Beta Documentation

Beta documentation and features are considered unreleased and have not been fully tested nor fully validated. The results are not guaranteed by ANSYS, Inc. (ANSYS) to be correct. You assume the risk of using beta features and beta documentation

At its discretion, ANSYS, Inc. (ANSYS) may release, change, or withdraw beta features in future revisions. Beta features are not subject to the ANSYS Class 3 error reporting system. ANSYS makes no commitment to resolve defects reported against beta features; however, your feedback will help us improve the quality of the product

ANSYS does not guarantee that input files used with beta features will run successfully from to version of the software, nor with the final released version of the features. You may need to the input files before running them on other versions.

Hydrodynamic Pressure Mapping (Beta Feature)

The Hydrodynamic Pressure Mapping ACT Extension is designed to facilitate the transfer of hydrodynamic pressures calculated in a Hydrodynamic Diffraction system to a Static Structural finite element analysis.

The hydrodynamic pressure terms are linear (first order), and include incident, diffracted, radiated and hydrostatic-varying components. The wave direction and frequency are selected from those included in the Hydrodynamic Diffraction calculation; incident wave amplitude and phase angle are also specified. Hydrostatic pressure may be added if required. The summed pressure terms are mapped on to triangular and quadrilateral panel elements in the Static Structural mesh.

The Hydrodynamic Pressure Mapping ACT Extension also allows the first order (Froude-Krylov, added mass) and second order (viscous drag) forces acting on Line Bodies (element types BEAM188 and PIPE288) to be included in a Static Structural analysis. The effect of ocean current may be accounted for in the drag calculation.

In addition to hydrodynamic pressures and beam loads, resultant translational and rotational accelerations about the structure center of gravity are determined for the balance of inertial loads in the Static Structural analysis.

Installing the Hydrodynamic Pressure Mapping ACT Extension

The Hydrodynamic Pressure Mapping ACT Extension is pre-installed with ANSYS Workbench, but must be loaded into Workbench before it can be used. To do so:

1. Start Workbench
2. Select **Extensions > Manage Extensions...**
3. In the Extensions Manager window, select the check box next to `AqwaLoadMapping`, then click **Close**.

The Hydrodynamic Pressure Mapping ACT Extension is configured so that it is saved with an ANSYS Workbench project. This means that you do not need to manually load the extension before you open a project or archive that was created with the extension loaded.

Hydrodynamic Pressure Mapping in the Workbench Context – Setting up a Workflow

Once the Hydrodynamic Pressure Mapping ACT Extension has been loaded into ANSYS Workbench, you will find a new *Static Structural with Hydrodynamic Pressure (Beta)* analysis system in the Toolbox pane. The Solution cell of a Hydrodynamic Diffraction system can be connected to the Hydrodynamic Pressure setup cell of the Static Structural with Hydrodynamic Pressure (Beta) system. A typical workflow is shown in Fig. 1. It is not necessary to have different Geometry sources in each system, but for the typical use case the Static Structural model will include internal and/or other structural components that are not required for the Hydrodynamic Diffraction calculation. The Static Structural with Hydrodynamic Pressure (Beta) system is otherwise identical to a regular Static Structural system.

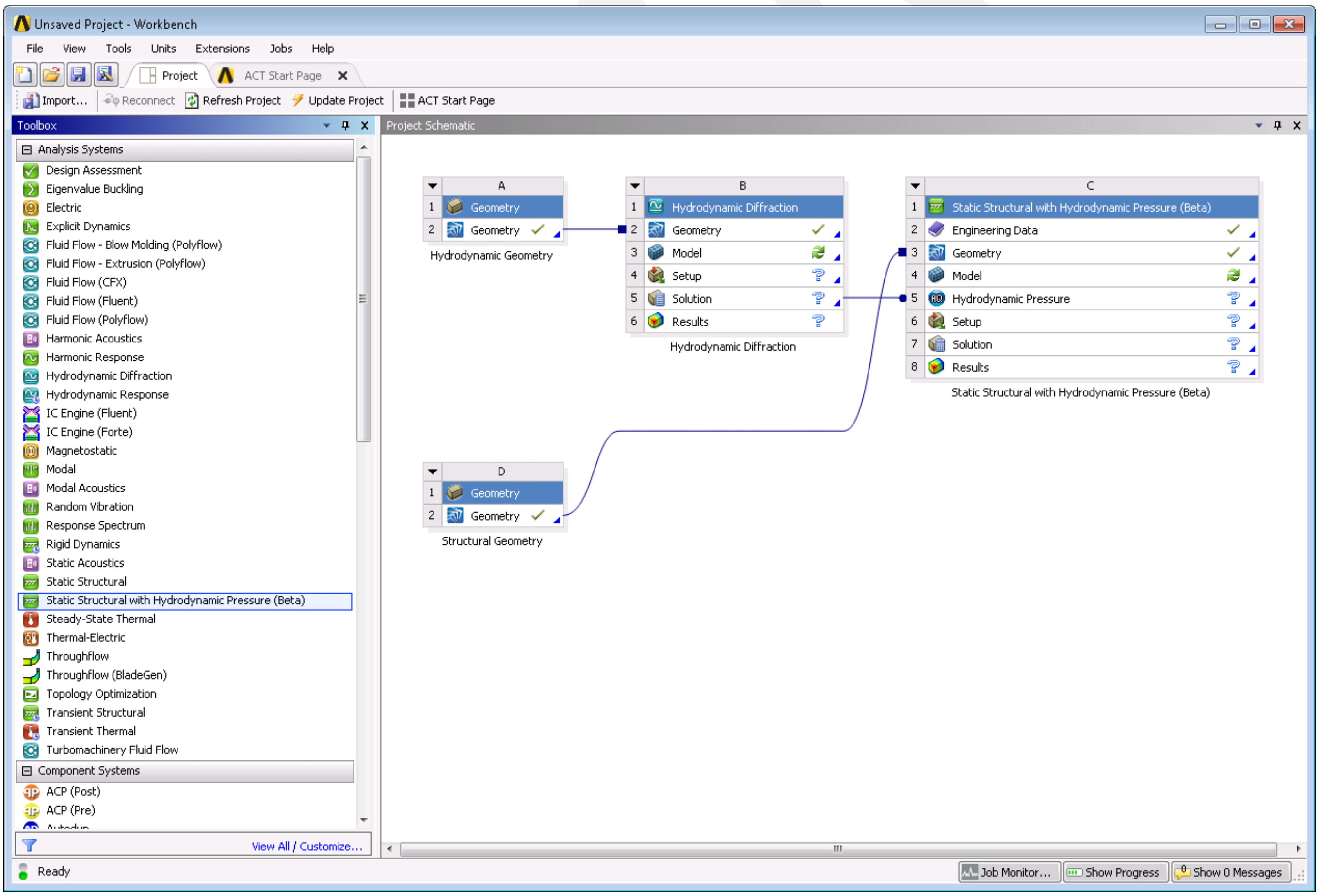

Figure 1: Typical workflow for Hydrodynamic Pressure Mapping

Opening the Static Structural with Hydrodynamic Pressure (Beta) system in the Mechanical editor, you will find an additional toolbar is available:

Clicking on the icon adds a Hydrodynamic Pressure object to the Static Structural analysis.

Alternatively, you can right-click on the Static Structural analysis and *Insert > Hydrodynamic Pressure (Beta)* from the context menu. Note that it is only possible to add one Hydrodynamic Pressure object to a Static Structural analysis.

Configuring the Hydrodynamic Pressure Object

In the Details of “Hydrodynamic Pressure (Beta)” panel (Fig. 2) you will find options that allow you to configure the Hydrodynamic Pressure object. These are described below.

Details of "Hydrodynamic Pressure (Beta)"

Details of Hydrodynamic Pressure	
Name	Hydrodynamic Pressure
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	Line Bodies
Hydrodynamic Properties	Tabular Data
Load Configuration	
Structure Name in HD Analysis	SemiSub
Structure Fixity in HD Analysis	Structure is Free to Move
Forward Speed in HD Analysis	2 [m sec^-1]
Wave Direction	30.0 [deg]
Wave Frequency / Period	0.2 [Hz] / 5.0 [sec]
Encounter Frequency	0.155611 [Hz]
Incident Wave Amplitude	2.4 [m]
Phase Angle	60 [deg]
Include Current	Yes
Current Profile	Tabular Data
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 Analysis
Structure Position Offset X	0 [m]
Structure Position Offset Y	0 [m]
Structure Position Offset Z	-22 [m]
Structure Rotation Offset RX	90 [deg]
Structure Rotation Offset RY	0 [deg]
Structure Rotation Offset RZ	0 [deg]
Imported Pressures	
Minimum Pressure	-30244 [Pa]
Maximum Pressure	33854 [Pa]
Imported Beam Loads	
Minimum Beam Load	4413.80039354982 [N m^-1]
Maximum Beam Load	58424.6879666466 [N m^-1]
Structure Acceleration at Center of Gravity	
CoG X Position	50 [m]
CoG Y Position	7.6735 [m]
CoG Z Position	1.2677E-05 [m]
In X Direction	0.031305 [m sec^-1 sec^-1]
In Y Direction	-0.0064155 [m sec^-1 sec^-1]
In Z Direction	-0.04545 [m sec^-1 sec^-1]
About X Axis	-0.0014739 [rad sec^-1 sec^-1]
About Y Axis	-0.0010069 [rad sec^-1 sec^-1]
About Z Axis	0.0025642 [rad sec^-1 sec^-1]

Figure 2: Details of "Hydrodynamic Pressure (Beta)"

Details of Hydrodynamic Pressure

The **Activity** field allows you to set the suppression state of the Hydrodynamic Pressure object. Set to Suppressed if you want to exclude the object from the analysis.

Model Type sets the types of loads that you want to map from the Hydrodynamic Diffraction analysis to the Static Structural analysis. Generally, it is possible to map pressures on to surfaces (Surface Bodies) or distributed loads on to beams (Line Bodies). The **Model Type** selection is limited to the geometry types that exist in the imported geometry.

Scope: Surfaces

When **Model Type** is set to *Surfaces Only* or *Surfaces and Line Bodies*, use Scope: Surfaces to select the surfaces that you want to map hydrodynamic pressures on to. Surfaces can be selected either by Named Selection, or by Geometry Selection from the graphical window.

Scope: Line Bodies

When **Model Type** is set to *Line Bodies Only* or *Surfaces and Line Bodies*, use Scope: Line Bodies to select the beams that you want to map hydrodynamic loads on to. Line Bodies can be selected either by Named Selection, or by Geometry Selection from the graphical window. You must also define viscous drag and inertia coefficients for the different cross sections that are included in your selection. These coefficients are used to calculate distributed loads on beams according to the Morison equation described in Section 5: Theory. The **Hydrodynamic Properties** table sets coefficients for each cross section type. For non-cylindrical cross sections (solid rectangular and rectangular tube) you must define separate viscous drag and inertia coefficients for each transverse direction.

Load Configuration

When your Hydrodynamic Diffraction analysis contains more than one structure, you must select the name of the structure (in the Hydrodynamic Diffraction system) that you intend to map loads from using the **Structure Name in HD Analysis** option.

The fixity of the selected structure in the Hydrodynamic Diffraction analysis is indicated by **Structure Fixity in HD Analysis**. When this field shows *Structure is Fixed in Place*, the radiated wave pressure component will be zero and no structure accelerations will be calculated. In this case, for consistency, the structure should also be fixed in the Static Structural analysis (for example, by a Fixed Support). If the *Structure is Free to Move* you are recommended to use the Weak Springs or Inertia Relief options (in the Static Structural Analysis Settings) to balance the resultant accelerations acting on the structure.

If a forward speed has been defined in the Hydrodynamic Diffraction analysis, this is shown in **Forward Speed in HD Analysis**.

The **Wave Direction** and **Wave Frequency / Period** fields are used to select a frequency/direction pair at which the hydrodynamic pressures acting on the structure will be evaluated. If a forward speed has been defined in the Hydrodynamic Diffraction analysis the **Encounter Frequency** for this frequency/direction pair is also shown. The **Incident Wave Amplitude** and **Phase Angle** fields define the other properties of the incident wave.

When **Model Type** is set to *Line Bodies Only* or *Surfaces and Line Bodies*, a profiled current can be included. To do so, set **Include Current** to *Yes* and use the **Current Profile** table to define current speed and direction data at depths below the waterline. If current speed and direction are entered for a single depth only, a 'slab' profile (constant with depth) is assumed; otherwise, the current velocity at the position of a Line Body element is linearly interpolated from the defined data.

<u>Mapping Configuration</u>

The meshes in the Static Structural and Hydrodynamic Diffraction systems are generally not coincident, which necessitates some approach to map the hydrodynamic pressures on to the structural mesh. The **Pressure Mapping** option can be set to *Interpolated* or *Direct*.

Using the interpolated method, hydrodynamic pressures at each node of the selected surfaces in the structural mesh are interpolated from the element-centered pressures calculated in the Hydrodynamic Diffraction analysis. This method can be used for models where forward speed is defined in the Hydrodynamic Diffraction analysis, and for models that include Line Bodies in the load mapping selection. However, the accuracy of the method is inherently limited by the interpolation of the pressure data and is dependent on the resolution of the mesh in the Hydrodynamic Diffraction analysis. Furthermore, the interpolated method cannot be used where the Hydrodynamic Diffraction analysis contains more than one structure.

Where the direct method is used, hydrodynamic pressure components at the position of each selected node are evaluated directly from the diffracting panel source strengths calculated in the Hydrodynamic Diffraction analysis. If the Hydrodynamic Diffraction analysis contains multiple structures, it is possible to use the direct method to map pressures on to one of those structures in the Static Structural system. This method also avoids the loss of accuracy due to interpolation, but cannot be used to map loads on to Line Bodies and is incompatible with Hydrodynamic Diffraction analyses that include forward speed.

Regardless of the selected method the mapped hydrodynamic loads will always include incident, diffracted and radiated components, as indicated by the **Include Incident**/**Diffracted**/**Radiated Wave** fields. Hydrostatic pressure can be optionally added by setting **Include Hydrostatic Pressure** to *Yes*. The hydrostatic-varying pressure component is always included when the interpolated method is used but is optional for the direct method. Second order pressure terms on diffracting panels are not calculated, but Line Body loads do include a second order viscous drag component.

Axis Transformation

Although you can share one Geometry source between the Hydrodynamic Diffraction and Static Structural systems, in the typical use case the Static Structural model will include internal and/or other structural components that are not required for the Hydrodynamic Diffraction calculation. Depending on the context, the geometries in the Static Structural and Hydrodynamic Diffraction systems may therefore employ different axis systems. To account for this, it is possible to define an axis transformation from the Static Structural analysis to the Hydrodynamic Diffraction analysis.

Where the axis systems are consistent between the two analyses, the **Static Structural Position** may be set as *Matches Hydrodynamic Diffraction Analysis*. Otherwise, this option should be set to *Differs from Hydrodynamic Diffraction Analysis*, and **Structure Position/Rotation Offsets** should be defined for the translational and rotational freedoms.

Output: Imported Pressures, Imported Beam Loads, and Structure Acceleration at Center of Gravity

When **Model Type** is set to *Surfaces Only* or *Surfaces and Line Bodies*, the **Minimum** and **Maximum Pressures** on diffracting panel elements are reported. When **Model Type** is set to *Line Bodies Only* or *Surfaces and Line Bodies*, the **Minimum** and **Maximum Beam Loads** (as Force per unit Length) are also shown. For all cases the position of the structure's center of gravity (in the Static Structural axis system), and the resultant translational and rotational accelerations about that center of gravity, are displayed.

Generating Hydrodynamic Pressures and Inertial Loads

Once the Hydrodynamic Pressure object has been configured as required, right-click on the Hydrodynamic Pressure object and select Generate to start the load transfer process. This involves three main steps:

- Using the ANSYS solver to determine the structural mass properties, which are compared to the mass properties from the Hydrodynamic Diffraction calculation;
- Reading the output of the Hydrodynamic Diffraction calculation to determine hydrodynamic pressures at the structural mesh nodes and calculate translational and rotational accelerations to balance inertial loads;

- Writing a hydrodynamic pressures file which is utilized when the Static Structural analysis is solved.

It should be noted that the Hydrodynamic Pressure Mapping ACT Extension will take into account any difference in the unit systems that are employed in the Hydrodynamic Diffraction and Static Structural systems.

The transferred pressures are displayed in the graphical window (Fig. 3), and can be compared to the Structure Panel Pressures (Fig. 4) shown in the Pressures and Motions result object in the upstream Hydrodynamic Diffraction system. The pressure Component Selection in the Pressures and Motions object may need to be matched to those included in the Hydrodynamic Pressure object for such a comparison to be made.

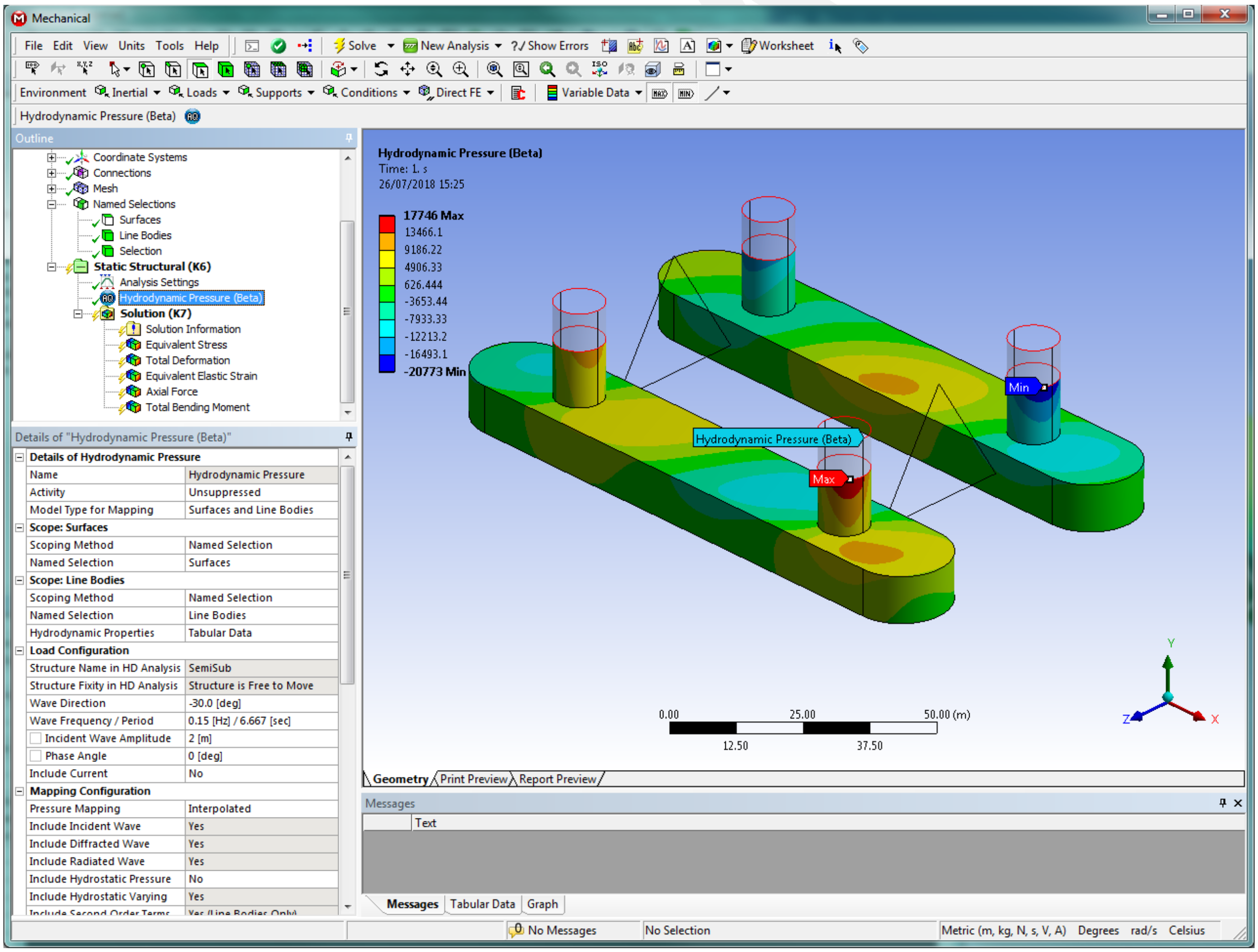

Figure 3: Hydrodynamic Pressures in the Static Structural system

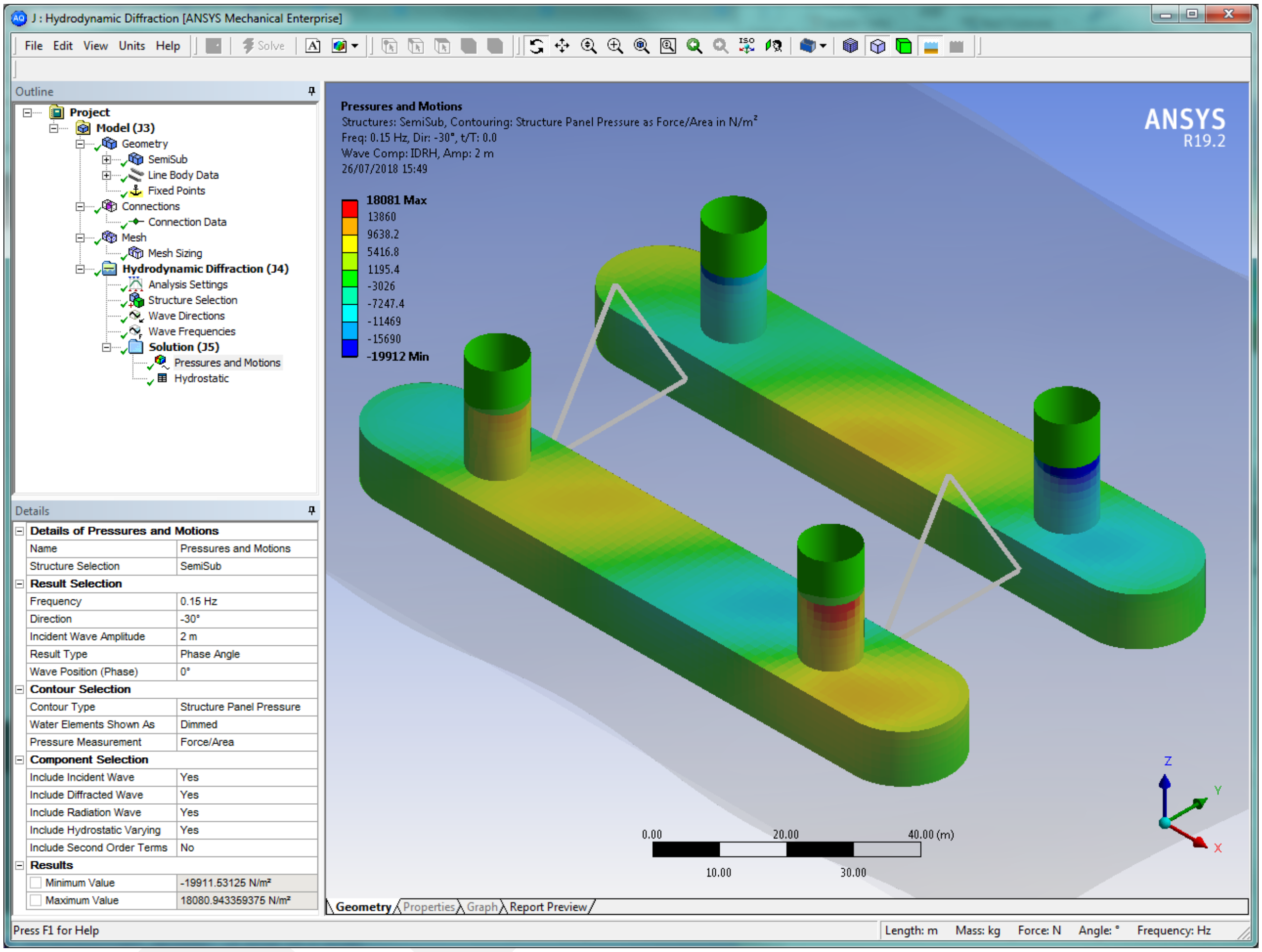

Figure 4: Hydrodynamic Pressures in the Hydrodynamic Diffraction system

Theory

This section discusses the theory applicable to the Hydrodynamic Pressure Mapping ACT Extension.

Hydrodynamic and Hydrostatic Pressures on Surfaces

The real and imaginary parts of the incident, diffracted, radiated and hydrostatic-varying pressure components calculated in the Hydrodynamic Diffraction analysis are used to evaluate the total hydrodynamic pressure at each structure node for the specified incident wave amplitude and phase angle:

$$P_\theta = a(P_r \cos\theta + P_i \sin\theta) \quad (1)$$

Where P_θ is the total hydrodynamic pressure at the required phase angle θ, with $\theta = 0°$ when the incident wave crest passes the center of gravity of the structure; a is the specified incident wave amplitude; and P_r and P_i are the real and imaginary parts, respectively, of the summed hydrodynamic (incident, diffracted, radiated and hydrostatic-varying) pressure components.

Where hydrostatic pressure is also included, the total pressure P is the summation of the hydrodynamic and hydrostatic parts:

$$P = P_\theta + P_s \quad (2)$$

Where the hydrostatic pressure P_s is calculated as:

$$P_s = \rho g z \quad (3)$$

In which ρ and g are the water density and acceleration due to gravity, respectively, as specified in the Hydrodynamic Diffraction analysis; and z is the depth of the structure node below the waterline in the Static Structural axis system (after any Axis Transformation has been applied).

Inertia and Viscous Drag Forces on Line Bodies

Distributed loads on submerged Line Bodies are calculated using the Morison equation:

$$F = \frac{1}{2}\rho D C_d (u_f - u_s)|u_f - u_s| + \rho A C_m \dot{u}_f - \rho A C_a \dot{u}_s \quad (4)$$

Where F is the total Morison force per unit length; D is the Line Body diameter (which should take into account any marine growth); C_d is the viscous drag coefficient; u_f is the flow velocity normal to the Line Body (comprising the diffracted wave particle velocity and any defined current); u_s is the structure velocity normal to the Line Body; $\dot{u}_f$ and $\dot{u}_s$ are the flow and structure accelerations, respectively, normal to the Line Body; A is the cross-sectional area of the Line Body; and C_a is the added mass coefficient, from which:

$$C_m = 1 + C_a \quad (5)$$

Where C_m is the inertia coefficient. Setting $C_m = 0.0$ 'turns off' loading due to the diffracted wave and added mass effects (second and third terms of Eq. 4); otherwise the valid range of values are $C_m \geq 1.0$. Similarly, viscous drag (first term of Eq. 4) can be ignored by setting $C_d = 0.0$. For non-cylindrical cross sections the Morison equation is applied separately in each direction normal to the Line Body.

The Hydrodynamic Pressure Mapping ACT Extension determines whether the ends of each beam element are above or below the local water surface and loads the Line Body accordingly. For elements that cut the water surface the loading is applied over the wetted length only.

Limitations

There exist some limitations to the geometry and model that can be used with the current (Beta) release of the Hydrodynamic Pressure Mapping ACT Extension. These are described below.

- The Beta version of the Hydrodynamic Pressure Mapping ACT Extension **will not be compatible with the full release version**. It will not be possible to open projects in later releases of the ANSYS software that were created in ANSYS Release 19.2 using the Hydrodynamic Pressure Mapping ACT Extension.

- It is not possible to map Hydrodynamic Pressure loads from a Hydrodynamic Response system (time response analysis or frequency statistical analysis).

- It is not possible to map loads on to more than one structure in the Static Structural analysis (though the upstream Hydrodynamic Diffraction analysis may contain more than one structure).

- It is not possible to map loads from a Hydrodynamic Diffraction analysis that includes multiple structures and forward speed.

- It is not possible to map loads on to Line Bodies from a Hydrodynamic Diffraction analysis that includes multiple structures.

- When the imported Geometry includes Line Bodies, the only Cross Sections that can be included are: Circular; Circular Tube; Rectangular; Rectangular Tube with equal wall thicknesses.

- Line Bodies of Model Type: Link (element type LINK167) are not currently supported.

- Where the Pressure Mapping method is set to Interpolated, you are limited to less than one million mesh nodes.

- Mapping on to the surfaces of solid bodies is not currently supported.

- In the structure mass checks, the only included mass is that which comes from:
    - The structural material itself;
    - Point Masses (direct or remote attachment);
    - Distributed Masses;
    - Additional Thickness definitions.
    - Any other mass (for example, due to Layered Sections or Command objects that define additional mass elements directly in APDL) will not be included. This may mean that the center of gravity about which rotational accelerations are applied is incorrect.

- When an internal Line Body is used to model the viscous drag acting on part of a structure, and that part of the structure is also modelled by diffracting panels, the wave particle kinematics around the Line Body will include the diffracted wave effects due to that part of the structure.