is usually centered at the body under analysis, and its outer boundary is specified by the *beach* parameter that must be inserted in the data tree. In this particular example, the beach parameter is set up so that the none-dissipative region coincides with the region with finer mesh resolution created around the body. Nevertheless, such a coincidence is in general not mandatory.

Open the *Problem description* container in the SeaFEM data tree and fill in the corresponding data as follows:

Problem description

- *Bathymetry*: Constant depth
- *Depth*: 10.0 m
- *Wave absorption*: Yes
- *X absorption reference*: 0 m
- *Y absorption reference*: 0 m
- *Absorption factor*: 1.0
- *Beach*: 3.0 m
- *Sommerfeld radiation condition*: Yes

2.4. Environment data

Next, environment data basically related to the waves spectra of the analysis must be specified. In the present case, the wave environment simply consist on a monochromatic wave whose properties must be specified as indicated here:

Environment data ► Waves

- *Wave spectrum type*: Monochromatic wave

- *Amplitude*: 0.1 m

- *Period*: 2.0 s

- *Direction*: 0.0 deg

- *Sway*: Yes

- *Heave*: Yes

- *Roll*: Yes

- *Pitch*: Yes

- *Yaw*: Yes

2.5. Time data

Apart from wave environment, analysis data concerning the time step and total simulation time must be introduced. For the present simulation these data reads as follows:

Time data

- *Simulation time*: 25 s

- *Time step*: 0.0 s

- *Output step*: 0.25

- *Start time recording*: 10 s

- *Initialization time*: 10 s

When time step is set to 0.0, it is actually evaluated internally by SeaFEM based on a stability criteria related to the *Free surface stability factor* entered in the *Numerical data* section of the data tree.

2.6. Boundary conditions

In this case, there exist three boundary conditions to be applied. They are similar to those used in [Tutorial 1](#).

First, a Body condition must be applied to the boundary surfaces of the floating cylinder (surfaces in cyan in the picture bellow). Body properties should read as follows:

Body data ► group:name ► Body properties

- *Mass*: vol*density

- *XG*: 0.0 m

- *YG*: 0.0 m

- *ZG*: 0.0 m

- *Radii of gyration*: 1.0 m

Body data ► group:name ► Degrees of freedom

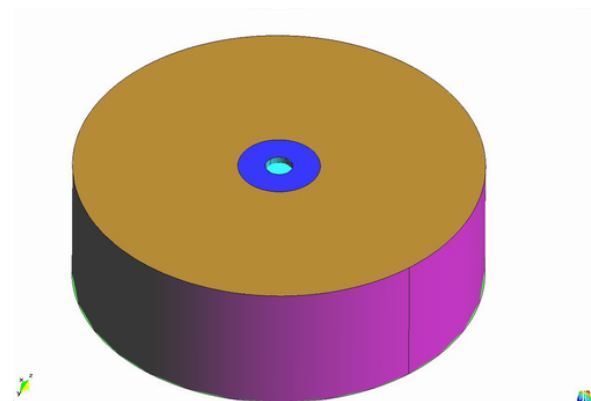
- *Surge*: Yes

Next, a Free surface condition must be applied to those surfaces located at $z=0$ (dark blue and brown surfaces in the picture bellow).

Boundary conditions ► Free surface

Finally, an Outlet boundary condition must be applied to the outer boundary of the computational domain (surface in magenta in the picture bellow). If a Sommerfeld radiation condition is to be applied, the outer surface should be of circular shape with center at the origin of coordinates.

Boundary conditions ► Outlet



Boundary conditions used in the floating cylinder analysis.

2.7. Numerical data

For the present simulation only the following numerical parameters are relevant:

Numerical data

- *Solver*: Conjugate gradient (tolerance = $1.0E-7$)

- *Preconditioner*: ILU

- *Solver tolerance*: $1.0E-7$

- *Solver max. iterations*: 1000

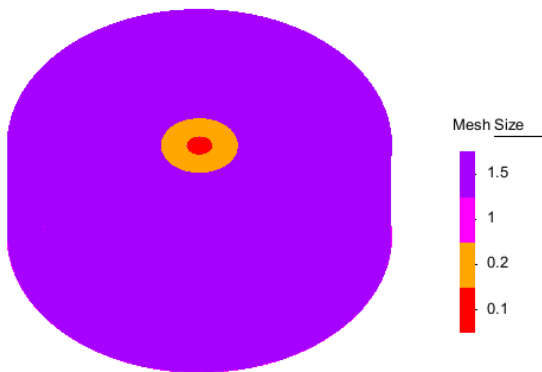
- *Free surface stability factor*: 1.0

2.8. Mesh

The last step before calculation is mesh generation. Mesh properties for the present analysis are summarized in what follows.

1. A minimum element size of 0.1 is assigned to the surfaces, lines and points of the bottom mounted cylinder.
2. An element size of 0.2 is assigned to the free surface region contained within the analysis area and to its outer perimeter.
3. An element size of 1.0 is assigned to the entire bottom surface of the computational domain.
4. Finally, an element size of 1.5 is assigned to the outer part of the free surface (out of the analysis area) and to the outlet boundary.

The following picture shows the result of these mesh size assignments:

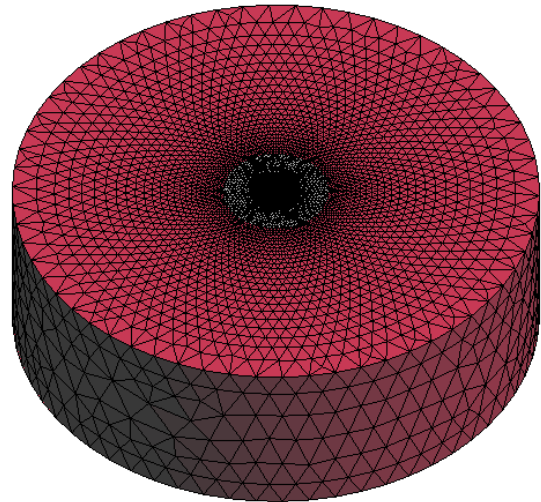


Mesh sizes assigned to the different regions of the computational domain.

Once mesh element sizes are assigned, we can proceed to mesh generation by using the following main menu option:

Mesh ► Generate mesh

The following window pops up, where maximum element size and mesh transition can be specified. By using a maximum element size of about 1.5 and an unstructured size transition of 0.1 a mesh is obtained consisting of about 30.000 nodes and 175.000 tetrahedra elements. The final resulting mesh is shown in the figure bellow.



Computational domain mesh used for the floating cylinder analysis.

2.9. Results

In this section the post-processing capabilities of SeaFEM will be described in some detail. The following topics will be addressed:

1. Visualization of wave elevation results.
2. Visualization of the pressure fields over the wetted surface of the floating structure.
3. We will also learn how to set-up an animation of the body's motion.
4. Graphics utils window will be used to readily visualize the time evolution of body movements and forces.

Once the calculation has finished, we can access the SeaFEM post-process module by using the following option in the main menu:

Postprocess ► Start

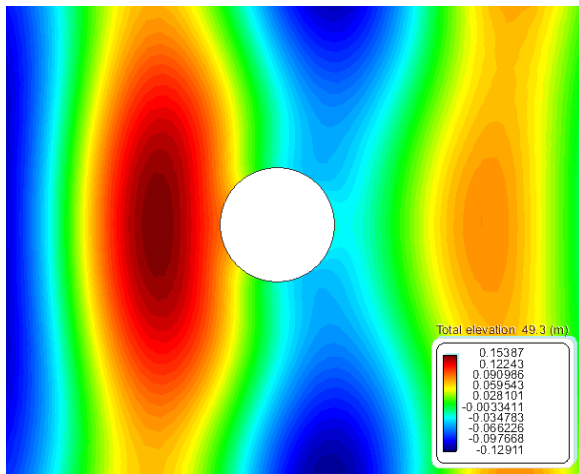
The main post-processing options of SeaFEM are readily available at one touch through the post-process panel located at the left hand side of the post-processing main window. Such a panel has 4 main areas. These are, from top to bottom:

1. The *Animations* panel mainly used to control the animation flow.
2. The *Meshes* panel that can be used to activate/deactivate various visualization options of separate meshes.
3. The *Results* panel that is used to select the results for to be visualized.
4. The *Preferences* panel used to control basic as well as advanced visualization options.

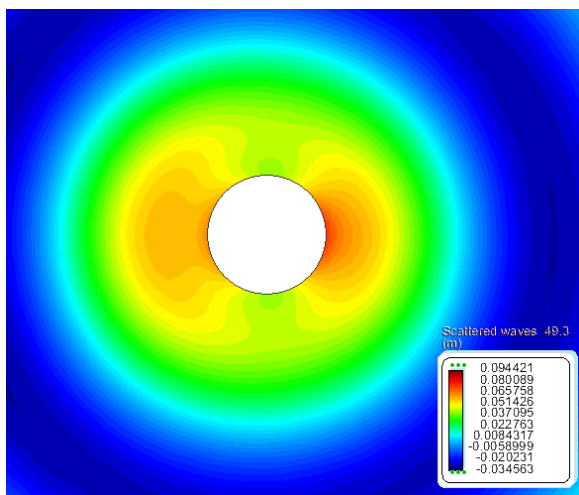
► Results visualization by Contour fill options.

In the two following figures, the contour fills corresponding to the total and scattered wave elevation results are shown respectively. The desired result is simply selected on the *Results* section of the post-process panel. The top view used in the present figures can be set by using the main menu option

View ► Rotate ► Plane XY (original)

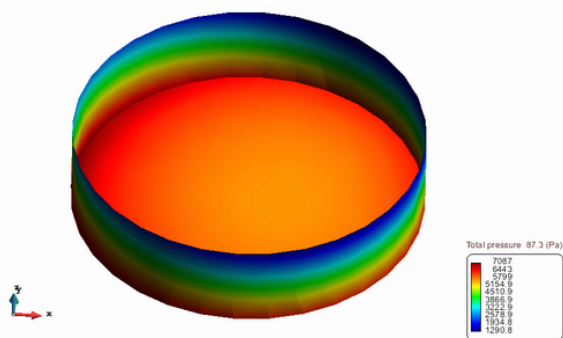


Total wave elevation (incident + scattered) results at time step $t = 49.3$ s.



Scattered wave elevation results.

Pressure field contour fills over the surface of the floating body can be visualized as well by selecting the corresponding result type within the *Results* panel.



Pressure (induced + hydrostatic) results over the body surface

• Set up of a body movements animation.

An animation of the body can be setup within the postprocessor. To this aim, the following option of the menu can be used.

Postprocess ► Computer animations

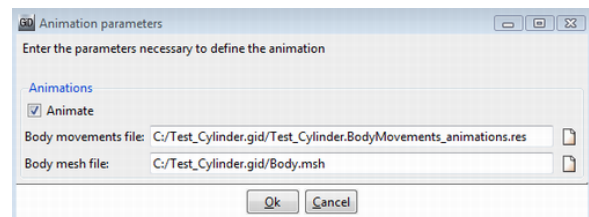
By using this option, a set of animation templates becomes available. Various default animation template files in XML

format (*.xml) can be found within the animations folder of the SeaFEM installation directory. These default templates are provided to be used to create new animations automatically. New template files can be added into the "animations" folder to perform special animations that fulfill the user particular demands. A detailed description of the XML syntax to be used in animation template files can be found in the "CompassLIB Postprocess" manual in <http://www.compassis.com/downloads/Manuals/CompassLIBPostprocessReference.pdf>.

In our particular case, the default "*Seakeeping animation*" template file can be used in order to setup an animation of the floating cylinder. By selecting the menu option

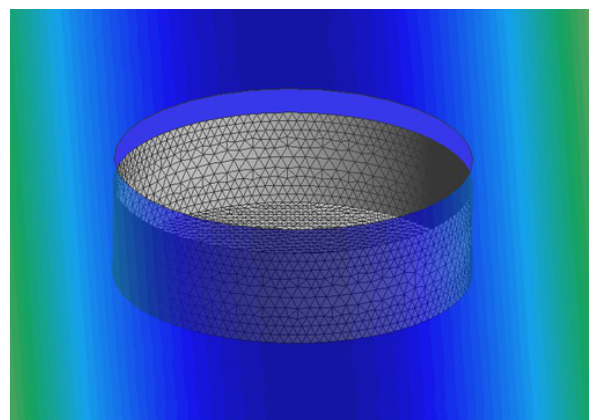
Postprocess ► Computer animations ► Seakeeping animation

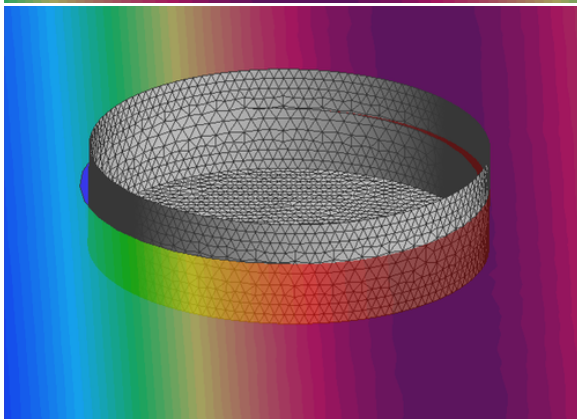
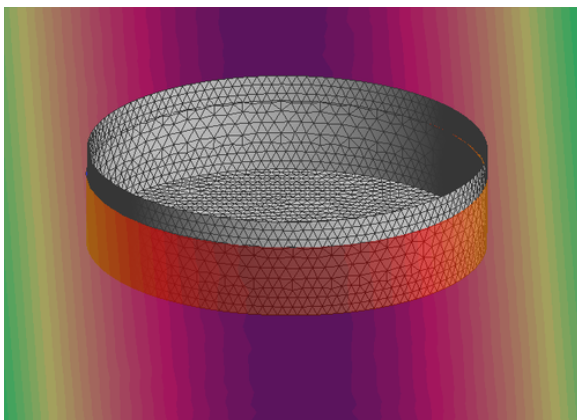
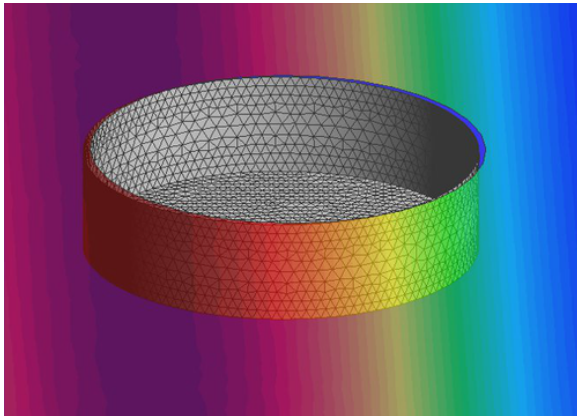
the following window will pop-up. The animation window may be filled by simply providing the *.BodyMovements_animations.res" file, and an auxiliary body mesh file (named here Body.msh) that must be previously created and exported using GiD mesh format in the SeaFEM preprocessor. Such an auxiliary mesh must exclusively contain the surface mesh of the body under consideration. It might be noted however, that such a body surface mesh may contain both the wetted and the dry parts of the floating structure.



After completion of the animations window information, the new body mesh will be loaded and added to the postprocessor meshes panel. Once this is done, an animation can be started so that the body mesh will be animated following the movement results contained in the *.BodyMovements_animations.res" file.

In the following set of figures, the animated position of the floating cylinder is shown for four different time steps. In these figures, the free surface is rendered transparent to allow visualization of the body mesh during the animation.



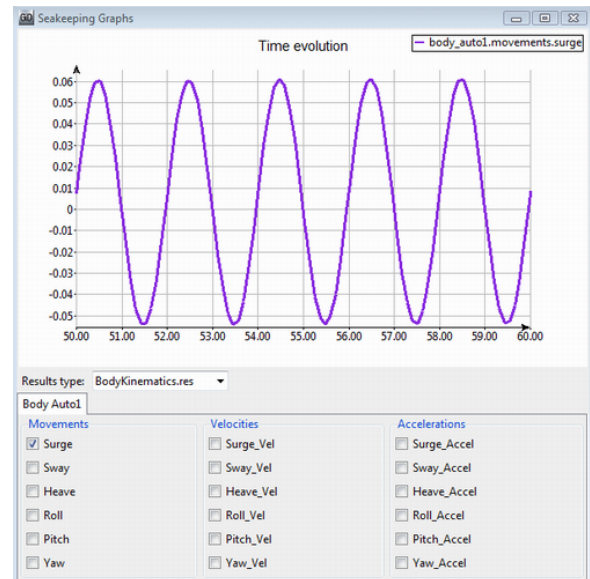


• Graph tools window

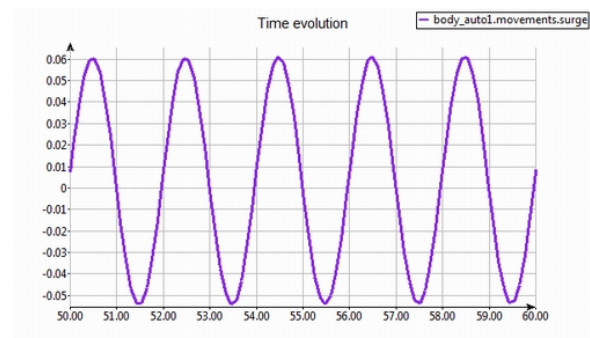
Body movements and loads can be also visualized by using the SeaFEM graphs utility. It can be accessed through the "Postprocess" menu, once the results have been loaded into the postprocessor.

Postprocess ► SeaFEM graphs

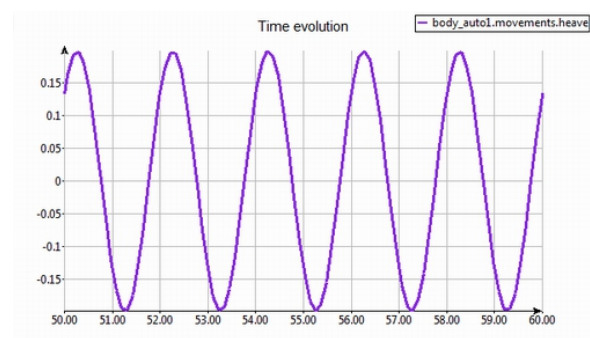
The SeaFEM graph utility looks like the figure below. Within the Results type combobox, the user can select which type of results wants to analyze (i.e. body movements, body forces, user defined results, etc.). Once the results type is selected, the available results are loaded in the bottom part of the graphs window, so that individual results can be selected separately for visualization.



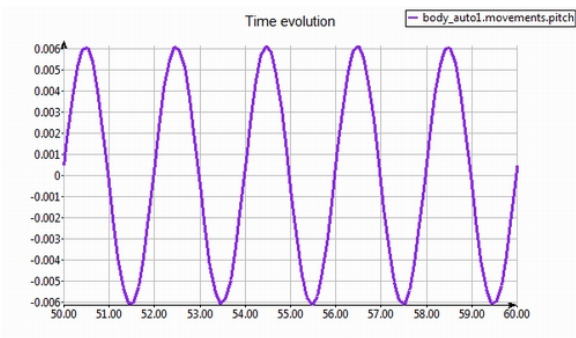
In the particular case of the present application example, Surge, Heave and Pitch body movements result in the following time evolution graphs.



Time evolution of the Surge body movement

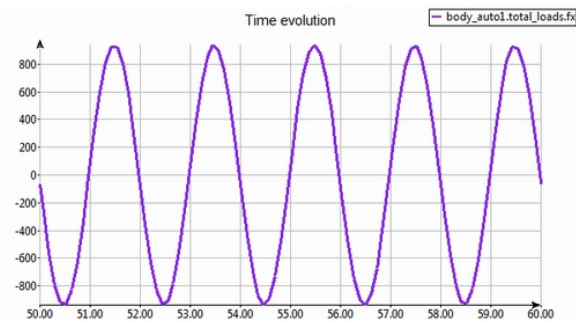


Time evolution of the Heave body movement

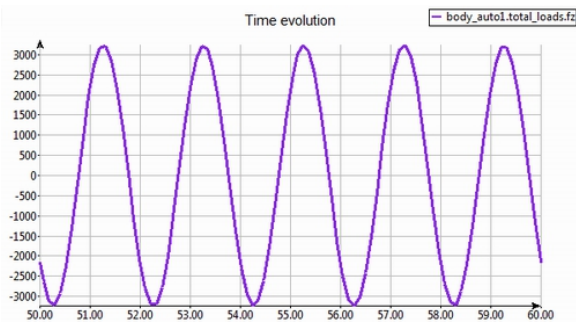


Time evolution of the Pitch body movement

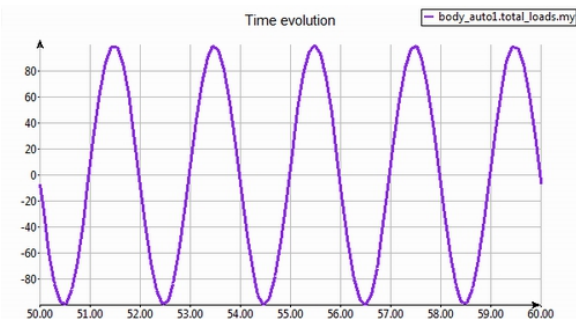
Total forces and moments acting on the body can also be visualized by using the same SeaFEM graphs utility.



Time evolution of the Fx body force



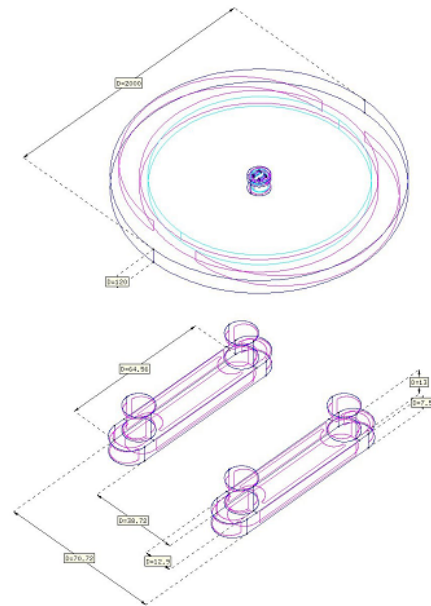
Time evolution of the Fz body force



Time evolution of the My moment

3. GVA 4000 Semisubmersible structure

In this Tutorial, a SeaFEM model is created in order to analyze and validate the RAO behaviour of a free-floating semi-submerged structure, while pulled by an external force. The following image shows the computational domain and a detail of the submerged part of the floating structure:



In order to perform the RAO analysis, a monochromatic wave spectrum is used. And the external force is represented using an elastic constrain.

The RAOs calculated in this tutorial are compared with the experimental results published in the work from "Claus G.F., Schmittner C. and Stutz K., Time-domain investigation of a semisubmersible in rogue waves. Proceedings of OMAE'02, 21st International Conference on Offshore Mechanics and Arctic Engineering, June 2002."

To automatically load the model, ready to be meshed and run, simply left-click on the button below.

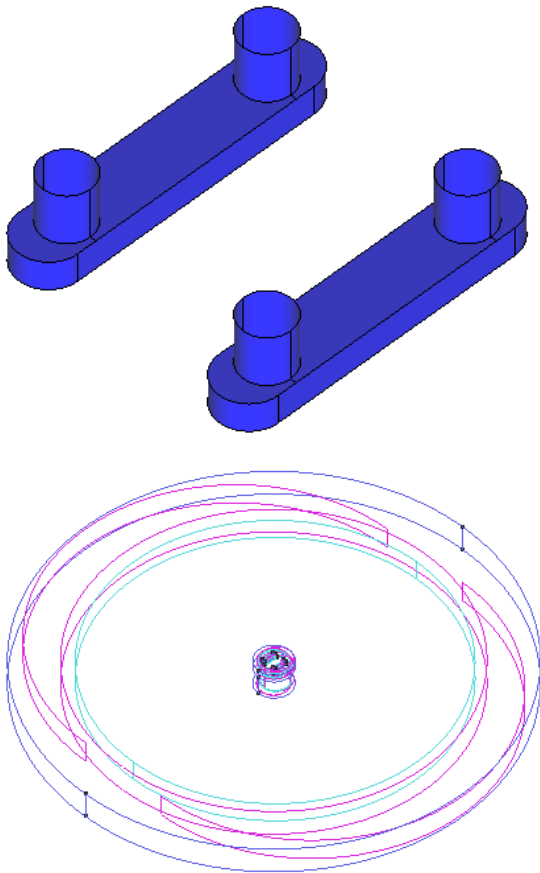
 **Load model**

On the contrary, to build the model from scratch proceed as indicated in the following sections of this tutorial.

3.1. Geometry definition and body data

In this tutorial, geometry creation is not described in detail. Although the geometry of the floating structure is simple enough as to be constructed using the CAD capabilities provided by the SeaFEM pre-processor, we will assume that the geometry was already created. Actually, the geometry of the present model can be downloaded from the Compass website.

The geometry of the submerged part of the GVA 4000 structure, and the corresponding computational domain are shown in the figure below.



(a) Submerged part of the GVA 4000 semisubmersible structure. (b) Computational domain.

The body has a characteristic length of 70 m and its height is 20 m . The centre of gravity is placed at the coordinates $(0,0,0.85)$ and the radii of inertia is $r_x=r_y=r_z=31.06\text{ m}$. The mass of the body is calculated internally to equal the mass of the water displaced by the structure. This is achieved defining the mass of the body as the product of water density and displacement volume. Such a function must be specified within the body properties. Summarizing, body properties for the submersible structure read as follows:

Body data ► group: name ► body properties

- Mass: vol*density
- XG: 0.0 m
- YG: 0.0 m
- ZG: 0.85 m
- Hydrostatics Type: Linear
- Radii of gyration: 31.06 m

Next, degrees of freedom are activated as follows:

Body data ► group: name ► degrees of freedom

- Surge: activated
- Sway: restricted
- Heave: activated
- Roll: restricted
- Pitch: activated
- Yaw: restricted

For the computational volume, the free-surface is placed as always at $z=0$. The water depth is set to $H=120\text{ m}$ and the domain has a radius of 1000 m . The beach boundary is placed 70 m away of the center of the body in order to avoid interferences with the results.

The *Body data* section of the SeaFEM data tree also provides the option to apply external forces and moments to a given body. In the present tutorial, we use this option to apply an elastic constraint to the floating structure so that it does not move away from its initial position. Specifically, an external force is applied to the body in a way that the magnitude of the force depends on the displacements of certain given points.

Body data ► group: name ► External loads ► External force X:

$$-0.5\text{E}5*\text{dx}[-40.28,27.36,-17.55]-0.5\text{E}5*\text{dx}[-40.28,-27.36,-17.55]\text{ N}$$

Body data ► group: name ► External loads ► External moment Y:

$$0.5\text{E}5*(\text{ZG}+16.7)*\text{dx}[-40.28,27.36,-17.55]+0.5\text{E}5*(\text{ZG}+16.7)*\text{dx}[-40.28,-27.36,-17.55]\text{ N}\cdot\text{m}$$

As can be appreciated, the force is applied in the X direction (movements in the Y direction are not allowed since sway degree of freedom was restricted), and made dependent on the displacement of the points located at coordinates $(-40.28, 27.36, -17.55)$ and $(-40.28, -27.36, -17.55)$ respectively. An elastic constraint of 0.5×10^5 is used. The sum of these two forces is equivalent to the action of a spring with an elastic modulus of 10^5 N/m . The given points are the outer edges of the submerged structure. For consistency, a moment along the Y direction is also added in order to counteract the moment produced by the elastic force (and to ensure that the applied forces do not generate vertical displacements).

3.2. General data

In the present tutorial, we are interested on the RAO behavior of a real structure. Hence, the seakeeping analysis of the structure subjected to the action of a white noise spectrum and without currents must be undertaken. To this aim, the general data of the problem must read as follows. Parameters not mentioned here may keep its default value.

General data ► Water density

- *Water density*: 1025 kg/m³

General data ► Problem setup ► Environment

- *Waves*: 1
- *Current*: 0

General data ► Problem setup ► Type of analysis

- *Seakeeping*: 1
- *Turning circle*: 0
- *Towing*: 0

General data ► Results ► Kinematics

- *Movements*: Yes
- *Velocities*: No
- *Accelerations*: No
- *Raos*: Yes

3.3. Problem description

As usual, a zone where no artificial dissipation is introduced must be created. The outer boundary of this region of no-dissipation is specified by the *beach* parameter that must be inserted in the data tree. Away from the beach wave absorption is induced numerically in order to stabilize the numerical results.

Hence, Beach parameter and other related data must be

inserted as indicated here:

Problem description

- *Bathymetry*: Constant depth
- *Depth*: 120.0 m
- *Wave absorption*: Yes
- *X absorption reference*: 0 m
- *Y absorption reference*: 0 m
- *Absorption factor*: 1.0
- *Beach*: 70.0 m
- *Sommerfeld radiation condition*: Yes

3.4. Environment data

Next, environment data to define the wave spectra of the analysis must be specified. Since in this tutorial we are interested in the RAOs results, a white noise spectrum must be used. The corresponding data is summarized below:

Environment data ► Waves

- *Wave spectrum type*: White noise
- *Amplitude*: 1.0 m
- *Direction*: 0.0 deg
- *Shortest period*: 8 s
- *Longest period*: 28 s
- *Number of wave periods*: 11

3.5. Time data

Apart from wave environment, analysis data concerning the time step and total simulation time must be introduced. For the present simulation these data reads as follows:

Time data

- *Simulation time*: 600 s

- Time step: 0.3 s
- Output step: 5 s
- Start time recording: 300 s
- Initialization time: 200 s

Since the computational method used within SeaFEM solves the problem in the time domain, initial conditions are important. Hence, in order to attain faster the harmonic steady state, it can be useful to introduce the incident potential smoothly. This is done by means of a cosine function that modulates the nominal incident potential during the initialization time. Moreover, the body is supposed to be initially at the equilibrium position with zero velocity. During the initialization period a dissipation term proportional to the body velocity is introduced. This dissipation term is smoothly decreased down to zero so that it disappears after the initialization period.

Keep in mind, that *Time data* introduced by the user in problems concerning RAOs analysis are just for the sake of guidance. Actually, SeaFEM internally evaluates optimal values for both, the *Simulation time* and the *time step* based on Fourier analysis requirements. Hence, at the beginning of the calculation, a message similar to that shown below will be prompted into the information file.

--> Time step has been switched to $dt = 0.21875$ s to fulfill Fourier analysis requirement

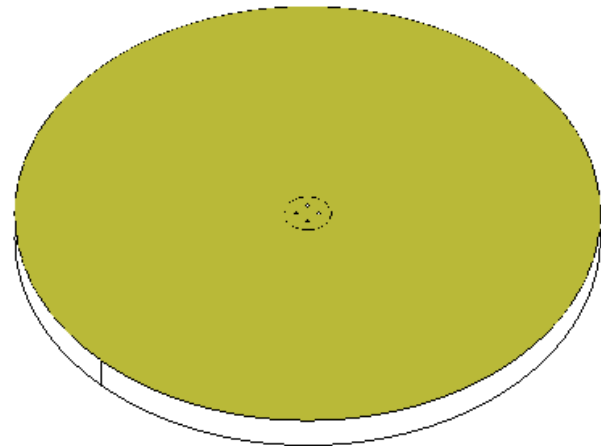
--> Simulation time has been modified to 412 s to fulfill Fourier analysis requirement

3.6. Boundary conditions

Apart from *Body data* condition which is assigned to the entire boundary surface of the floating structure as shown in section [Geometry definition and body data](#), three boundary conditions must be specified to set up a well-defined problem for RAOs analysis of the GVA 4000 structure. These are the *Free surface*, the *Outlet* and the *Bottom* boundary conditions. All of them are shown in the given order in the following images:

Boundary conditions ► Free surface

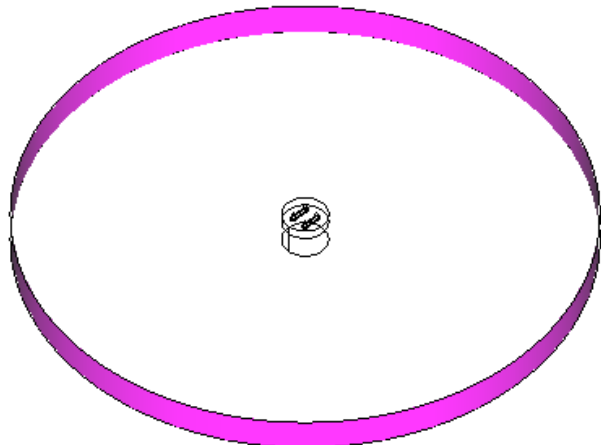
The Free surface boundary condition must be applied to the entire surface of the computational domain located at the plane $z=0$.



Free surface boundary condition

Boundary conditions ► Outlet

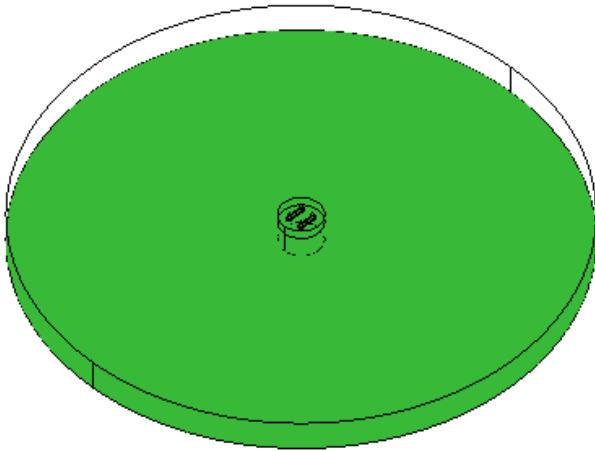
As usual, the Outlet condition is applied to the outer boundary of the computational domain.



Outlet boundary condition

Boundary conditions ► Bottom

Finally the Bottom condition, as the name indicates, is applied to the bottom surface of the computational domain.



Bottom boundary condition

3.7. Numerical data

For the present simulation only the following numerical parameters are relevant. Parameters not referred here may keep their default values.

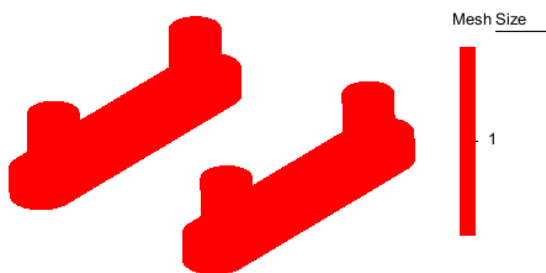
Numerical data

- Solver: Deflated conjugate gradient
- Preconditioner: ILU
- Solver tolerance: 1.0E-7
- Solver max. iterations: 1000
- Free surface stability factor: 1.0

3.8. Mesh

The last step before calculation is mesh generation. Mesh properties for the present analysis are summarized in what follows.

1. A minimum element size of 1 m. is assigned to the surfaces, lines and points of the semisubmersible structure.



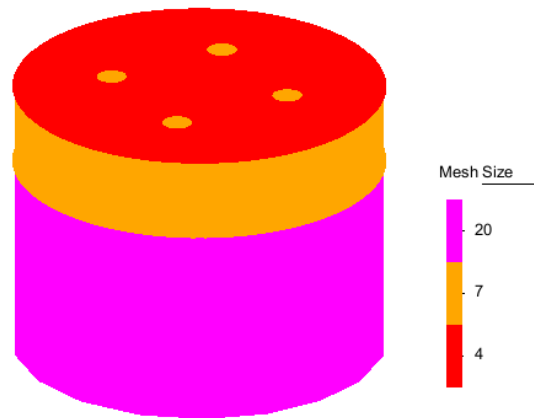
Elements size assigned to the body's surface

2. An element size of 4 m. is assigned to the free surface region

close to the floating structure.

3. An element size of 7 m. is assigned to the surfaces of the intermediate region that encloses the floating structure.

4. A larger element size about 20 m. is assigned to the surfaces of the cylindrical region located below the floating structure up to the bottom of the computational domain.



Element sizes assigned to the region close to the body in order to achieve better control on mesh size transition.

No other mesh parameters are assigned to the remaining part of the computational domain since the largest element size that will be allowed during the meshing process is also 20 m.

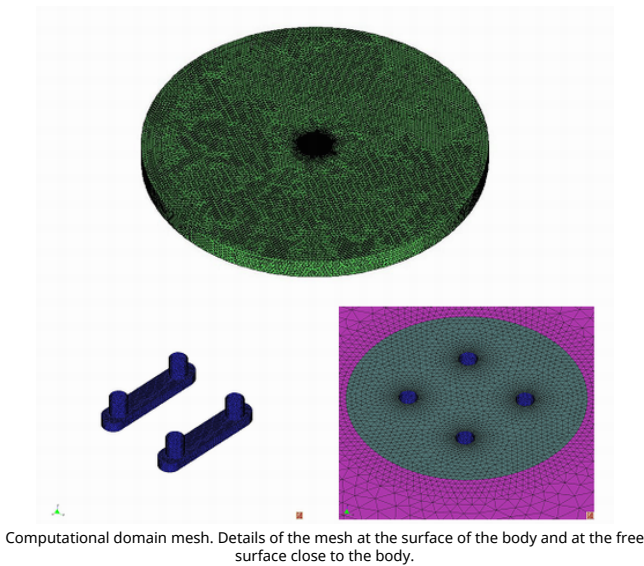
Once mesh element sizes are correctly assigned, we can proceed to mesh generation by using the following main menu option with the corresponding maximum size and mesh transition:

Mesh ► Generate mesh

Maximum element size: 20

Unstructured size transition: 0.3

The resulting mesh contains about 152,000 nodes and 870,000 tetrahedra elements. The final resulting mesh is shown in the figure below.



3.9. Results

The objective of this computation using white noise spectrum, is to be able to obtain the corresponding RAOs (Response Amplitude Operators).

For doing so, the calculation is launched using the *Start Calculation* icon, or the main menu option:

Calculate ► Calculate

During the simulation, it is possible to obtain information of the process evolution either by clicking in the *Calculation Information* icon, or by accessing the main menu option:

Calculate ► View process info

Once the calculation has finished, we can access the SeaFEM post-process module by using the following option in the main menu:

Postprocess ► Start

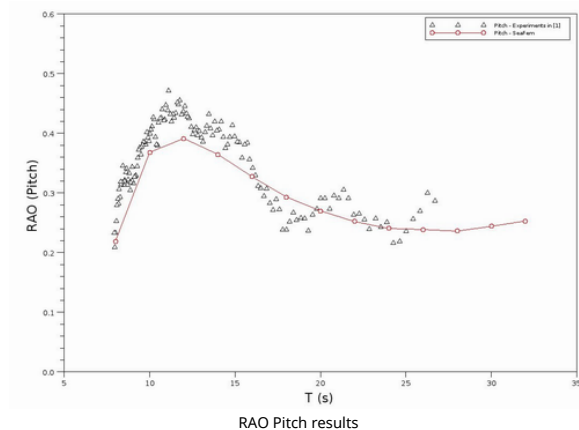
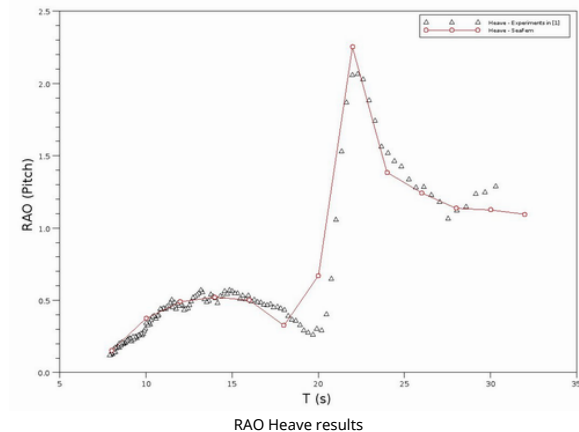
Once the post-process module is loaded, it is possible to visualize as usual the various field results (i.e. velocity, pressure over body, wave elevation ...) by selecting the corresponding results on the left hand side panel. Nevertheless, for the present case RAOs results constitute the most significative output of the calculation. RAOs may be plotted by using the SeaFEM graphs util.

Postprocess ► SeaFEM graphs

Furthermore, you can also access RAOs results in text file format by opening the following file in the current project directory.

\$(projectname)\\$(projectname).Raos.res

The following images show the results obtained, compared to the ones obtained from the literature.



4. TLP Tutorial

The main purpose of this tutorial is to test a real TLP platform corresponding to a 5 MW wind turbine using SeaFEM. Actually, the characteristics of the TLP used herein correspond to that used by the NREL (National Renewable Energy Laboratory), and the final goal is to compare the results obtained using SeaFEM against those reported by NREL. For more details, see reference [1].

The actual TLP structure under analysis has three major components: the wind turbine, the floating platform and the mooring system. In the present analysis, only the floating platform is physically modeled, while the mooring system and wind turbine effects are taken into account by using appropriate mass, damping and stiffness matrices. The wave environment used within the present analysis consists on a white noise spectrum. It allows to obtain response amplitude operator (RAO) results at different frequencies. The range under consideration is between 2,09 rad/s and 0,31 rad/s. Hence, the shortest period is about 3 s. and the largest one is about 20 s.

To automatically load the model, ready to be meshed and run, simply left-click on the button below.

 [Load model](#)

On the contrary, to build the model from scratch proceed as indicated in the following sections of this tutorial.

4.1. Geometry definition

This section is devoted to the description of the geometry generation process. SeaFEM has plenty of tools for geometry

generation. Hence, the way to construct the geometry that is shown in this tutorial may constitute just one of many possibilities. For a more detailed description of geometry generation tools and other pre-processing capabilities of SeaFEM, we strongly recommend the user to take a look at the GiD pre-processing manual (see <http://www.gidhome.com/support>).

For the present case, both the body and the computational domain are cylindrical, and therefore they can be easily created by means of the geometry generation tools available in the GUI of SeaFEM. Before starting to construct the geometry, it is important to remark that the entire computational domain in SeaFEM must be located below the undisturbed water surface (i.e $z \leq 0$). In the present case, the domain will be also centered at the origin (0,0,0).

In order to construct the geometry for the present analysis, simply follow the steps described below.

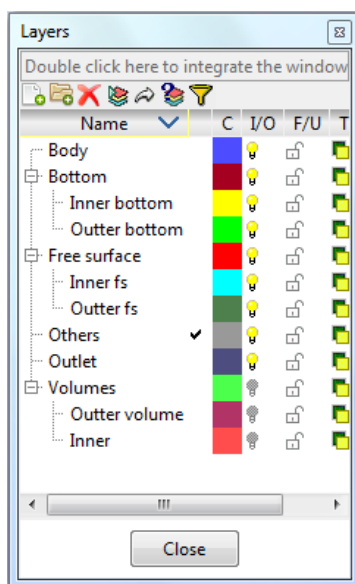
1. Open the layers window and create a layer called *Body*:

Utilities ► Layers

- Right click on the layers window
- Select the option New layer
- Rename the layer to *Body*

2. Repeat step 1 up to five times to created new layers named *Bottom*, *Free surface*, *Others*, *Outlet*, *Volumes*.

3. Create child layers as to mimic the layers structure shown in the picture.



Layers structure

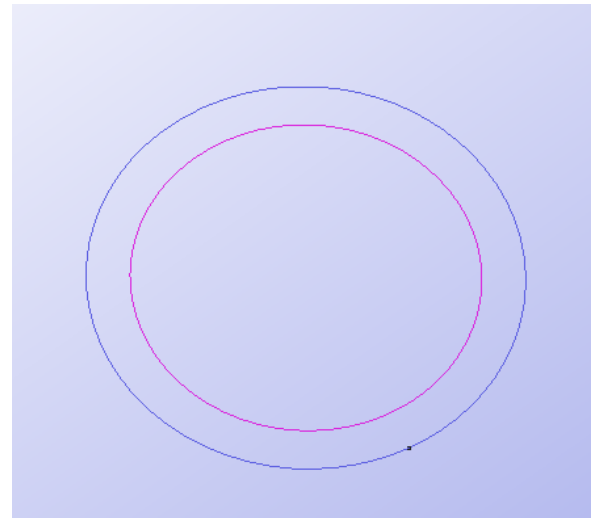
4. Start to create the geometry of the floating platform.

Geometry ► Create ► Object ► Circle

- Enter center: 0.0 0.0 0.0

- Enter normal: Positive Z

- Enter radius: 9.0

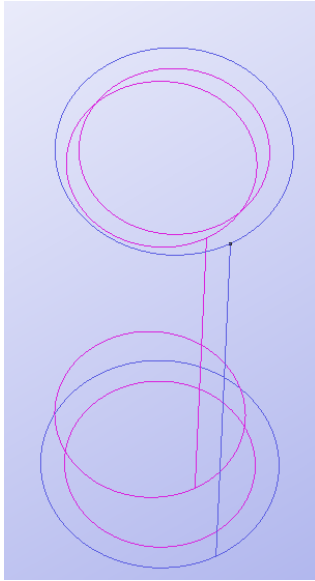


Floating platform circle created.

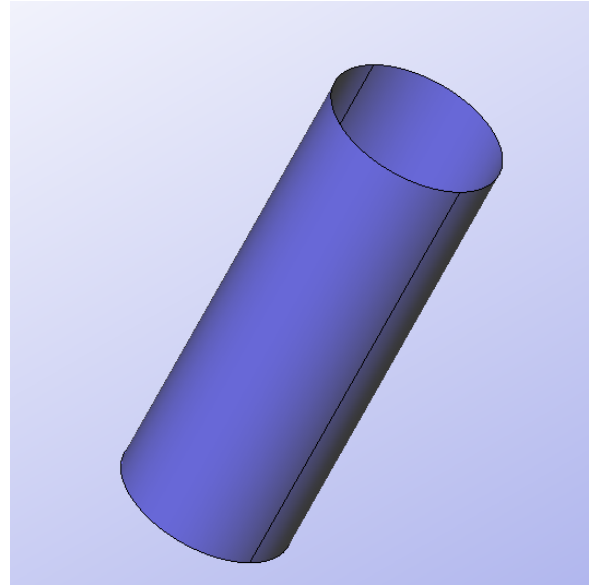
5. Create the cylindrical surface of the floating platform by copying and extruding the base circle.

Utilities ► Copy

- Entity type: surfaces
- Transformation: translation
- First point: 0.0 0.0 0.0
- Second point: 0.0 0.0 -47.89
- Do extrude: Surfaces
- Select the circle surface and proceed.



Floating platform cylinder.



Cylinder resembling the floating platform under analysis.

6. Clean and organize the geometry of the floating structure.

Geometry ► Delete ► Surfaces

- Select the top surface of the cylinder and proceed to delete.

- Activate the layer named *Body*

- Send all surfaces of the body to the corresponding layer

Send to ► Surfaces

- Select the surfaces of the cylinder and proceed.

Next, simply proceed to generate the computational domain. It represents the significative region of the sea surrounding the body structure under analysis (i.e. the TLP structure). Such a region is actually composed by two distinct areas; the first one is the area close to the model and is named analysis area while the second one is the region located far from the structure which is called waves absorption area or beach area. To construct the computational domain, follow the steps described below:

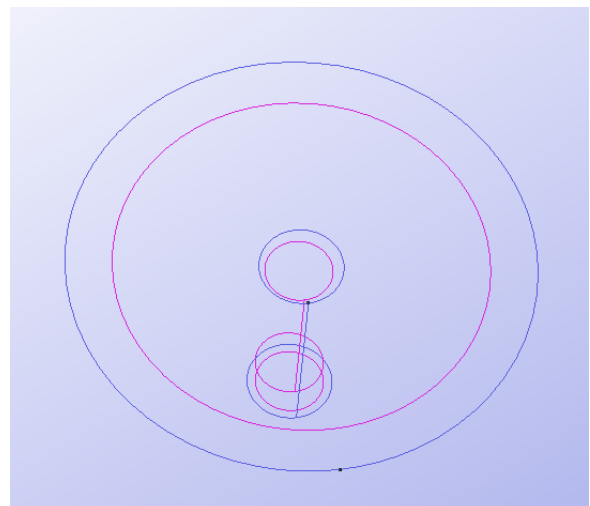
7. Create a circle delimiting the analysis area.

Geometry ► Create ► Object ► Circle

- Enter center: 0.0 0.0 0.0

- Enter normal: Positive Z

- Enter radius: 50 m



Analysis zone circle created.

8. Clean the geometry.

Geometry ► Delete ► Surfaces

- Select the surface of the recently created analysis area.

9. Regenerate the free surface region of the analysis area encompassing the hole corresponding to the floating platform's undisturbed waterline.

Geometry ► Create ► NURBS surface ► By countour

- Select the top circles of both, the platform and the analisis region and proceed with the NURBS generation process.

10. Create a circle delimiting the absorption area.

Geometry ► Create ► Object ► Circle

- Enter center: 0.0 0.0 0.0

- Enter normal: Positive Z

- Enter radius: 600 m

11. Create the outter boundary of the computational domain.

Utilities ► Copy

- Entity type: surfaces

- Transformation: translation

- First point: 0.0 0.0 0.0

- Second point: 0.0 0.0 -200.0

- Do extrude: Surfaces

- Select the outter circle of the computational domain and proceed.

12. Clean the geometry.

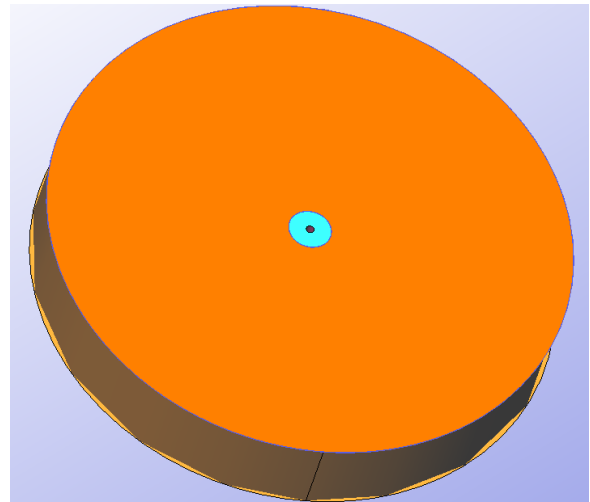
Geometry ► Delete ► Surfaces

- Select the top surface of the computational domain and proceed.

13. Regenerate the free surface of the absorption area encompassing the hole corresponding to the analysis area.

Geometry ► Create ► NURBS surface ► By contour

- Select the top circles of the analysis zone and the waves absortion area.



Surface created between two circles, the analysis zone and the wave absortion area.

14. Organize geometry by layers.

- For each surface, activate the corresponding destination layer

Send to ► Surfaces

- Select the corresponding surface and proceed to send it to the corresponding layer (deactivate the option *Also lower entities* if you want to move the surface but not its associated lines and points).

15. Next, we must create the volume necessary to complete the control volume of the simulation.

Geometry ► Create ► Volume ► By countour

- Select the all surfaces of the model and proceed to create the control volume.

16. Send the volume to the corresponding layer.

- Select the "Inner volume" layer.

- Deselect the option "*Also lower entities*".

- Use the option

Send to ► Volumes

- Select the inner volume and proceed.

- Repeat the process with the Outer volume.

4.2. Body data

Once the geometry has been created, a *Body data* condition can be applied to the floating body structure to define its properties. To this aim, select the option

Body data

in the data tree and assign the corresponding condition to the surfaces that define the floating TLP platform.

Next, fill in the Body data of the recently assigned body condition as indicated below.

Body data ► group: name ► body properties

- *Mass*: 0.0

- *XG*: 0.0 m

- *YG*: 0.0 m

- *ZG*: 0.0 m

- *Hidrostatics Type*: Linear

- *Radii of gyration R11*: 8.3 m

- *Radii of gyration R22*: 8.3 m

- *Radii of gyration R33*: 7.2 m

The mass and gravity center values used in this case need a more detailed explanation. As it is seen, all these values are set to zero in the SeaFEM data tree. This is so, because the true values of these parameters are actually specified using the Tcl extension of SeaFEM. Such a procedure may seem complicated at first sight, but it is done in this way to facilitate comparison against NREL results. This procedure has also other advantages for this particular case as it is explained next.

The first thing that must be noted here is that, by default all properties and results in SeaFEM are referred to the gravity center of the body under analysis. Nevertheless, this is not

always the most convenient since many properties of the structure may be referred to a different point. This is the present case for instance, where all properties are referred in NREL to the origin of coordinates (0.0, 0.0, 0.0). Hence, SeaFEM provides the opportunity to define the properties of the structure (mass, stiffness and damping) using a generic reference point. To this aim, some functions are provided through the Tcl extension of SeaFEM that permit to introduce mass, stiffness and damping matrices of various structural elements (as explained in detail in the user manual [2]) being those actually modelled or not in SeaFEM. In the present case, in addition to the TLP platform mass matrix, wind turbine and mooring system contributions will be added using the Tcl extension mentioned above.

To this aim, write down the following TdynTcl code in a text file, and provide such a file as a simulation parameter.

General data ► Tcl data ► Use Tcl external script: Yes

```
proc TdynTcl_InitiateProblem { } {
TdynTcl_Add_Mass_Matrix 1 [list 0.0 0.0 0.0] [list \
8.6E+06 0 0 0 -349.16E+06 0 \
0 8.6E+06 0 349.16E+06 0 0 \
0 0 8.6E+06 0 0 0 \
0 349.16E+06 0 14754.48E+06 0 0 \
-349.16E+06 0 0 0 14754.48E+06 0 \
0 0 0 0 361.4E+06 ]
TdynTcl_Add_Mass_Matrix 1 [list 0.0 0.0 0.0] [list \
0.7E+06 0 0 0 44.3E+06 0 \
0 0.7E+06 0 -44.3E+06 0 6.6E+06 \
0 0 0.7E+06 0 -6.6E+06 0 \
0 -44.3E+06 0 3499E+06 0 0 \
44.3E+06 0 -6.60E+06 0 3560E+06 0 \
0 6.60E+06 0 -513.3E+06 0 101.2E+06 ]
TdynTcl_Add_Damping_Matrix 1 [list 0.0 0.0 0.0] [list \
0.04E+06 0 -0.01E+06 -0.25E+06 4.00E+06 0.08E+06 \
0 0 0 -0.11E+06 -0.18E+06 -0.05E+06 \
-0.01E+06 0 0 -0.04E+06 -0.92E+06 -0.33E+06 \
0.27E+06 -0.1E+06 0 16.17E+06 50.30E+06 13.88E+06 \
3.42E+06 0.06E+06 -1.00E+06 -23.92E+06 400.1E+06 59.01E+06 \
0.05E+06 -0.02E+06 0.22E+06 11.08E+06 -52.60E+06 101.2E+06]
TdynTcl_Add_Stiffness_Matrix 1 [list 0.0 0.0 0.0] [list \
0 0 0 0.3E+06 0.2E+06 0 \
0 0 0 -0.1E+06 0.3E+06 -0.07E+06 \
0 0 0 -0.3E+06 -0.4E+06 0 \
0 0 0 8.5E+06 -22.4E+06 59.7E+06 \
0 0 0 26.8E+06 28.9E+06 -4.1E+06 \
0 0 0 -1.2E+06 1.1E+06 -4.8E+06 ]
TdynTcl_Add_Stiffness_Matrix 1 [list 0.0 0.0 0.0] [list \
```

```
0.247E+06 0 1.53E+06 0 -7.26E+06 0 \
0 0.214E+06 0 6.27E+06 0 -0.511E+06 \
1.53E+06 0 70.4E+06 0 -44.6E+06 0 \
0 6.23E+06 0 15400E+06 0 -408E+06 \
-6.94E+06 0 -31.8E+06 0 15400E+06 0 \
0 0 0 -345E+06 0 94.7E+06 ]
```

```
TdynTcl_Message "TdynTcl_DefineBodyData finished!!!" notice
}
```

In the code above, the following matrices are introduced in the given order:

- Mass matrix of the TLP platform.
- Mass matrix of the wind turbine.
- Damping matrix due to the wind turbine.
- Stiffness matrix of the wind turbine.
- Stiffness matrix due to the mooring system.

Note that all these matrices are reference to the origin of coordinates, and that output results will be referred to the (XG, YG, ZG) point specified in the SeaFEM data tree (being in this case also the origin of coordinates).

Finally, degrees of freedom of the floating structure must be activated as follows:

Body data ► group: name ► degrees of freedom

- Surge: activated
- Sway: restricted
- Heave: activated
- Roll: restricted
- Pitch: activated
- Yaw: restricted

The *Body data* section of the SeaFEM data tree also provides the option to apply external forces and moments to a given body. Use the following function for the Z-component of the external force, which is the force applied to the legs of the TLP structure tensioned in dynamic equilibrium.

Body data ► group: name ► External loads ► External force Z:

Mass*g-vol*density*g

4.3. General data

Once the geometry is ready, it is necessary to set up the properties and conditions that define the problem under consideration. First of all, select to perform a seakeeping analysis. For this, open the Start Data window and select the following options:

Data ► Start data

- Simulation dimension = 3D
- Seakeeping analysis environment = Waves
- Seakeeping analysis type = Seakeeping
- Units system = Int. system (SI) - N,m,Pa
- Geometry units = m

Seakeeping analyses undertaken using SeaFEM always consider that the fluid medium is sea water. Nevertheless, water density can be adjusted to match the actual fluid properties variation. For the present analysis, water density is taken as $\rho = 1025 \text{ kg/m}^3$.

General data ► Water density ► Water density: 1025 kg/m3

Ensure that the results file is set to binary 2 option:

General data ► Results ► Results file: Binary 2

Since we are mainly interested in the RAOs results of the TLP structure, ensure that RAOs results output is activated:

General data ► Results ► Kinematics ► Raos: Yes

Note that RAOs results option becomes available only when the *Wave spectrum type* option is set to *White noise* within the *Environment data* section of the data tree.

Environment data ► Waves ► Wave spectrum type

4.4. Problem description

As mentioned in the *Geometry definition* section of the tutorial, a zone where no artificial dissipation is introduced is defined for numerical convenience. Such a no-dissipation region is usually centered at the body under analysis, and its outer boundary is specified by the beach parameter that must be inserted in the data tree. In this particular example, the beach parameter is set at a distance $d = 30 \text{ m}$.

Open the Problem description section of the SeaFEM data tree and fill in the data as follows:

Problem description

- *Bathymetry*: constant depth.

- *Depth*: 200.0 m.

- *Waves absorption*: Yes.

- *X absorption reference*: 0.0 m.

- *Y absorption reference*: 0.0 m.

- *Absorption factor*: 1.0

- *Beach*: 30.0 m.

- *Sommerfeld radiation condition*: Yes.

- *Time step*: 0.0 s.

- *Output step*: 0.5 s.

- *Start time recording*: 350 s.

- *Initialization time*: 300 s.

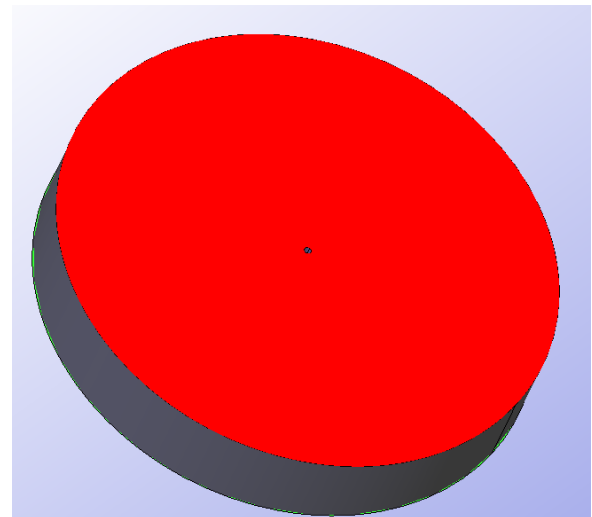
Remember that *Time data* parameters introduced by the user in problems concerning RAOs analysis are just for the sake of guidance. Actually, SeaFEM internally evaluates optimal values for both, the *Simulation time* and the *time step* based on Fourier analysis requirements.

4.7. Boundary conditions

Apart from Body data condition which is assigned to the entire boundary surface of the floating structure (see section) three boundary conditions must be specified to set-up a well-defined problem; these are the free surface, the outlet and the bottom boundary conditions.

Boundary conditions ► Free surface

This boundary condition must be applied to the two surfaces located at the top of the computational domain.



Free surface boundary condition

Boundary conditions ► Outlet

Apply the outlet condition to the outer surface located around the computational domain.

4.5. Environment data

In the *Environment data* section of the data tree the user defines the waves spectra properties that characterize the problem under analysis. Various predefined types of spectra are available, as for instance the monochromatic wave, white noise, pierson moskowitz, jonswap, and jonswap 2 spectra. In addition, custom wave spectra can be defined by the user as well as real spectra that can be directly read from text files. The actual parameters to be entered depends on the type of spectra selected. In the present case a *White noise* spectra is used since we are mainly interested in the RAOs behavior of the TLP platform. To this aim, open the Environment data section of the data tree and setup the problem as follows:

Environment data ► Waves

- *Wave spectrum*: White noise.

- *Amplitude*: 1 m.

- *Direction*: 0.0 deg.

- *Shortest period*: 3 s.

- *Longest period*: 20 s.

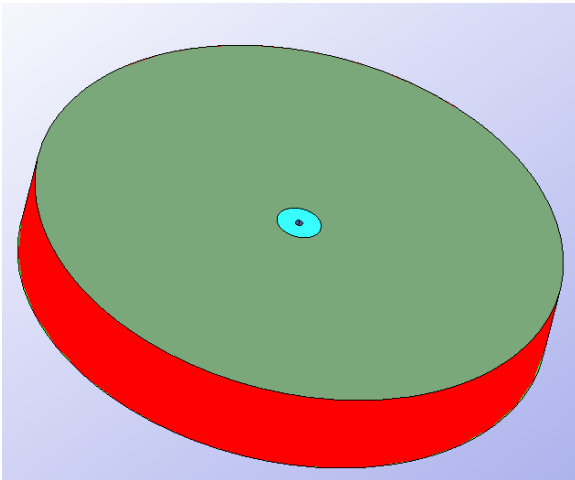
- *Number of waves periods*: 40.

4.6. Time data

Next, analysis data concerning the time step and total simulation time must be introduced. For the present simulation these data reads as follows:

Problem description

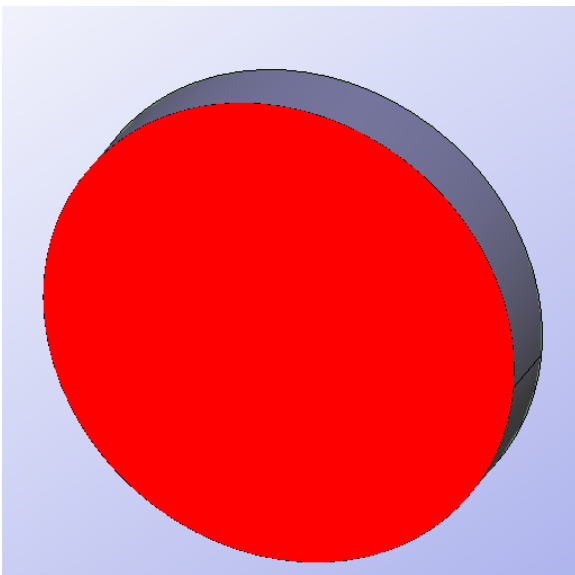
- *Simulation time*: 10000 s.



Outlet boundary condition

Boundary conditions ► Bottom

Apply this boundary condition to the two bottom surfaces of the computational domain.



Bottom boundary condition

4.8. Numerical data

The last configuration step of the analysis consists on specifying the numerical data. Processor unit and number of CPUs, solver and preconditioner, and many other parameters may be defined within this section of the data tree.

For the present simulation only the following numerical parameters need to be modified.

Numerical data

- *Processor unit:* CPU+GPU (if available in the computer)
- *Solver:* Deflated conjugate gradient
- *Preconditioner:* Diagonal

- *Solver tolerance:* 1.0E-7
- *Solver max. iterations:* 1000
- *Free surface stability factor:* 0.5
- *Damping factor:* 0.0
- *Dynamic solver max. iter.:* 100
- *Dynamic solver relaxation:* 0.25
- *Max. iter. time step:* 20
- *Tolerance:* 1.0e-4
- *Alpha B-N:* -0.1

4.9. Mesh

Before starting the calculation the computational mesh must be generated. In order to do this it is necessary to assign different element sizes to points, lines, surfaces and volumes that constitute the computational domain. As usual, the mesh size is smaller around the body with the aim of having better discretization close to the analysis region. Further, the mesh size increases when approaching the absorption area.

To reproduce the computational mesh used for the present analysis, proceed to assign mesh sizes as shown in what follows.

Mesh ► Unstructured ► Assign sizes on surfaces and descends

Assign a size of about 1.0 m to the surfaces of the floating cylinder and to the analysis area free surface.

Once mesh element sizes are assigned, it has to be ensured that the meshing preferences are correctly chosen. To this aim, open the following option of the main menu and inspect some of the meshing preferences.

Utilities ► Preferences ► Meshing

- *Unstructured mesher - Volume:* Advancing front
- *Automatic correct sizes:* None

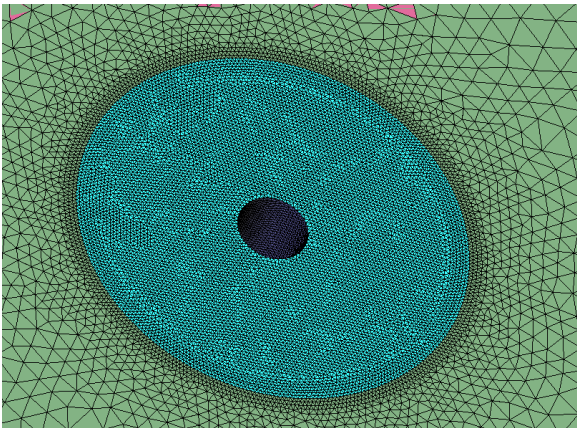
Then we can proceed to mesh generation by using the following option of the main menu and using the reported meshing settings.

Mesh ► Generate mesh

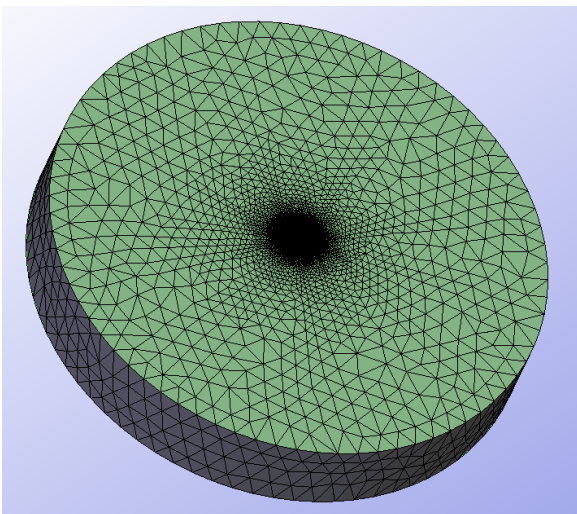
- *Maximum element size:* 40

- Unstructured size transitions: 0.1

The mesh finally obtained must be similar to that shown in the figure bellow. It can be appreciated how the mesh is gradually refined when approaching the body surface.



Detail of the mesh close the TLP structure



Global view of the computational mesh

4.10. Results

After the meshing process the SeaFEM calculation can be started using the following option of the main menu.

Calculate ► Calculate

After finishing the calculation, load the postprocess module of SeaFEM

Potsprocess ► Start

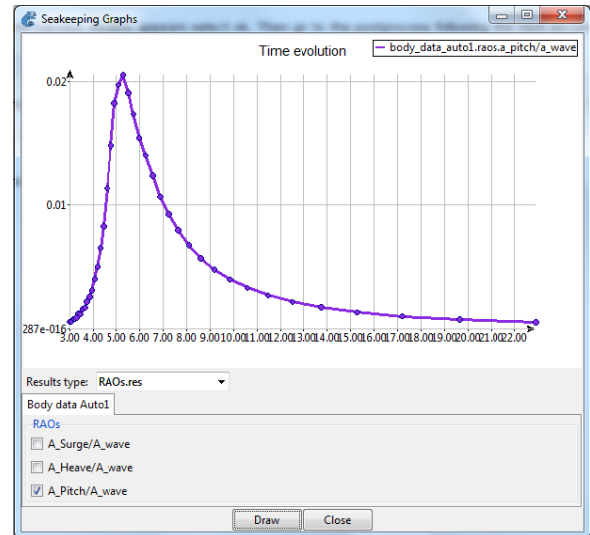
For the sake of comparison against the NREL results, proceed to represent the pitch RAO results of SeaFEM.

View Results ► SeaFEM Graphs

- Select the RAOs.res results file

- Select Pitch results option.

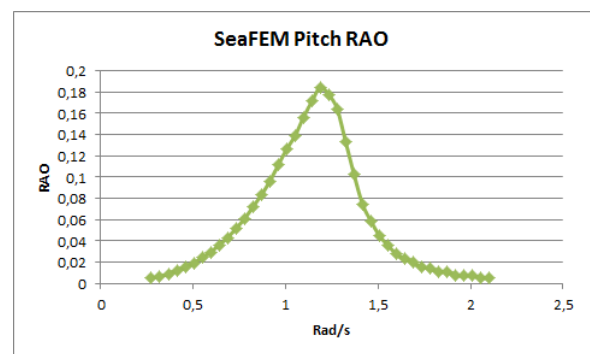
The graph obtained should look similar to that shown in the figure bellow.



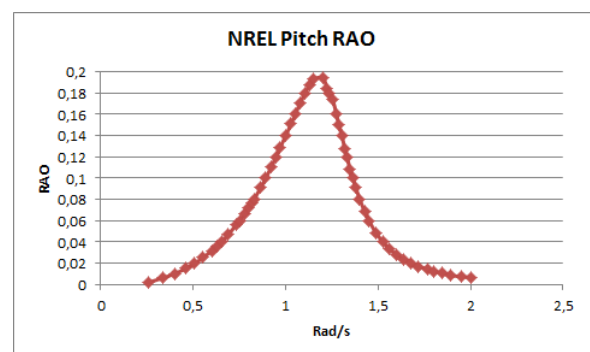
Pitch RAO results

RAOs results are reported in rad/m and are given as a function of time.

The *Filename.RAOs.res* file can be edited externally to obtain for instance the displacement at the end of the TLP structure. To this aim, we must simply multiply the Pitch RAOs results by the corresponding gyration radius (i.e. 9.0 m). This can be useful to compare the SeaFEM obtained results with some other results. It has been done in this case to compare the results obtained with SeaFEM calculus against the results reported in the NREL study. The following graphs represent the pitch RAO obtained using SeaFEM and the pitch RAO reported in the NREL study.



SeaFEM pitch RAO graph.



NREL pitch RAO graph.

As we can see the results are very similar in both studies, being

the max RAO obtained in SeaFEM about 0,184592, and in the NREL case about 0,19413, hence resulting in a difference of about 0,0095382 (i.e. 0.05%).

4.11. Bibliography

[1] Matha, D. Model development and loads analysis of an offshore wind turbine on a tension leg platform, with comparison to other floating turbine concepts. *NREL/SR-500-45891 Subcontract Report*, February 2010.

[2] Compass Ingeniería y Sistemas. SeaFEM reference. Solver for seakeeping and maneuvering problems. <http://www.compassis.com/compass/en/Soporte>.

5. ISSC TLP structure

The main object of this tutorial is to explain the basic use of the linear solver capabilities (frequency domain analysis) of SeaFEM by recreating the validation case of the ISSC TLP structure also used for validation purposes of the time domain module of SeaFEM. Also, by doing this, one can compare the results obtained with other programs.

To automatically load the model, ready to be meshed and run, simply left-click on the button below.



On the contrary, to build the model from scratch proceed as indicated in the following sections of this tutorial.

5.1. Problem description

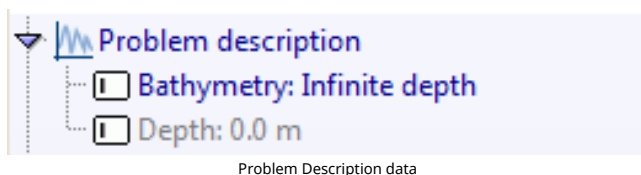
First of all problem description data concerning the Linear module of SeaFEM is described. Note that the Linear module of SeaFEM deals with frequency domain analysis.

Data tree ► Problem description

One of the main characteristics of the linear module of SeaFEM is the ease of use to prepare a new analysis. In this sense, only two parameters are required within the problem description section of the data tree:

-Bathymetry: Infinite depht. If Constant depth is chosen, then the depth has to be given.

-Depth: Only available if constant depth is chosen. If this is the case, then the depth should be inserted.



Problem Description data

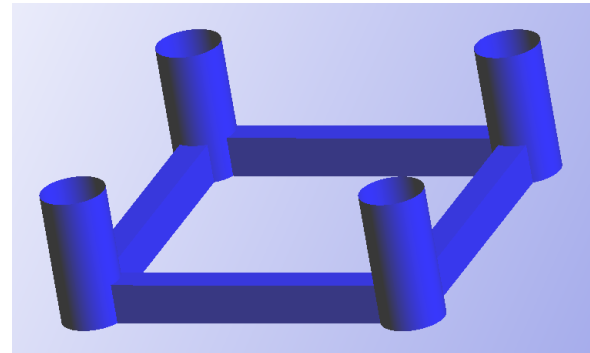
5.2. Geometry definition

In this section of the tutorial we will briefly present the geometry used for the analysis. Such a geometry is exactly the same as that of the ISSC TLP structure used in the time domain validation case and can be found at the Compassis's webpage

within the product's support section:

<http://www.compassis.com/compass/en/Soporte>

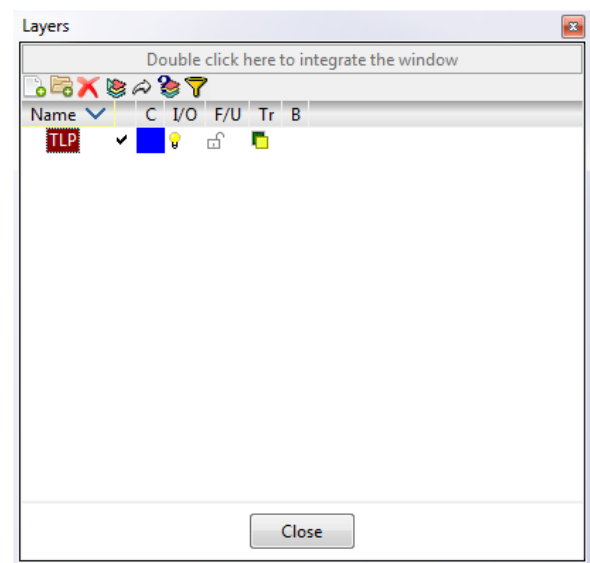
Once downloaded, the geometry can be inserted into a new SeaFEM project by importing the corresponding iges file. The following image shows the shape of the TLP structure:



ISSC TLP Structure geometry.

Once the geometry has been inserted, it is generally recommended to create a layer for each body. This is especially helpful when various bodies are present in a single project. In this case, up to this point, just one layer must be present within the layers window

Utilities ► Layers



Layers for the TLP Validation Case.

Finally, the normals of the geometry must be checked so that they point outwards of the body. If this is not done, the results will be incorrect.

5.3. General data

Once the geometry has been generated, it is time to set up the analysis. First of all, select to perform a frequency domain analysis. To this aim, open the Start Data window and select the following options:

Data ► Start data

- Simulation dimension = 3D

- Analysis domain = Frequency domain
- Seakeeping analysis environment = Waves
- Type of analysis = Seakeeping
- Units system = Int. system (SI) - N,m,Pa
- Geometry units = m

The next step is to define the general data of the problem under consideration. To do so, go to the General data menu in the Data tree:

Data Tree ► General data

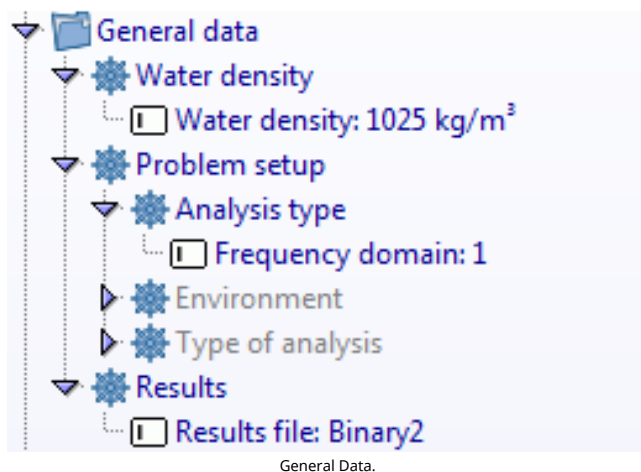
Seakeeping analyses undertaken using the linear frequency domain module allways consider that the fluid medium is sea water. Nevertheless, water density can be adjusted to match the actual fluid under consideration. For the present analysis, water density is taken as $\rho = 1025 \text{ kg/m}^3$.

General data ► Water density ► Water density: 1025 kg/m3

Ensure that the results file is set to binary 2 option:

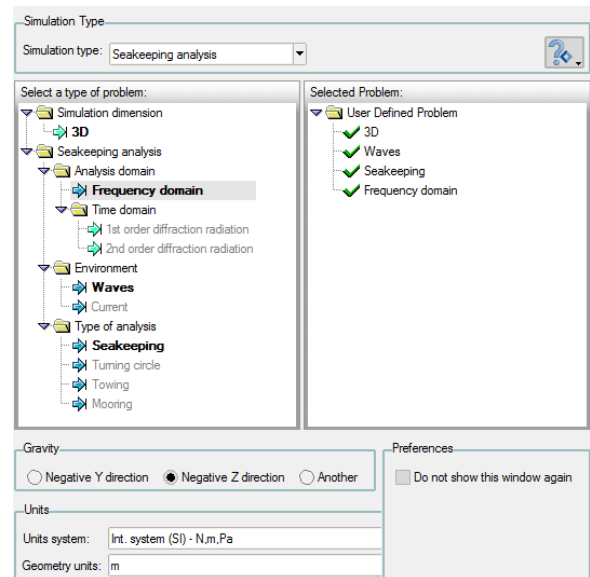
General data ► Results ► Results file: Binary 2

Frequency domain results can be stored in several different formats; Default (which is the same as binary2), binary1, ASCII or Nemoh formats. If Nemoh is chosen, all results files will be stored using the native Nemoh format and placed under the Nemoh_output folder. For the present validation case, use default or binary2 options which correspond to the native output format of SeaFEM.



As a final remark, note that most of the general simulation parameters can also be set from the *Start Data* window which prompts when the program is started, or when clicking the following option within the menu bar.

Data ► Start data



SeaBEM Start Data window

5.4. Environment data

In the *Environment data* section of the data tree the user defines the wave properties that characterize the problem under consideration. For the present case, the following values must be introduced to setup the problem:

Environment data ► Waves

- Shortest period: 8 s.
- Longest period: 15 s.
- Number of wave periods: 10
- Lower heading: 0 deg
- Upper heading: 90 deg
- Number of wave headings: 5
- Speed: 0 m/s
- Speed direction: 0 deg

5.5. Body data

In this section the body boundary conditions are defined:

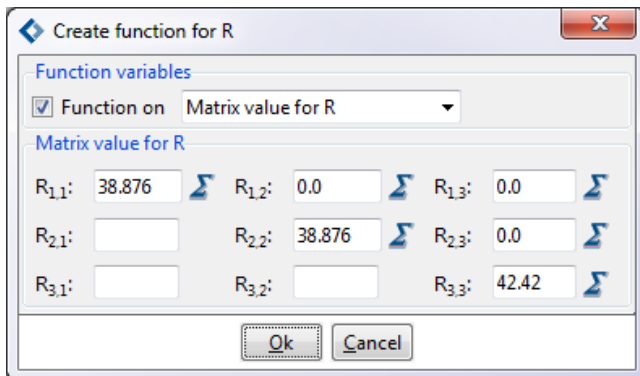
Data tree ► Body data

The main body properties are:

- Mass: 0.0 Kg. This way the mass is calculated using the displacement caused by the body.

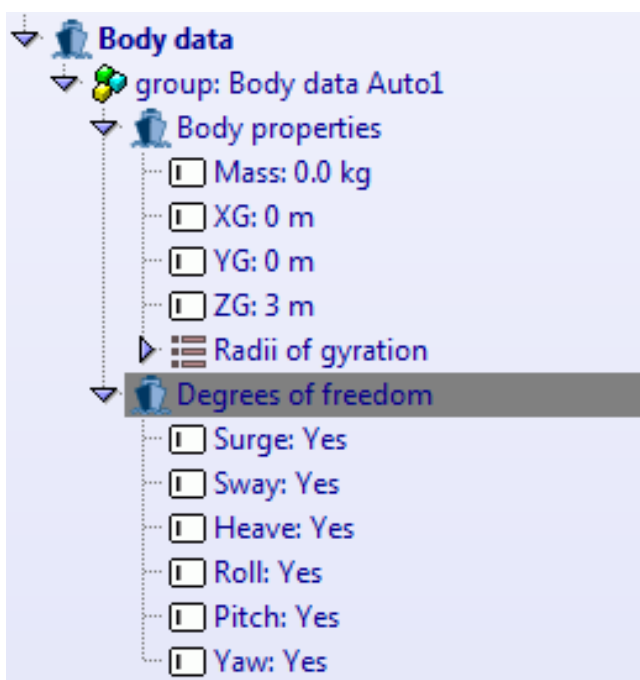
The center of gravity: (XG=0m, YG=0m, ZG=3m)

- The radii of gyration has the following values:



Radii of gyration Matrix

- For this case, the free degrees of freedom are:



Body Data configuration menu.

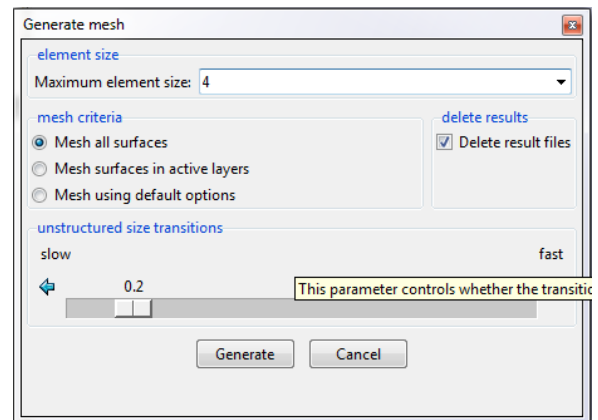
5.6. Mesh

Finally, before beginning the calculation, the mesh has to be defined. This case can be calculated with a relatively simple mesh without having to define any mesh data. Frequency domain calculations require a mesh of quadrilateral elements. Nevertheless, there is no need to change the element type () within the mesh properties menu since SeaFEM automatically manages these mesh requirements. Without further ado, the following data is used:

Mesh ► Element type

- Open the mesh generation menu found under the "mesh" main menu: . The following window will appear:

Mesh ► Generate mesh



Mesh generation menu.

As it can be observed, the element size will be 4, with a transition size of 0.2.

- After creating the mesh, the number of nodes should be around 1187, and the number of elements around 1167.

5.7. Results

This final section shows some of the results ascertained after executing the frequency domain solver.

After meshing the body, we can proceed to execute the solver for calculation. This is done by executing the following menu sequence:

Calculate ► Calculate

or by pressing the icon shown below which is located within the SeaFEM toolbar:



Once the calculation has started, information on how it's doing can be found by going to the following menu's option:

Calculate ► View process info...

or by pressing the following icon in the toolbar:



If the execution finishes ok, a message will be prompted that will also ask if you wish to go to the postprocessor. As the default type of results was chosen, press postprocess. Another way to postprocess is by going to the following menu's option:

Postprocess ► Start

or by pressing the following icon:



If the type of results option chosen was not default or Binary2, the traditional post has to be used. To do this use:

Postprocess ► Traditional post

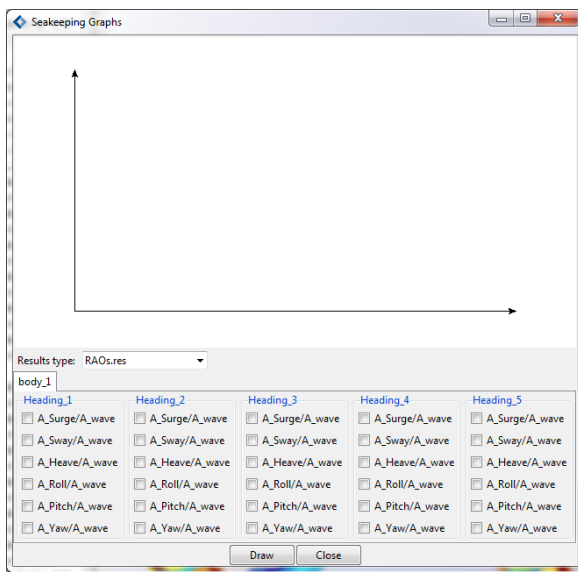
To see graph results when using the default (binary2) postprocess format, use the following option of the postprocess menu:

Postprocess ► SeaFEM graphs

On the contrary, if using the traditional post, go to:

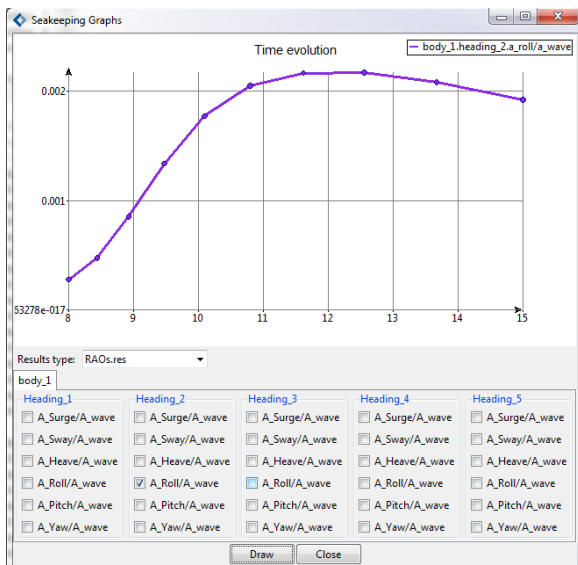
View Results ► SeaFEM Graphs

With this, the following window should pop up:



Seakeeping graphs.

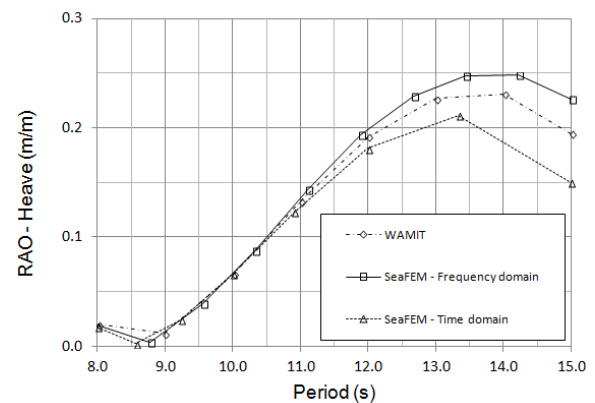
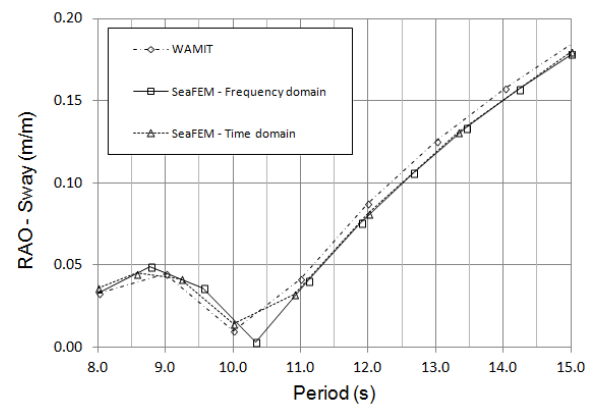
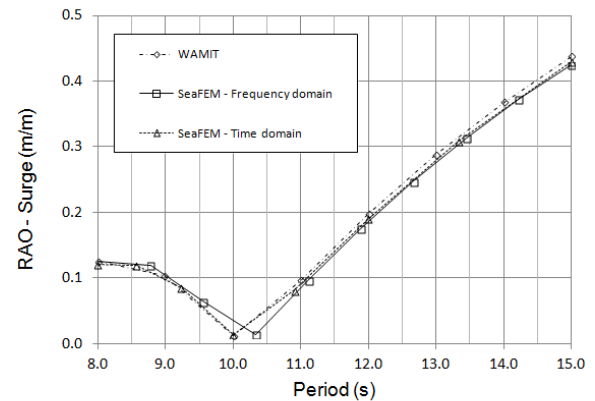
For the sake of comparison the results concerning a heading direction about 22.5 degrees will be used. Hence, select "A_Roll/A_wave" within the second heading results box. By clicking on draw, the following graph is obtained:

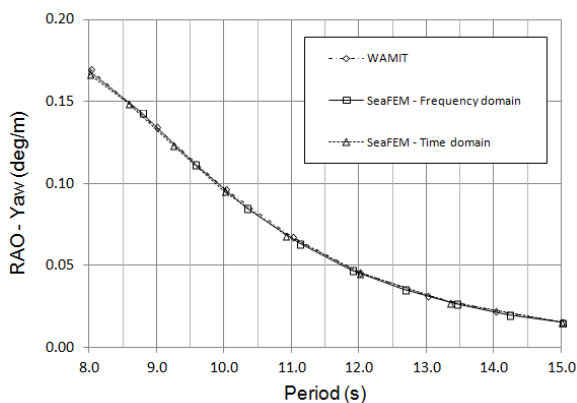
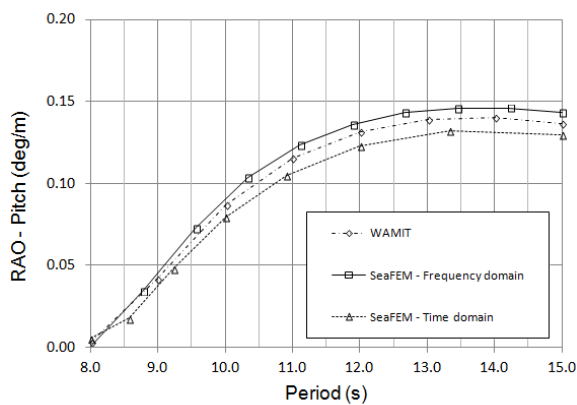
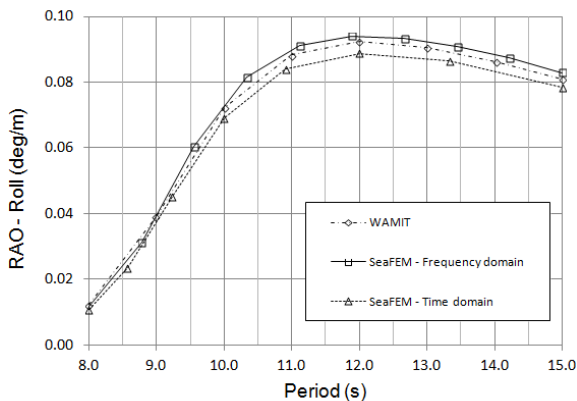


RAO of roll motion for a wave direction of 22.5 degrees using SEABEM.

For the sake of comparison, the output of SeaFEM is

postprocessed and plotted together with the results of WAMIT. The corresponding graphs are shown below. It must be taken into account that WAMIT provides RAOs results in non-dimensional form for both, translation and rotational degrees of freedom. For the latter, this is achieved by multiplying RAOs results by a characteristic length provided as an input of the model under analysis. In this particular case, such a characteristic length is half the separation between the legs of the TLP structure, and is about $L = 43.125$ m. Hence, for proper comparison, SeaFEM results concerning RAOs rotational degrees of freedom are also multiplied by the same characteristic length.





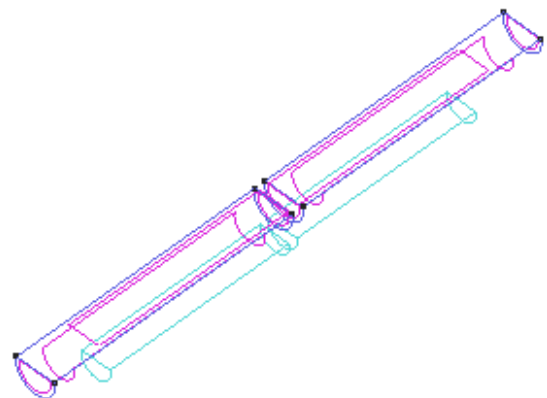
6.1. Geometry definition and body data

In this tutorial, geometry creation is not described in detail. Actually, the geometry of the Pelamis device is simple enough as to be constructed using the CAD capabilities provided by the SeaFEM pre-processor and explained in detail in previous tutorials.

The Pelamis considered here consists of two circular cylinders connected by a ball joint with an additional restriction that prevents any relative rotation between the two segments along their common longitudinal axis. An energy extracting system is used to extract energy from the relative pitch velocity between the two cylinders. Such an energy extracting system acts in turn as pitch damper. The main characteristics of each Pelamis segment are summarized in the table below.

Length	34.0 m.
Beam	5.5 m.
Draft	3.404 m.
Distance between segments	1.5 m.
XG (from stern side)	17.0 m.
YG	0.0 m.
ZG	-1.0 m.
Radii of inertia Rxx	2.4 m.
Radii of inertia Ryy = Rzz	12.3 m.

The geometry of the submerged part of the Pelamis, and the corresponding computational domain are shown in the figure below.

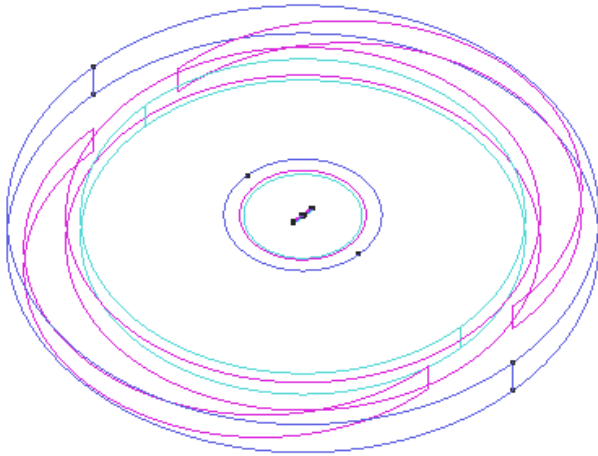


Geometry of the submerged part of the Pelamis device considered in the present analysis

6. Multibody analysis of a Pelamis device

The next tutorial illustrates the use of SeaFEM's multibody capabilities for the analysis of complex devices involving the mechanical interaction of various moving parts. To this aim, the analysis of a device of the Pelamis type is undertaken. The Pelamis is a design conceived for extracting energy from waves. It consists of a set of bodies with a snake shape connected by mechanical joints, whose actual shape is rearranged due to the action of waves. Energy can be extracted from the relative movement across consecutive segments of the device.

In this tutorial, a simplified version of the Pelamis, consisting of just two articulated segments, is modelled under the action of irregular sea waves.



Computational domain used for the analysis of the Pelamis device. Domain's depth is taken to be $h = 100.0$ m. A near field region is considered with a diameter $D = 400$ m.

Body properties must be specified separately for each segment of the Pelamis by applying a Body condition to each one of the parts. Body properties should read as follows:

Body data ► group:Segment1 ► Body properties

- Mass: $\text{vol}(1) \cdot \text{density}$
- XG: -17.75 m
- YG: 0.0 m
- ZG: -1.0 m
- Radii of gyration Rxx: 2.4 m
- Radii of gyration Ryy = Rzz: 12.3

Body data ► group:Segment1 ► Degrees of freedom

- Surge: Yes
- Sway: Yes
- Heave: Yes
- Roll: Yes
- Pitch: Yes
- Yaw: Yes

Body data ► group:Segment2 ► Body properties

- Mass: $\text{vol}(2) \cdot \text{density}$
- XG: 17.75 m
- YG: 0.0 m
- ZG: -1.0 m
- Radii of gyration Rxx: 2.4 m
- Radii of gyration Ryy = Rzz: 12.3

Body data ► group:Segment2 ► Degrees of freedom

- Surge: Yes
- Sway: Yes
- Heave: Yes
- Roll: Yes
- Pitch: Yes
- Yaw: Yes

Real PTO models of actual Pelamis devices are quite complex and detailed information is usually not publicly available. Hence, in this tutorial a simplified PTO model is going to be used. In this sense, the PTO is assumed to work as a pitch damper, where the reaction of the dampers produces a pitch moment in the opposite direction of the relative pitch velocity, and whose magnitude is proportional to the relative velocity. The mathematical model describing this PTO system is:

$$M_y^{(1)} = -K_{\text{PTO}}(\omega_y^{(1)} - \omega_y^{(2)})$$

$$M_y^{(2)} = -K_{\text{PTO}}(\omega_y^{(2)} - \omega_y^{(1)})$$

$$P = K_{\text{PTO}}(\omega_y^{(2)} - \omega_y^{(1)})^2$$

where $M_y^{(i)}$ and $\omega_y^{(i)}$ represent the pitch moment and angular velocity for segment (i), K_{PTO} is a constant value depending on the PTO design, and P is the instantaneous extracted power. In the present tutorial K_{PTO} value was assumed to be $5.0\text{E}7$ N·m/rad/s

The moments exerted by the PTO system over each one of the Pelamis parts can be specified in the form of an external moment within the Body data definition of the corresponding Body group.

Body data ► group:Segment1 ► External loads

- External moment Y: $-5.0E7*(wy(1)-wy(2))$

Body data ► group:Segment2 ► External loads

- External moment Y: $-5.0E7*(wy(2)-wy(1))$

6.2. General data

In the present tutorial, we are interested on the seakeeping analysis of the Pelamis device. Hence, the floating bodies must be subjected to the action of an irregular waves spectrum. In particular, a Jonswap's type spectrum is considered without currents. General data defining the problem can be summarized as follows. Parameters not mentioned here may keep its default value.

General data ► Water density

- Water density: 1025 kg/m^3

General data ► Problem setup ► Environment

- Waves: 1

- Current: 0

General data ► Problem setup ► Type of analysis

- Seakeeping: 1

- Turning circle: 0

- Towing: 0

General data ► Results ► Kinematics

- Movements: Yes

- Velocities: Yes

- Accelerations: Yes

- Raos: No

General data ► Results ► User defined ► Custom output 1

- Active: Yes

- Name: My_PTO

- Expression: $-5.0E7*(wy(1)-wy(2))$

General data ► Results ► User defined ► Custom output 2

- Active: Yes

- Name: Instantaneous power

- Expression: $5.0E7*(wy(2)-wy(1))^2$

Note the usage of two custom outputs defined by the user that are intended to keep track of the moment acting on each segment of the Pelamis due to the damping action of the PTO and to record the instantaneous power extracted also by the idealized PTO system under consideration.

6.3. Problem description

As usual, a zone where no artificial dissipation occurs is defined for numerical convenience. Such a none-dissipative region is usually centered at the body under analysis, and its outer boundary is specified by the *beach* parameter that must be inserted in the data tree. In this particular example, the beach parameter is set up so that the none-dissipative region coincides with the region with finer mesh resolution created around the body (near field region). Nevertheless, such a coincidence is in general not mandatory.

Open the *Problem description* container in the SeaFEM data tree and fill in the corresponding data as follows:

Problem description

- Bathymetry: Constant depth

- Depth: 100 m

- Wave absorption: Yes

- X absorption reference: 0 m

- Y absorption reference: 0 m

- Absorption factor: 1.0

- Beach: 200 m

- Sommerfeld radiation condition: Yes

- Initialization time: 10 s

6.4. Environment data

The wave environment under consideration corresponds to an irregular sea modelled with a Jonswap spectrum with a mean wave period of 11.0 s., a significant wave height of 2.5 m., and a wave spreading angle of 30.0 degrees. The mean direction of propagation is along the x axis, which is aligned with the longitudinal direction of the device. A total number of 55 monochromatic waves have been used to reproduce the irregular sea under consideration. All actual properties of the spectrum under consideration are summarized as follows (parameters not reported in the following list should take their default values).

Environment data ► Waves

- Wave spectrum type: Jonswap

- Mean wave period: 11 s

- Significant wave height: 2.5 m

- Shortest period: 4 s

- Longest period: 22 s

- Number of waves periods: 11

- Mean wave heading: 0 deg

- Spreading angle: 30 deg

- Number of waves headings: 5

6.5. Time data

Apart from wave environment, analysis data concerning the time step and total simulation time must be introduced. For the present simulation these data reads as follows:

Time data

- Simulation time: 1000 s

- Time step: 0.0 s

- Output step: 0.5

- Start time recording: 10 s

When time step is set to 0.0, it is actually evaluated internally by SeaFEM based on a stability criteria related to the *Free surface stability factor* entered in the *Numerical data* section of the data tree. A simulation time about 1000 s may be used for the sake of computation time economy. Nevertheless, longer time series are usually required, specially when the model under consideration is intended for the assessment of wave power extraction. In this sense, it can be useful to keep track of the time evolution of the mean generated power since it provides a good measure of the convergence of the simulation results (see the results section of this tutorial for further details).

6.6. Mooring system

The mooring system used to avoid drift effects on the device consists of three single-segment catenary lines, all of them connected to the front of the hull of the first segment of the Pelamis. The three lines then spread towards the sea bottom with a spreading angle of 120 degrees between the lines. The three catenary lines are identical and their main characteristics are summarized in the table below.

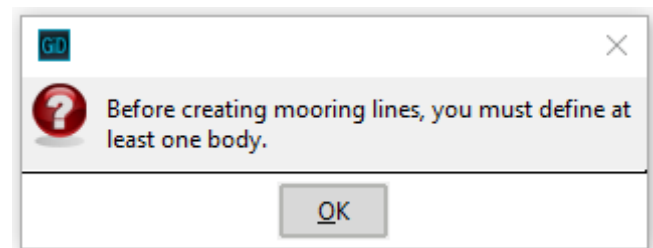
Length	267.4 m.
Cross-section area	7.85E-3 m ²
Effective linear density	525.64 kg/m
Young modulus	210 GPa

The graphic user interface of SeaFEM provides tools to ease the setup of mooring systems, from very simple moorage lines like those used in the present tutorial to very complex mooring systems consisting of several multi-segmented lines with additional buoys and weights designed for fine station-keeping control.

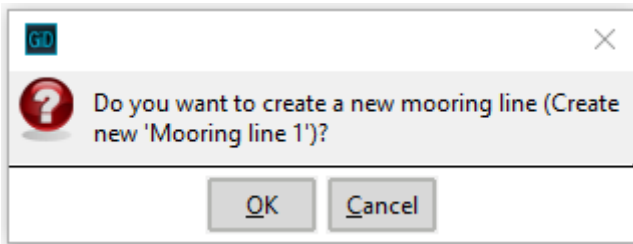
Here, we will be briefly present the various steps necessary to define the mooring lines used to connect the Pelamis to the sea bottom.

- To define a new mooring system in SeaFEM, double-click on the option of the data tree. Note that to define mooring data at least one body must be already defined. If not, the following warning message would appear.

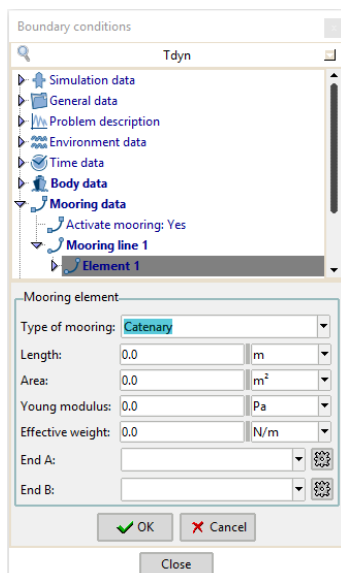
Mooring data



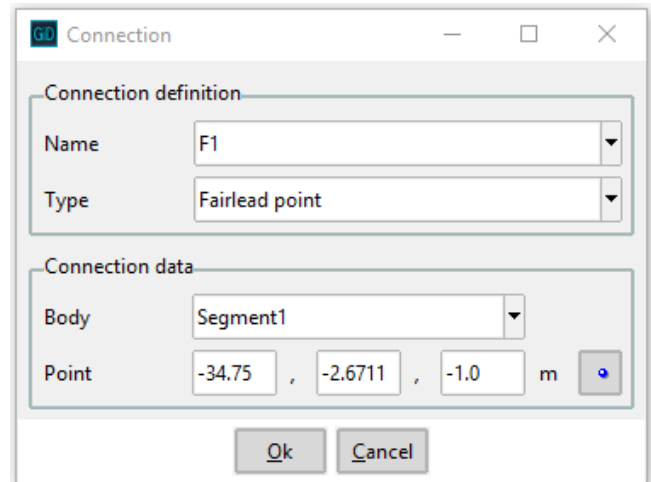
- The first time mooring data definition is invoked, the following message will appear asking for confirmation.



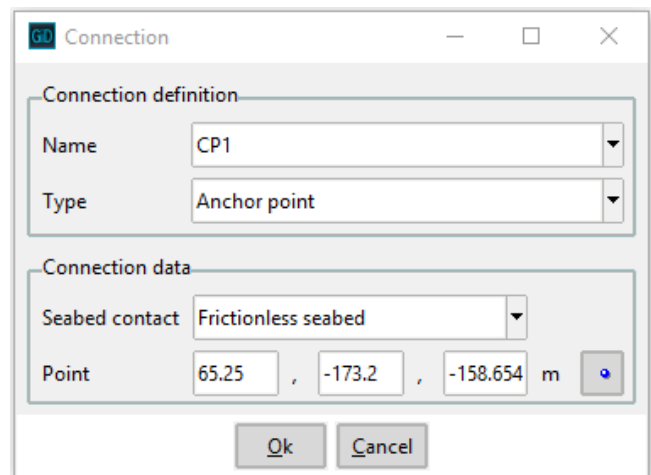
- Once accepted, mooring capabilities will become active, the first mooring line containing a single element will be automatically created and the *Mooring element* definition window will be opened as shown in the figure below. Within this window, the user must specify the mooring line segment properties (as reported in the summary table at the beginning of this section) and the corresponding connection points.



- To create a connection point, left-click the *Edit link* icon next to the *End A/B* window entries. When one or more connection points have been already defined, they will be available in the drop down list allowing their direct selection. The new connection point creation window looks like those in the following images, where data defining the first fairlead and anchor points have been entered respectively. For further details on all the options available for the definition of different types of connection points see the SeaFEM's users manual.

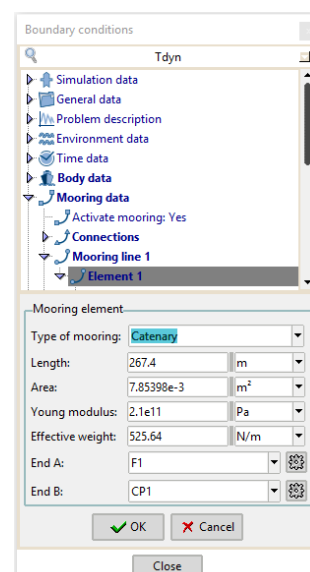


Fairlead point definition's window (point where the mooring segment is directly attached to the body)



Anchor point definition's window (point where the mooring segment is fixed to the seabed)

- After definition of the two connections points of the mooring segment, and after introducing all the properties that define the corresponding mooring element, the definition's windows should look like this:



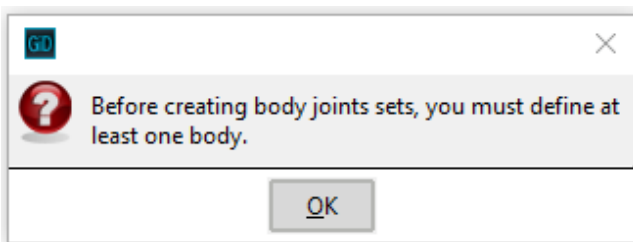
New mooring lines can be defined from scratch, or may be created by copying and editing an already existing line. For the latter option, just right-click on the existing mooring line group and select *Copy* option. Next, the name of the new mooring line can be edited as well as the connection points and mooring element properties.

6.7. Multibody analysis definition

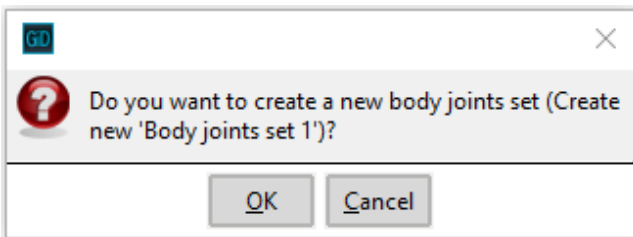
To end with the setup of a multibodies analysis, it is necessary to define the links between bodies that actually define the required mechanical constraints. As in the case of the mooring system, SeaFEM provides GUI tools to ease the definition of complex multibody systems. In this section it will be explained how to define a simple connection between the two cylindrical segments of the Pelamis.

- To define a new body joint in SeaFEM, double-click on the option of the data tree. Note that to define a body joint at least one body must be already defined (it is possible to define mechanical constraints involving only a single body using some of the body joint options, all of them based on the same langrange multipliers formulation). If not, the following warning message would appear.

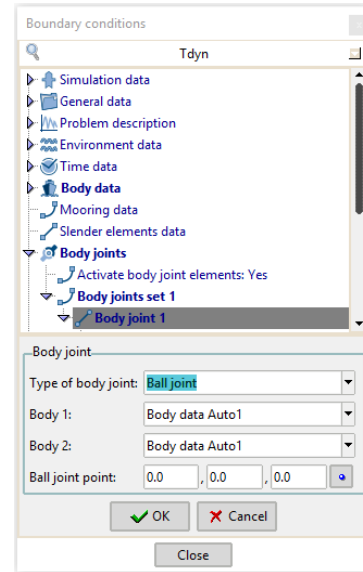
Body joints



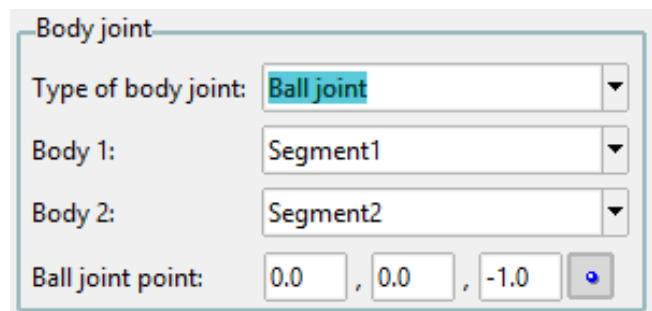
- The first time body joints option is invoked, the following message will appear asking for confirmation.



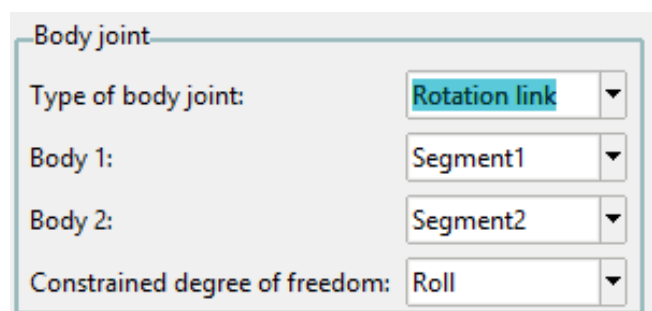
- Once accepted, body joint capabilities will become active, a first set of body joints containing a single body joint will be automatically created and the *Body joint* definition window will be opened as shown in the figure below. Within this window, the user must specify the type of connection and the bodies involved in the definition of the current joint. Depending on the type of body joint selected different parameters may be required to finalize the set up of the corresponding mechanical constraints.



- In the following figures the definitions of a Ball joint and a simple Rotation link between the two cylindrical segments of the Pelamis device are shown.



Definition of a Ball joint between segments 1 and 2 of the Pelamis device under analysis in the present tutorial



Definition of a simple Rotation link that enforces the Roll movement to be the same in both segments of the Pelamis

6.8. Boundary conditions

As usual, several boundary conditions must be applied to finalize the set up of the SeaFEM analysis. In addition to the Body conditions described in detail in section *Geometry definition* and *body data*, the following boundary conditions must be applied.

First, a *Free surface* condition must be applied to those surfaces located at $z=0$

Boundary conditions ► Free surface

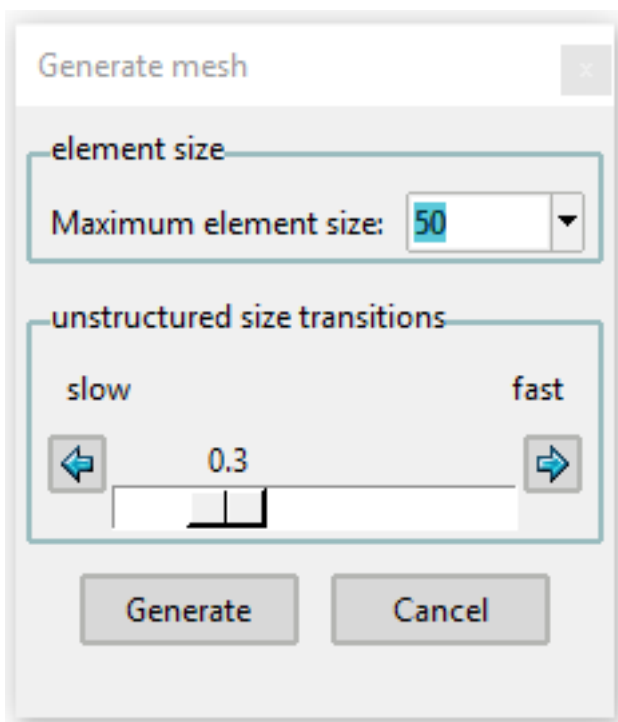
Finally, an *Outlet* boundary condition must be applied to the

outer boundary of the computational domain. If a Sommerfeld radiation condition is to be applied, the outer surface should be of circular shape with center at the origin of coordinates.

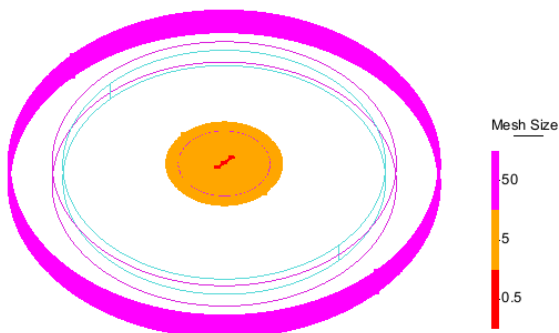
Boundary conditions ► Outlet

6.9. Mesh

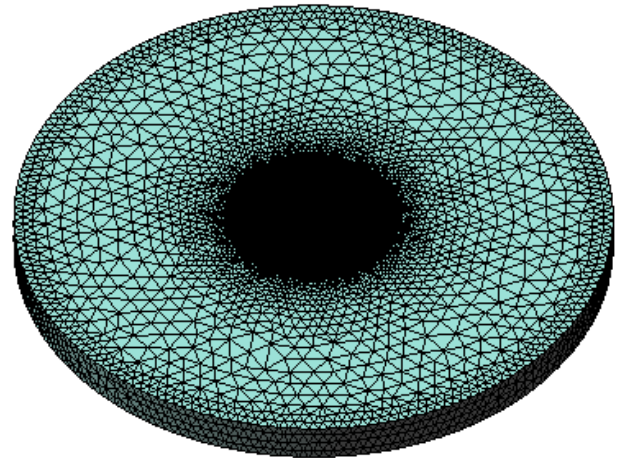
The computational domain considered for the present analysis has 100 m. depth. A mesh size of 0.5 m. has been used for the device and its surroundings. A mesh size of about 5.0 m. is used in the near field region. A maximum mesh size about 50 m. and an unstructured mesh size transition of 0.3 is used for mesh generation as shown below. The damping zone is located starting at 200 m. away from the device. The resulting mesh consists of 20,436 nodes and 105,046 linear tetrahedra.



Maximum element size and unstructured mesh size transition used during the generation of the finite element mesh for the analysis of the Pelamis device under consideration in the present tutorial



Element sizes explicitly assigned to various geometrical entities of the bodies and the computational domain for fine tuning of the generated mesh.



Isometric global view of the finite element mesh used in the simulation of the seakeeping behavior of a Pelamis (multibody) device

6.10. Results

This final section shows some of the results obtained after executing the time domain solver.

After meshing the model we can proceed to execute the solver for calculation. This is done by executing the following menu sequence:

Calculate ► Calculate

or by pressing the icon shown below which is located within the SeaFEM toolbar:



Once the calculation has started, information on how it's doing can be found by going to the following menu's option:

Calculate ► View process info...

or by pressing the following icon in the toolbar:



If the execution finishes ok, a message will be prompted that will also ask if you wish to go to the postprocessor. Postprocess can be also accessed at any time by using the main menu option:

Postprocess ► Start

or by pressing the following icon:



When using the default (binary2) postprocess format, time evolution graphical results can be visualized by using the following option of the main postprocess menu:

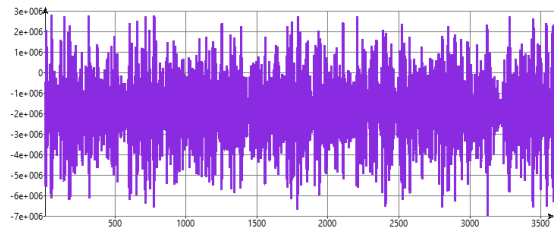
Postprocess ► SeaFEM graphs

On the contrary, if using the traditional post, go to:

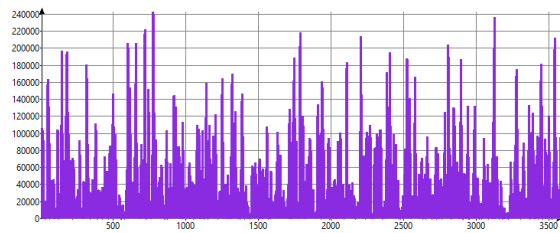
View Results ► SeaFEM Graphs

Pitch moment on body 1 due to the action of the PTO and the

instantaneous extracted power are shown in the following figures as they appear in the graphical results window of the SeaFEM postprocess.

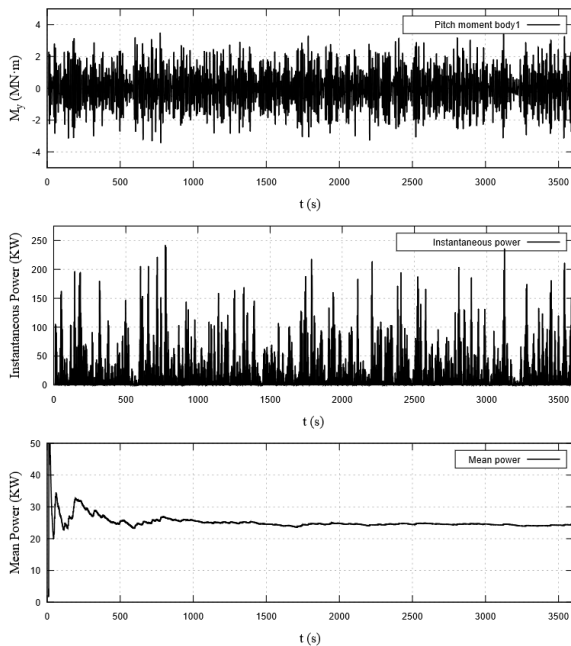


Pitch moment on body 1 (N-m)



Instantaneous extracted power (W)

As explained in the [Time data](#) section of this tutorial, large time series may be simulated for an accurate assessment of the wave power extraction. And it may be a good practice to plot the time evolution of the mean generated power since it will usually provide a good measure of the simulation's convergence. To this aim, the instantaneous power data recorded in the custom output results file can be edited and operated to obtain the mean generated power. Pitch moment, instantaneous power and mean power are plotted together in the following graph.



Time evolution of some quantities that are of interest in the analysis of the Pelamis device analyzed in the present tutorial. a) Pitch moment acting on segment 1 of the Pelamis due to the action of the PTO. b) Instantaneous power extracted from waves by means of the idealized PTO considered in this tutorial. c) Time evolution of the mean power extracted from waves. It can be observed that after 1500 s, the mean power remains almost constant indicating that the simulation has converged for the wave spectrum under consideration

7. Analysis of a floating OWC device

The next tutorial illustrates the use of SeaFEM capabilities for the analysis of oscillating water column (OWC) devices. OWCs are a particular type of wave energy converter (WEC) designed to harness the energy from a water column that oscillates due to the action of waves inside of a pressurized chamber. Waves actually force the water column to act like a piston forcing the air to flow out and back into the chamber. This causes a continuous flow flowing through a bidirectional turbine that finally converts pneumatic power into electricity.

In this tutorial, a simplified version of a floating OWC device is considered. It consists of three tied chambers with squared-shape cross-section. Emphasis is on explaining how to set up pressurized free surface (PFS) boundary conditions, and how to take into account in the dynamics of the floating body the forces acting on the device due to the internal pressure of the chambers.

7.1. Geometry definition and body data

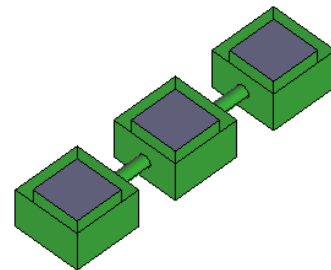
In this tutorial, geometry creation is not described in detail since it is simple enough as to be constructed using the CAD capabilities provided by the SeaFEM pre-processor explained in previous tutorials.

The OWC design considered in this tutorial consists on a scale model of a device with three square-shaped columns opened to

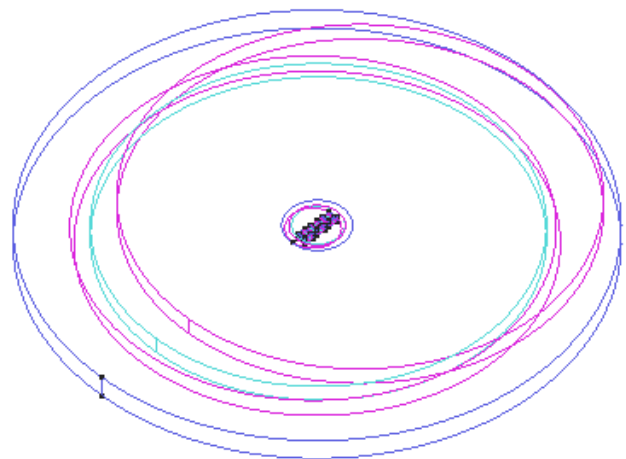
the sea water at their bottom. The main characteristics of the device are summarized in the following table.

Total length	1.408 m.
Draft	0.1995 m.
XG	0.0 m.
YG	0.0 m.
ZG	-0.052 m.
Radii of inertia Rxx	0.150 m.
Radii of inertia Ryy	0.410 m.
Radii of inertia Rzz	0.546 m.
Separation between chambers	0.2 m.
Inner chamber cross section	0.0576 m ²

The geometry of the submerged part of the OWC device, and the corresponding computational domain are shown in the figure below.



Geometry of the submerged part of the floating OWC device. It can be observed how the three independent chambers are joined by rigid arms so that the device as a whole behaves as a rigid body



Computational domain used for the analysis of the OWC device. Domain's depth is taken to be $h = 0.8$ m. A near field region is considered with a diameter $D = 44$ m.

Since the three chambers are rigidly connected, body properties can be specified considering the whole device as being a single rigid body. To this aim, just one Body condition is applied to all surfaces defining the geometry of the OWC device (inner and outer walls of the three chambers and their joining arms). Body properties should read as follows:

Body data ► group:Body data Auto1 ► Body properties

- Mass: vol*density

- XG: 0.0 m

- YG: 0.0 m

- ZG: -0.052 m

- Radii of gyration Rxx: 0.150 m

- Radii of gyration Ryy: 0.41 m

- Radii of gyration Rzz: 0.546 m

Body data ► group:Segment1 ► Degrees of freedom

- Surge: Yes

- Sway: Yes

- Heave: Yes

- Roll: Yes

- Pitch: Yes

- Yaw: Yes

7.2. General data

As in the previous tutorial, we are interested in the seakeeping behavior of the floating structure under the action of an irregular sea state. Hence, the floating body is going to be subjected to the action of an Jonswap spectrum. General data defining the problem can be summarized as follows. Parameters not mentioned here may keep its default value.

General data ► Water density

- Water density: 1025 kg/m³

General data ► Problem setup ► Environment

- Waves: 1

- Current: 0

General data ► Problem setup ► Type of analysis

- Seakeeping: 1

- Turning circle: 0

- Towing: 0

General data ► Results ► Kinematics

- Movements: Yes

- Velocities: Yes

- Accelerations: Yes

- Raos: No

7.3. Problem description

As usual, a zone where no artificial dissipation occurs is defined for numerical convenience. Such a none-dissipative region is usually centered at the body under analysis, and its outer boundary is specified by the *beach* parameter that must be inserted in the data tree.

Open the *Problem description* container in the SeaFEM data tree and fill in the corresponding data as follows:

Problem description

- Bathymetry: Constant depth

- Depth: 0.8 m

- Wave absorption: Yes

- X absorption reference: 0 m

- Y absorption reference: 0 m

- Absorption factor: 1.0

- Beach: 3 m

- Sommerfeld radiation condition: Yes

7.4. Environment data

The wave environment under consideration is scaled in accordance to the geometry of the model under consideration. It actually corresponds to an irregular wave state described by a unidirectional Jonswap spectrum with a mean wave period of 1.14 s. and a significant wave height of 0.028 m. The mean

direction of propagation is at 45 degrees about the x axis, misaligned with respect to the longitudinal axis of the floating structure. A total number of 25 monochromatic waves have been used to reproduce the irregular sea under consideration. All actual properties of the spectrum under consideration are summarized as follows (parameters not reported in the following list should take their default values).

Environment data ► Waves

- Wave spectrum type: Jonswap
- Mean wave period: 1.14 s
- Significant wave height: 0.028 m
- Shortest period: 0.52 s
- Longest period: 2.508 s
- Number of waves periods: 25
- Mean wave heading: 45 deg
- Spreading angle: 0 deg
- Number of waves headings: 1

7.5. Time data

Apart from wave environment, analysis data concerning the time step and total simulation time must be introduced. For the present simulation these data reads as follows:

Time data

- Simulation time: 150 s
- Time step: 0.0 s
- Output step: 0.5
- Start time recording: 0 s
- Initialization time: 10 s

When time step is set to 0.0, it is actually evaluated internally by SeaFEM based on a stability criteria related to the *Free surface stability factor* entered in the *Numerical data* section of the data tree.

7.6. Mooring system

The mooring system used to avoid drift effects on the device consists of four single-segment catenary lines, each one anchored to one bottom corner of the structure. The four lines then spread towards the sea bottom. The four catenary lines are identical and their main characteristics are summarized in the table below.

Mooring element type	Catenary
Length	3.75 m.
Cross-section area	5.6332E-6 m ²
Effective linear weight	0.378544 N/m
Young modulus	210 GPa

SeaFEM provides a simple tool that may help on the design of very simple mooring lines like those used in the present analysis. To access this tool, simply resort to the following main menu option:

Geometry ► Create ► Catenary NURBS...

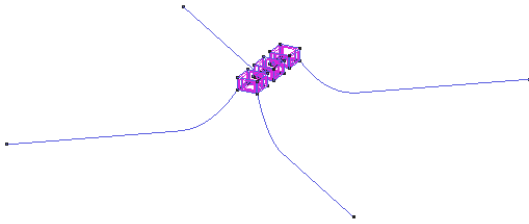
This tool consists of a window form where the user must enter all the parameters that define the mooring line (i.e. end points Xi and Xe, effective linear weight W, cross-section area A, Young's modulus E and length L). Currently, only elastic catenary lines can be modeled using this tool, but two types of seabed interaction can be taken into account. The first one is actually a "no seabed" option so that the resulting catenary will freely slack between the two specified end point. The second type of seabed interaction corresponds to the "sandy bed" option and the resulting catenary will not slack but will partially lie at the ground level given by the second end point.

Lines Definition	
Pick Ini.	Pick End
Xi (m)	Xe (m)
Yi (m)	Ye (m)
Zi (m)	Ze (m)
W (N/m)	L (m)
A (m2)	
E (N/m2)	Type
	Seabed parameter
	Tension (N)
<div> <div>Create</div> <div>Close</div> </div>	

Catenary creation window. This tool is provided in SeaFEM to help users define simple mooring lines based on elastic catenaries. It works by providing the properties of the mooring segment whose elastic catenary equilibrium solution is further plotted for the ease of visualization

Once all the parameters are introduced and the 'Create' button is invoked, the program solves the elastic catenary problem to find the equilibrium configuration of the mooring line. This configuration is finally plotted in the graphic user interface for visual inspection. This tool is useful as far as it allows to visualize the initial configuration of simple mooring systems (i.e. those composed of single-segment lines) thus providing the opportunity to detect obvious design errors (especially those errors related with the coordinates of the fairlead and anchor

points) before introducing all mooring information in the data tree. The geometry of the mooring lines that is plotted in the GUI also eases the definition of the mooring geometry in the data tree since the end points of each mooring element can be selected interactively. Note: it is recommended to isolate in an independent layer the mooring geometry generated by the 'Catenary NURBS' option. This further allows to freeze this auxiliary layer when meshing the model.



Initial equilibrium configuration of the 4-lines mooring system used for the present analysis of the OWC device

7.7. Boundary conditions

As usual, several boundary conditions must be applied to finalize the set up of the SeaFEM analysis. In addition to the Body conditions described in detail in section [Geometry definition and body data](#), the following boundary conditions must be applied.

First, a *Free surface* condition must be applied to those surfaces located at $z=0$.

Boundary conditions ► Free surface

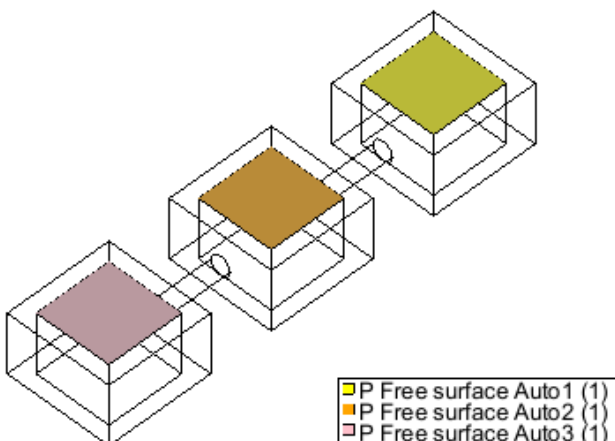
It is important to note that the free surface boundary condition must include also the free surface patches located inside of the chambers, where 'Pressurized free surface' conditions will be also applied later.

An *Outlet* boundary condition must be applied to the outer boundary of the computational domain. If a Sommerfeld radiation condition is to be applied, the outer surface should be of circular shape with center at the origin of coordinates.

Boundary conditions ► Outlet

Finally, 'Pressurized free surface' boundary conditions must be applied on the top of each oscillating water column. An independent condition must be applied to each column as shown in the figure below.

Boundary conditions ► P Free surface



Pressurized free surface boundary conditions applied to the top of the oscillating water columns. Note that a 'P Free surface' conditions is applied separately to each one of the columns

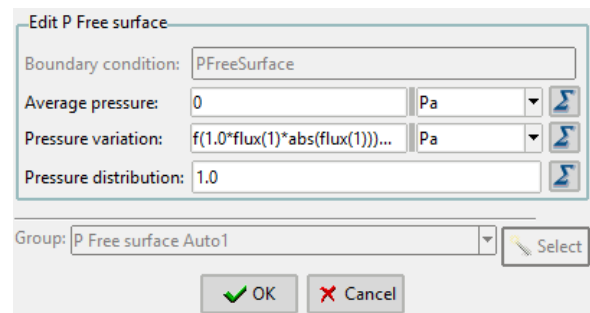
'P Free surface' boundary condition applies to each node of its corresponding surface patch a pressure value given by:

$$P = (P_{\text{aver}} + P_{\text{var}}(t)) \cdot P_{\text{dist}}(x,y)$$

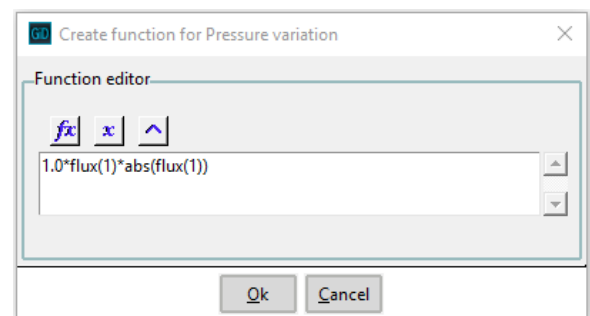
where P is the pressure to be applied, P_{ave} is an average pressure value which is constant in time and uniform in space, $P_{\text{var}}(t)$ is a pressure variable in time but uniform in space, and $P_{\text{dist}}(x,y)$ is a distribution function constant in time and uniform in the x - y plane. The pressure enforced by the user through this condition may account for any conceivable effect. Nevertheless, the basic expression in the case of an OWC device will be of the form:

$$P = C \cdot \text{flux} \cdot \text{abs}(\text{flux})$$

where C is a constant that characterizes the turbine to which the chamber is connected, and flux is an internal variable calculated by SeaFEM that corresponds to the air flux induced by the oscillating water column. For the present analysis the constant C is equal to $14.7E6$, the average pressure is zero and the pressure distribution function takes a uniform value equal to 1.0. The next two figures show how to fill the forms that define the 'P Free surface' boundary condition.



Editor window for the P Free Surface boundary condition



Function editor window showing the expression of the pressure variation function enforced on the top free surface of the first oscillating water column. Equivalent expressions must be used for the rest of the chambers just changing the chamber identification index

7.8. Body external loads

In addition to the environment wave loads, the dynamic behavior of a floating OWC device is also influenced by the forces and moments exerted over the structure due to the overpressure that exists inside of the column chambers. At each time step, SeaFEM evaluates a first order approximation for such forces and moments. These magnitudes are internally accessible during the calculation and can be applied as external forces on the structure to take into account their influence on the dynamics of the floating body. In order to solve the dynamic problem, all external forces and moments must be referred to

the center of gravity of the body. Hence, the total external force and moment acting on the floating structure are given as a first approximation by the following expressions:

$$F = \sum_i f_i$$

$$M = \sum_i m_i + \sum_i r_i \times f_i$$

where $i = 1, \dots, n$ refers to each one of the existing chambers and r_i refers to the position vector from the center of gravity to the center of the corresponding column's free surface. Hence, in the present case the components of the external forces and moments acting on the OWC device read as follows:

$$F_x = f_{1x} + f_{2x} + f_{3x}$$

$$F_y = f_{1y} + f_{2y} + f_{3y}$$

$$F_z = f_{1z} + f_{2z} + f_{3z}$$

$$M_x = M_{1x} + M_{2x} + M_{3x} + y_1 \cdot f_{1z} - z_1 \cdot f_{1y} + y_2 \cdot f_{2z} - z_2 \cdot f_{2y} + y_3 \cdot f_{3z} - z_3 \cdot f_{3y}$$

$$M_y = M_{1y} + M_{2y} + M_{3y} + z_1 \cdot f_{1x} - x_1 \cdot f_{1z} + z_2 \cdot f_{2x} - x_2 \cdot f_{2z} + z_3 \cdot f_{3x} - x_3 \cdot f_{3z}$$

$$M_z = M_{1z} + M_{2z} + M_{3z} + x_1 \cdot f_{1y} - y_1 \cdot f_{1x} + x_2 \cdot f_{2y} - y_2 \cdot f_{2x} + x_3 \cdot f_{3y} - y_3 \cdot f_{3x}$$

Using the SeaFEM's syntax and the actual values of r_i for each column, the expressions to be entered in the 'External loads' fields of the 'Body data' section in the data tree are as follows:

$$F_x = Fx_Pfs(1) + Fx_Pfs(2) + Fx_Pfs(3)$$

$$F_y = Fy_Pfs(1) + Fy_Pfs(2) + Fy_Pfs(3)$$

$$F_z = Fz_Pfs(1) + Fz_Pfs(2) + Fz_Pfs(3)$$

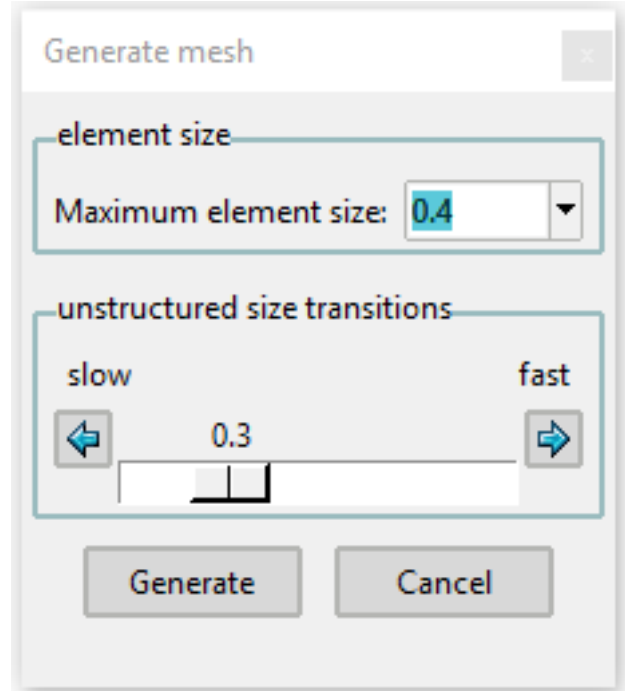
$$M_x = Mx_Pfs(1) + Mx_Pfs(2) + Mx_Pfs(3) - 0.052 * Fy_Pfs(1) - 0.052 * Fy_Pfs(2) - 0.052 * Fy_Pfs(3)$$

$$M_y = My_Pfs(1) + My_Pfs(2) + My_Pfs(3) + 0.052 * Fx_Pfs(1) + 0.536 * Fz_Pfs(1) + 0.052 * Fx_Pfs(2) + 0.052 * Fx_Pfs(3) - 0.536 * Fz_Pfs(3)$$

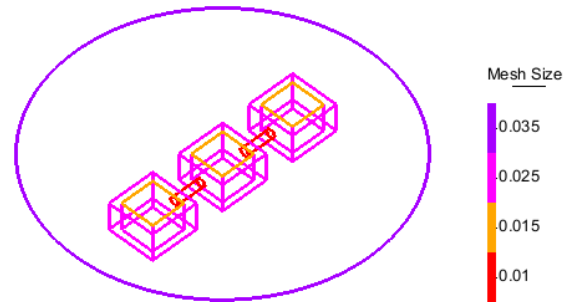
$$M_z = Mz_Pfs(1) + Mz_Pfs(2) + Mz_Pfs(3) - 0.536 * Fy_Pfs(1) + 0.536 * Fy_Pfs(3)$$

7.9. Mesh

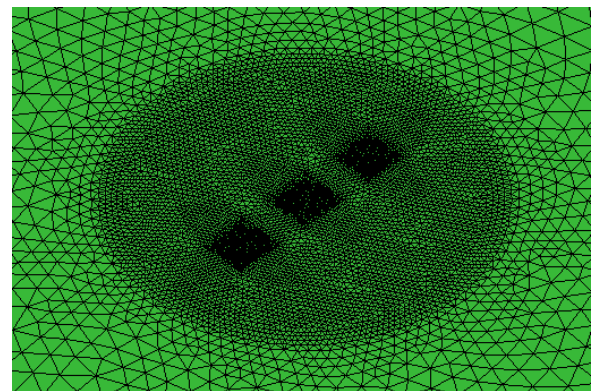
The computational domain for the present analysis corresponds to a control volume scaled in accordance to the model of the OWC device. Element sizes have been applied to the body of the OWC device and to the near field region in order to control the resolution of the finite element mesh which finally consists of 51,067 nodes and 281,466 linear tetrahedra.



Maximum element size and unstructured mesh size transition used during the generation of the finite element mesh for the analysis of the floating OWC device under consideration in the present tutorial



Element sizes assigned to the body of the OWC device and to the perimeter of the near field region



Detailed view of the finite element mesh used in the simulation of the seakeeping behavior of an OWC floating device

7.10. Results

This final section shows some of the results obtained after executing the time domain solver.

After meshing the model we can proceed to execute the solver

for calculation. This is done by executing the following menu sequence:

Calculate ► Calculate

or by pressing the icon shown below which is located within the SeaFEM toolbar:



Once the calculation has started, information on how it's doing can be found by going to the following menu's option:

Calculate ► View process info...

or by pressing the following icon in the toolbar:



If the execution finishes ok, a message will be prompted that will also ask if you wish to go to the postprocessor. Postprocess can be also accessed at any time by using the main menu option:

Postprocess ► Start

or by pressing the following icon:



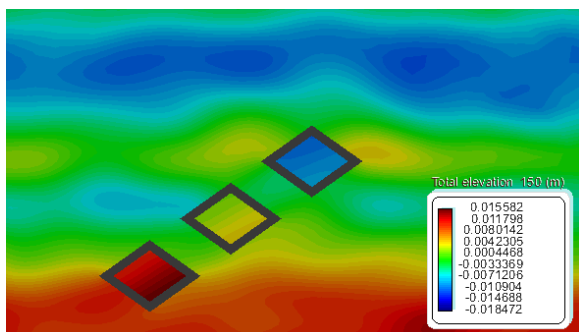
When using the default (binary2) postprocess format, time evolution graphical results can be visualized by using the following option of the main postprocess menu:

Postprocess ► SeaFEM graphs

On the contrary, if using the traditional post, go to:

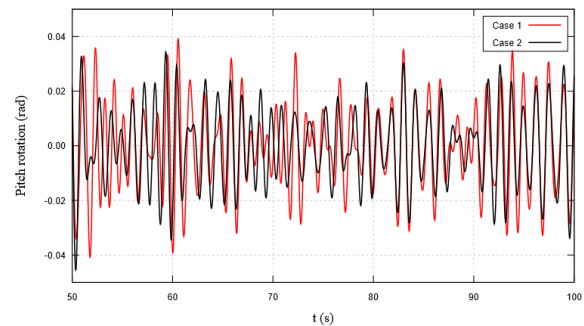
View Results ► SeaFEM Graphs

In the following figure, it can be observed the total wave elevation inside of each one of the three chambers of the OWC device as well as in the near field region around the floating structure.



Total wave elevation in the region close to the OWC floating structure and inside of the three chambers of the device.

Pitch rotation of the floating OWC device is compared in two distinct models in the following graph. In the first case, external forces due to the pressure inside of the chambers are not taken into account. In the second case, these external forces are added to the dynamics of the floating structure to assess their influence on the global movement of the device.



Pitch rotation of the floating OWC device. In Case 1 the forces due to the internal pressure of the chambers is not taken into account. In Case 2 these external forces are added to the dynamics of the device to assess their actual influence on the global behavior of the floating structure.