Introduction to Hydrodynamic Analysis with Ansys Aqwa

Workshop 09.1: Load Mapping to Ansys Mechanical for a Truss Spar

Release 2021 R2

Ansys

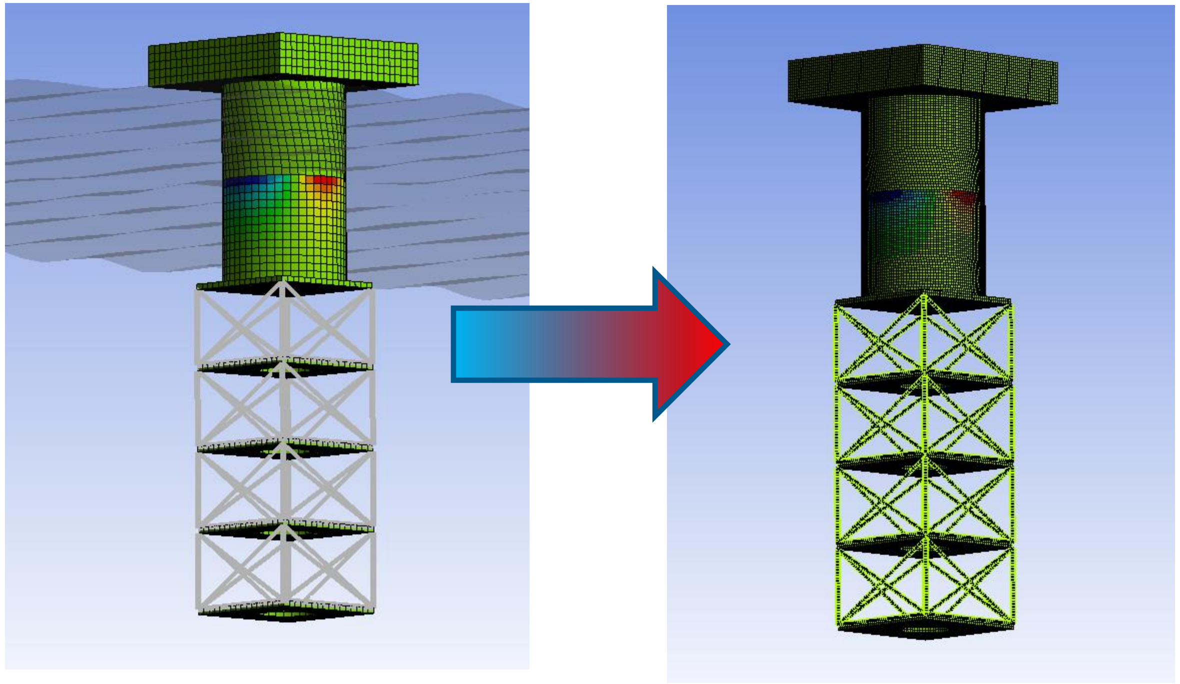
Load Mapping to Ansys Mechanical

The goal of this workshop is to undertake load mapping from a Hydrodynamic Diffraction system to a Static Structural system.

The model is the truss spar used in Workshop 5.

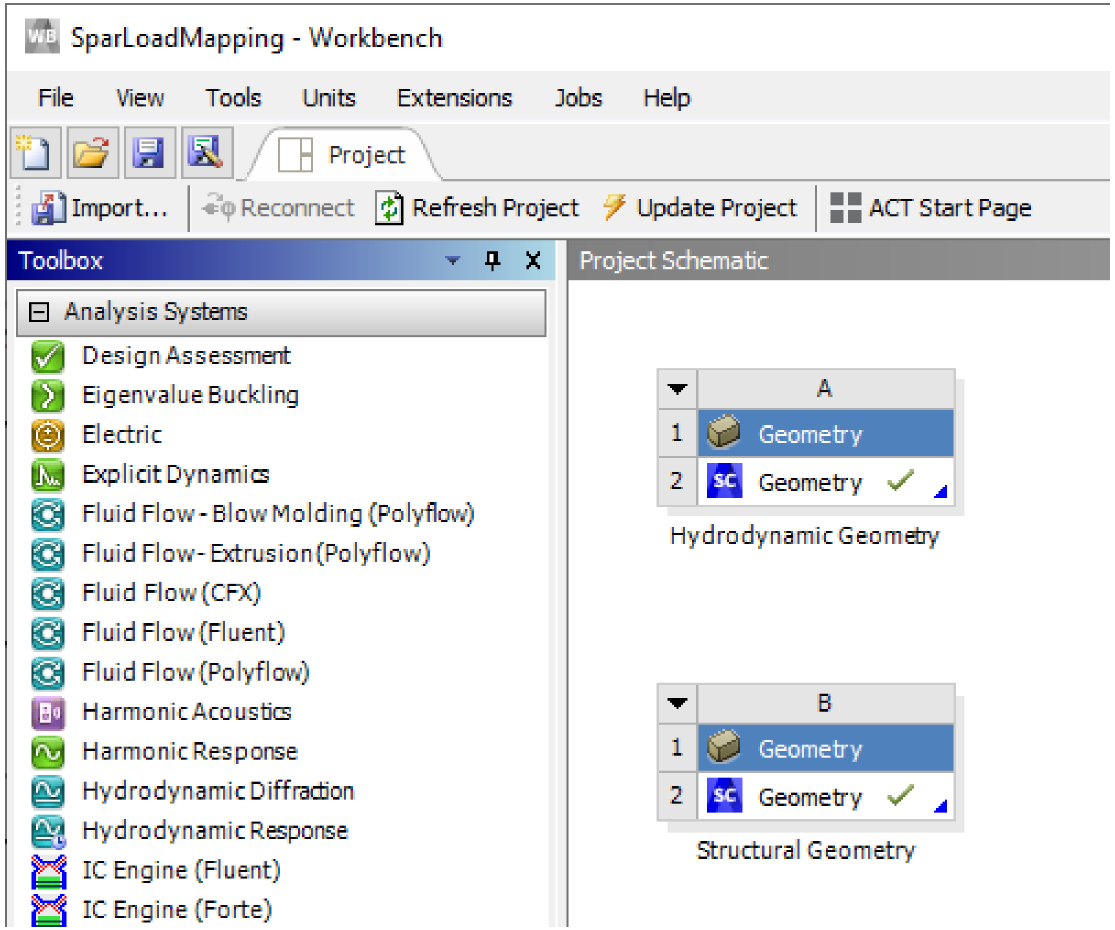
The steps of this exercise include:

- Set up the radiation/diffraction analysis
- Set up the structural model for the linear structural analysis
- Transfer the wave load to the structural model
- Review results



Project Schematic

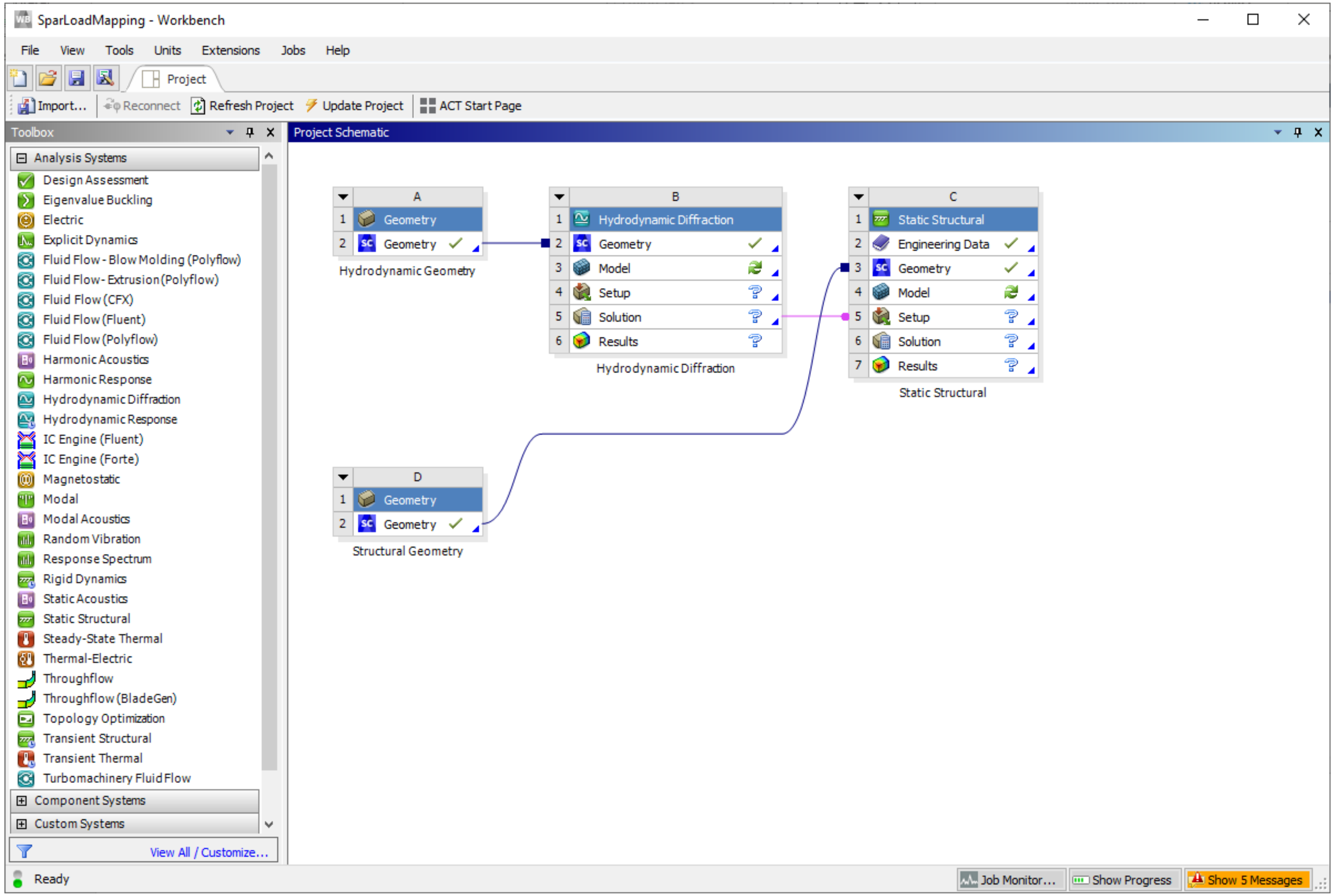
- Open Workbench
- Open the Workbench project **SparLoadMapping.wbpj**
- This project just contains two predefined Geometry cells
- The coordinate system for the Structural Geometry differs from that of the Hydrodynamic Geometry (Z position of origin)



Project Schematic

Complete the project workflow:

- **Drag and drop a Hydrodynamic Diffraction system on to the Hydrodynamic Geometry**
- Drag and Drop a Static Structural system on to the Structural Geometry
- **Drag the Hydrodynamic Diffraction Solution cell on to the Setup cell of the Static Structural system**
- Workbench will move the systems around to make the workflow as clear as possible



Radiation/Diffraction Analysis

- First we will set up the Hydrodynamic Diffraction analysis to generate the pressures and loads to be transferred to Mechanical.

- From the Project Schematic double-click on the Model cell of the Hydrodynamic Diffraction system.

Radiation/Diffraction Analysis

Set up the Mass and Inertia properties in the HD system.

Usually we would have the structure mass properties available to us; in this case the values have been pre-calculated by Ansys.

- Right-click on the Truss_Spar object and insert a Point Mass
- Go to the point mass properties and set Mass Definition to Manual, Define Inertia Values By to Direct Input of Inertia
- Enter the mass and inertia properties shown here. This point mass represents the contribution of the surface bodies only – the line bodies have their own mass and inertia properties (calculated by Aqwa)

Details

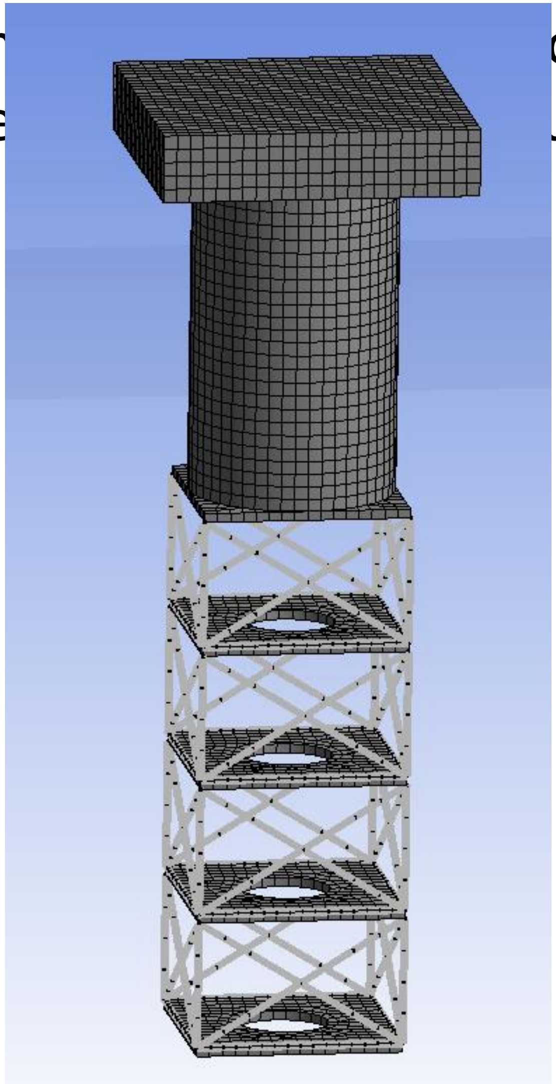
Details of Point Mass	
Name	Point Mass
Visibility	Visible
Activity	Not Suppressed
Point Mass Properties	
Mass Definition	Manual Definition
X	0.0 m
Y	0.0 m
Z	-7.71 m
Mass	39683000 kg
Inertia Properties	
Define Inertia Values By	Direct Input of Inertia
Kxx	69.3441289522382 m
Kyy	69.3441289522382 m
Kzz	28.241295086793 m
Ixx	190820000000 kg.m²
Ixy	0.0 kg.m²
Ixz	0.0 kg.m²
Iyy	190820000000 kg.m²
Iyz	0.0 kg.m²
Izz	31650000000 kg.m²

Radiation/Diffraction Analysis

When transferring pressures to a FEA model a finer mesh is norm ove
the load mapping process. For this example we will use larger me up
the solution process.

- Set Defeaturing Tolerance to 2m
- Set Maximum Element Size to 5m
- Generate mesh

Details	
Details of Mesh	
Defaults	
Control Type	Basic Controls
Mesh Parameters	
Defeaturing Tolerance	2 m
Maximum Element Size	5 m
Maximum Allowed Frequency	0.25 Hz
Meshing Type	Program Controlled
Generated Mesh Information	
Total Nodes	5451
Total Elements	5340
Diffracting Nodes	3137
Diffracting Elements	3140
Line Body Nodes	339
Line Body Elements	193
Field Points	0



Radiation/Diffraction Analysis

Select the Analysis Settings object and set the options as shown to help speed up the solution. QTFs are not required if the model is only to be used for pressure mapping.

Details	
Details of Analysis Settings	
Name	Analysis Settings
External Operation before Solving	None
External Operation after Solving	None
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 Fre...	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 Datab...	No
Generate AHD Pressure Output	No

Radiation/Diffraction Analysis

At this stage we need to define the load cases that we would like to analyse in the FEA model. Normally we would define many frequencies, but for brevity in this exercise we are going to define just one wave frequency.

- Go to the Wave Directions object. Leave the default settings since there is little computational overhead in the number of wave directions solved for in Aqwa.
- Go to the Wave Frequencies object. Select Manual Definition for the Range and provide a single period of 6 seconds.

Details

Details of Wave Directions	
Name	Wave Directions
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

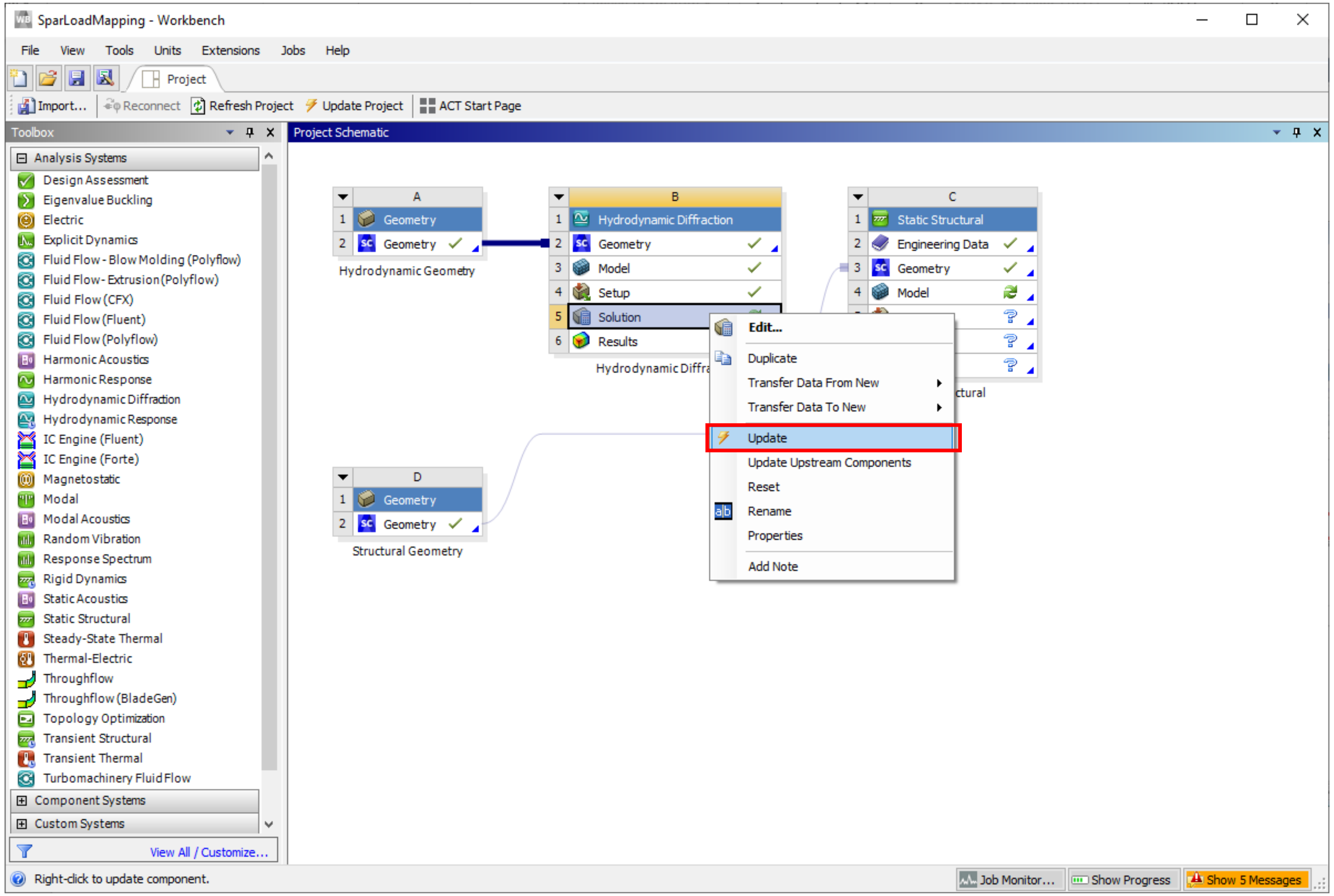
Details

Details of Wave Frequencies	
Name	Wave Frequencies
Intervals Based Upon	Frequency
Incident Wave Frequency/Period Definition	
Range	Manual Definition
Definition Type	Single
Lowest Frequency Definition	Manual Definition
Lowest Frequency	0.16667 Hz
Longest Period	6 s
Additional Frequencies A	
Additional Range	None

Radiation/Diffraction Analysis

We need to instigate the Hydrodynamic Diffraction solve from the Project Schematic page, so that Workbench will transfer the Hydrodynamic Diffraction results files to the Static Structural system.

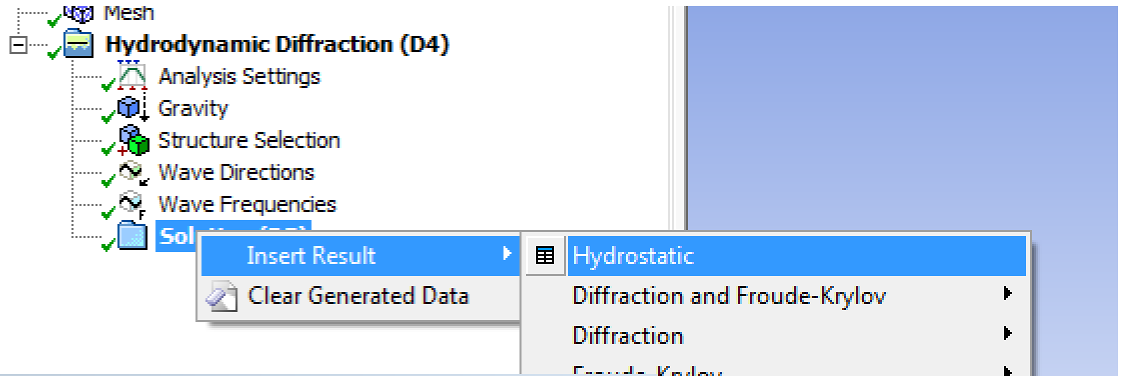
- Go back to the Project Schematic, right-click on the Hydrodynamic Diffraction Solution cell, and select Update.
- Once the solve is completed, return to the Aqwa Workbench editor.



Radiation/Diffraction Analysis

Review Results:

- Right-click on Solution to insert a Hydrostatic result and select the Truss_Spar as Structure.
- From the Tool Bar insert a Pressures and Motions result [Pressures and Motions] and set up as shown.
- Evaluate All Results [Evaluate All Results]
- Review results
- Save the Project and close HD



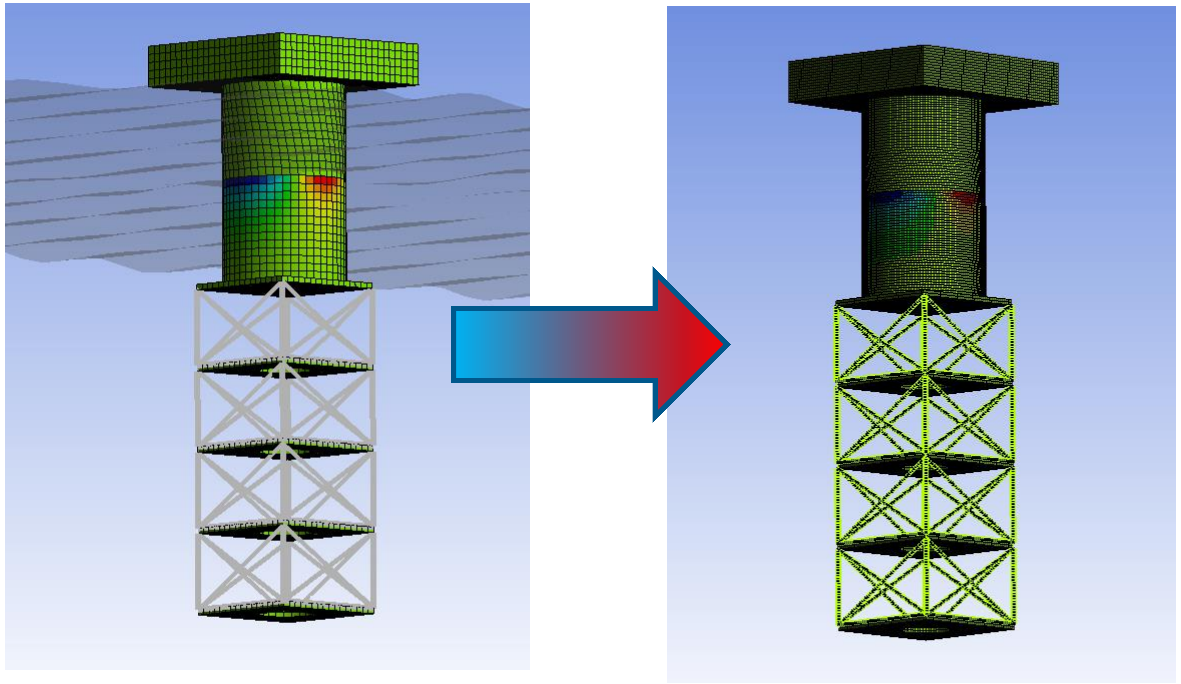
Details

Details of Pressures and Motions	
Name	Pressures and Motions
Structure Selection	Truss_Spar
Result Selection	
Frequency	0.16667 Hz
Direction	-180°
Incident Wave Amplitude	1 m
Result Type	Phase Angle
Wave Position (Phase)	0°
Contour Selection	
Contour Type	Structure Panel Pressure
Water Elements Shown As	Dimmed
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	No

Summary

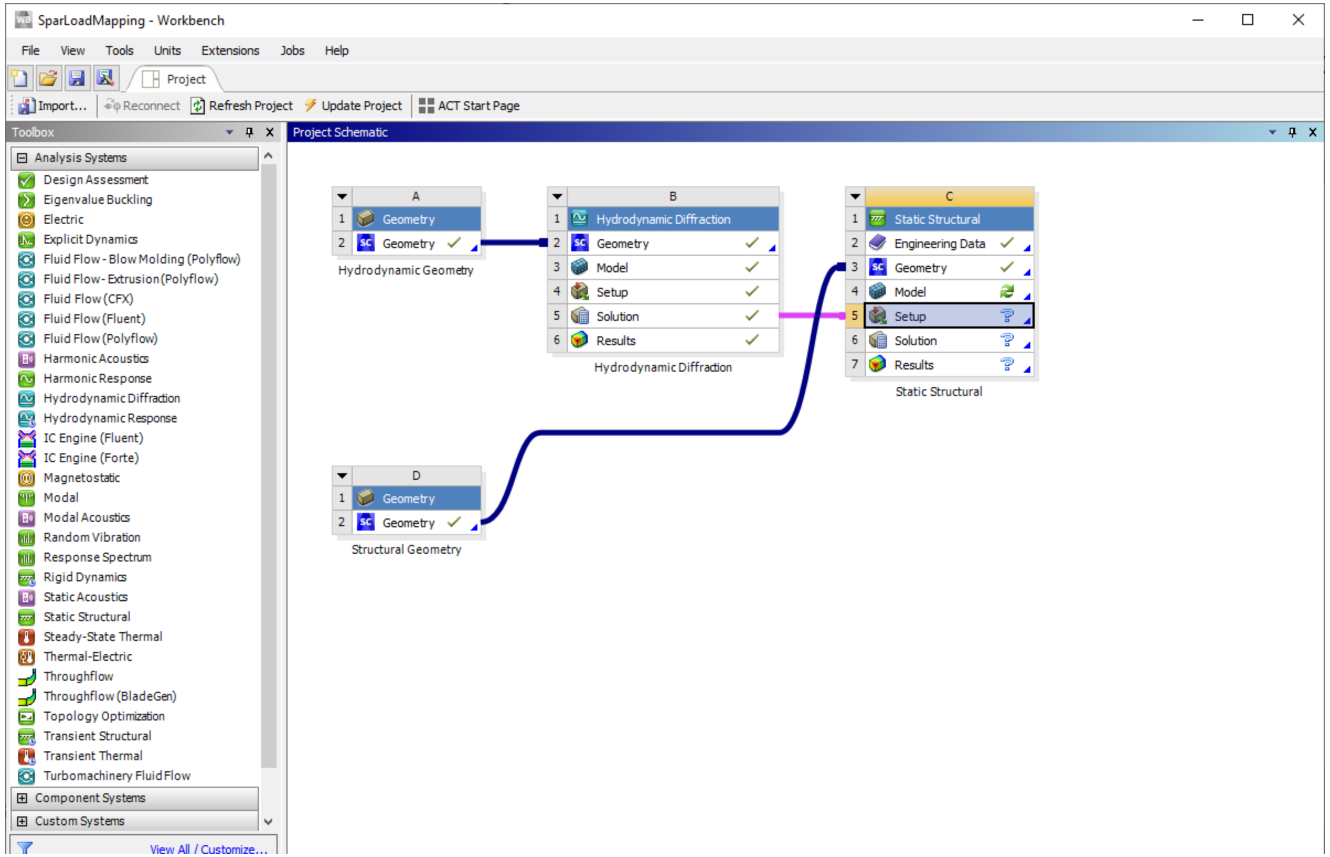
After solving the Hydrodynamic Diffraction solution we have available the pressures and motions for a selection of wave load cases.

The next step will be to transfer those loads to the FEA model.



Linear Structural Analysis

- Close the Aqwa Workbench editor. (Depending on your licensing arrangement, it may not be possible to have both Aqwa and Mechanical editors open at the same time.)
- From the Project Schematic double-click on the Setup cell of the Static Structural system.

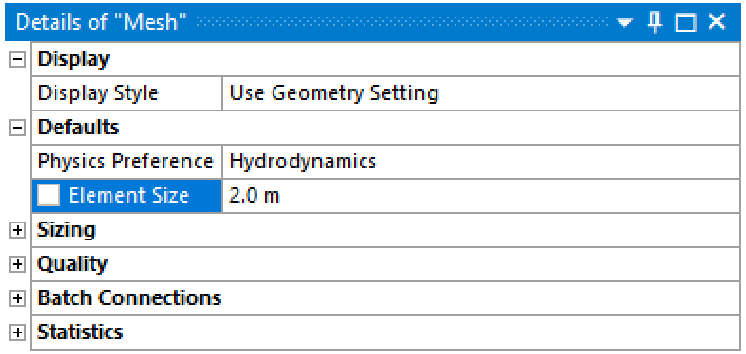


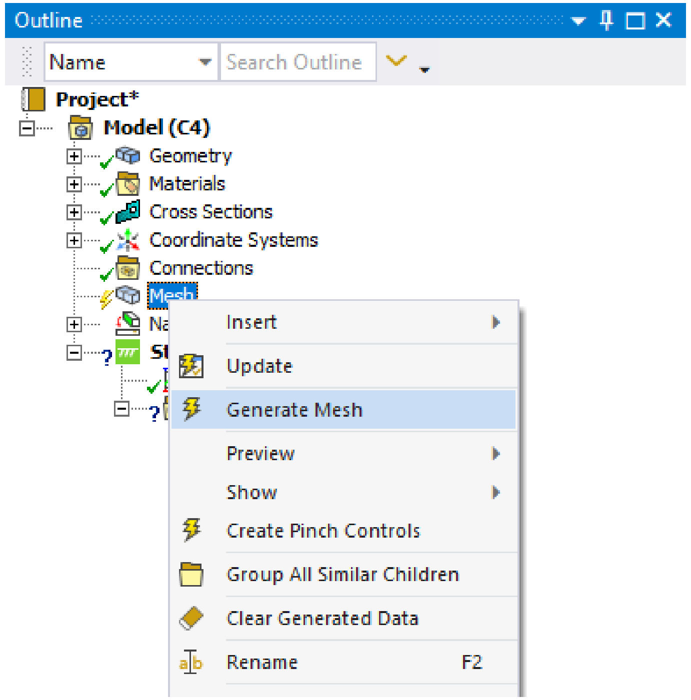
Ansys

Linear Structural Analysis

Create the structural mesh:

- Select the Mesh object in the Outline tree
- Set the Physics Preference to Hydrodynamic and the Element Size to 2.0 m.
- Right-click on the Mesh object and generate the mesh



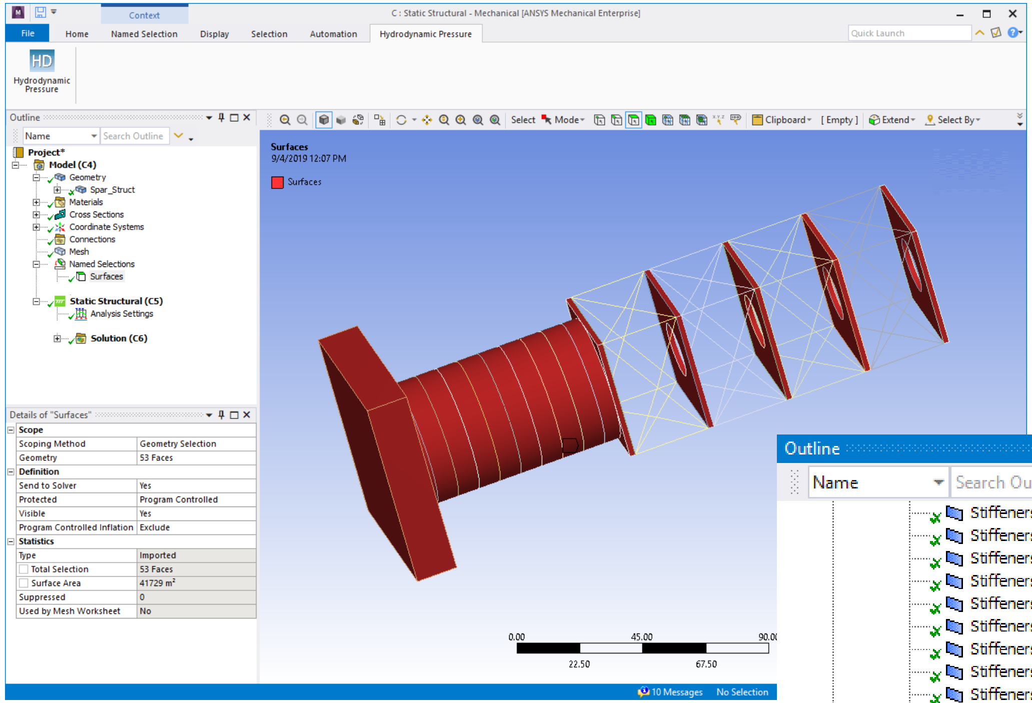


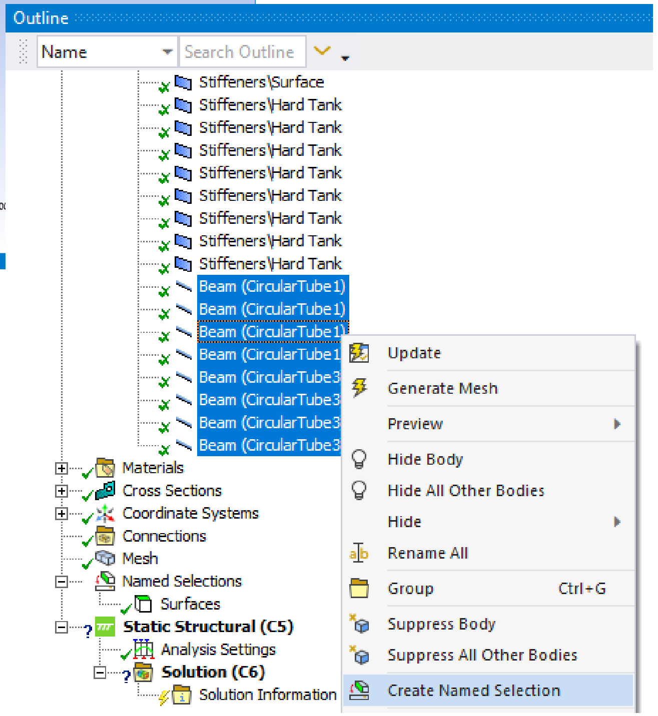
Linear Structural Analysis – Named Selections

We will need to select the external surfaces, and the beams, separately for the load mapping case definition.

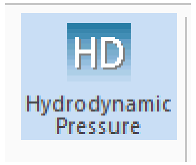
The external surfaces have already been grouped in the ‘Surfaces’ Named Selection

In the Outline tree, under Geometry, select all 8 Beam bodies, right-click and select Named Selection. Call the new Named Selection ‘Beams’





Linear Structural Analysis – Hydrodynamic Pressure

- Click on the Static Structural analysis in the Outline tree.
- On the top toolbar, click on the Hydrodynamic Pressure icon 

- A Hydrodynamic Pressure object will be added to the tree. In the background, the results files from the Hydrodynamic Diffraction analysis will have been copied into the Static Structural files directory and the hydrodynamic database will be read into Mechanical.

Linear Structural Analysis – Hydrodynamic Pressure

Click on the Hydrodynamic Pressure object. This object allows you to:

- Select the surfaces and/or line bodies on to which the hydrodynamic loads are mapped
- Select a wave frequency and direction from those available in the hydrodynamic database
- Set the wave amplitude and phase angle, and define a current for line body loads
- Select the mapping method and some pressure components
- Define an axis transformation, to account for different coordinate systems between Aqwa and Mechanical geometries
- Review the resulting pressures/loads/accelerations

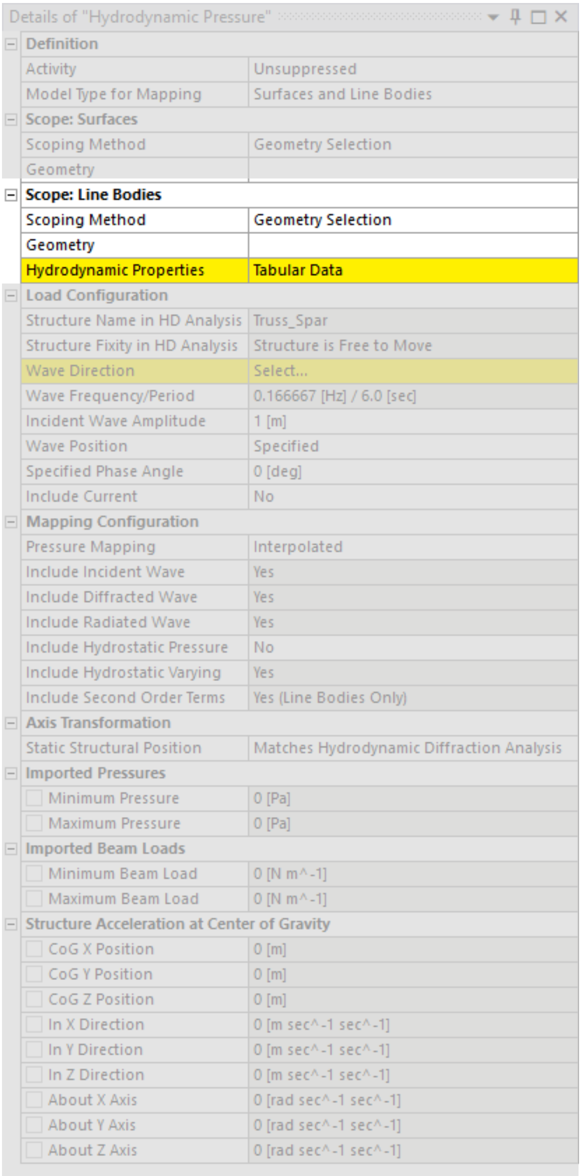
Details of "Hydrodynamic Pressure"

Definition	
Activity	Unsuppressed
Model Type for Mapping	Surfaces and Line Bodies
Scope: Surfaces	
Scoping Method	Geometry Selection
Geometry	
Scope: Line Bodies	
Scoping Method	Geometry Selection
Geometry	
Hydrodynamic Properties	Tabular Data
Load Configuration	
Structure Name in HD Analysis	Truss_Spar
Structure Fixity in HD Analysis	Structure is Free to Move
Wave Direction	Select...
Wave Frequency/Period	0.166667 [Hz] / 6.0 [sec]
Incident Wave Amplitude	1 [m]
Wave Position	Specified
Specified Phase Angle	0 [deg]
Include Current	No
Mapping Configuration	
Pressure Mapping	Interpolated
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiated Wave	Yes
Include Hydrostatic Pressure	No
Include Hydrostatic Varying	Yes
Include Second Order Terms	Yes (Line Bodies Only)
Axis Transformation	
Static Structural Position	Matches Hydrodynamic Diffraction Analysis
Imported Pressures	
Minimum Pressure	0 [Pa]
Maximum Pressure	0 [Pa]
Imported Beam Loads	
Minimum Beam Load	0 [N m^-1]
Maximum Beam Load	0 [N m^-1]
Structure Acceleration at Center of Gravity	
CoG X Position	0 [m]
CoG Y Position	0 [m]
CoG Z Position	0 [m]
In X Direction	0 [m sec^-1 sec^-1]
In Y Direction	0 [m sec^-1 sec^-1]
In Z Direction	0 [m sec^-1 sec^-1]
About X Axis	0 [rad sec^-1 sec^-1]
About Y Axis	0 [rad sec^-1 sec^-1]
About Z Axis	0 [rad sec^-1 sec^-1]

Linear Structural Analysis – Hydrodynamic Pressure

- In Scope: Surfaces, change the Scoping Method to Named Selection. Set the Named Selection to ‘Surfaces’.

- In Scope: Line Bodies, change the Scoping Method to Named Selection. Set the Named Selection to ‘Beams’.
- A new Hydrodynamic Properties field will appear: click on Tabular Data, and review the tube properties (drag and added mass coefficients) for the tube cross sections included in the selection. Click ‘Apply’ to accept the pre-defined coefficient values.



Ansys

Linear Structural Analysis – Hydrodynamic Pressure

- Under Load Configuration, we just need to select a Wave Direction from those directions that were included in the Hydrodynamic Diffraction calculation.
- Select 180°.
- The Incident Wave Amplitude and Phase Angle can be left as 1 m and 0° for now, but try changing them once the workshop is completed to see the effect.
- Leave the Include Current option as No.

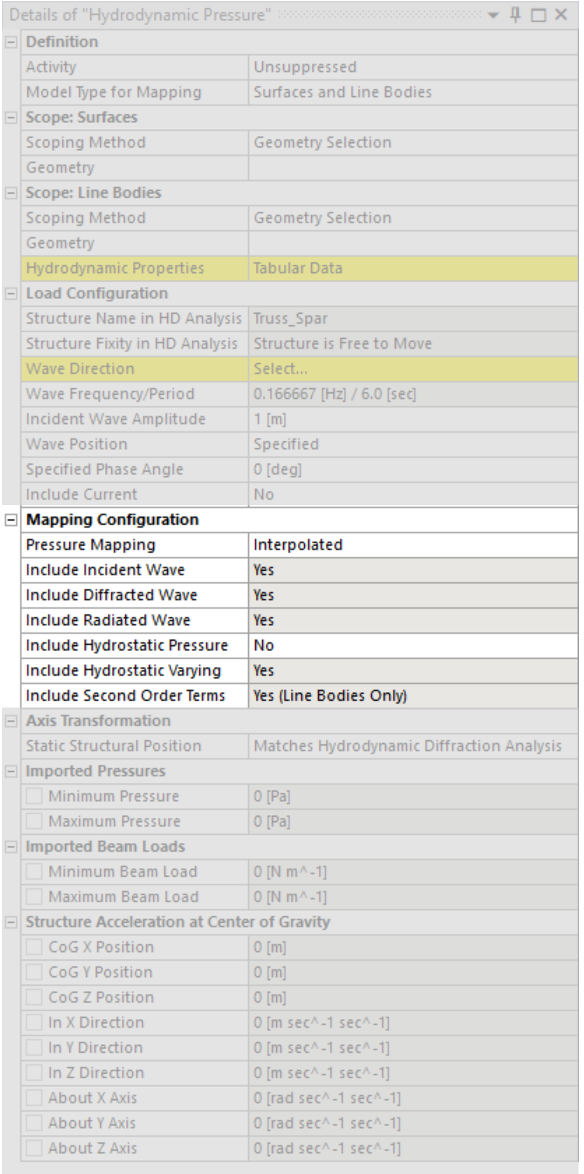
Details of "Hydrodynamic Pressure"

Definition	
Activity	Unsuppressed
Model Type for Mapping	Surfaces and Line Bodies
Scope: Surfaces	
Scoping Method	Geometry Selection
Geometry	
Scope: Line Bodies	
Scoping Method	Geometry Selection
Geometry	
Hydrodynamic Properties	Tabular Data
Load Configuration	
Structure Name in HD Analysis	Truss_Spar
Structure Fixity in HD Analysis	Structure is Free to Move
Wave Direction	Select...
Wave Frequency/Period	0.166667 [Hz] / 6.0 [sec]
Incident Wave Amplitude	1 [m]
Wave Position	Specified
Specified Phase Angle	0 [deg]
Include Current	No
Mapping Configuration	
Pressure Mapping	Interpolated
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiated Wave	Yes
Include Hydrostatic Pressure	No
Include Hydrostatic Varying	Yes
Include Second Order Terms	Yes (Line Bodies Only)
Axis Transformation	
Static Structural Position	Matches Hydrodynamic Diffraction Analysis
Imported Pressures	
Minimum Pressure	0 [Pa]
Maximum Pressure	0 [Pa]
Imported Beam Loads	
Minimum Beam Load	0 [N m^-1]
Maximum Beam Load	0 [N m^-1]
Structure Acceleration at Center of Gravity	
CoG X Position	0 [m]
CoG Y Position	0 [m]
CoG Z Position	0 [m]
In X Direction	0 [m sec^-1 sec^-1]
In Y Direction	0 [m sec^-1 sec^-1]
In Z Direction	0 [m sec^-1 sec^-1]
About X Axis	0 [rad sec^-1 sec^-1]
About Y Axis	0 [rad sec^-1 sec^-1]
About Z Axis	0 [rad sec^-1 sec^-1]

Linear Structural Analysis – Hydrodynamic Pressure

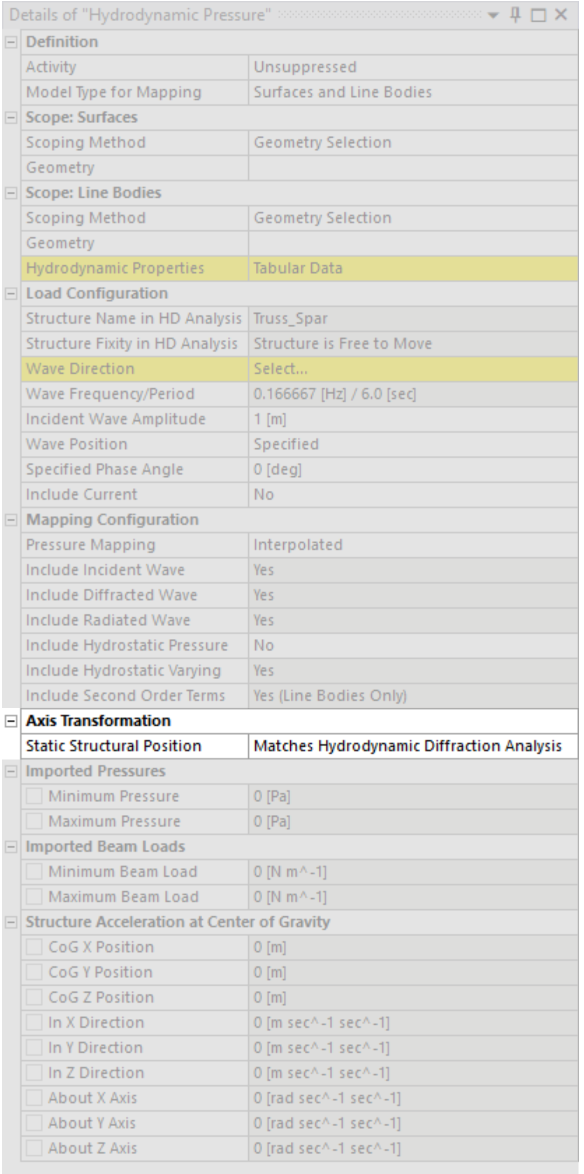
- Under Mapping Configuration, we must leave the Pressure Mapping method as Interpolated. The other option – Direct mapping – cannot be used with line bodies.

- The pressure components are mostly shown for information only (they cannot be turned off) but we can choose to include or exclude Hydrostatic pressure. Leave this as No for now, but try adding the Hydrostatic component once the workshop is completed to see the effect.



Linear Structural Analysis – Hydrodynamic Pressure

- The axis system in Aqwa is defined with the global vertical origin (Z = 0) positioned at the waterline, but in the structural geometry we have Z = 0 at the lowest point on the structure.
- To account for this difference in axis systems set Static Structural Position to Differs from Hydrodynamic Diffraction Analysis, and define a Structure Position Offset Z of -160 m.
- Note that axis system rotations can also be defined (in some structural models Y will point vertically upwards, but in Aqwa it is always Z that is vertical).



Linear Structural Analysis – Hydrodynamic Pressure

- The completed Hydrodynamic Pressure definition is shown here.

- Right-click on the Hydrodynamic Pressure object and click Generate. The process may take a couple of minutes.

- Once it is complete, you should see a distribution of pressures displayed in the Graphical window. The Imported Pressures/Beam Loads and Structure Accelerations should also be shown in the Hydrodynamic Pressure object details.

Details of "Hydrodynamic Pressure"

Definition	
Activity	Unsuppressed
Model Type for Mapping	Surfaces and Line Bodies
Scope: Surfaces	
Scoping Method	Named Selection
Named Selection	Surfaces
Scope: Line Bodies	
Scoping Method	Named Selection
Named Selection	Beams
Hydrodynamic Properties	Tabular Data
Load Configuration	
Structure Name in HD Analysis	Truss_Spar
Structure Fixity in HD Analysis	Structure is Free to Move
Wave Direction	-180.0 [deg]
Wave Frequency/Period	0.166667 [Hz] / 6.0 [sec]
Incident Wave Amplitude	1 [m]
Wave Position	Specified
Specified Phase Angle	0 [deg]
Include Current	No
Mapping Configuration	
Pressure Mapping	Interpolated
Include Incident Wave	Yes
Include Diffracted Wave	Yes
Include Radiated Wave	Yes
Include Hydrostatic Pressure	No
Include Hydrostatic Varying	Yes
Include Second Order Terms	Yes (Line Bodies Only)
Axis Transformation	
Static Structural Position	Differs from Hydrodynamic Diffraction A...
Structure Position Offset X	0 [m]
Structure Position Offset Y	0 [m]
Structure Position Offset Z	-160 [m]
Structure Rotation Offset RX	0 [deg]
Structure Rotation Offset RY	0 [deg]
Structure Rotation Offset RZ	0 [deg]

Linear Structural Analysis – Weak Springs

In the Analysis Settings for Static Structural, set Weak Springs On. This is used to prevent singularities in the solution since there are no other constraints on the model.

Details of "Analysis Settings"

Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	On
Spring Stiffness	Program Controlled
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off

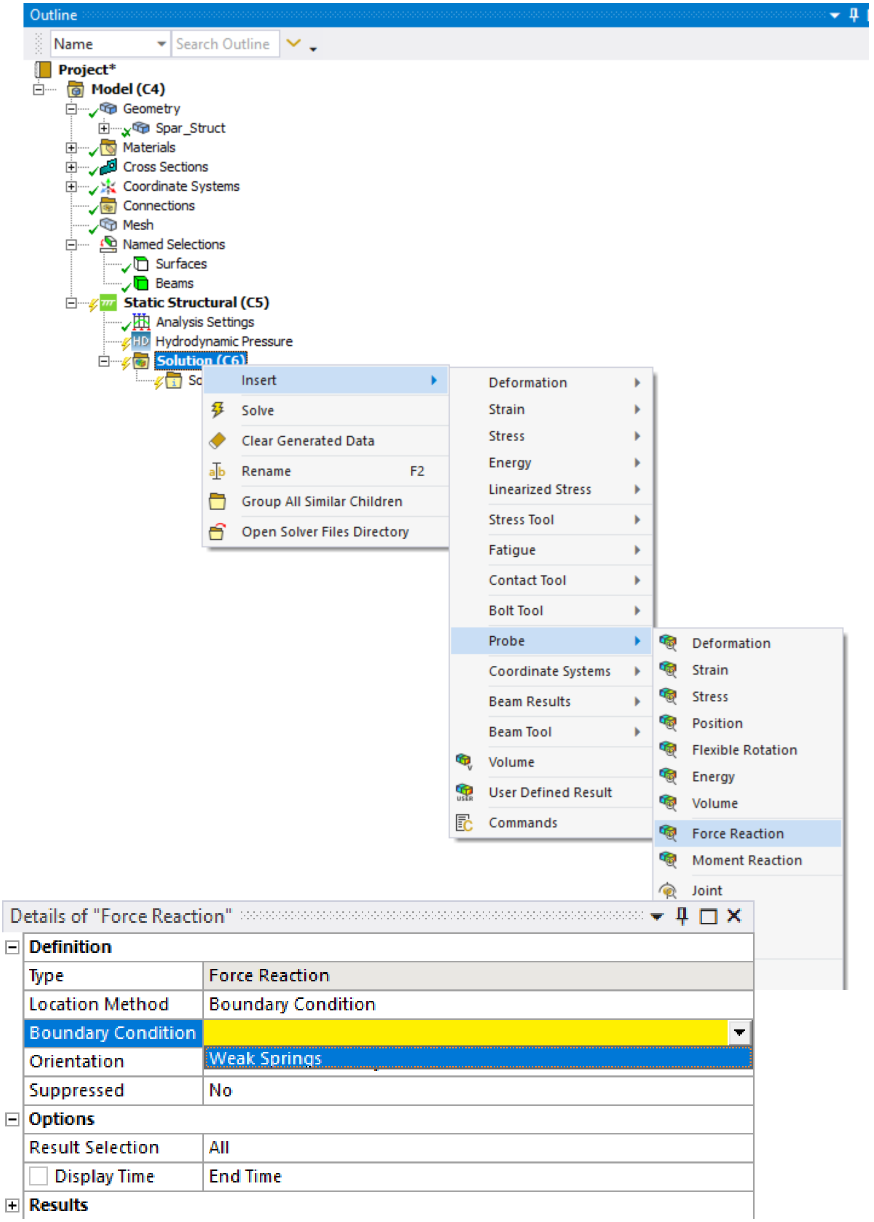
Linear Structural Analysis

The Static Structural analysis is now ready to be solved – click Solve on the top toolbar.

To see how well the load mapping has been undertaken we can:

- Check the reactions at the weak spring supports
- Look at the mapped pressures and compare with those displayed in the Hydrodynamic Diffraction system

Right-click on the Solution for Static Structural and insert a Force Reaction probe. Select Weak Springs as the Boundary Condition



Results Review

- Open the Aqwa Hydrodynamic Diffraction system and select the Pressures and Motions result. The settings that were previously input correspond to the zero phase condition we have just reported in the Mechanical model.

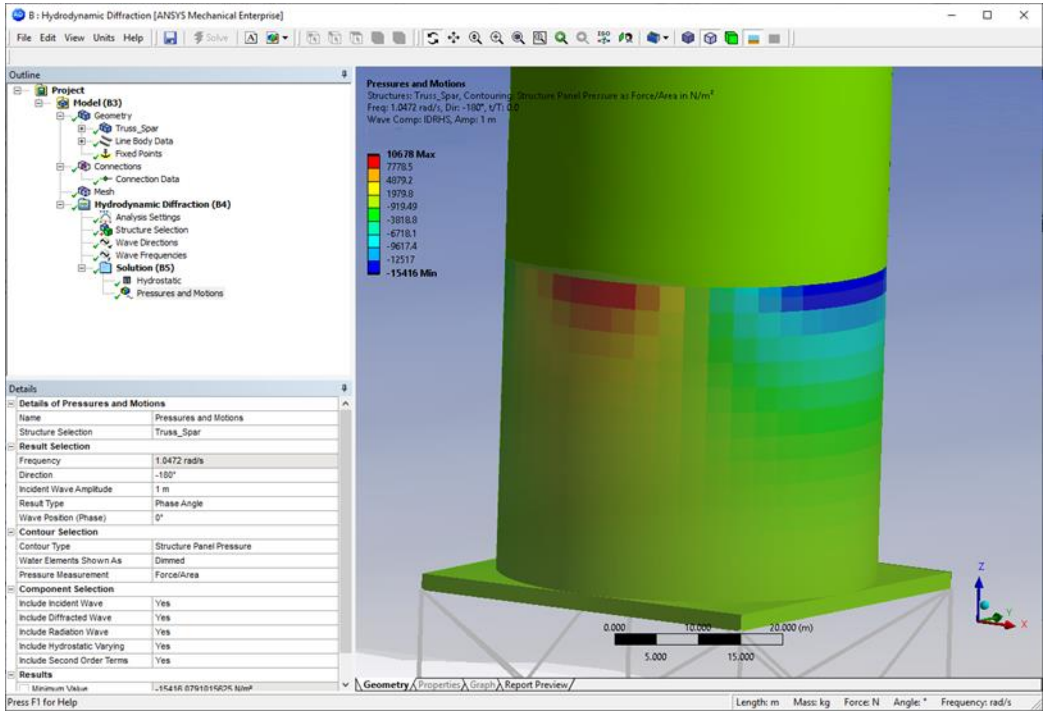
Details

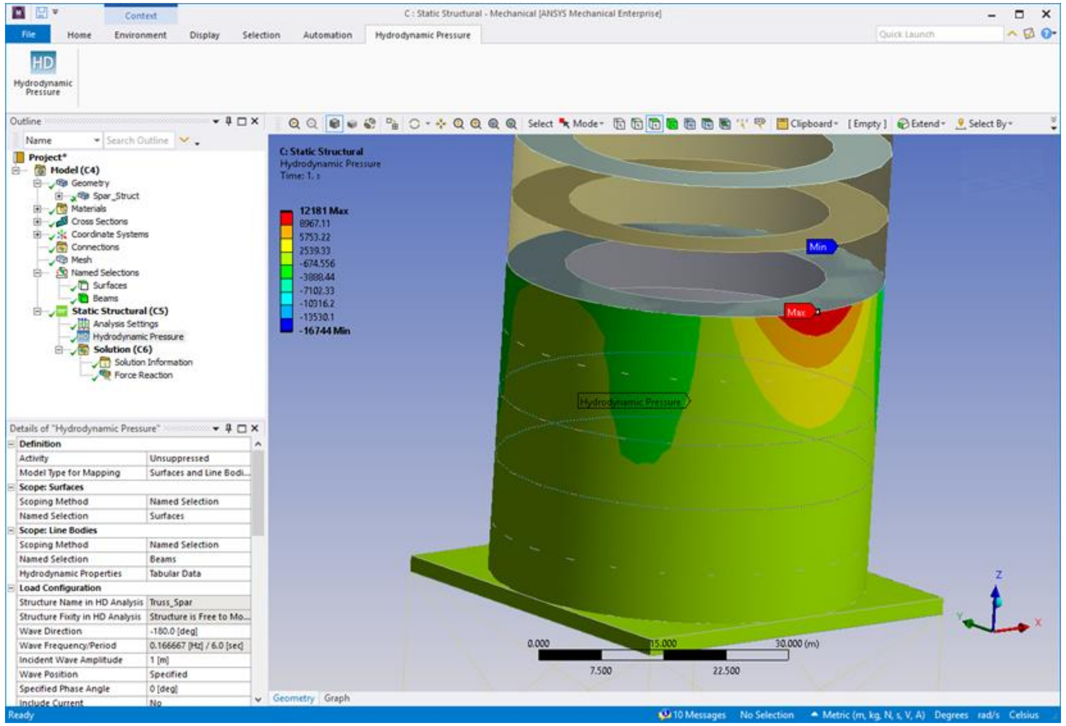
Details of Pressures and Motions	
Name	Pressures and Motions
Structure Selection	Truss_Spar
Result Selection	
Frequency	0.16667 Hz
Direction	-180°
Incident Wave Amplitude	1 m
Result Type	Phase Angle
Wave Position (Phase)	0°
Contour Selection	
Contour Type	Structure Panel Pressure
Water Elements Shown As	Dimmed
Pressure Measurement	Force/Area

Pressures and Motions
Structures: Truss_Spar, Contouring: Structure Panel Pressure as Force/Area in N/m^2
Freq: 0.16667 Hz, Dir: -180°, t/T: 0.0
Wave Comp: IDRHS, Amp: 1 m

Results Review

- You will observe the same general pattern of the loading. Also note that the maximum and minimum values reported are similar, but not exactly the same. This is due in part to the coarse mesh we employed in the Aqwa model, and the interpolation that is undertaken during the mapping process. Much better agreement would have been obtained with a more refined hydrodynamic mesh.





Results Review

- Back in the Static Structural system, click Evaluate All Results to update the Force Reaction result.
- Select the Force Reaction in the Outline tree. In the Tabular Data we can see the reaction forces in X, Y and Z – these represent the out-of-balance forces on the model.
- The X, Y and Z values are relatively small, compared to the self-weight of the structure (approx. 577 MN).

Results Review

- We can also check the magnitude of the out-of-balance forces with the total wave loading force generated by Aqwa.
- In the Aqwa Workbench editor, right-click on the HD Solution and Insert Result > Diffraction and Froude-Krylov > Force/Moment vs Frequency
- Set result Component to Global X. Evaluate All Results
- The force amplitude for the frequency of 0.16667 Hz (6 seconds) is around 6.3 MN. This is significantly larger that the maximum out-of-balance force in Mechanical.