Introduction

1 Introduction

This report shows an example of the steps to take for setting up a Fluid Structure Interaction analysis. The interaction will be between the codes **SeaKeeping** and **RamSeries,** both developed and commercialized by Compass IS www.compassis.com

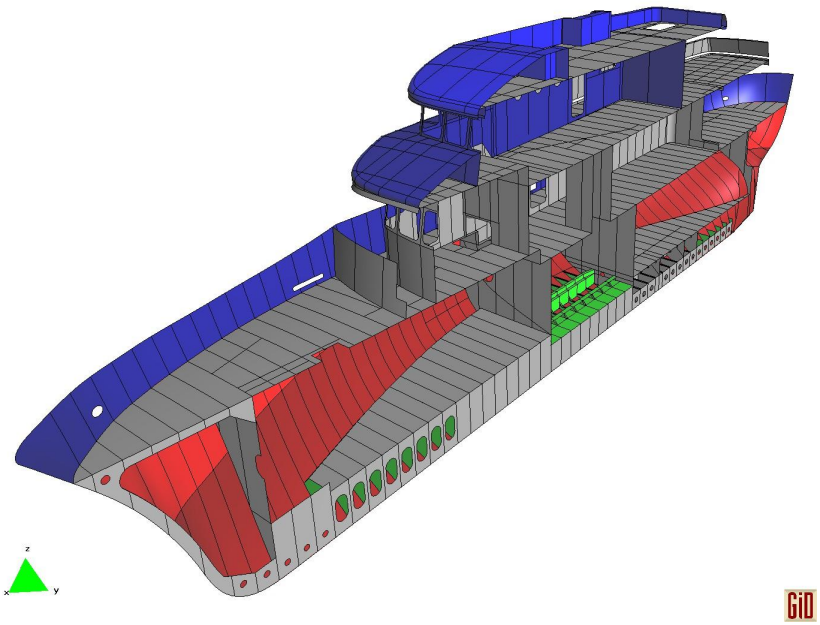
In this particular case, the interaction will be performed via a results file, obtained from a previous sea keeping analysis. These results - *Total Pressures* - will be the load for the structural analysis.

Both the SeaKeeping and RamSeries FSI analysis models will be fully explained in this report.

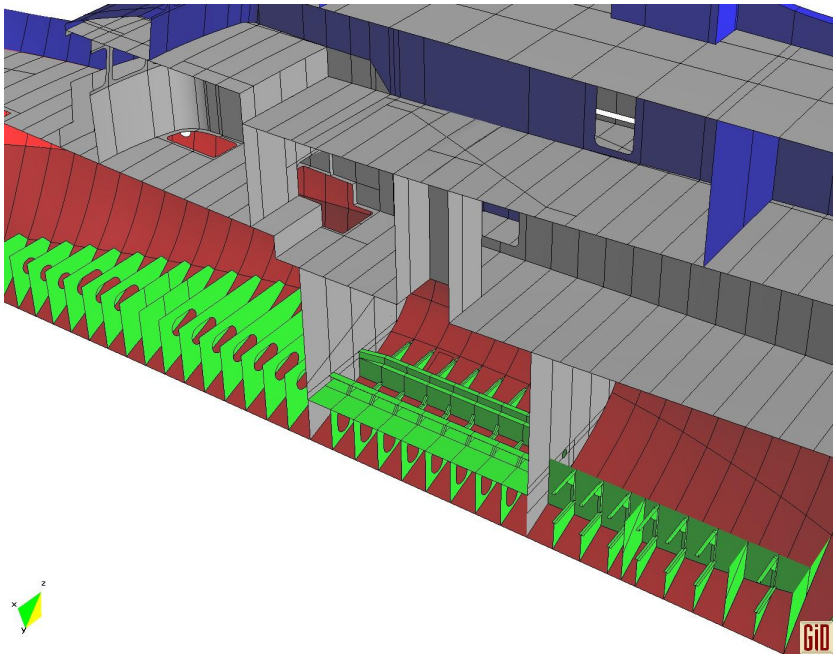
The model used for the example is a ship, provided by *Isonaval SA*.

2 CAD Model

The complete CAD model, which use has been possible thanks to the courtesy of *Isonaval SA*, is shown in the following images:



CAD Model



The main dimensions of the model are the following:

- L_{pp} = 26 m
- B = 6 m
- T = 2.4 m

3 SeaKeeping Model

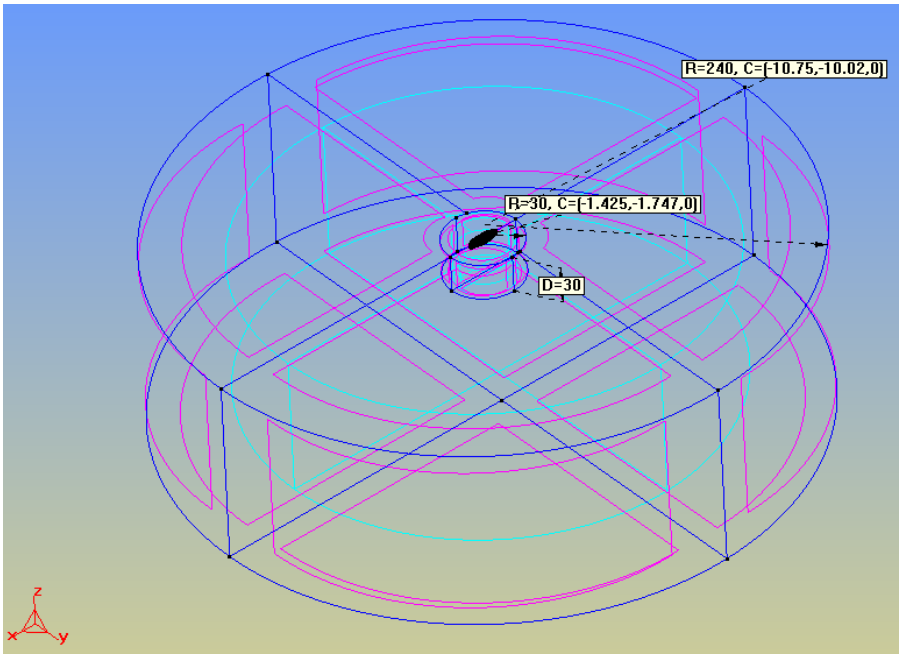
A sea-keeping analysis has been performed first of all, in order to obtain the time-domain pressure distribution over the hull, due to the waves´dynamics.

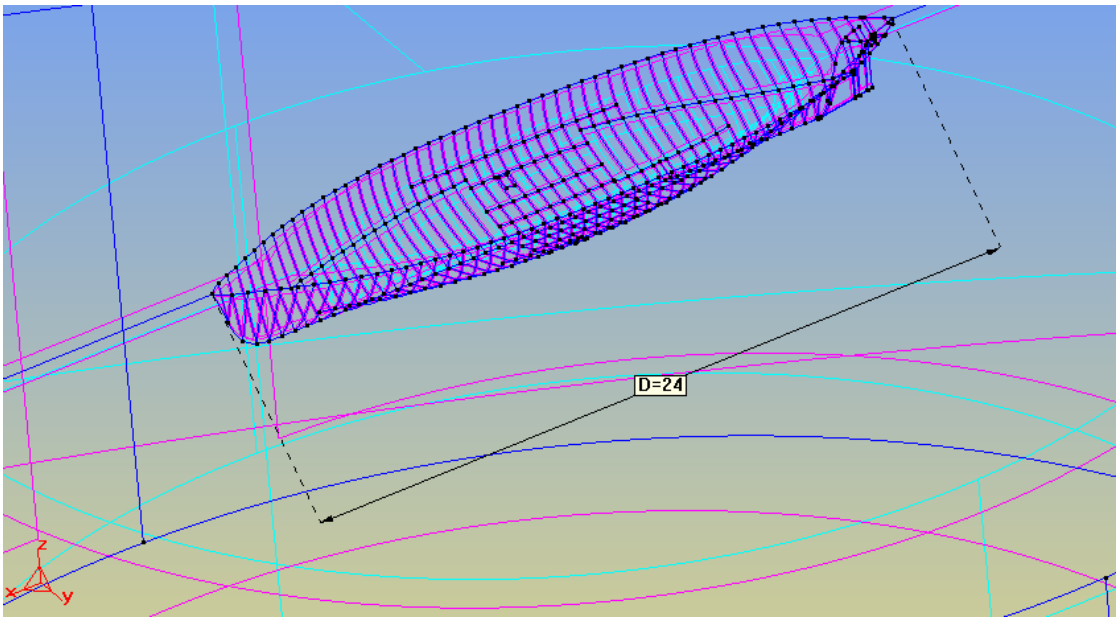
This analysis has been performed with the code **SeaKeeping** developed by Borja Serván (*CIMNE/ Compass IS*), and included in the Multi-Physiscs environment interface **CompassFEM**.

3.1 Domain Setting

The model for SeaKeeping requires an special configuration of the analysis domain, in order to optimize the calculation. The following images show the main dimensions of the domain:

Domain Setting

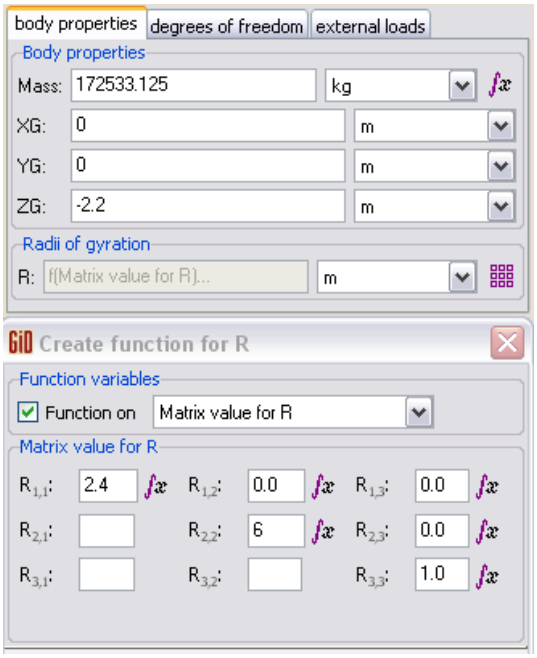




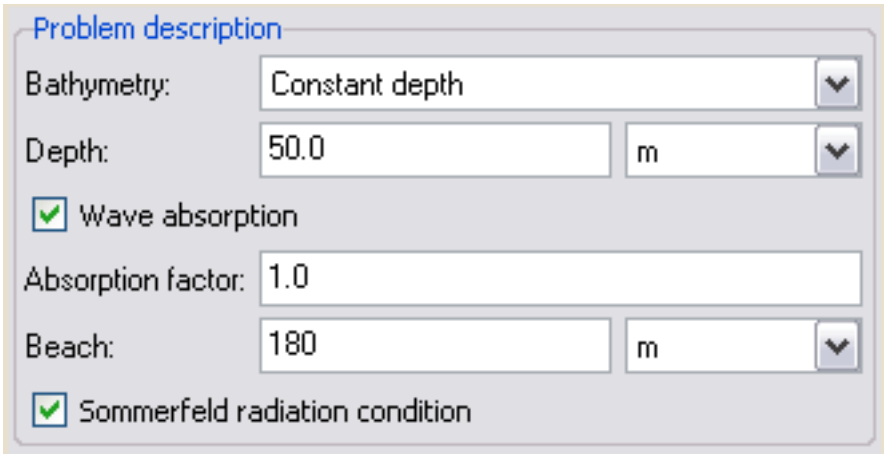
3.2 Ship Data

The data corresponding to the ship condition, and depth, are the following:

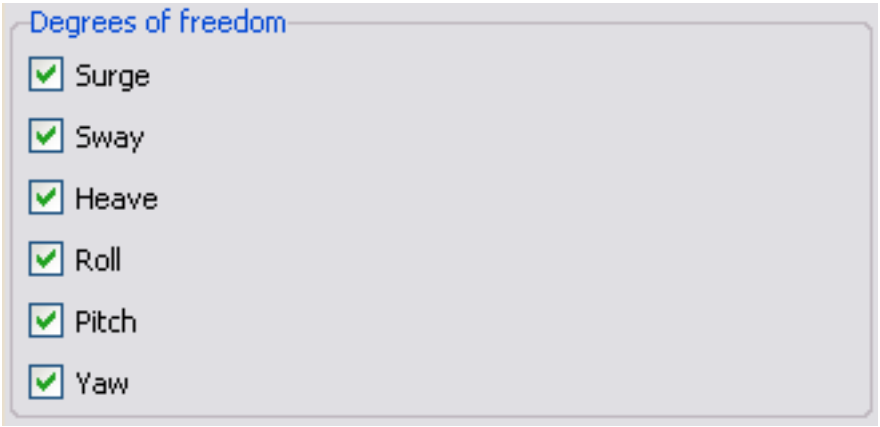
Ship Data



There are some parameters to define, which are related to the **Bathymetry**:



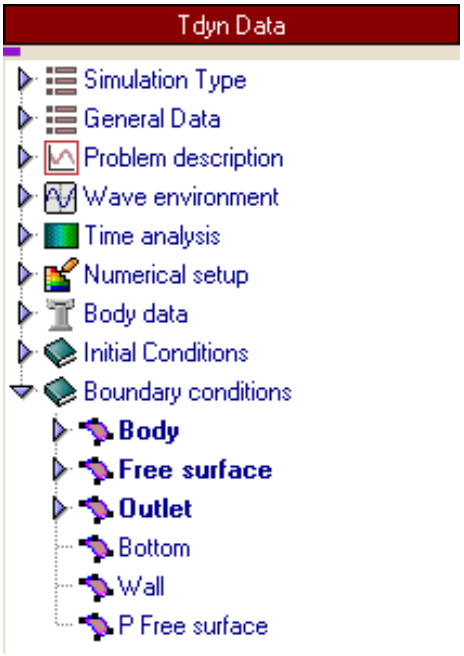
All **degrees of freedom** have been released:

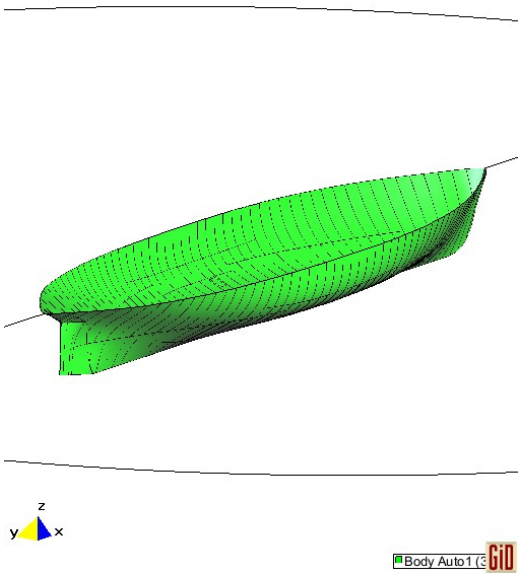


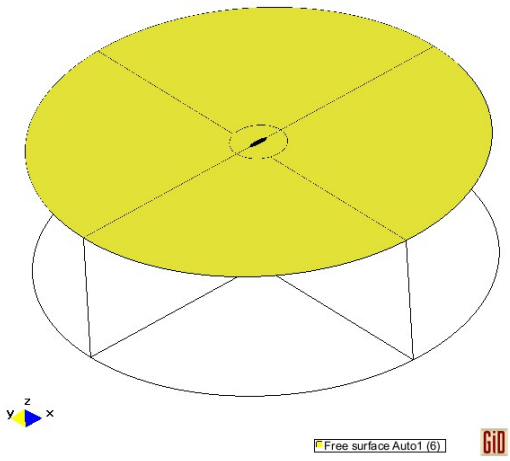
Boundary Conditions

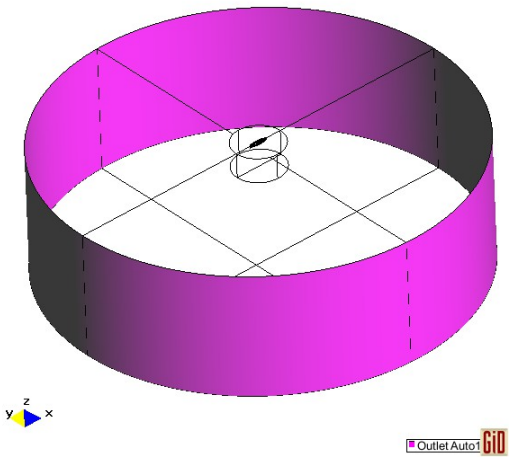
3.3 Boundary Conditions

The basic Boundary Conditions to apply, are the following:







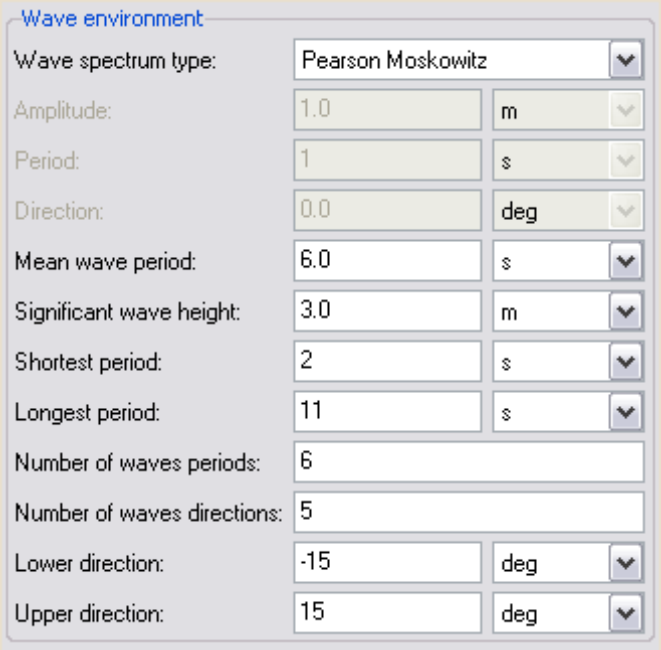


From up to down, and right to left, the images show the boundary conditions applied: **Body**, **FreeSurface**, and **Outlet**.

3.4 Wave Environment

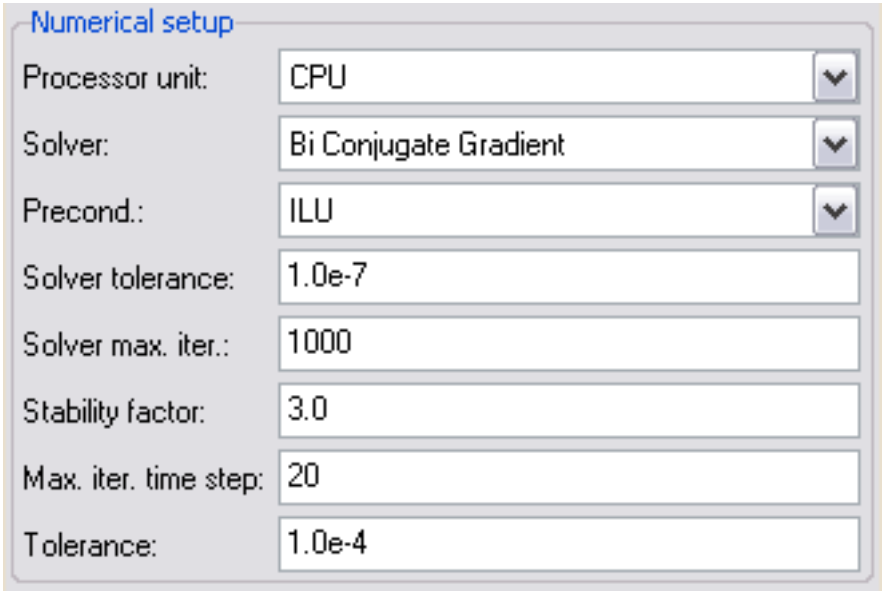
The Wave environment has been set to the following values:

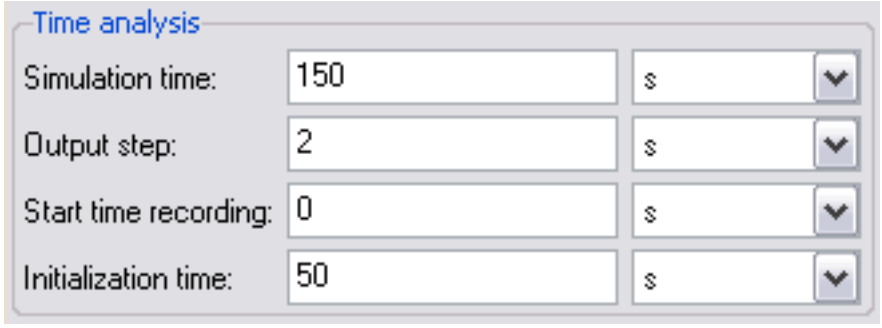
Wave Environment



3.5 Numerical Data

The **numerical data** and **simulation parameters**, are the following:

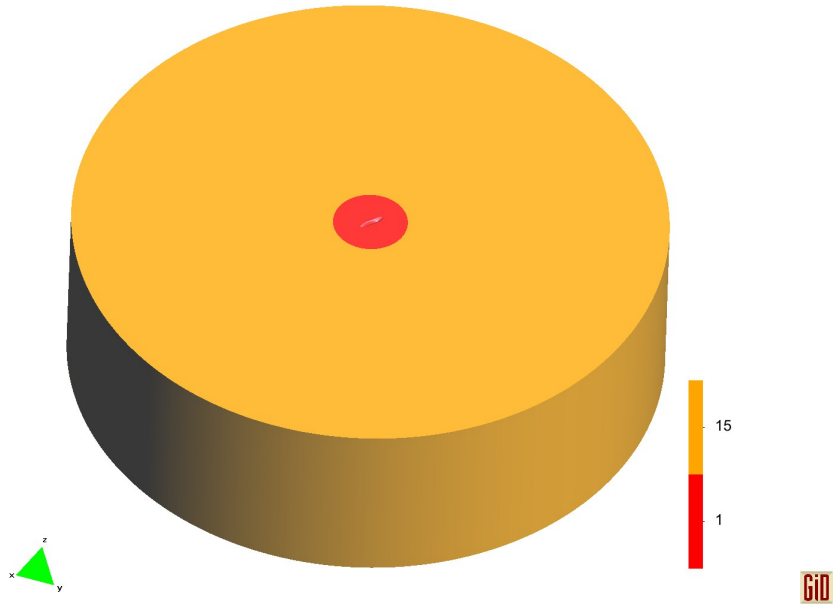


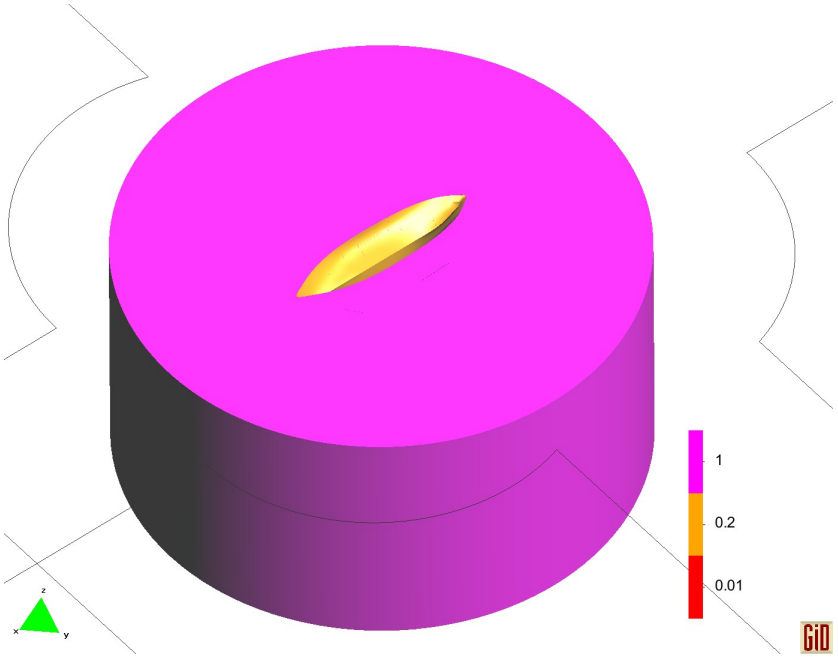


Mesh

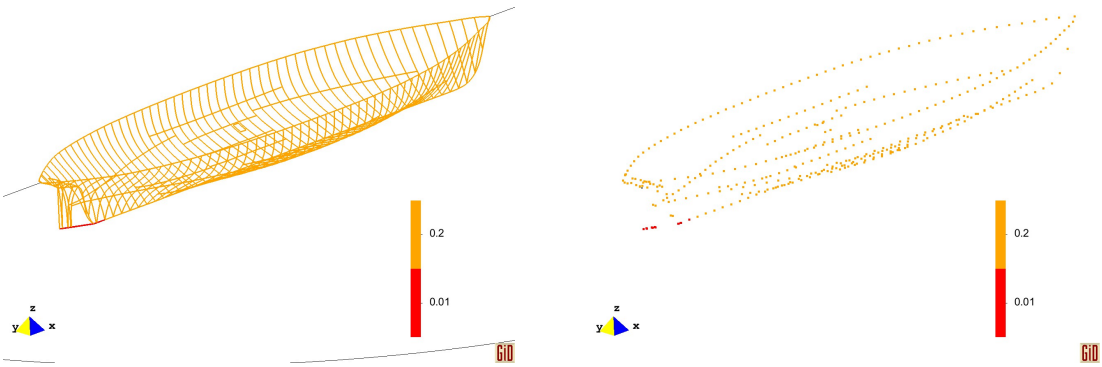
3.6 Mesh

Some care must be taken when assigning the mesh sizes:



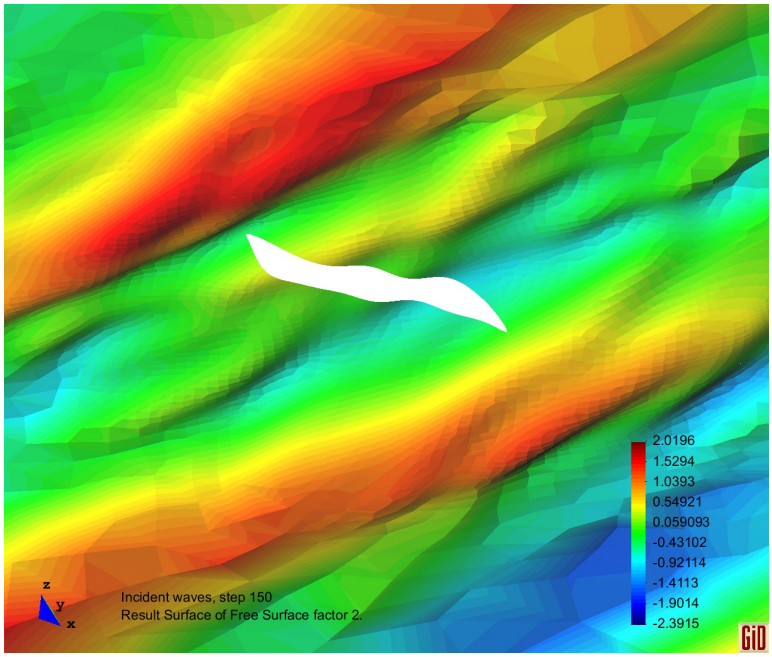


Mesh



3.7 Results

There are different types of interesting results, which are worth being analyzed.

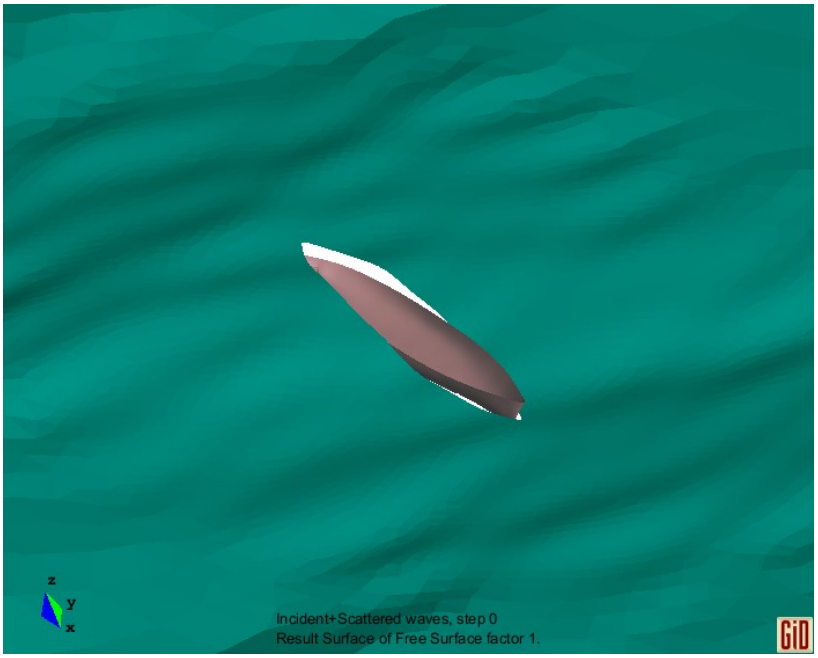


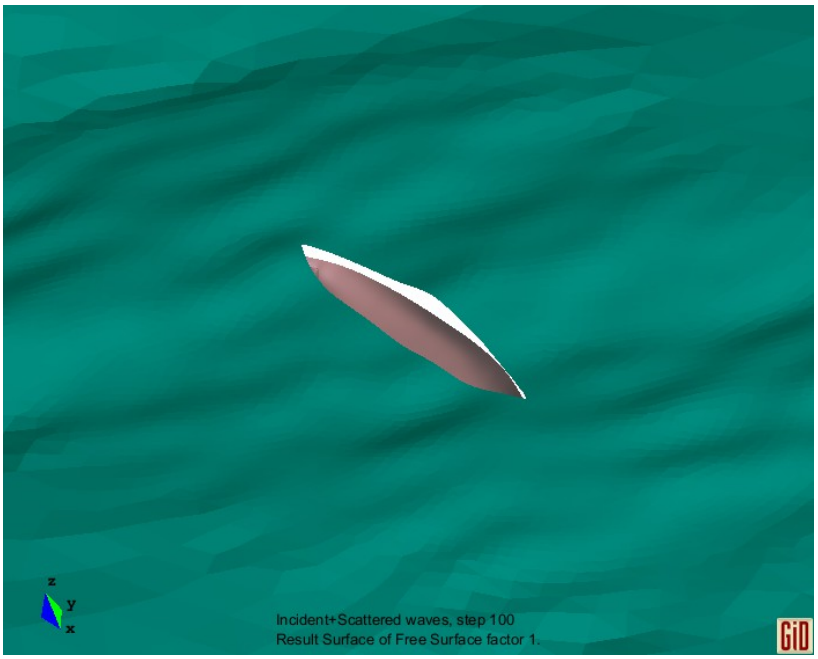
Free Surface (Wave Elevations [m])

Some stages of the analysis are shown below:

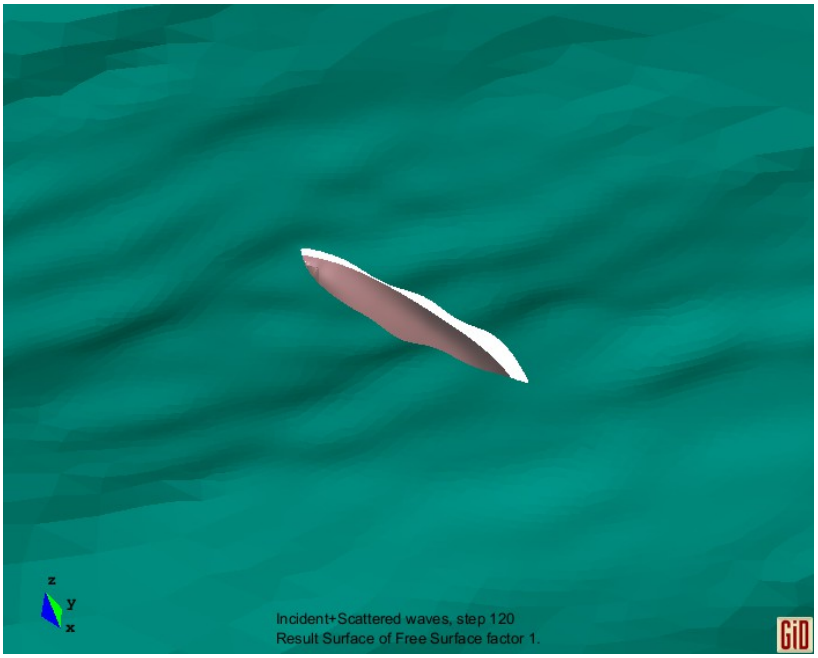
- Free surface

Results

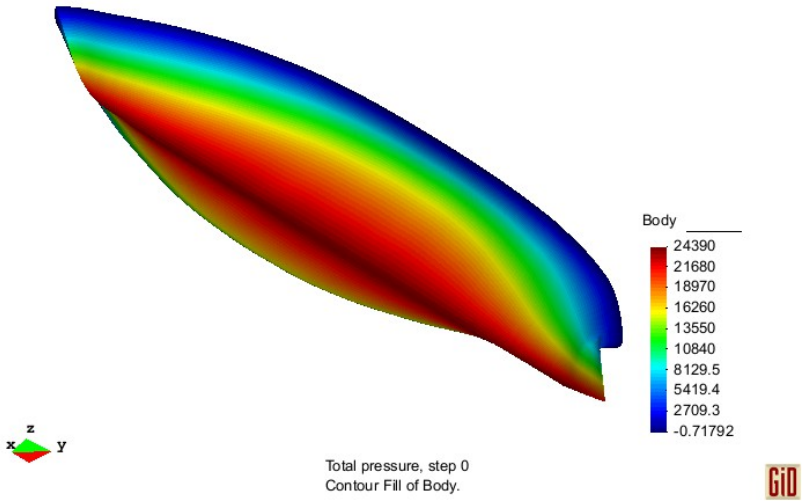

z
y
x
Incident+Scattered waves, step 0
Result Surface of Free Surface factor 1.
GiD


z
y
x
Incident+Scattered waves, step 100
Result Surface of Free Surface factor 1.
GiD

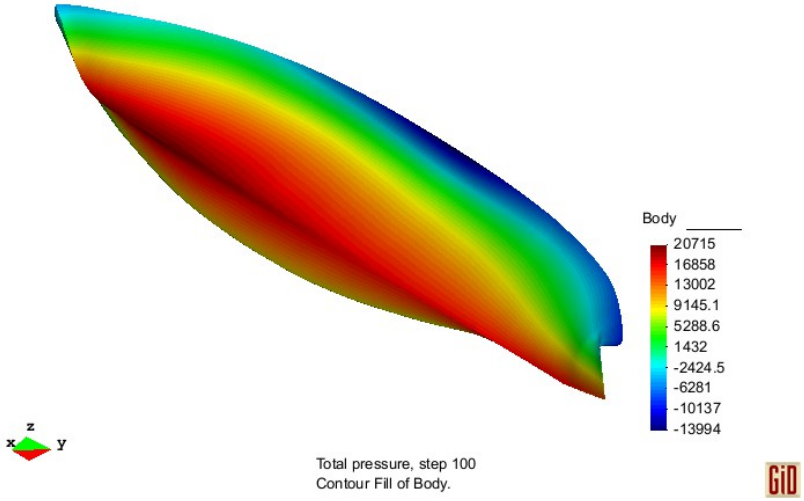
Results

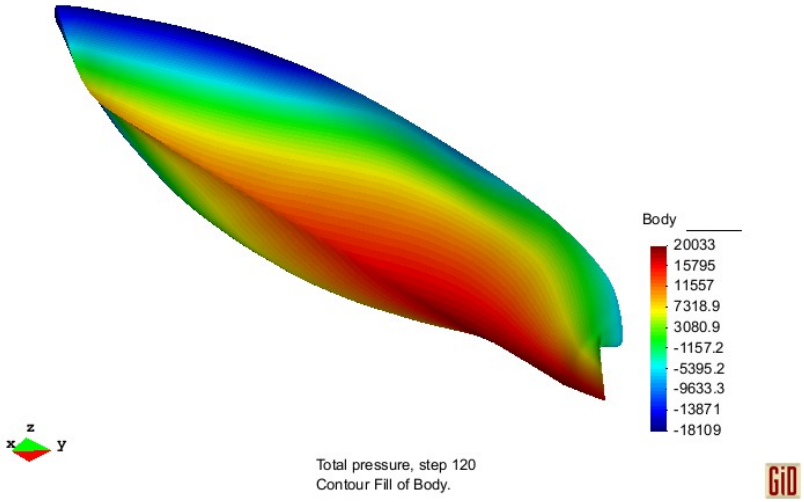


- Total Pressures



Results

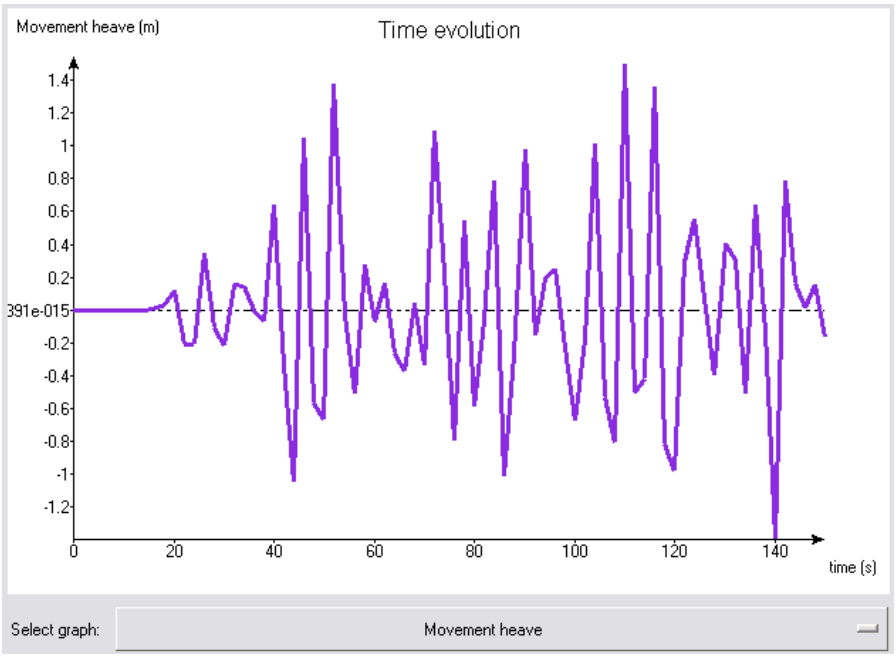


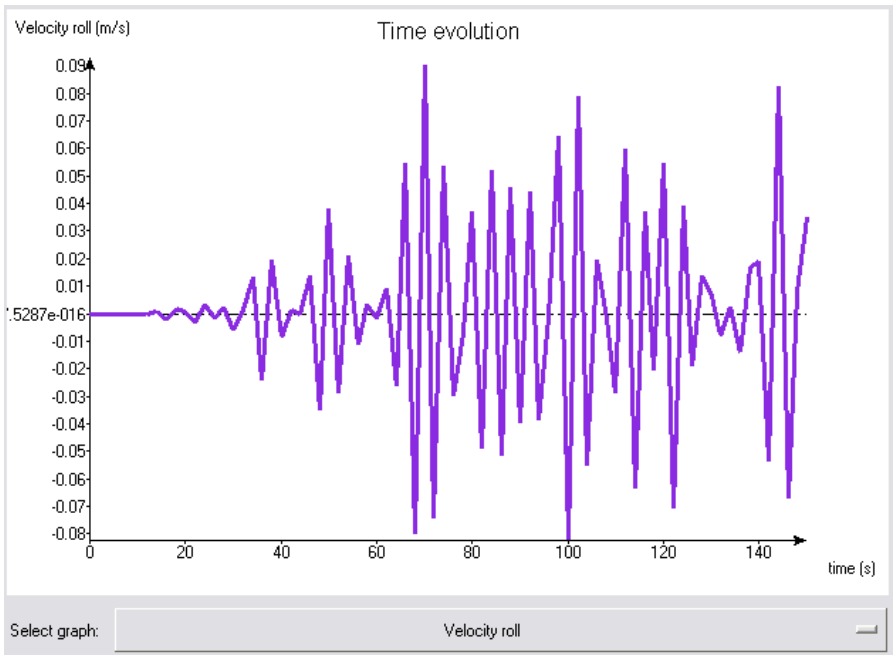


This **Total Pressure** result is the one used for performing the **FSI analysis**.

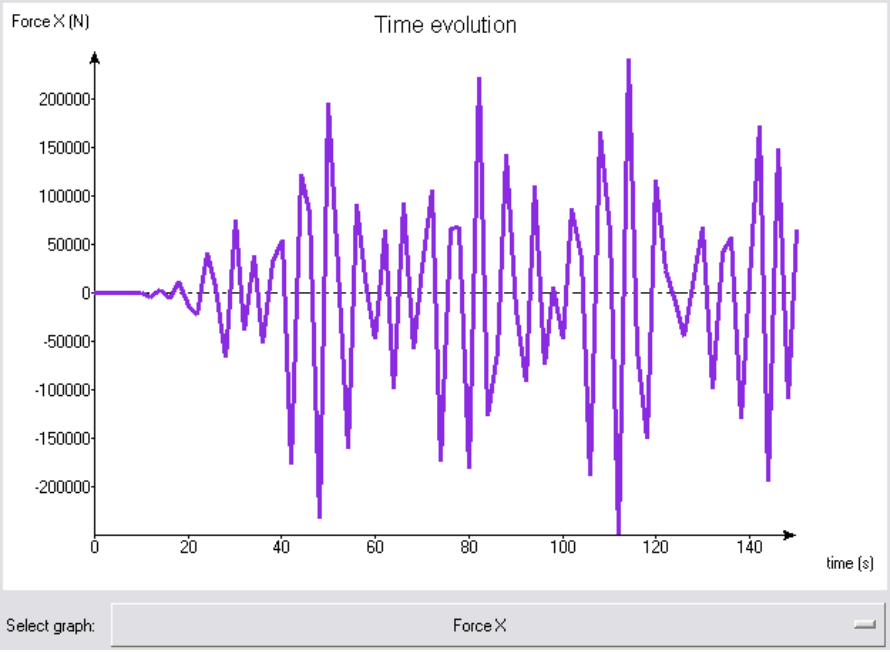
Also, **plots** for the forces hull, velocity, and movements of the hull can be displayed. Some of them are showed in the following images, as example:

Results


Movement heave (m)
Time evolution
1.4
1.2
1
0.8
0.6
0.4
0.2
391e-015
-0.2
-0.4
-0.6
-0.8
-1
-1.2
0
20
40
60
80
100
120
140
time (s)
Select graph:
Movement heave


Velocity roll (m/s)
Time evolution
0.09
0.08
0.07
0.06
0.05
0.04
0.03
0.02
0.01
'.5287e-016
-0.01
-0.02
-0.03
-0.04
-0.05
-0.06
-0.07
-0.08
0
20
40
60
80
100
120
140
time (s)
Select graph:
Velocity roll

Results



4 FSI Model

A Fluid-Structure Interaction (FSI) calculation has been performed with the FEM structural analysis code RamSeries (also developed and commercialized by *Compass IS*).

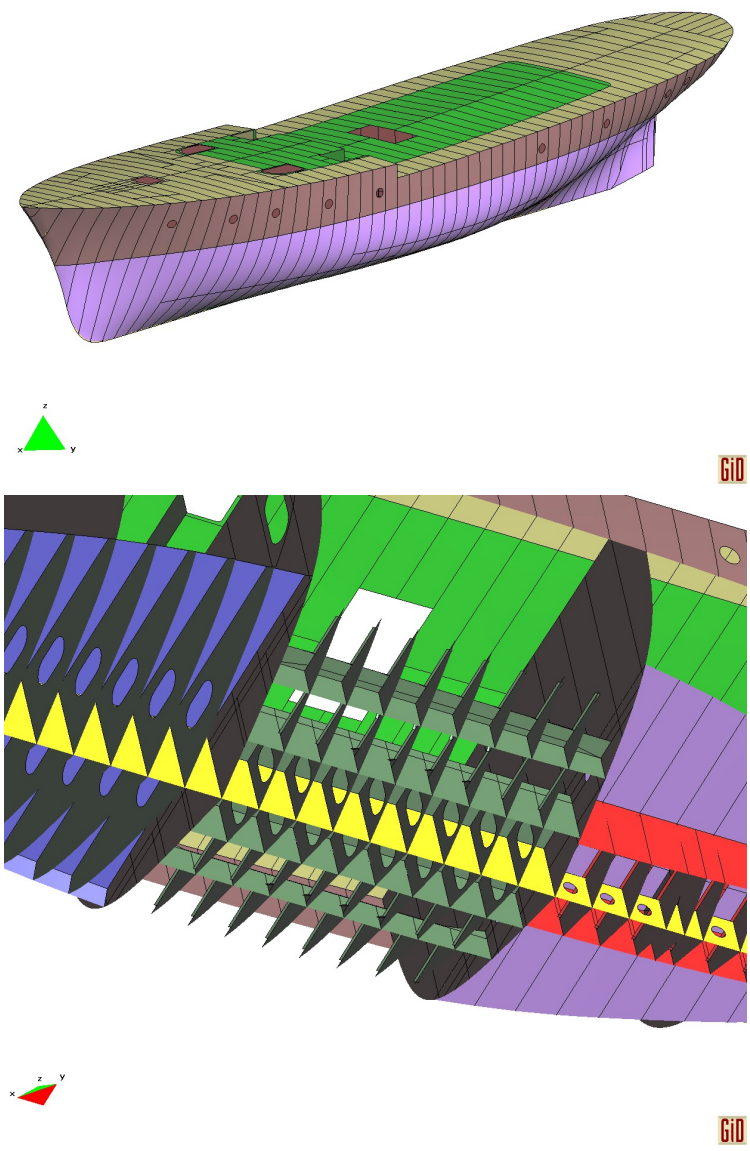
In this analysis, the pressure distribution obtained from the previous calculation (SeaKeeping) has been used as a dynamic load.

This can be performed thanks to the interpolation algorythms implemented by Compass, which allow to interchange information (pressures, deformations,..) between both the fluid (SeaKeeping, in this case) and the structural finite element meshes.

4.1 Domain Setting

Due to the size and complexity of the hull structure model, only the lower part of it will be included for the analysis.

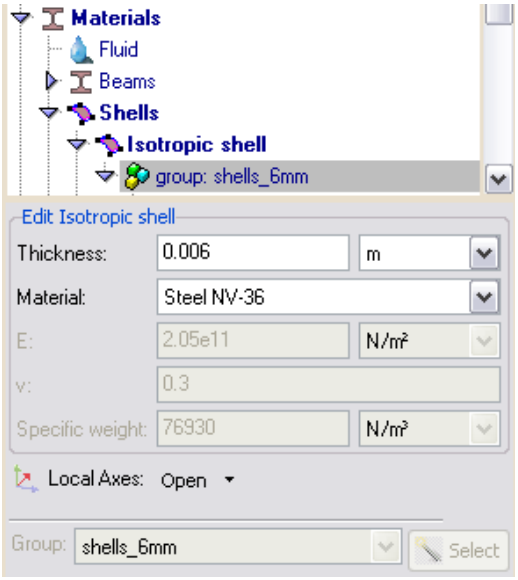
Domain Setting

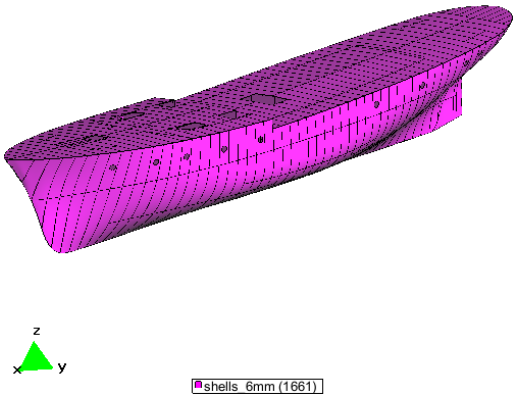


4.2 Materials

For simplicity, the same material has been used for the whole model.

Materials





4.3 Boundary Conditions

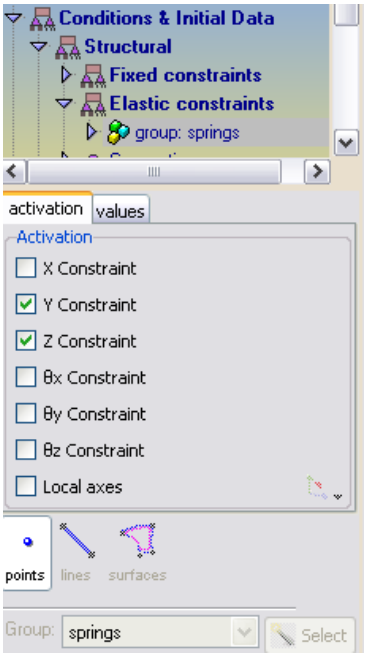
Elastic constraints have been used in this case, as boundary conditions.

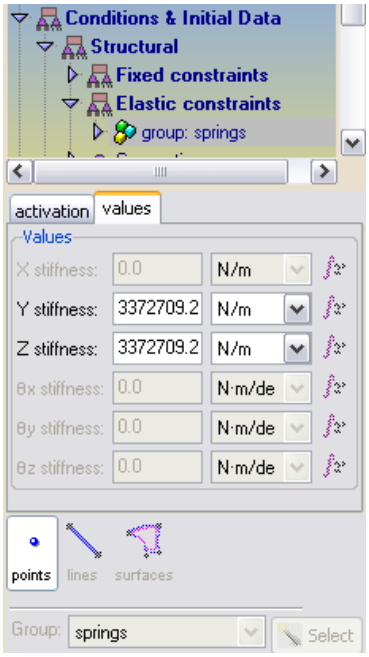
This is somehow artificial but, since it does not introduce excesive ficticious loads on the structure, it has been considered as a reasonable solution.

The spring stiffness has been calculated to allow a maximum displacement of $\delta=1.0\ m$ (in both vertical and laterall directions), for a force of $F=3372709.2\ N$, which is three times the displacement ($F=3*\Delta$) of the ship for the load situation considered.

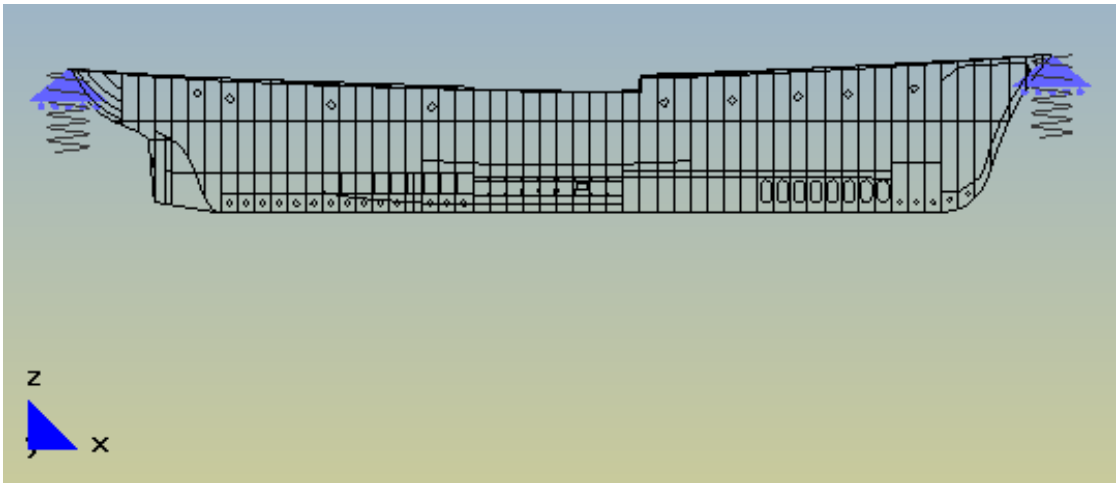
Therefore:

$$\kappa = F/\delta = 3372709.2\ N/m$$



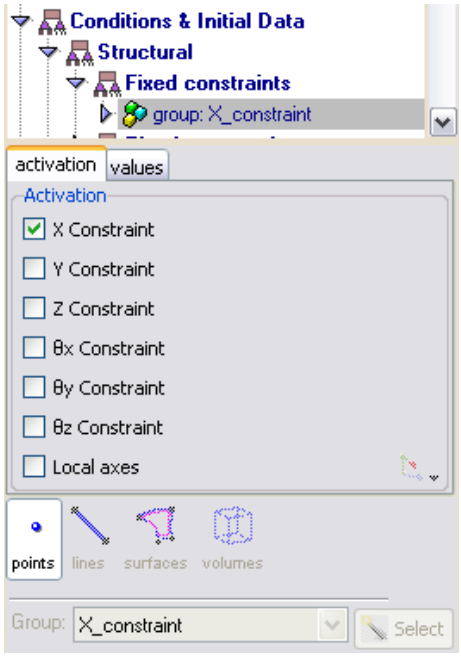


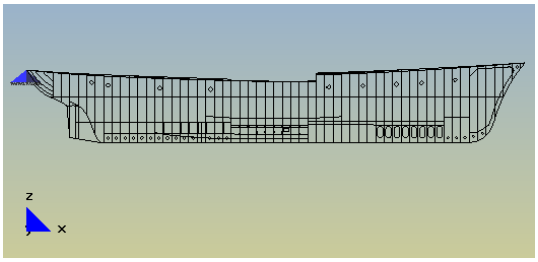
Boundary Conditions



The condition has been applied to two points, one in the bow and the other in the stern, being both in the longitudinal mid-plane of the ship.

Also, a fixed constraint has been applied at the stern, limiting the movement along the longitudinal direction.



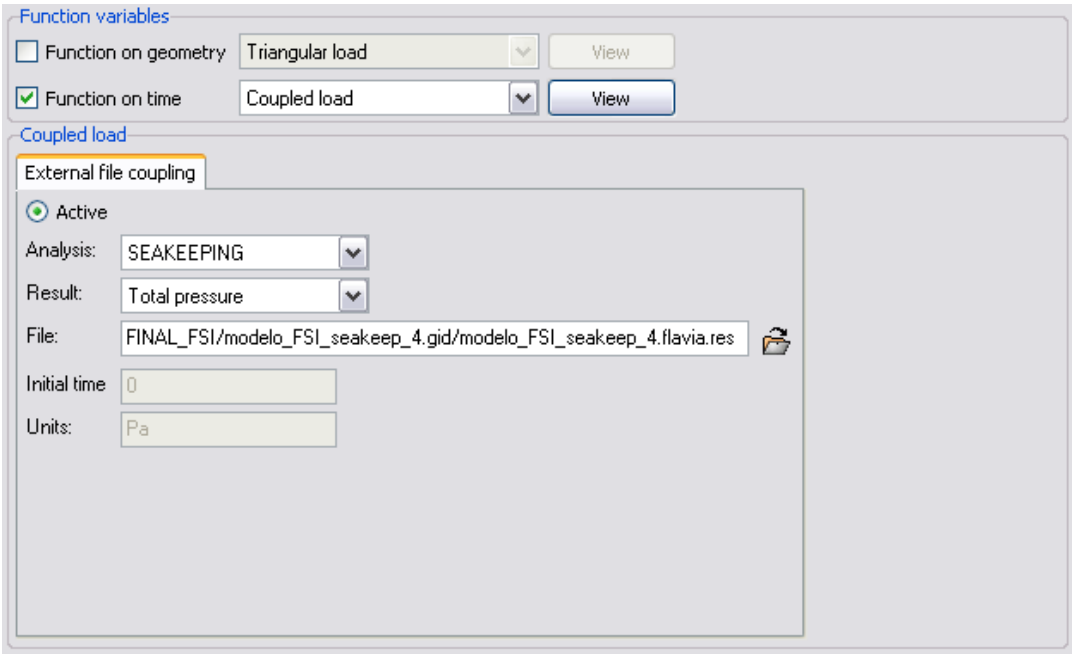


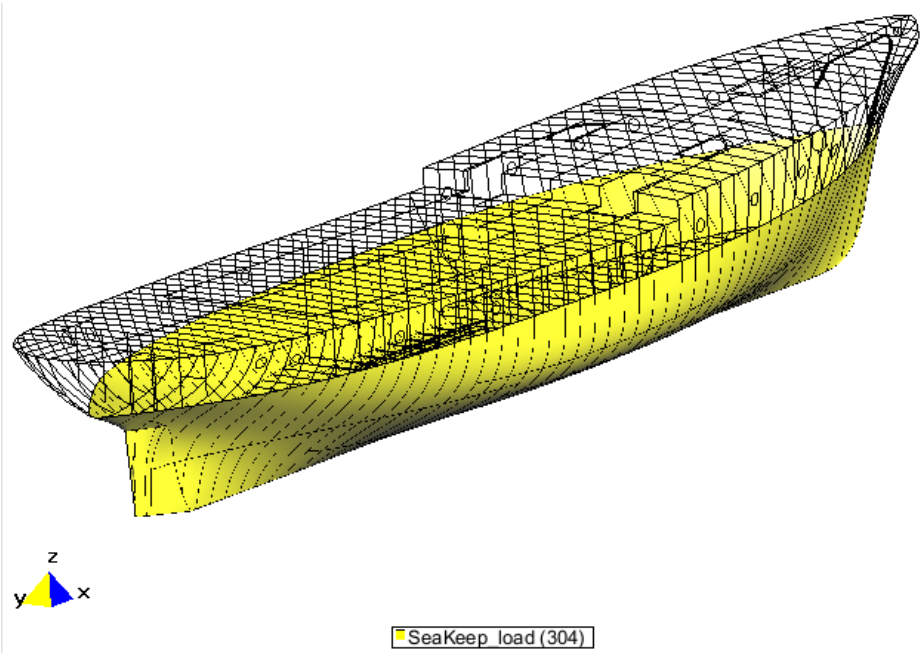
4.4 Applied Loads

FSI Load:

This load is imposed to the hull surface below the floating line, which is the corresponding part of the geometry where pressure results coming from SeaKeeping code are available. A mesh interpolation is internally performed, for transferring the results from the SeaKeeping model mesh, to the structure mesh.

Applied Loads





Self Weight:

A load has been applied in order to simulate the weight corresponding to the depth for wich the SeaKeeping analysis has been performed. The SeaKeeping code outputs the corresponding displacement, and a distributed weight over the structure geometry has been calculated and applied:

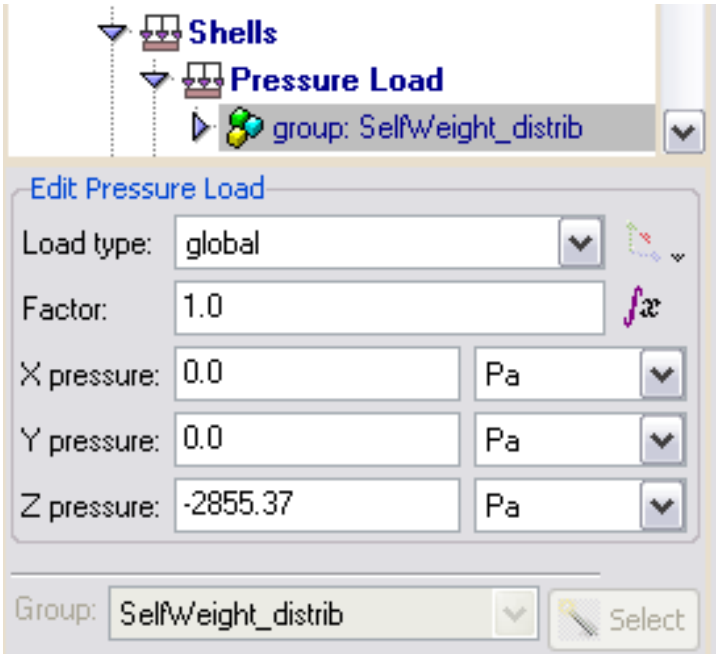
Displacement: Δ **= 1686354.6 N**

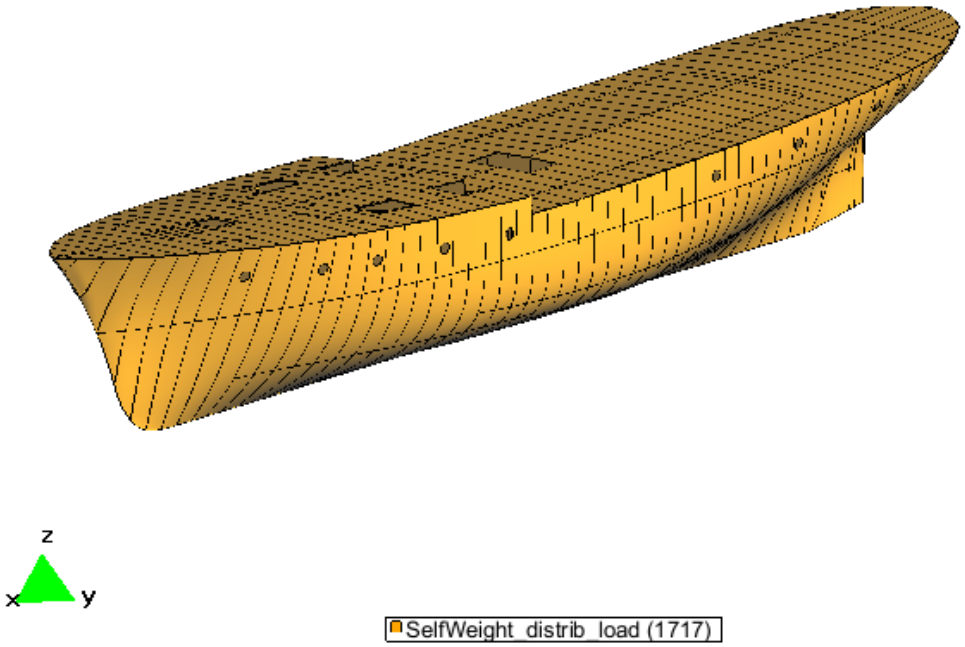
Total model area: **A = 590.59 m^2**

Load: **q = Δ/A = 2855.37 N/m^2**

Therefore, the applied load is, finally:

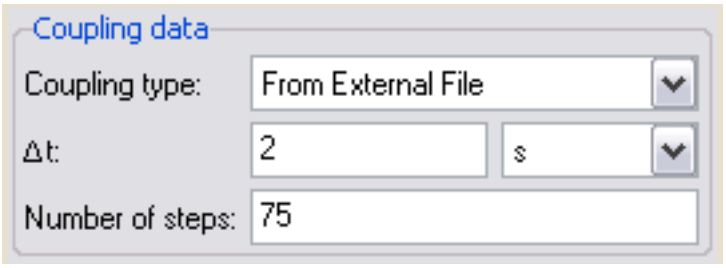
Applied Loads





4.5 FSI Settings

In order to being able to perform the coupled analysis, some specific data must be inserted:

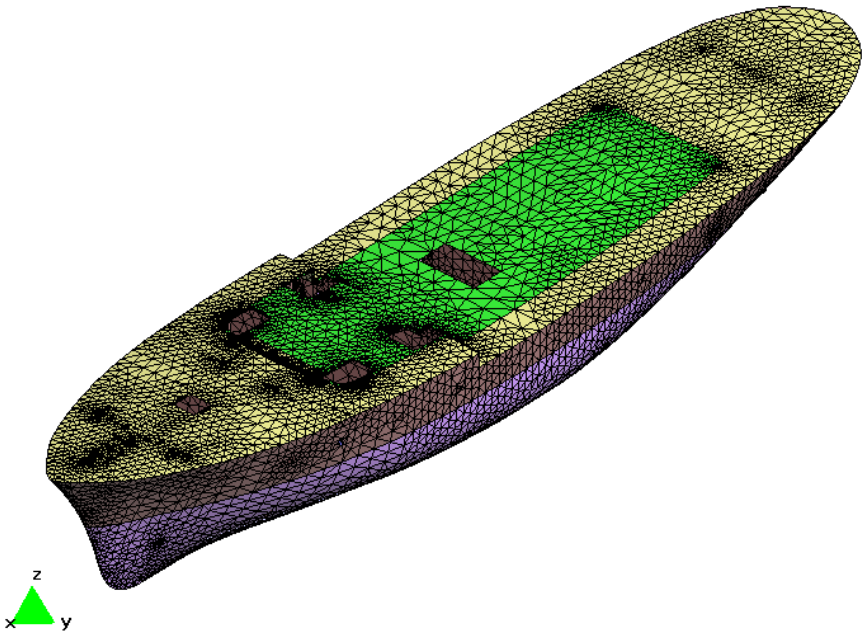


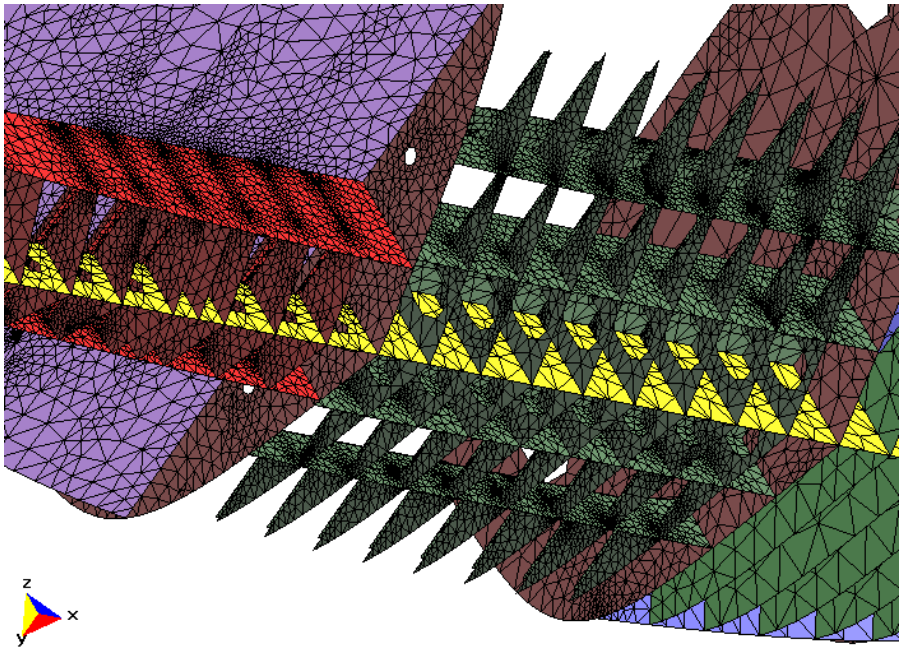
Mesh

4.6 Mesh

An unstructured, triangular mesh has been generated, just indicating an overall maximum size. This size is **0.6 m**, with a mesh transition of **0.6**.

The total number of nodes is **25009**, and the number of elements is **52771**.



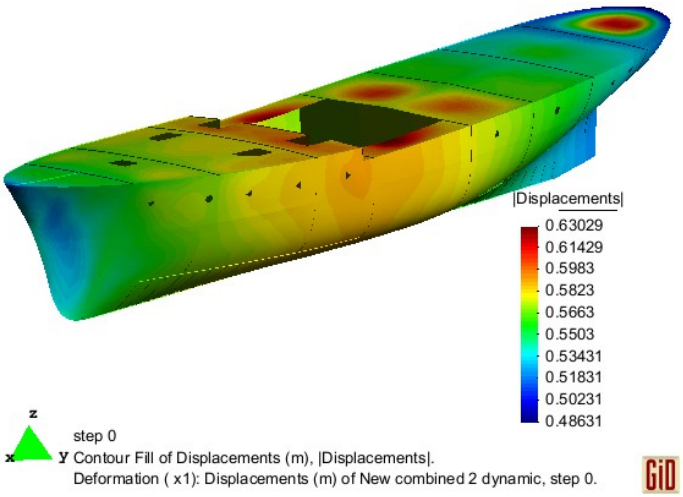


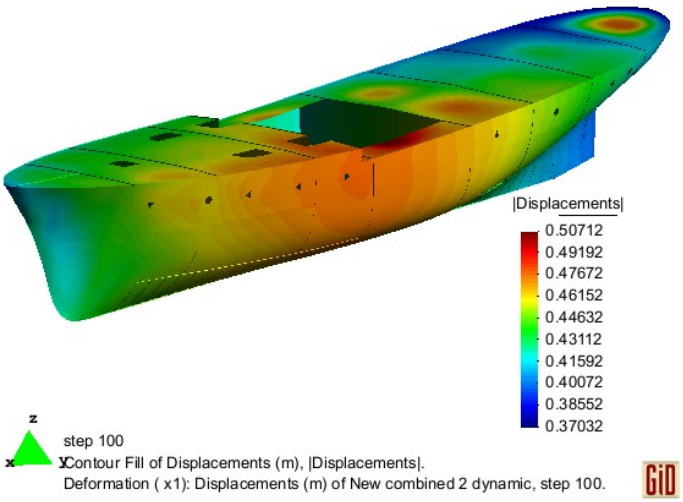
4.7 Results

Displacements

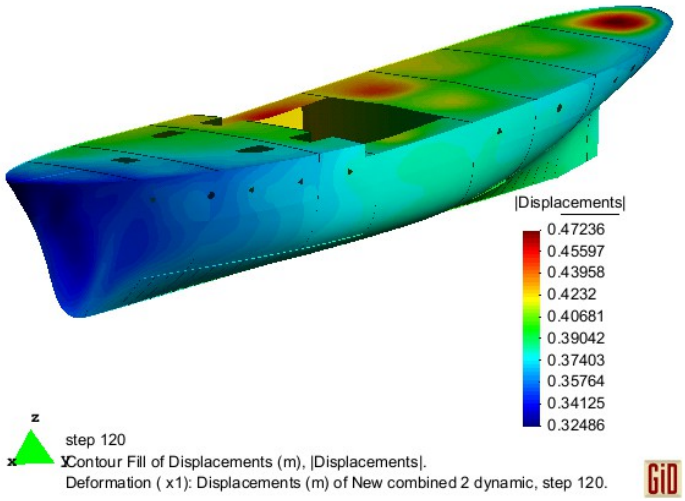
Diferent stages of the analysis are shown below:

Results


|Displacements|
0.63029
0.61429
0.5983
0.5823
0.5663
0.5503
0.53431
0.51831
0.50231
0.48631
z
x
y
step 0
Contour Fill of Displacements (m), |Displacements|.
Deformation (x1): Displacements (m) of New combined 2 dynamic, step 0.

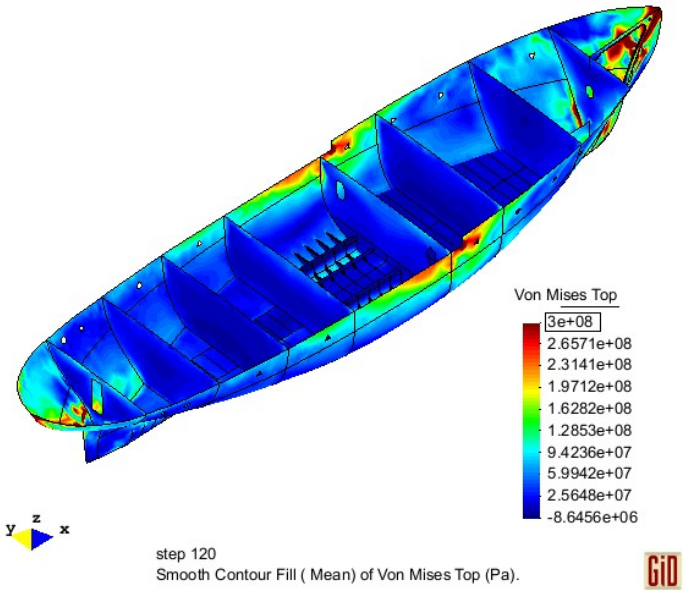

|Displacements|
0.50712
0.49192
0.47672
0.46152
0.44632
0.43112
0.41592
0.40072
0.38552
0.37032
z
x
y
step 100
Contour Fill of Displacements (m), |Displacements|.
Deformation (x1): Displacements (m) of New combined 2 dynamic, step 100.

Results

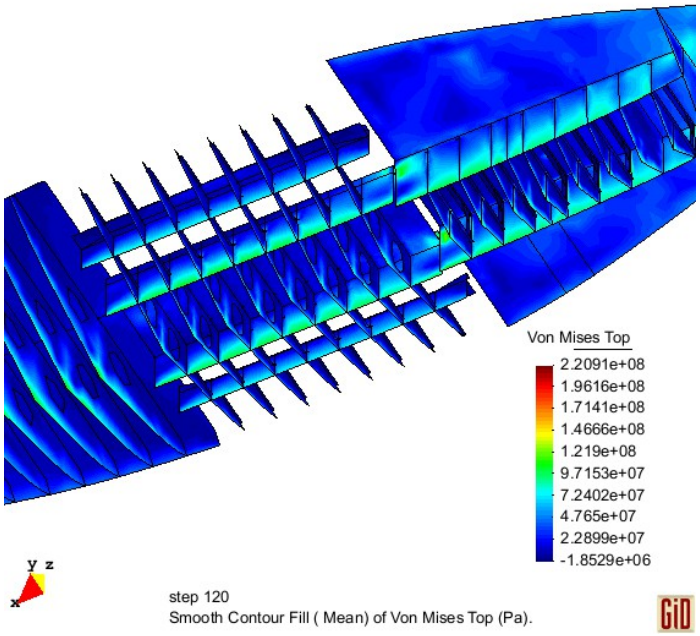


Stresses

Stress (Von Mises) results are shown below, for a certain step of the analysis:



Results


Von Mises Top
2.2091e+08
1.9616e+08
1.7141e+08
1.4666e+08
1.219e+08
9.7153e+07
7.2402e+07
4.765e+07
2.2899e+07
-1.8529e+06
y z
x
step 120
Smooth Contour Fill (Mean) of Von Mises Top (Pa).