ANSYS

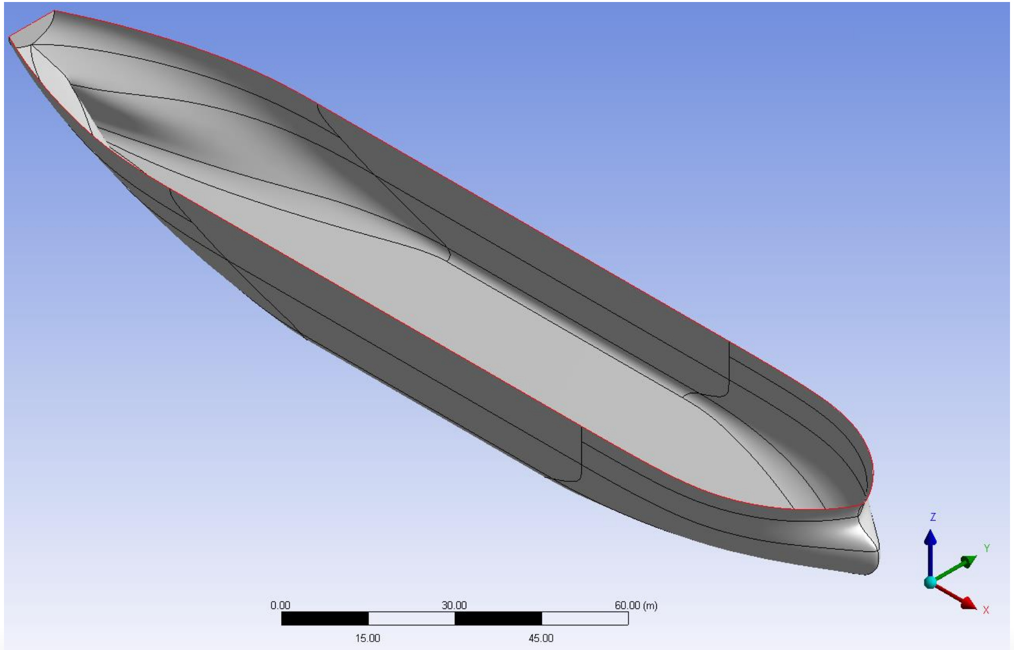
Workshop 1: Aqwa Basics – Hydrodynamic Diffraction

Introduction to Hydrodynamic Analysis with ANSYS Aqwa

ANSYS Release 19.2

Ship Hydrodynamic Diffraction

The goal of this workshop is to create a hydrodynamic radiation/diffraction model of a ship, which is then meshed, solved and typical results produced



Insert HD System and Associate Geometry

Create Hydrodynamic Diffraction (HD) System (as shown in Lecture 1)

Import CAD geometry and/or create model directly

Repair as necessary

Position the model(s) in the correct vertical and horizontal locations required for the analysis

Cut the model(s) at the water line to provide the required delineation between above and below water (no element splitting)

Combine bodies associated with a single vessel to form one part.

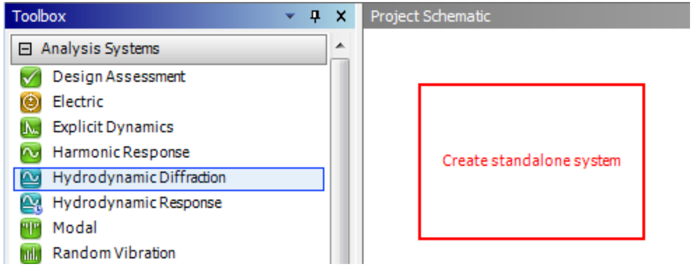
Create HD System

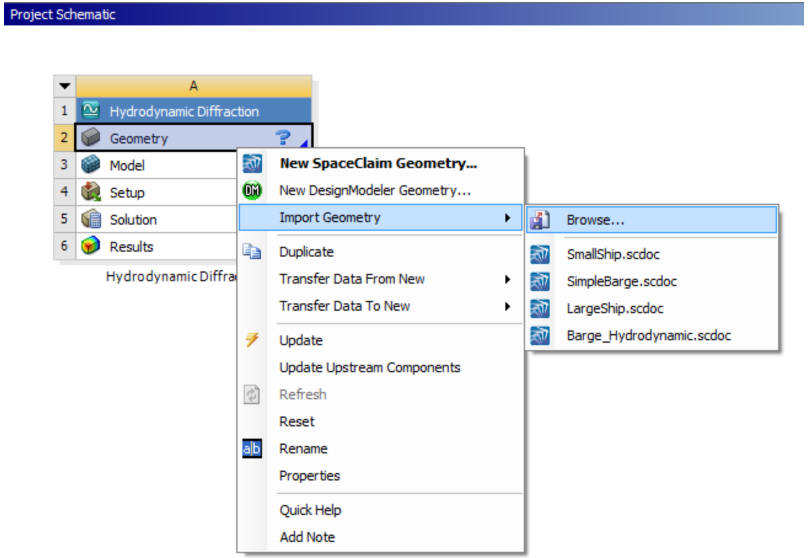
Start Workbench.

Add HD system by drag & drop onto WB Project Schematic page.

RMB on Geometry cell, select Import Geometry, and navigate to the file ship.igs, and Open this file.

On the Project Schematic, double-click on the Geometry cell to open SpaceClaim (this will take a few moments).

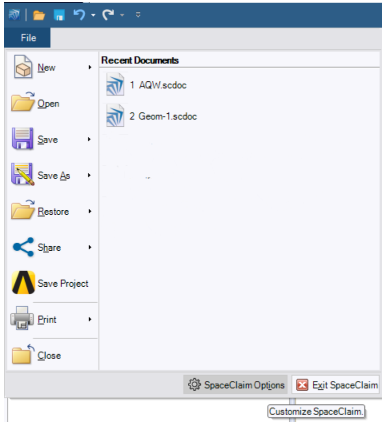


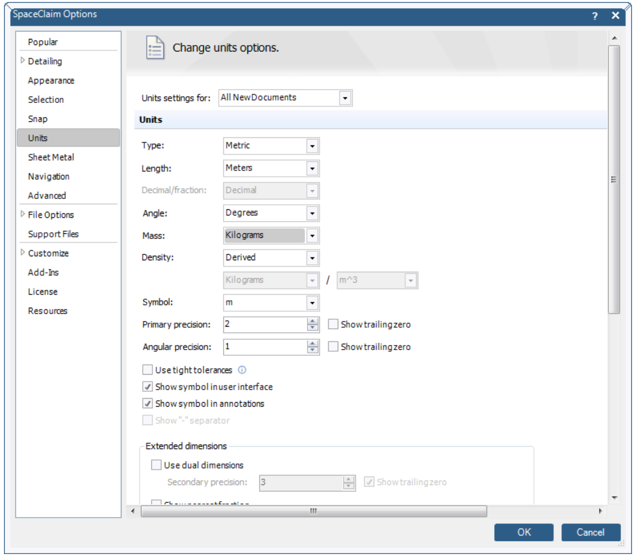


Set Geometry Units

We are going to work in SI units. Check to see if these are the default units for SpaceClaim.

Click on File and select SpaceClaim Options.

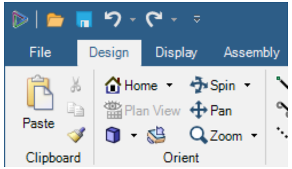




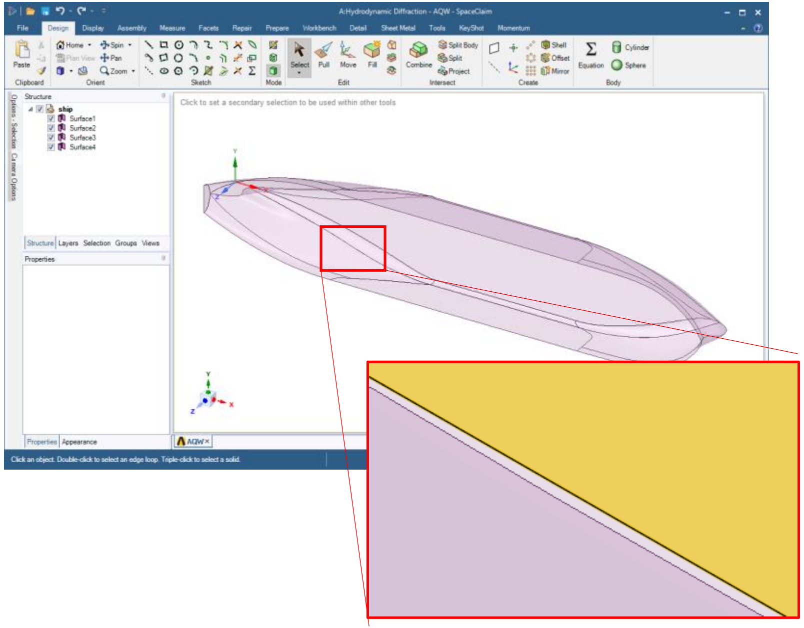
Select Units, set Units settings for All New Documents. Set Units Type to Metric and ensure length unit is Meters.

Repair Geometry

Select the Design tab and click on the Home icon in the top ribbon.



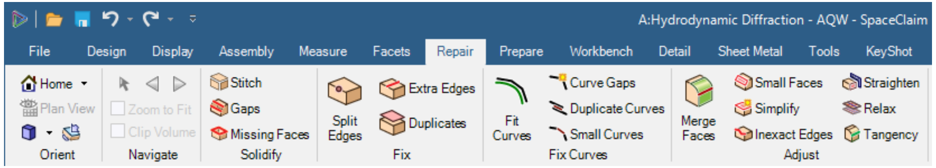
You will now see the geometry displayed in the graphics window.



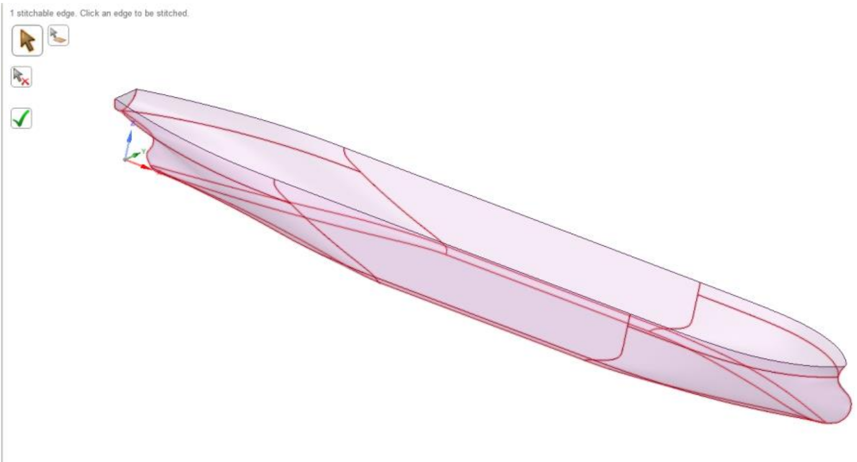
Many 3D CAD geometries are not directly suitable for simulation usage, with overlapping sections, missing parts, etc. SpaceClaim provides many tools to repair these automatically. We will use the Stitch tool to repair the geometry.

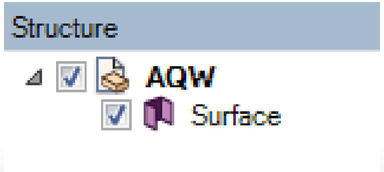
Repairs

Select Repair tab and click on the Stitch tool in the top ribbon.



SpaceClaim will automatically identify any surfaces that can be connected together to form a single surface body. Click the Complete button to accept all repairs. 

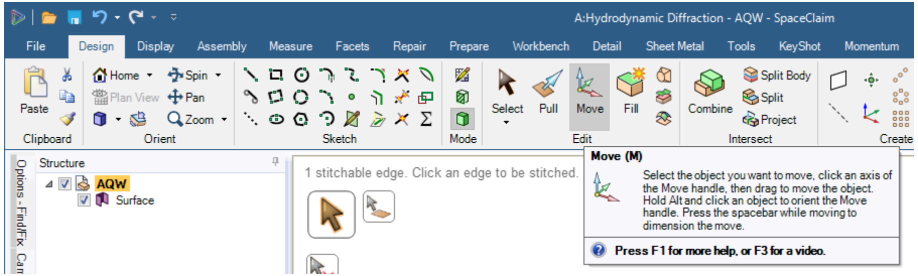




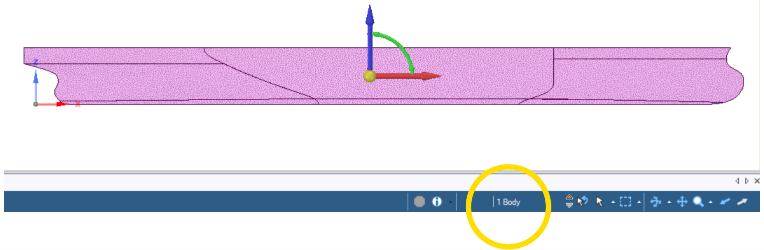
Correct Model Position

The global origin of the model is at the keel of the ship. Aqwa requires the global origin to be at the still water level. We can make use of the Move tool to position the vessel in the correct position.

Select Design tab and click on the Move tool in the top ribbon.

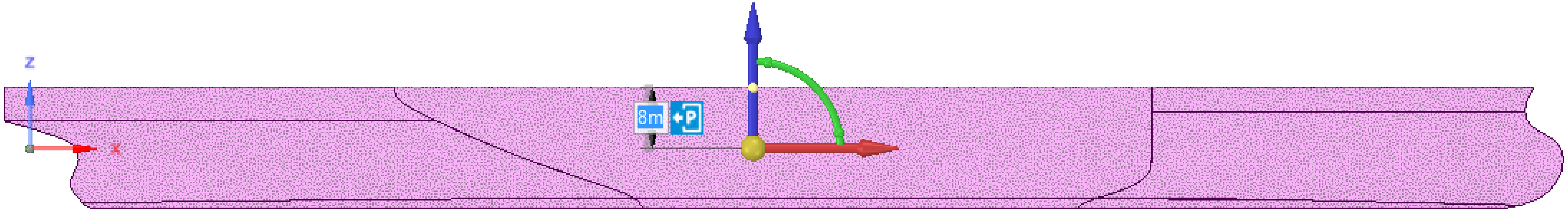


Triple-click on the vessel in the graphics window to select the associated body. You can confirm what has been selected by checking the information tab in the bottom margin.



Correct Model Position

The axes that appear allow us to move or rotate the body. Click on the blue arrow and drag the body down by a small amount. A blue dimension box will appear. Type in 8.



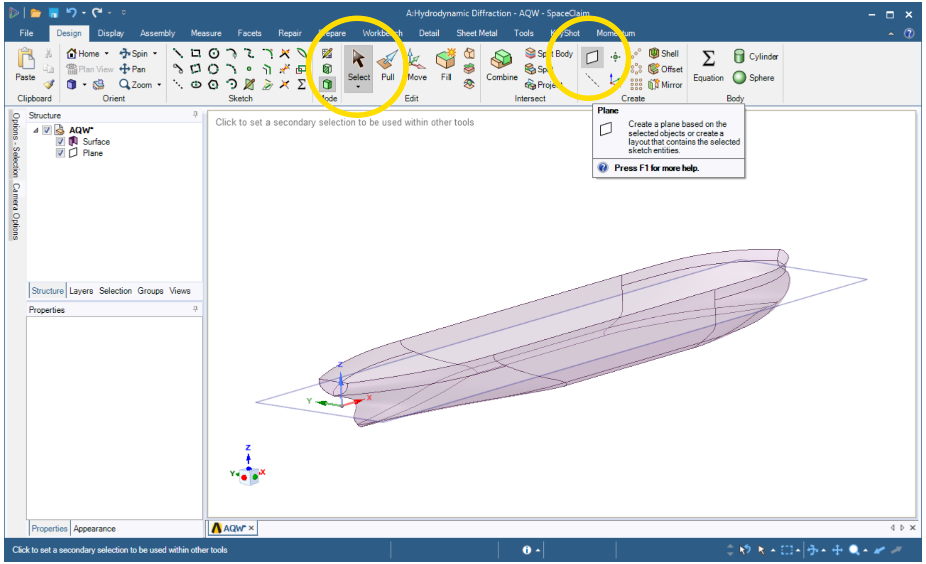
Hit Esc key twice to exit the tool.

Define the Waterline

We now need to cut the model at the water line so that we can mesh without elements crossing this line.

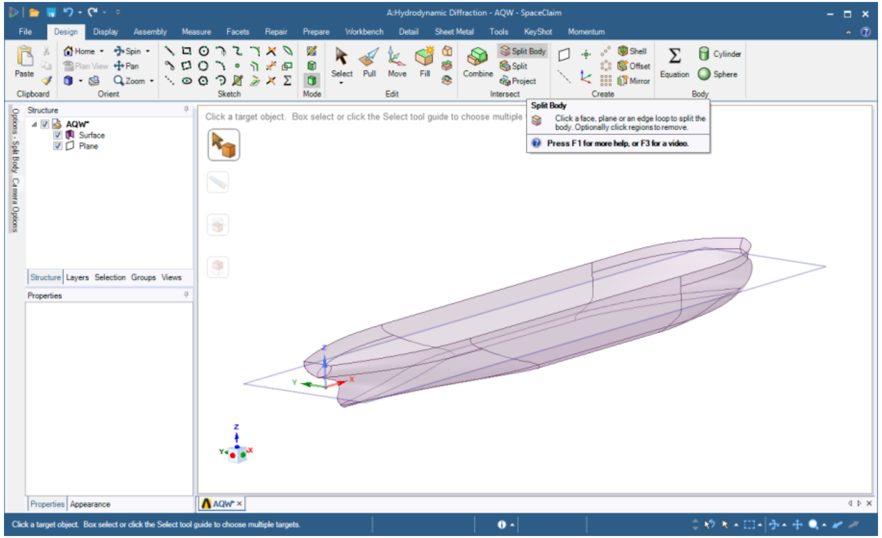
The Split Body tool allows a body to be split using a face, plane or edge loop. We will use the global XY plane, as required by Aqwa.

Choose a view of the geometry so that the global axis triad can be seen. First select Design tab and click on the Select tool in the top ribbon. Then click on the World Origin X axis, then Ctrl click on the Y axis. Finally click on the Plane tool in the top ribbon.



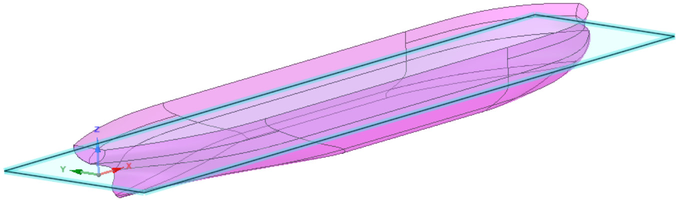
Split Model at the Waterline

Select the Split Body tool on the Design tab.



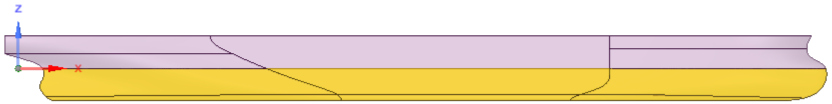
Click on the surface body in the graphics window. This will highlight the body.

Then select the plane that we have just defined as the cutter. Note you can either select this in the graphics window, of select the plane in the Structure tree.



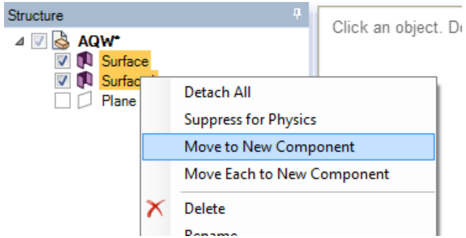
Hit Esc key to exit the tool.

Define the Waterline

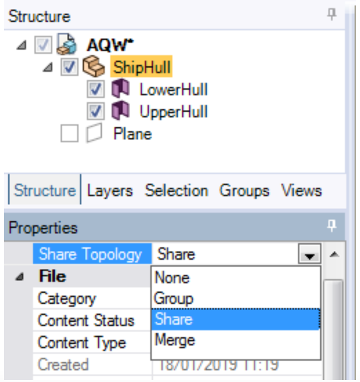


We now have two surface bodies, one above and one below the water line.

Aqwa requires that an individual vessel shares topology for any constituent bodies or components.



Combine the two surface bodies by selecting both in the Structure tree, right-click and choose Move to New Component. Rename Component1 and the two surface bodies as shown.



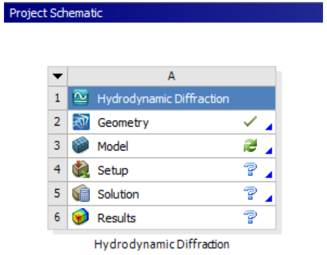
Click on ShipHull in the Structure tree, and in the Properties panel set Share Topology to Share.

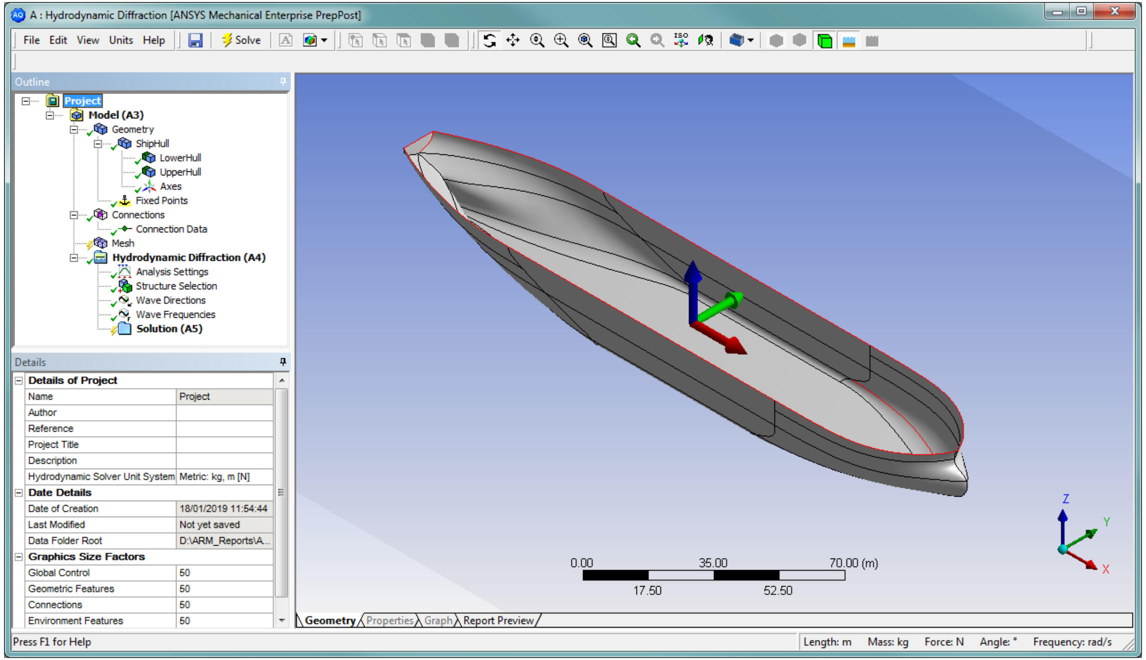
Exit SpaceClaim.

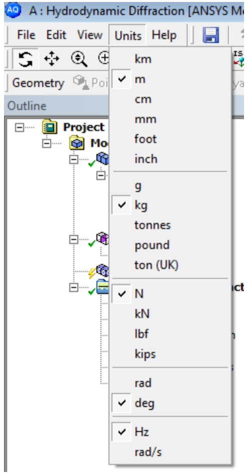
Adding Aqwa Specific Parameters

Double-Click on Model Cell on Project Schematic.

Check that units are set to m, kg and N.







Global Parameters

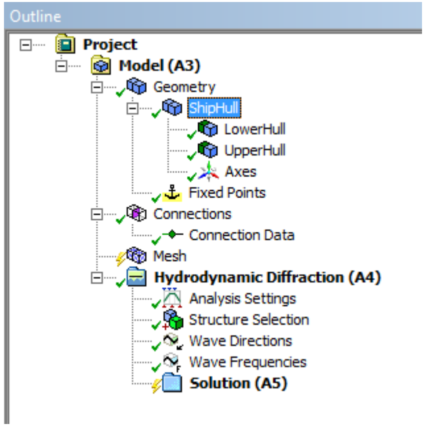
Selecting the Geometry object in the tree provides access to some global data in the details window, such as Water Depth, Water Density and Gravity. Set the Water Depth to 500m. Note that the default import preferences include Surface Bodies and Line Bodies, but also Solid Bodies (which are actually invalid in Aqwa).

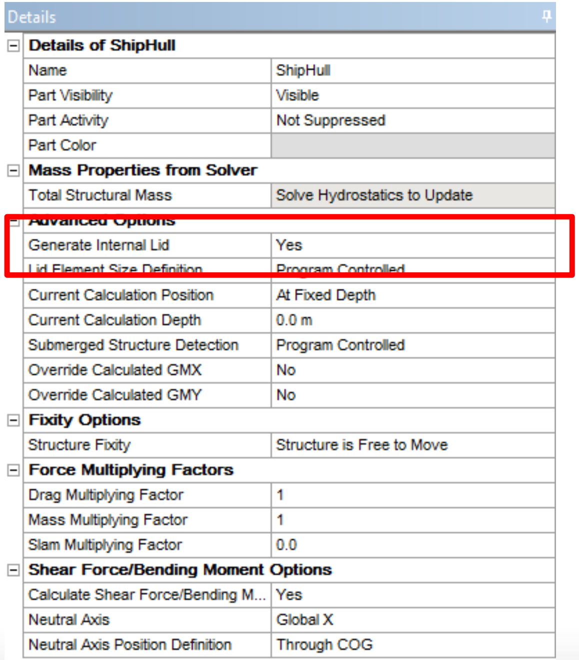
Details

Details of Geometry	
Name	Geometry
Attached Assembly Path	D:\ARM_Reports\AQWA_TO_...
Environment Constants	
Water Depth	1000 m
Water Density	1025 kg/m³
Gravity	9.80665 m/s²
Water Size X	820 m
Water Size Y	260 m
Stability/Time Response-Specific Options	
Tube Drag Coefficients	Defined in Line Body Details
Seabed Inline Friction Coefficient	0.0
Seabed Lateral Friction Coefficient	0.0
Composite Cable Seabed Definition	
Seabed Type	No Composite Cable Seabed
Import Preferences	
Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes

Vessel/Structure Parameters

Selecting a vessel or structure in the tree provides access to data specific to that vessel, such as whether it should be included in the simulation, is free or fixed, whether an internal lid is required, and additional calculation information.





Set Generate Internal Lid Option in Details of ShipHull to “Yes”. This will remove any potential irregular frequencies, which occur due to the source distribution approach employed by Aqwa.

Analysis Settings

These options control how the analysis is to proceed, and what types of results are to be reported and stored.

They relate directly to the options used in the Aqwa analysis that are described in the Aqwa Reference Manual.

The Wave Grid Size Factor is used in determining the size of the diffracted water surface plot in the visualization window with respect to the size of the model.

Set Ignore Modelling Rule Violations to “Yes” to ignore mesh quality check violations (refer to Lecture 8: Appendix Slides for more information on the mesh quality checks).

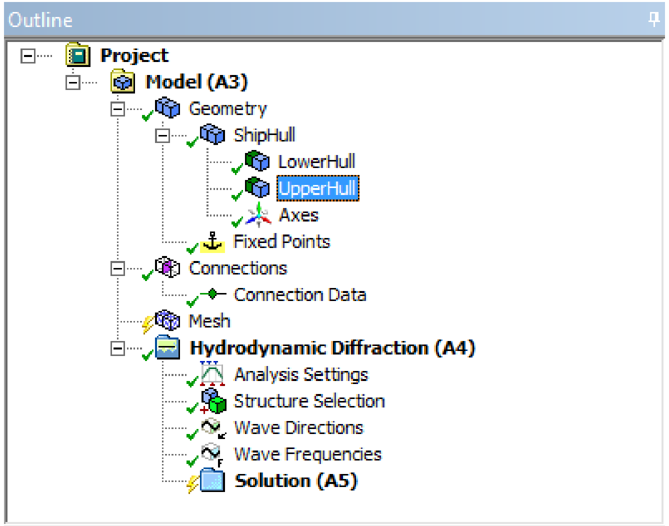
Set Calculate Full QTF Matrix to “No” to speed up the calculation.

Details

Details of Analysis Settings	
Name	Analysis Settings
Parallel Processing	Program Controlled
Generate Wave Grid Pressures	Yes
Wave Grid Size Factor	2
Common Analysis Options	
Ignore Modelling Rule Violations	Yes
Calculate Extreme Low/High Freq...	Yes
Include Multi-Directional Wave Int...	Yes
Near Field Solution	Program Controlled
Linearized Morison Drag	No
QTF Options	
Calculate Full QTF Matrix	No
Output File Options	
Source Strengths	No
Potentials	No
Centroid Pressures	No
Element Properties	No
ASCII Hydrodynamic Database	No
Example of Hydrodynamic Databa...	No

Bodies within Vessels/Structures

Selecting a body within a vessel or structure in the tree provides access to data specific to that body, such as whether it should be included in the simulation and whether it is diffracting or non-diffracting. When first imported, any surface body below the water line is assumed to be diffracting (Surface Type: Program Controlled).



Details

Details of UpperHull	
Name	UpperHull
Body Visibility	Visible
Body Activity	Not Suppressed
Body Color Definition	Inherited from Part
Structure Type	Physical Geometry
Surface Type	Program Controlled

Provide Additional Aqwa Elements

DesignModeler or SpaceClaim can be used to create models consisting of panels and Morison type elements, such as cylindrical tubes.

Some additional element types can be added directly within HD

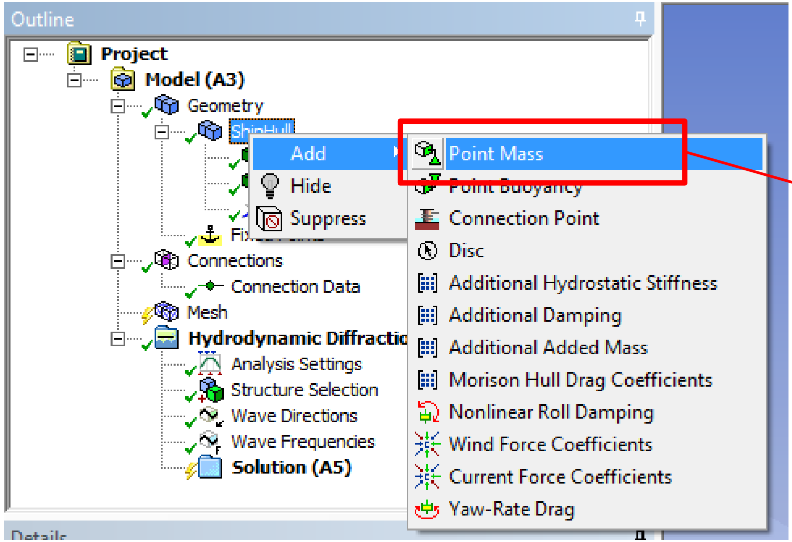
- Point mass
- Point buoyancy
- Discs

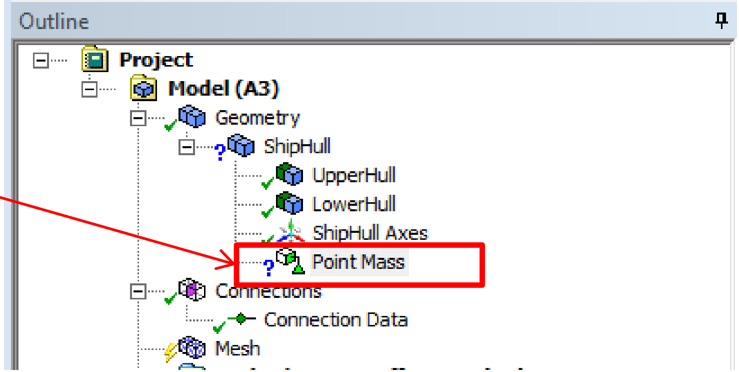
As a minimum we must provide a point mass to describe the mass matrix for the vessel.

Provide Additional Aqwa Elements

For each vessel/structure defined provide a point mass element

This is inserted in the tree by selecting the required vessel, and right clicking on Add > Point Mass





Point Mass Properties

The program can compute the mass based upon the displacement of the vessel, or this can be defined directly in the details window.

The mass inertia matrix must ALWAYS be defined, either via Radius of Gyration or direct input

Values required are highlighted in yellow

Details

Details of Point Mass	
Name	Point Mass
Visibility	Visible
Activity	Not Suppressed
Mass Properties	
Mass Definition	Program Controlled
X	0.0 m
Y	0.0 m
Z	0.0 m
Mass	1 kg
Inertia Properties	
Define Inertia Values By	Radius of Gyration
Kxx	0.0 m
Kyy	0.0 m
Kzz	0.0 m
Ixx	0.0 kg.m²
Ixy	0.0 kg.m²
Ixz	0.0 kg.m²
Iyy	0.0 kg.m²
Iyz	0.0 kg.m²
Izz	0.0 kg.m²

Point Mass Input

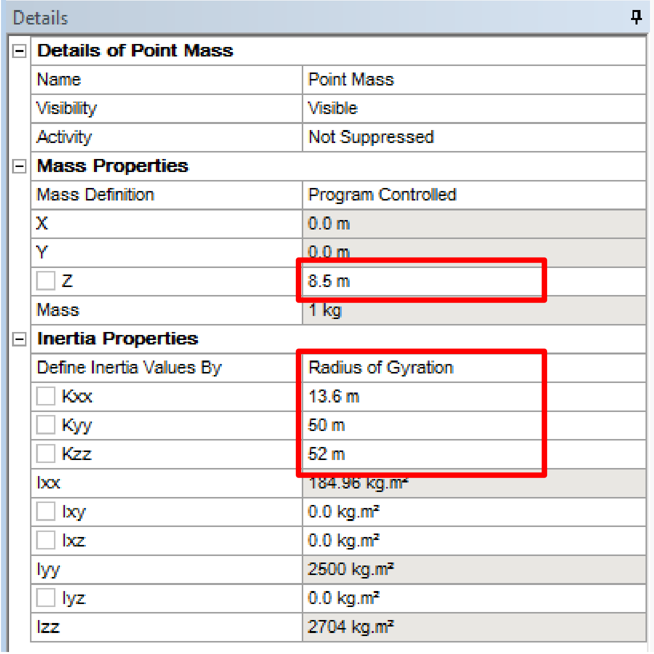
If Mass Definition is Program Controlled the mass will equal the displacement.

Set k_{XX}, k_{YY} and k_{ZZ} to standard default values (beam is 40m, length 200m)

For a regular ship;

- k_{XX} = 0.34*Beam
- k_{YY} = 0.25*Length
- k_{ZZ} = 0.26*Length

Set the VCG (Z coordinate) to 8.5 m for this exercise



Details

Details of Point Mass	
Name	Point Mass
Visibility	Visible
Activity	Not Suppressed
Mass Properties	
Mass Definition	Program Controlled
X	0.0 m
Y	0.0 m
Z	8.5 m
Mass	1 kg
Inertia Properties	
Define Inertia Values By	Radius of Gyration
Kxx	13.6 m
Kyy	50 m
Kzz	52 m
Ixx	184.96 kg.m²
Ixy	0.0 kg.m²
Ixz	0.0 kg.m²
Iyy	2500 kg.m²
Iyz	0.0 kg.m²
Izz	2704 kg.m²

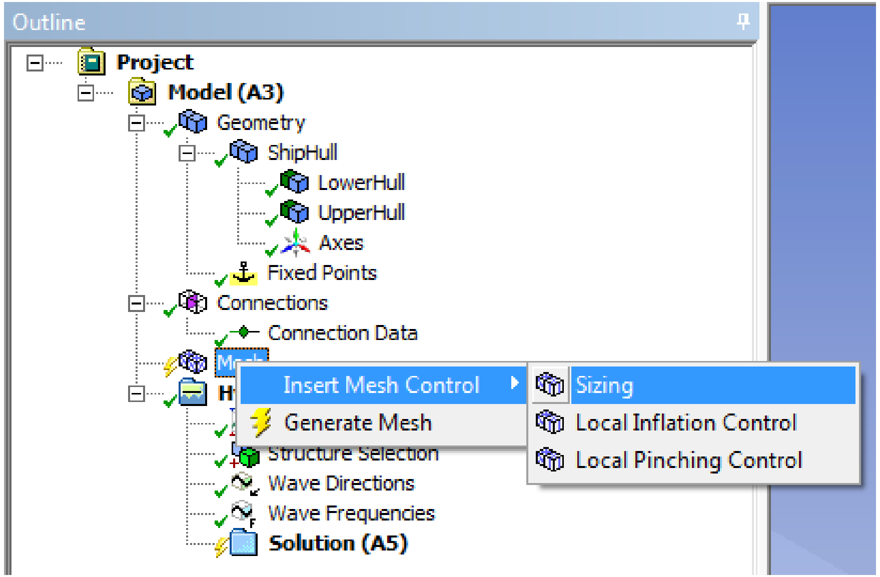
Meshing

The vessel(s)/structure(s) may now be meshed. When Mesh is selected in the tree some additional toolbar items appear.

Generate Mesh invokes the meshing tool using the parameters defined in the Details window.

If *localized* control of meshing is required, a Mesh Control may be utilized. This will not be used in this exercise, so do not include a Mesh Control.

Both these items can also be accessed by RMB on the Mesh object in the tree.



Mesh Control

In the details window the Global Control is set to Basic Controls by default.

If more control is required for the meshing then the Global Control can be set to Advanced Controls. This exposes additional options for the meshing process. This is not going to be covered for this training.

Note that the ANSYS Meshing System can be used upstream of an Aqwa HD System. This is the preferred approach if more advanced meshing is required.

Details

Details of Mesh	
Defaults	
Global Control	Basic Controls
Mesh Parameters	
Defeaturing Tolerance	2 m
Max Element Size	5 m
Max Allowed Frequency	0.211 Hz (Estimated from input)
Meshing Type	Program Controlled

Details

Details of Mesh	
Defaults	
Control Type	Advanced Controls
Mesh Parameters	
Maximum Allowed Frequency	0.211 Hz (Estimated from input)
Meshing Type	Program Controlled
Sizing	
Relevance	0
Advanced Size Function	Off
Relevance Centre	Coarse
Element Size Definition	Program Controlled
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Centre	Coarse
Minimum Edge Length	5.97125764634216E-04 m
Inflation	
Use Inflation	No
Defeaturing	
Use Pinching	No
Sheet Loop Removal	No
Automatic Mesh-Based Defeatur...	Yes
Defeaturing Tolerance Definition	Program Controlled
Defeaturing Tolerance	0.4210312540626 m

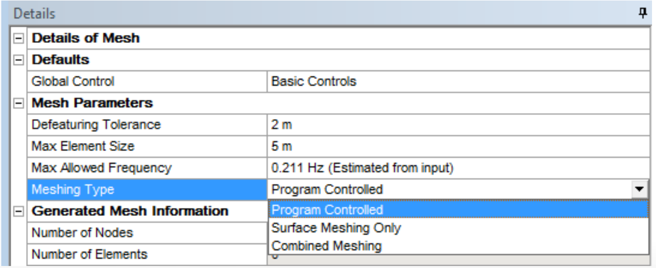
Meshing Selections

When Basic Controls is selected, the Maximum Element Size, Defeaturing Tolerance and Meshing Type may be defined.

The Defeaturing Tolerance controls how small details are treated. If the detail is smaller that this tolerance then a single element may span over it.

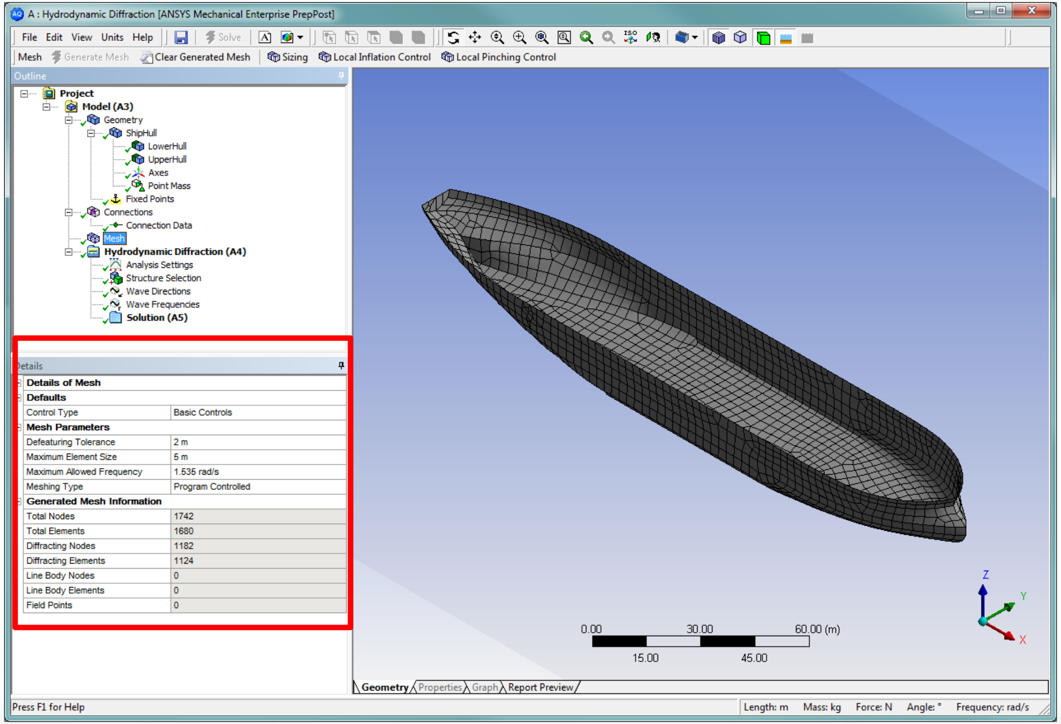
The Maximum Element Size controls the sizes of elements generated. This determines the maximum wave frequency that can be utilized in the diffraction analysis.

If Program Controlled meshing is chosen the program will use a surface mesher for vessels only containing surfaces (panels), and a combined mesher if the vessel also contains line elements. These may be specifically requested using the drop down menu.



Meshing

When Generate Mesh is selected the meshing tool is instigated and a mesh using the parameters defined is created. Mesh information is given in the details panel.



Meshing

From the Units menu, change the frequency unit from Hz to rad/s.

In the Mesh Details panel adjust the Defeaturing Tolerance to 0.5m and the Max Element Size to 2.5m (note that the defeaturing tolerance must not be greater than 0.6 x Max Element Size).

Click Generate Mesh.

You will notice that the maximum allowed wave frequency now becomes in the order of 2 rad/s, however the number of elements is more than tripled (which increases the solver time significantly).

For a quick solution set the Max Element Size back to 5m and Defeaturing Tolerance to 2m.

Click Generate Mesh again.

Analyze

The model is now ready to be analyzed. The following additional items will be seen in the tree, under Hydrodynamic Diffraction:

- **Structure Selection**
- **Wave Directions**
- **Wave Frequencies**
- **Solution**

Structure Selection

Structure Selection enables the definition of Interacting Structure Groups (for multi-body problems). By default all vessels are assumed interacting.

Details

Details of Structure Selection	
Name	Structure Selection
Structures to Exclude	None
Group of Structures	
Interacting Structure Groups	None
Structure Ordering	
Structure 1	ShipHull

Wave Directions

This objects permits wave directions to be defined, either as a range or with individual values. We can also add a forward speed correction to the calculation by changing the Type option.

For this analysis leave Wave Directions as already defined.

Details

Details of Wave Directions	
Name	Wave Directions
Visibility	Visible
Type	Range of Directions, No Forward Speed
Required Wave Input	
Wave Range	-180° to 180°
Interval	45°
Number of Intermediate Directions	7
Optional Wave Directions A	
Additional Range	None
Optional Wave Directions B	
Additional Range	None
Optional Wave Directions C	
Additional Range	None
Optional Wave Directions D	
Additional Range	None

Wave Directions

Direction Number	Wave Direction (°)
1	-180
2	-135
3	-90
4	-45
5	0.0
6	45
7	90
8	135
9	180

Wave Frequencies

This object permits wave frequencies to be defined, either as a range or with individual values.

The initial maximum frequency is determined by the mesh size; attempting to change this to a higher frequency will produce an error. If higher frequencies are required the mesh size will need to be reduced.

Frequencies may be Program Controlled (with equal frequency or period interval) or manually defined. Choose Manual Definition, since we know for this vessel that there is a resonant frequency in roll at 0.174 rad/s.

Set Number of Intermediate Values to 8 (normally this would be much higher, typically 50 or more). Then include an Additional Frequency: set the Additional Range to Single, Lowest Frequency Definition to Manual, and the Lowest Frequency to 0.174 rad/s.

Details

Details of Wave Frequencies	
Name	Wave Frequencies
Intervals Based Upon	Frequency
Incident Wave Frequency/Period Definition	
Range	Manual Definition
Definition Type	Range
Lowest Frequency Definition	Program Controlled
Lowest Frequency	0.1 rad/s
Longest Period	62.83185 s
Highest Frequency Definition	Program Controlled
Highest Frequency	1.32551 rad/s
Shortest Period	4.74022 s
Number of Intermediate Values	8
Interval Frequency	0.13617 rad/s
Additional Frequencies A	
Additional Range	Single
Lowest Frequency Definition	Manual Definition
Lowest Frequency	0.174 rad/s
Longest Period	36.11026 s
Additional Frequencies B	
Additional Range	None
Additional Frequencies C	
Additional Range	None
Additional Frequencies D	
Additional Range	None

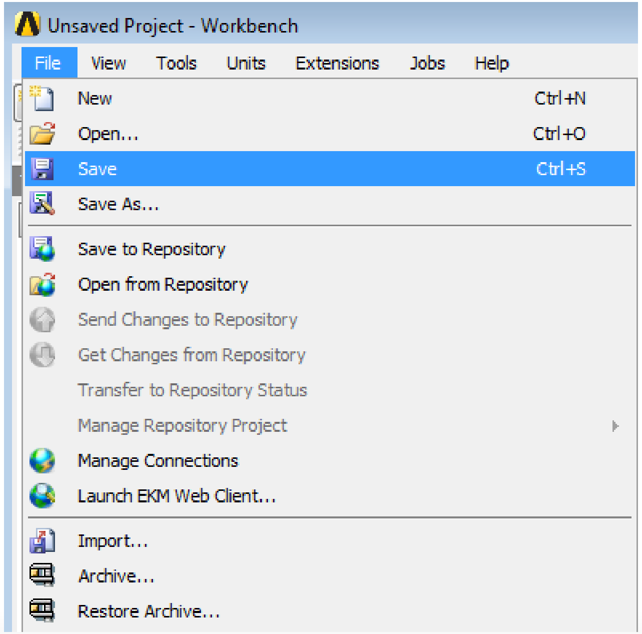
Wave Frequencies

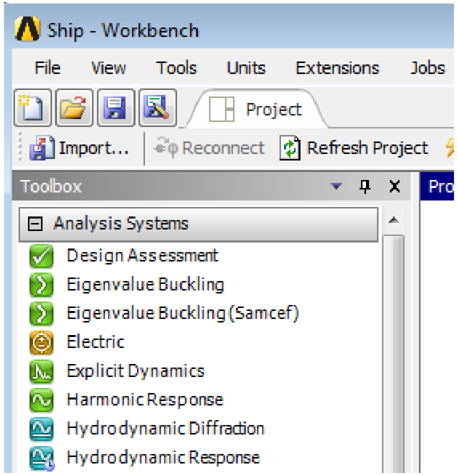
Number	Wave Frequency (rad/s)
1	0.1
2	0.174
3	0.23617
4	0.37233
5	0.5085
6	0.64467
7	0.78084
8	0.917
9	1.05317
10	1.18934
11	1.32551

Save Project

Save the project from the Workbench Project Page, File > Save

Browse to the training working directory and save the project as Ship.wbpj. The title on the Project Page will reflect this change.



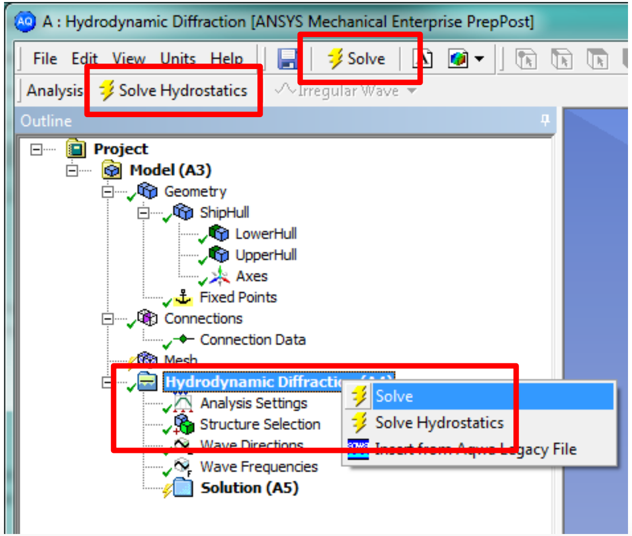


Solve

Two Analysis options are available:

- **Calculate hydrostatics only (compute displacement and small angle stability parameters)**
- **Compute full hydrodynamic properties and results**

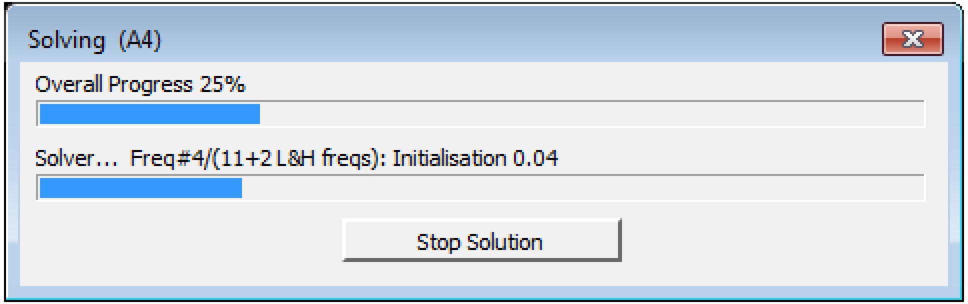
These are available through the toolbar menu items, or using the context sensitive menu on the Analysis object (RMB click)



Solve

If a full analysis is chosen the basic hydrostatic solution is internally run first to ensure consistent mass and displacement. Any Point Mass objects with Mass Definition: Program Controlled will be updated.

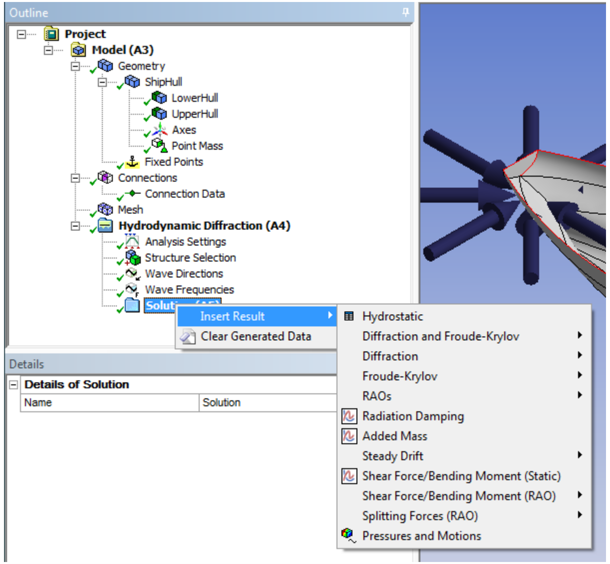
Whilst a solve is being processed a progress bar appears to indicate how far the calculation has reached.

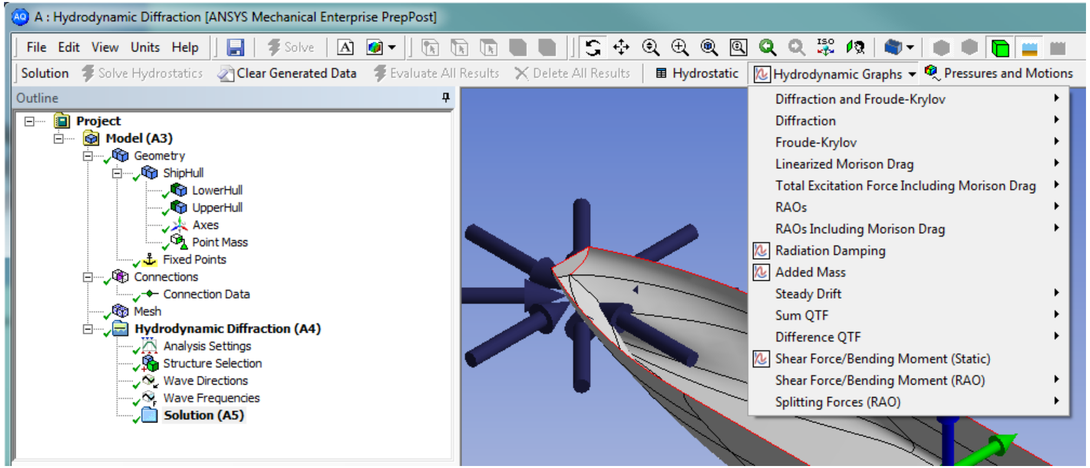


Review Results

Results can be added when the Solution object is selected. This can be done before or after an analysis has been undertaken.

Available sets of results are: Hydrostatic table, Hydrodynamic graphs, Pressures and Motions contour plots





Single Ship Model

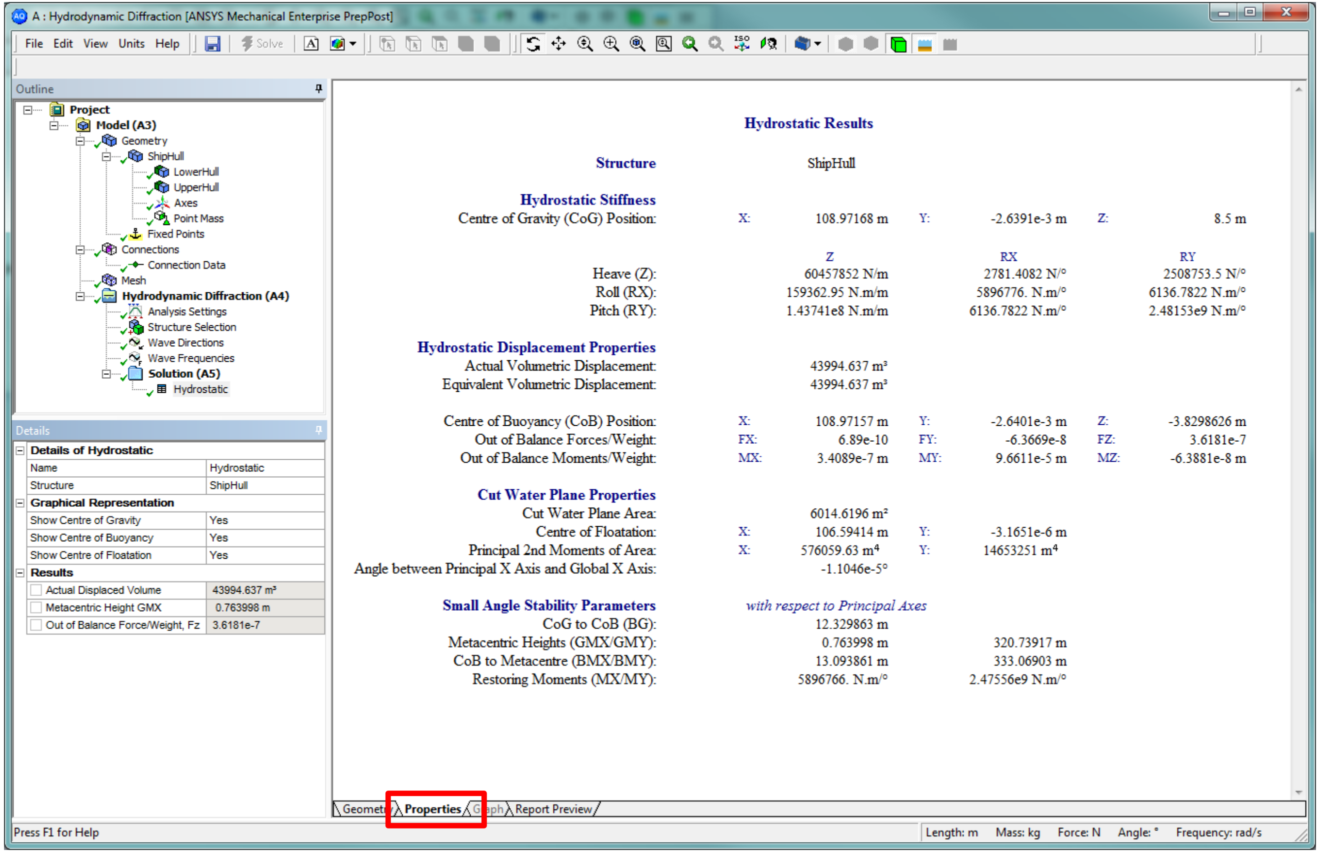
Insert results (details on following slides)

- **Hydrostatic Table**
- **Hydrodynamic Graphs**
- **Pressures and Motions**

For each result set the required structure must be selected

Hydrostatics

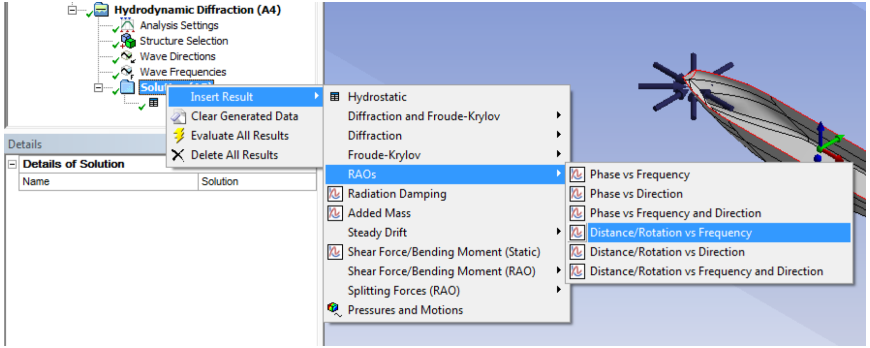
When selected these results appear on the Properties tab.

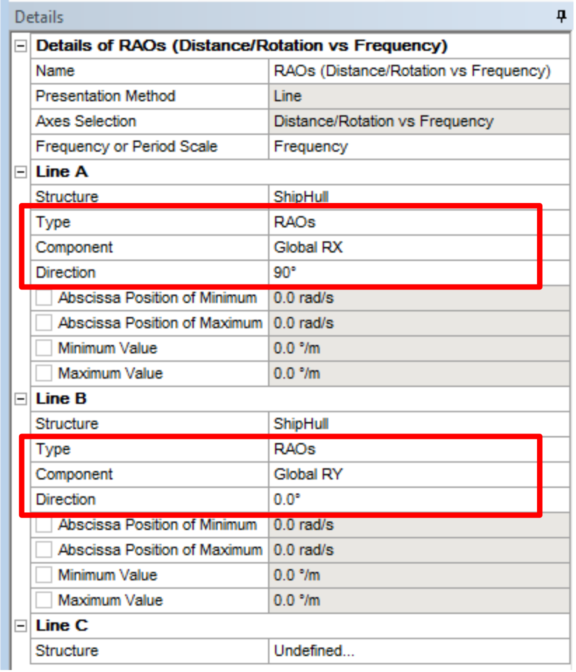


Hydrodynamic Graphs

Graphs allow computed parameters to be plotted.

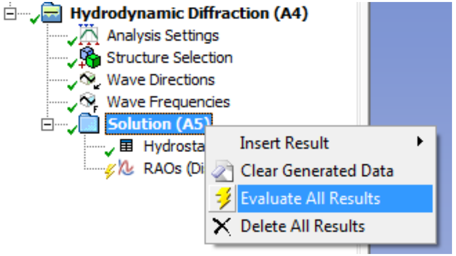
Multiple graphs may appear on a single plot, and multiple plots may be requested by inserting additional Hydrodynamic Graph objects in the tree. Note that for some results there are options related to where the results are reported for and what they are relative to. This will be covered in the next workshop.



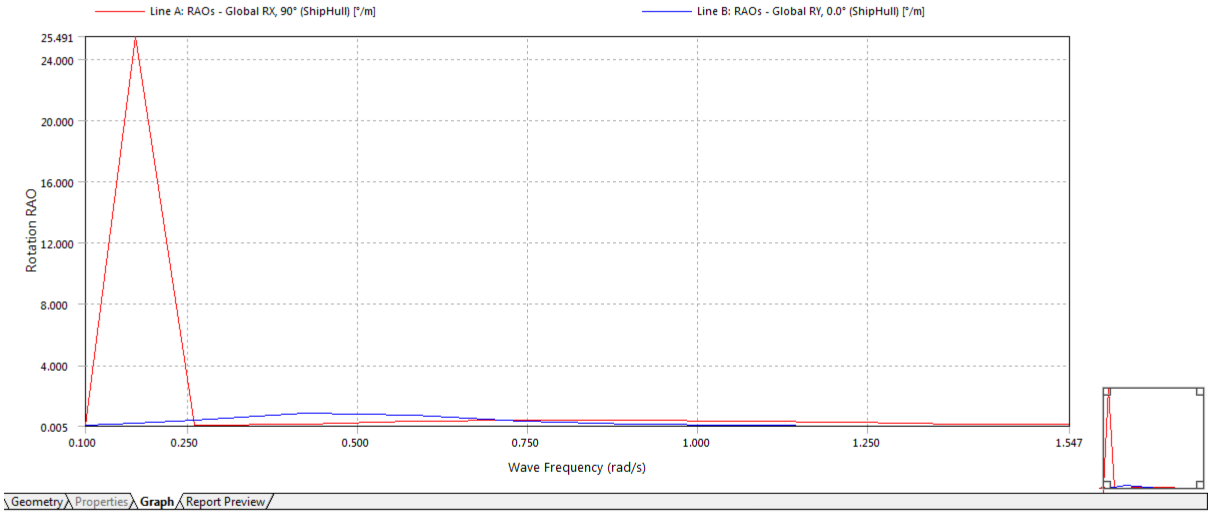


Hydrodynamic Graphs

Evaluate All Results must be selected for graphical results to be presented/updated if created after the solve



Note the very high roll value at 0.174 rad/sec. We are going to reduce this later by including some additional damping



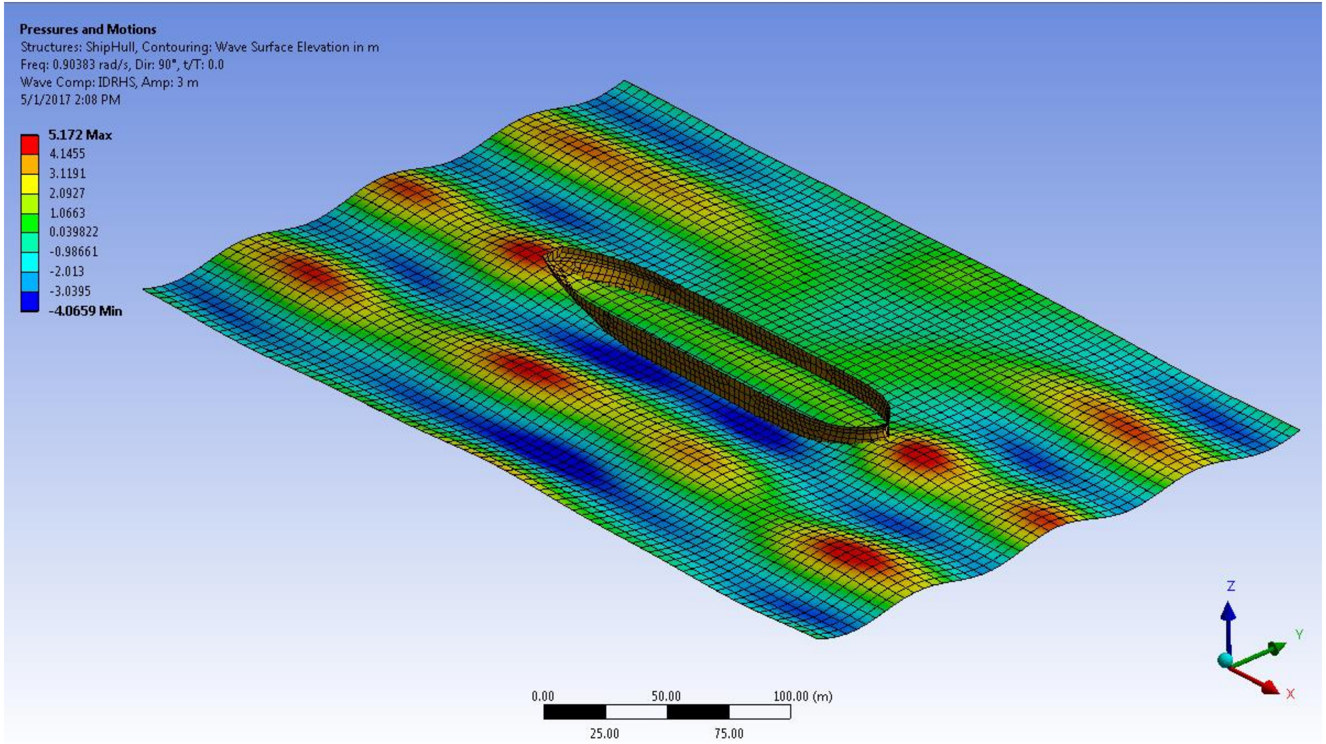
Tabulated Results Data

Wave Frequency (rad/s)	Line A (°/m)	Line B (°/m)
0.1	0.10238	0.08935
0.174	25.49074	0.18916
0.26076	0.05092	0.38704
0.42153	0.16051	0.84057
0.5823	0.3035	0.75095
0.74307	0.38887	0.34989
0.90383	0.39426	0.1681
1.0646	0.33913	0.06439
1.22536	0.24649	0.02597
1.38613	0.15232	0.01144
1.5469	0.10852	4.68144e-3

ANSYS

Hydrodynamic Pressures and Motions

This object presents plots of wave contours and hull pressures



Hydrodynamic Pressures and Motions

These are used to select the wave frequency and direction to utilize in the plot. Choose higher frequencies for more interesting plots

Result type may be Amplitude, Phase Angle, Minimum or Maximum

- Choose Phase Angle if specific wave position required

If Phase Angle is chosen then Wave Position may be 0°, 90°, Specified or Sequence

- Choose Sequence if animation is required

If Sequence is chosen then Number of Steps specifies how many wave positions to display in the animation

Details

Details of Pressures and Motions	
Name	Pressures and Motions
Structure Selection	ShipHull
Result Selection	
Frequency	0.90383 rad/s
Direction	90°
Incident Wave Amplitude	3 m
Result Type	Phase Angle
Wave Position (Phase)	Sequence
Number of Steps	12
Contour Selection	
Contour Type	Wave Surface Elevation
Structure Elements Shown As	Opaque
Component Selection	
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiation Wave	Yes
Include Second Order Terms	Yes

Hydrodynamic Pressures and Motions

Hull pressures may be interpolated or individual panel results. Can also show wave elevation, resultant (RAO) displacement or air gap.

Water Elements Shown As dims or shows the wave surface.

Choose either Head of Water or Force/Area for the hull contours using Pressure Measurement.

Include or exclude wave components

Change Incident Wave Amplitude to see the effect of Second Order Terms

Details

Details of Pressures and Motions	
Name	Pressures and Motions
Structure Selection	ShipHull
Result Selection	
Frequency	0.90383 rad/s
Direction	90°
Incident Wave Amplitude	3 m
Result Type	Phase Angle
Wave Position (Phase)	Sequence
Number of Steps	12
Contour Selection	
Contour Type	Structure Interpolated Pressure
Water Elements Shown As	Opaque
Pressure Measurement	Force/Area
Component Selection	
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiation Wave	Yes
Include Hydrostatic Differential	Yes
Include Second Order Terms	Yes

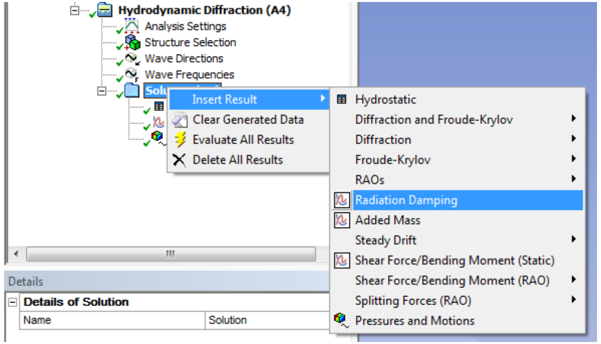
ANSYS

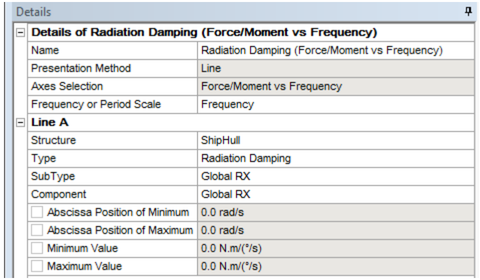
Additional Roll Damping

We saw earlier very large roll motion RAOs for beam waves. This is quite common for ship-shaped models due to the lack of viscous effects and vortex shedding that provide much of the roll damping.

Bilge keel damping can be included in subsequent Hydrodynamic Response analyses. To simulate the damping effects in the Hydrodynamic Diffraction system we can provide additional linear damping.

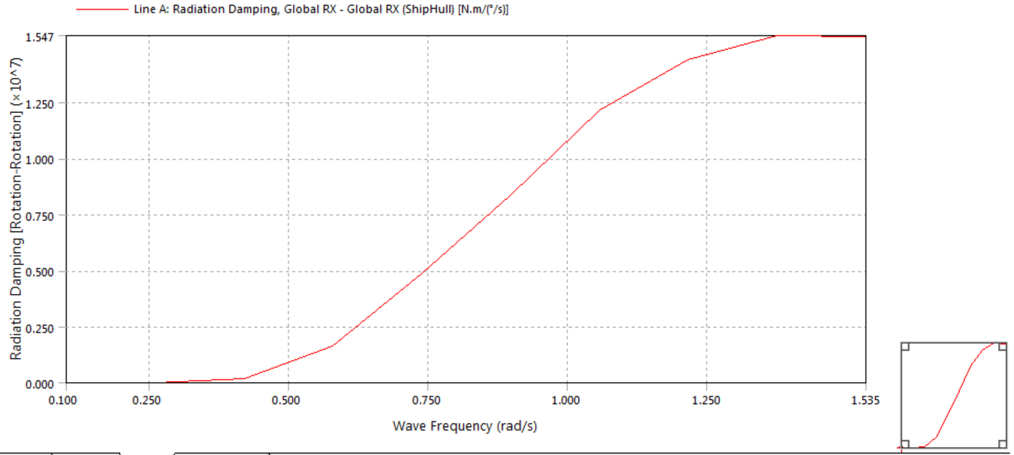
To see the level of radiation damping computed at the resonant frequency include a Radiation Damping result. Choose Sub Type and Component as Global RX (this will be the 4th element on the leading diagonal of the damping matrix).





Additional Roll Damping

At the resonant frequency of 0.174 rad/s we have negligible damping: 156 Nm/(°/s) against a peak of 1.547e+07 Nm/(°/s)



Geometry | Properties | Graph | Report Preview

Tabulated Results Data

Wave Frequency (rad/s)	Line A (N.m/(°/s))
0.1	3.44193
0.174	156.31455
0.25948	3549.55655
0.41896	168848.35292
0.57844	1632474.30779
0.73791	4872772.73078
0.89739	8374604.69301
1.05687	12140456.66036
1.21635	14411275.7871
1.37582	15471821.61642
1.5353	15441536.10473

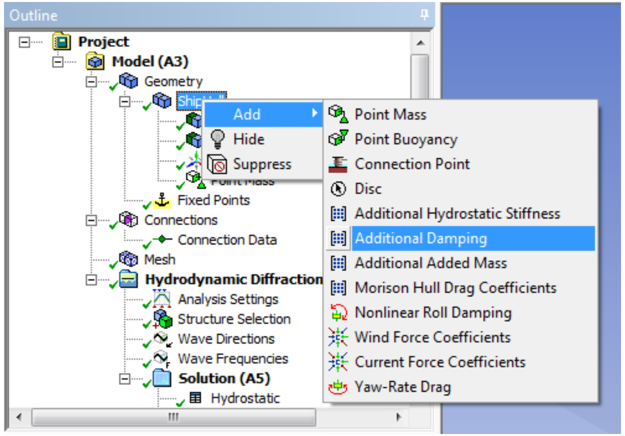
Additional Roll Damping

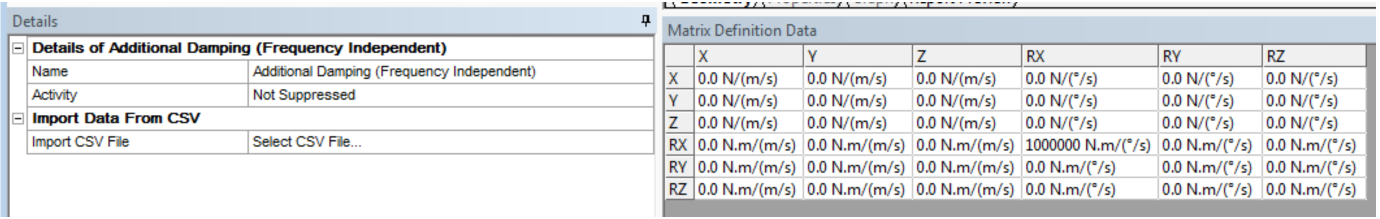
We are going to add some additional roll damping of 1.0e+06 Nm/(°/s).

Select ShipHull in the Outline tree and RMB Add > Additional Damping.

In the Matrix Definition Data panel enter 1e6 in the RX/RX cell, as shown.

This will invalidate the existing results. Re-solve the Hydrodynamic Diffraction analysis.





Details

Details of Additional Damping (Frequency Independent)	
Name	Additional Damping (Frequency Independent)
Activity	Not Suppressed
Import Data From CSV	
Import CSV File	Select CSV File...

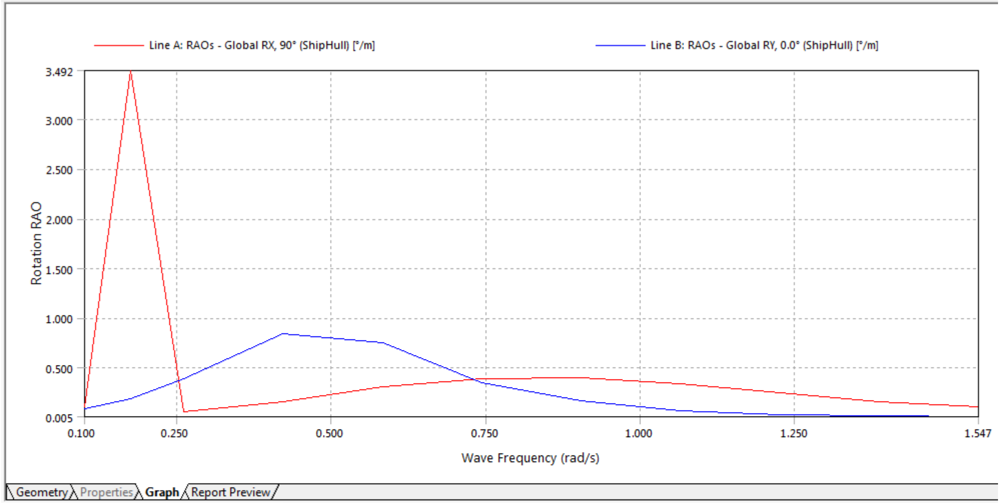
Matrix Definition Data

	X	Y	Z	RX	RY	RZ
X	0.0 N/(m/s)	0.0 N/(m/s)	0.0 N/(m/s)	0.0 N/(°/s)	0.0 N/(°/s)	0.0 N/(°/s)
Y	0.0 N/(m/s)	0.0 N/(m/s)	0.0 N/(m/s)	0.0 N/(°/s)	0.0 N/(°/s)	0.0 N/(°/s)
Z	0.0 N/(m/s)	0.0 N/(m/s)	0.0 N/(m/s)	0.0 N/(°/s)	0.0 N/(°/s)	0.0 N/(°/s)
RX	0.0 N.m/(m/s)	0.0 N.m/(m/s)	0.0 N.m/(m/s)	1000000 N.m/(°/s)	0.0 N.m/(°/s)	0.0 N.m/(°/s)
RY	0.0 N.m/(m/s)	0.0 N.m/(m/s)	0.0 N.m/(m/s)	0.0 N.m/(°/s)	0.0 N.m/(°/s)	0.0 N.m/(°/s)
RZ	0.0 N.m/(m/s)	0.0 N.m/(m/s)	0.0 N.m/(m/s)	0.0 N.m/(°/s)	0.0 N.m/(°/s)	0.0 N.m/(°/s)

Additional Roll Damping

Look at the RAO plot previously included in the Solution. You will see that the peak roll response is now much more reasonable. As expected, the additional damping has little effect away from the resonant frequency.

Save the project and close Aqwa.



Tabulated Results Data

Wave Frequency (rad/s)	Line A (°/m)	Line B (°/m)
0.1	0.10234	0.08935
0.174	3.49168	0.18916
0.26076	0.05089	0.38704
0.42153	0.1605	0.84057
0.5823	0.30348	0.75095
0.74307	0.38882	0.34989
0.90383	0.39419	0.1681
1.0646	0.33906	0.06439
1.22536	0.24643	0.02597
1.38613	0.15229	0.01144
1.5469	0.1085	4.68144e-3