Introduction to Hydrodynamic Analysis with Ansys Aqwa

# Workshop 05.1: Aqwa Basics – Hydrodynamic Response

Release 2021 R2

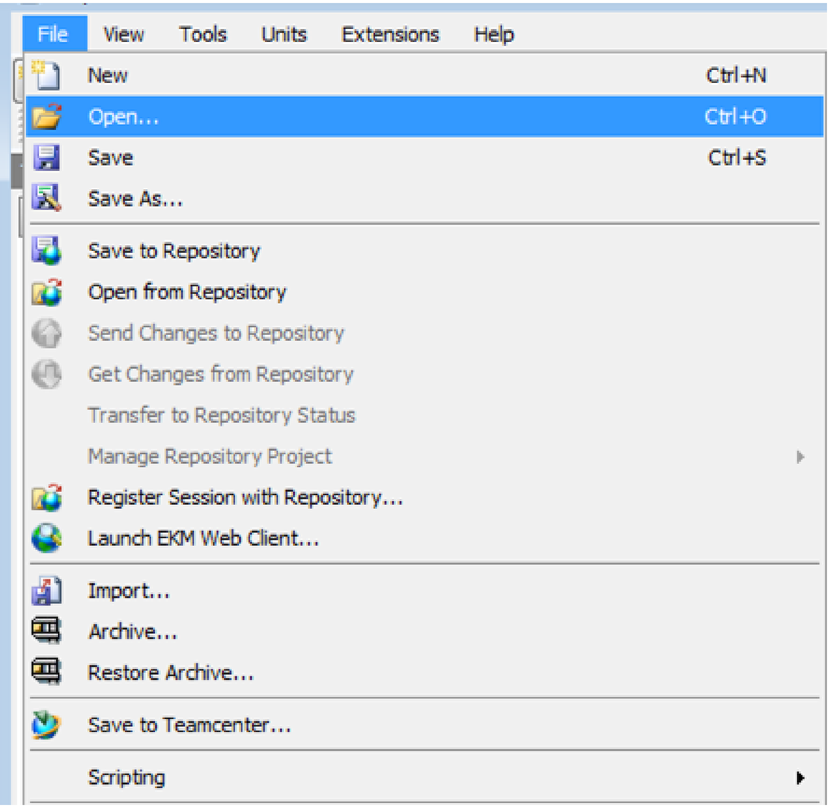
Ansys

# Hydrodynamic Response (HR) Simulation

- The goal of this workshop is to create a mooring system for a ship and run a series of static and dynamic response analyses

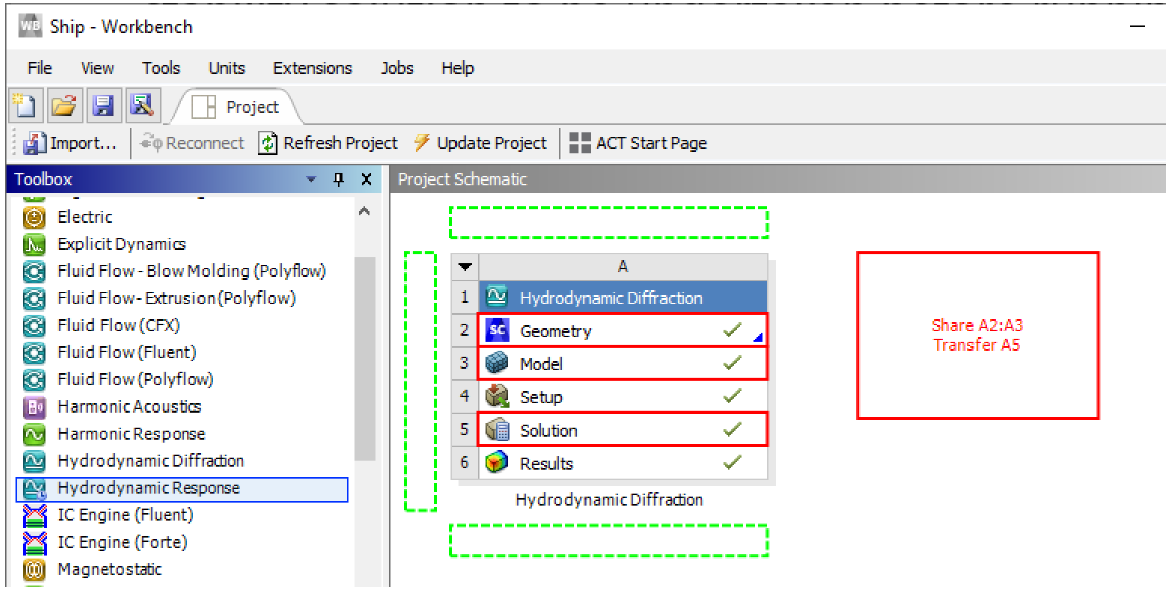
**Open the Ship project previously created in Workshop 04.1**

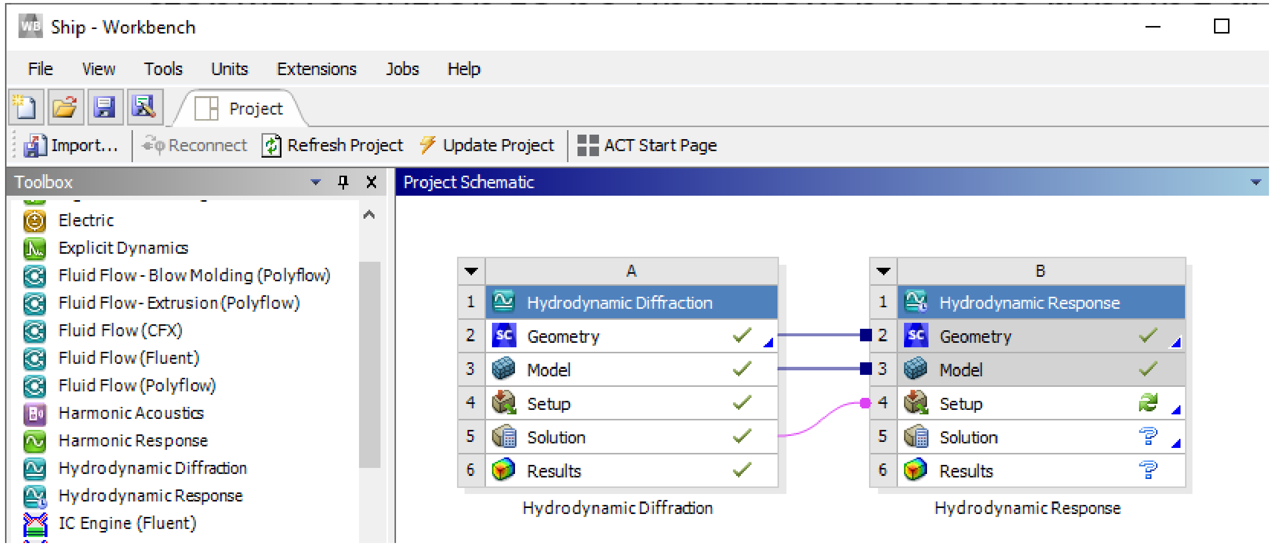
**Save As… to create a copy – you will need to re-use the completed project from Workshop 04.1 again in the next session**



# Adding the Static HR System

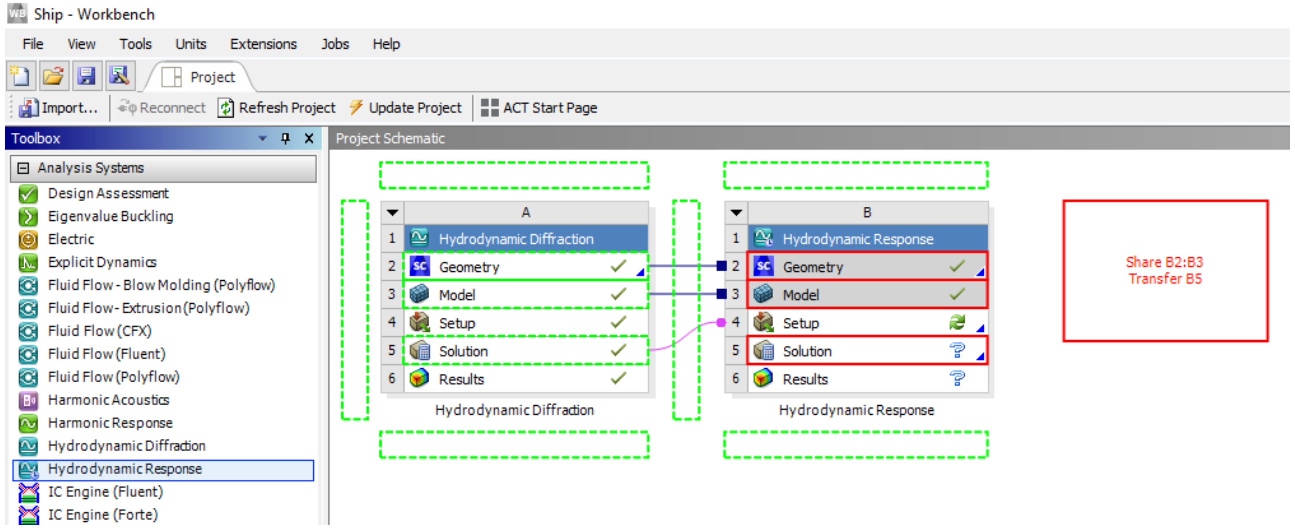
- Drag and drop the Hydrodynamic Response to the WB Project Schematic  while sharing the solution from Hydrodynamic Diffraction.  This will become the static stability solution to be undertaken before running dynamic simulations



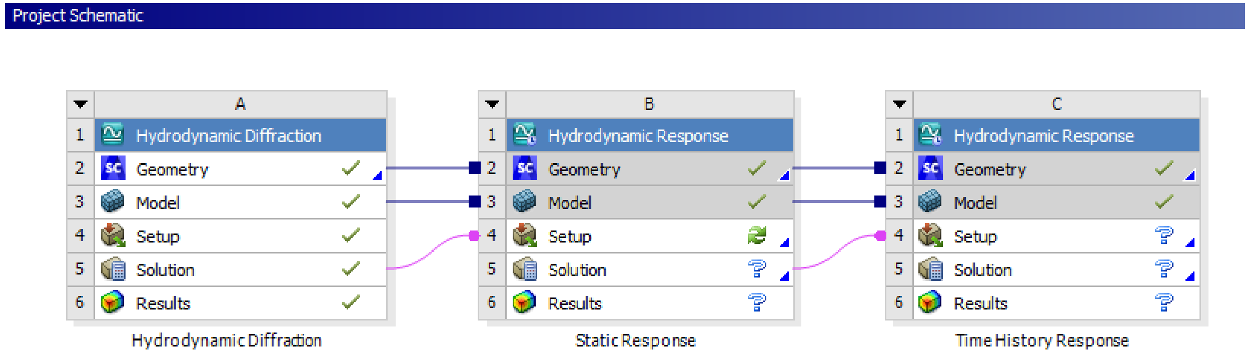



# Adding a Dynamic HR System

- A second HR system is added that will be for the time history dynamic analysis

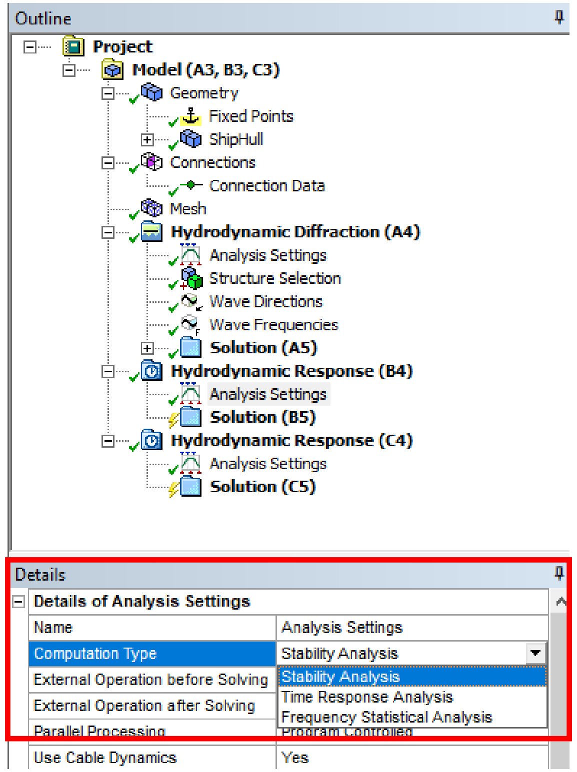


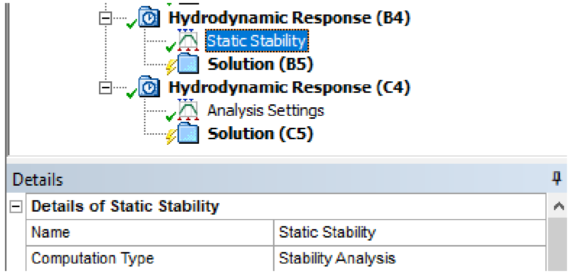

**Drag and drop the new HR system on the Solution cell of the existing HR system, and double click on the Setup cell. Note that we can re-name the systems by RMB clicking on the system header and selecting Rename**



# Set up Static Stability Response System

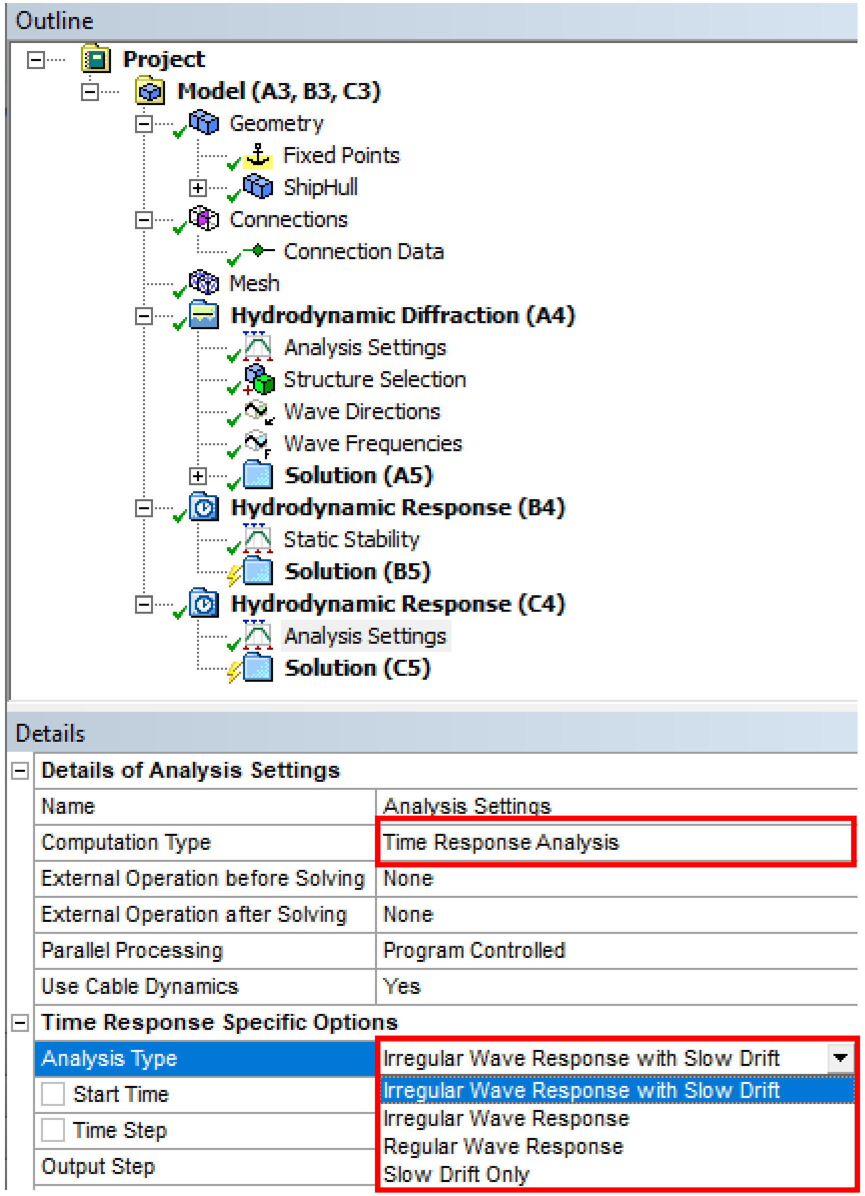
- In the project tree, under analysis setting of Hydrodynamic Response (B4) check that the analysis type is set as Stability Analysis (this is the default when the upstream system is a Hydrodynamic Diffraction analysis)
- This will allow us to determine the starting position of the system to be used for the Time Domain analysis.
- Rename this Analysis system to Static Stability by selecting the Hydrodynamic Response item in the tree and changing the Name in the Details panel.



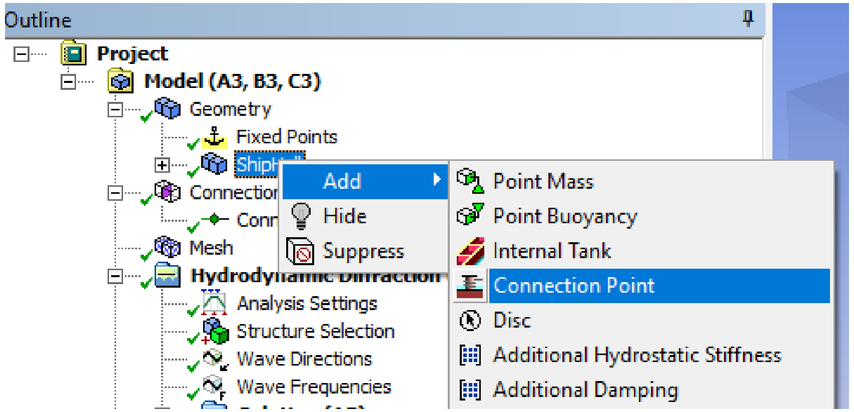



# Set up Time Response System

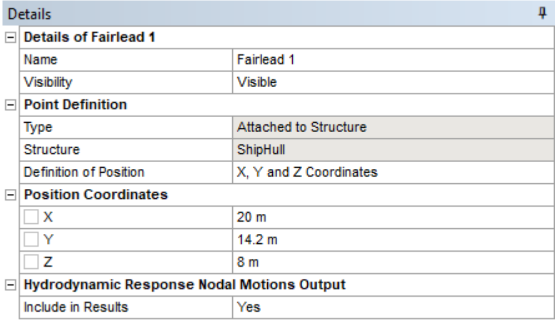
- In the project tree, under analysis setting of Hydrodynamic response (C4), rename this Analysis system to Time Domain Response.
- Note that the Computation Type of Time Response Analysis is the default when the upstream system is a  Static Stability Analysis.
- We will set the Analysis Type at a later stage



# Add Fairlead Connection Points



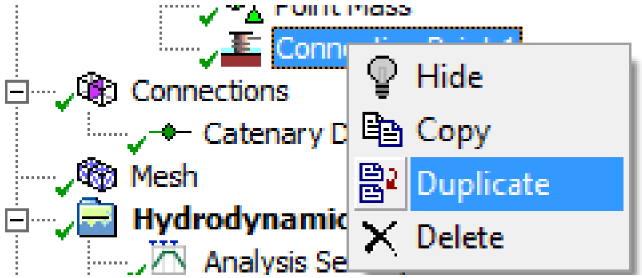

- Add four fairlead locations for the mooring system
- Select Geometry > ShipHull > Add > Connection Point
- Enter X,Y,Z coordinates in details panel as below (in global system!)



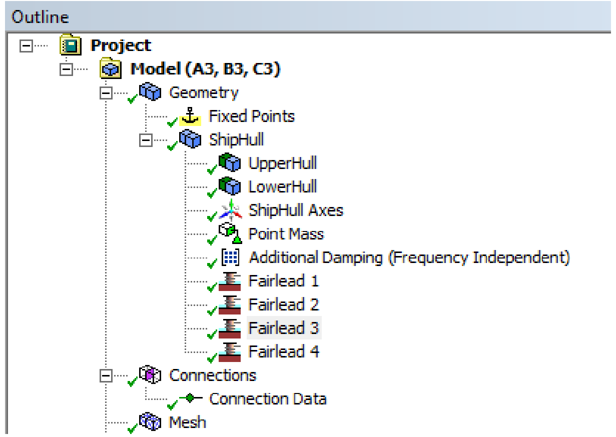
Details

| Details of Fairlead 1 | |
|---|---|
| Name | Fairlead 1 |
| Visibility | Visible |
| **Point Definition** | |
| Type | Attached to Structure |
| Structure | ShipHull |
| Definition of Position | X, Y and Z Coordinates |
| **Position Coordinates** | |
| X | 20 m |
| Y | 14.2 m |
| Z | 8 m |
| **Hydrodynamic Response Nodal Motions Output** | |
| Include in Results | Yes |

- Add 3 other connection points (can use Duplicate as an alternative)
  - Point 2 20.0, -14.2, 8.0
  - Point 3 180.0, 16.0, 8.0
  - Point 4 180.0, -16.0, 8.0

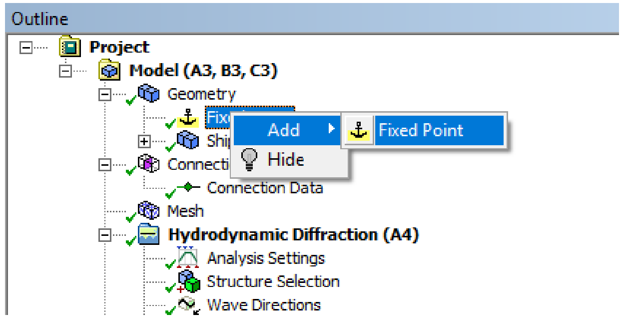


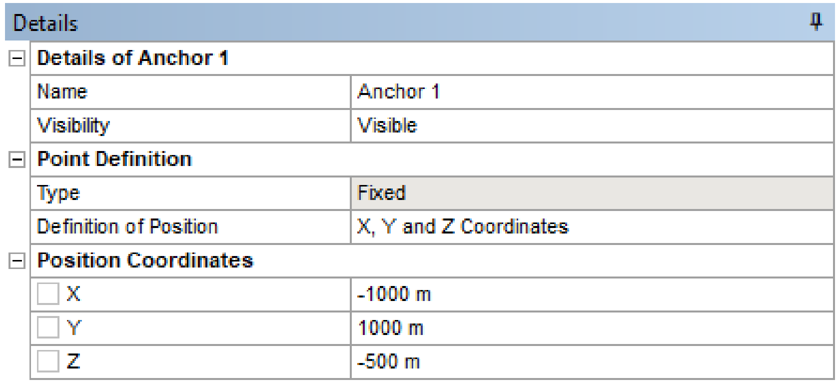

- Rename Connection Points as Fairlead 1 through 4
- Note the Nodal Motions Output is set to Yes by default. This allows relevant results to be reported at these points

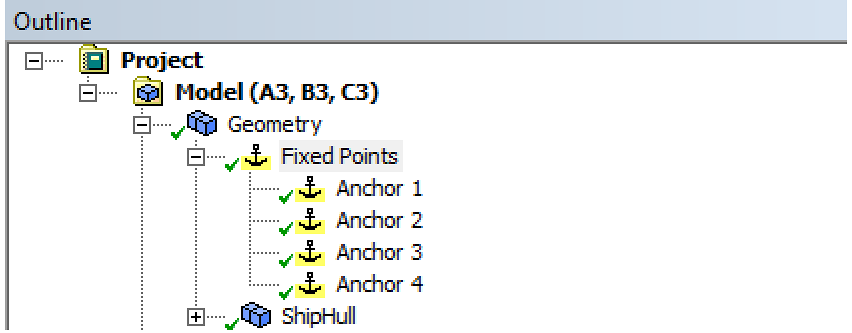


# Add Anchor Fixed Points

- Add four anchor locations for the mooring system
- Select Model > Fixed Point > Add > Fixed Point
- Enter X,Y,Z coordinates in details panel as below
- Repeat for other 3 fixed points
    - Point 2 -1000,-1000,-500
    - Point 3 1000,1000,-500
    - Point 4 1000,-1000,-500
- Rename Fixed Points as Anchor 1 through 4

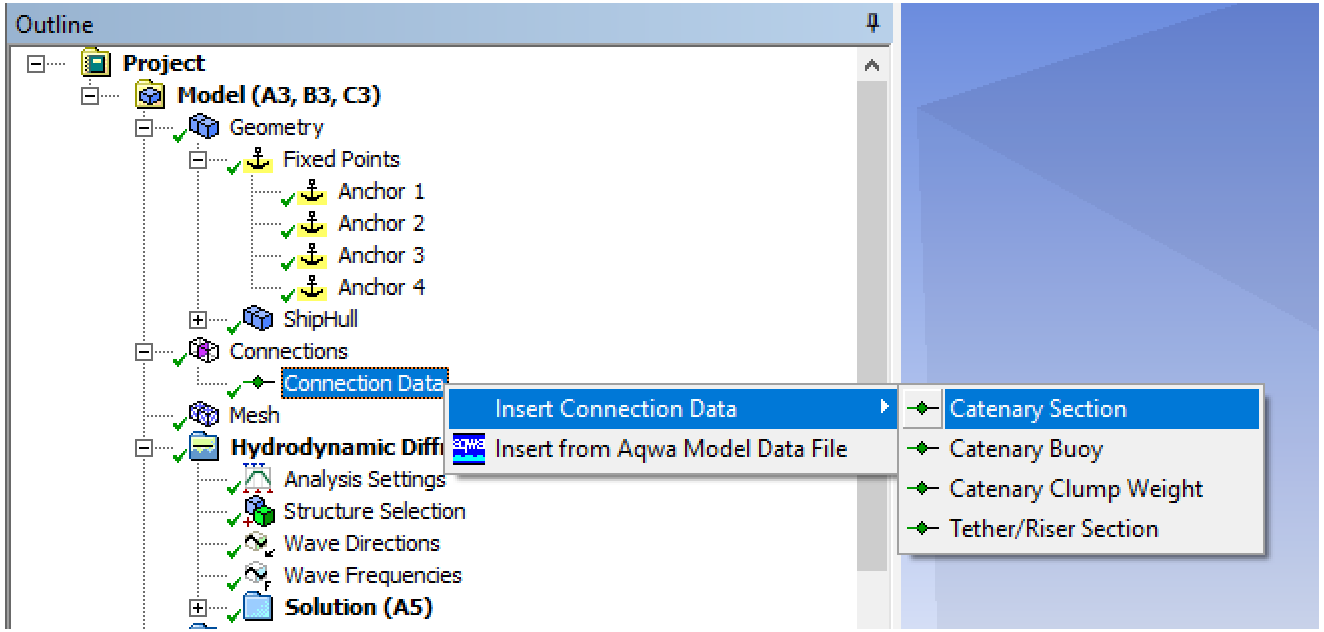


# Set Up Mooring Line Properties

- We are going to use composite catenary lines for the mooring system. These are lines made up of one or more segments (or sections) with varying properties e.g. chain/wire/chain
- Select Connections > Connection Data > Insert Connection Data > Catenary Section. Note we can also define intermediate buoys and/or clump weights for catenary lines.



# Mooring Line Properties

- Provide data for Mass/Unit Length, Equivalent Cross Sectional Area, Stiffness, Maximum Tension and Equivalent Diameter as below.
  - Note we can also define bending stiffness, non-linear axial properties
- Repeat for two additional sections (use the Duplicate function to save time)

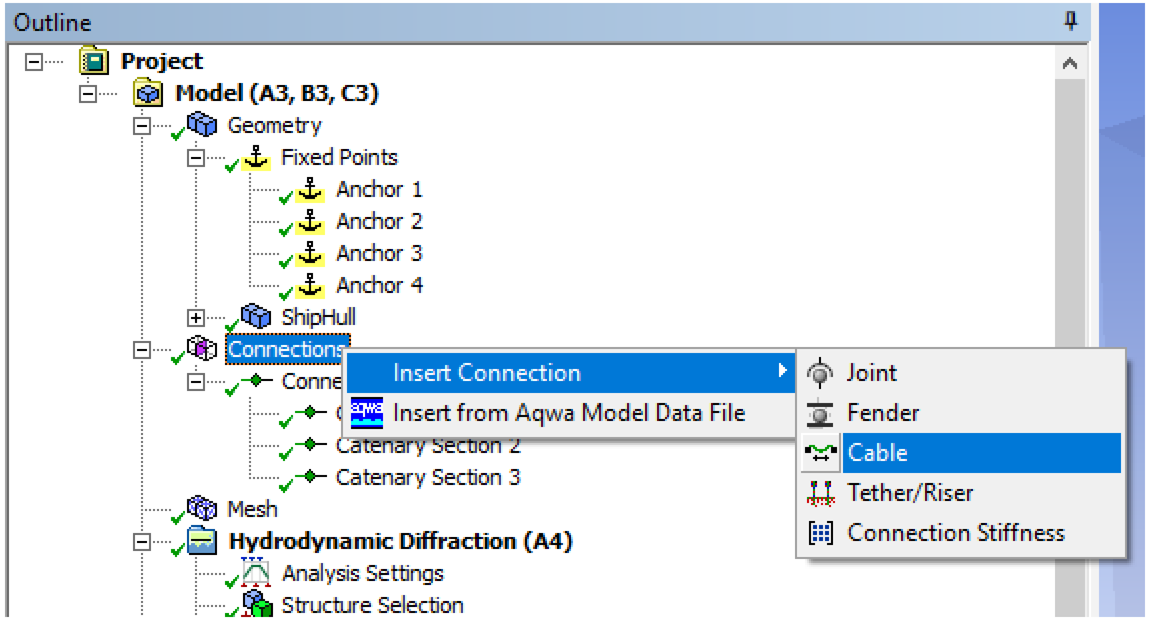
| Details | |
|---|---|
| **Details of Catenary Section 1** | |
| Name | Catenary Section 1 |
| **Section Properties** | |
| Mass/Unit Length | 150 kg/m |
| Equivalent Cross-Sectional A... | 0.01 m² |
| Stiffness, EA | 600000000 N |
| Maximum Tension | 7500000 N |
| Bending Stiffness, EI | 0.0 N.m² |
| Axial Stiffness Coefficient k1 | 0.0 N |
| Axial Stiffness Coefficient k2 | 0.0 N |
| Axial Stiffness Coefficient k3 | 0.0 N |
| **Section Hydrodynamic Properties** | |
| Added Mass Coefficient | 1 |
| Transverse Drag Coefficient | 1 |
| Equivalent Diameter | 0.1 m |
| Longitudinal Drag Coefficient | 0.025 |

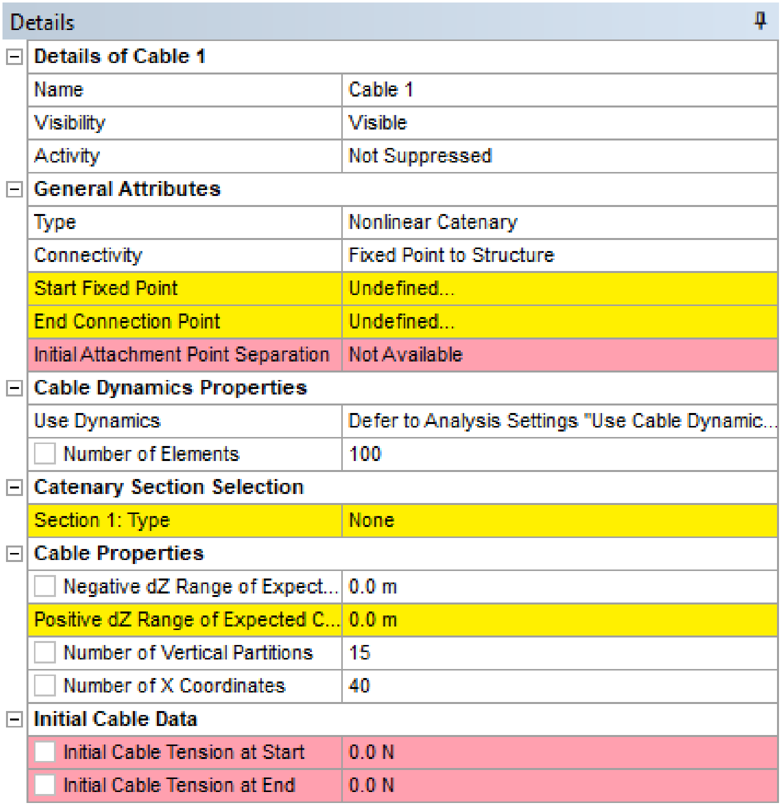
| Details | |
|---|---|
| **Details of Catenary Section 2** | |
| Name | Catenary Section 2 |
| **Section Properties** | |
| Mass/Unit Length | 120 kg/m |
| Equivalent Cross-Sectional A... | 0.01 m² |
| Stiffness, EA | 900000000 N |
| Maximum Tension | 7500000 N |
| Bending Stiffness, EI | 0.0 N.m² |
| Axial Stiffness Coefficient k1 | 0.0 N |
| Axial Stiffness Coefficient k2 | 0.0 N |
| Axial Stiffness Coefficient k3 | 0.0 N |
| **Section Hydrodynamic Properties** | |
| Added Mass Coefficient | 1 |
| Transverse Drag Coefficient | 1 |
| Equivalent Diameter | 0.1 m |
| Longitudinal Drag Coefficient | 0.025 |

| Details | |
|---|---|
| **Details of Catenary Section 3** | |
| Name | Catenary Section 3 |
| **Section Properties** | |
| Mass/Unit Length | 170 kg/m |
| Equivalent Cross-Sectional A... | 0.01 m² |
| Stiffness, EA | 600000000 N |
| Maximum Tension | 7500000 N |
| Bending Stiffness, EI | 0.0 N.m² |
| Axial Stiffness Coefficient k1 | 0.0 N |
| Axial Stiffness Coefficient k2 | 0.0 N |
| Axial Stiffness Coefficient k3 | 0.0 N |
| **Section Hydrodynamic Properties** | |
| Added Mass Coefficient | 1 |
| Transverse Drag Coefficient | 1 |
| Equivalent Diameter | 0.1 m |
| Longitudinal Drag Coefficient | 0.025 |

# Mooring Line Connections

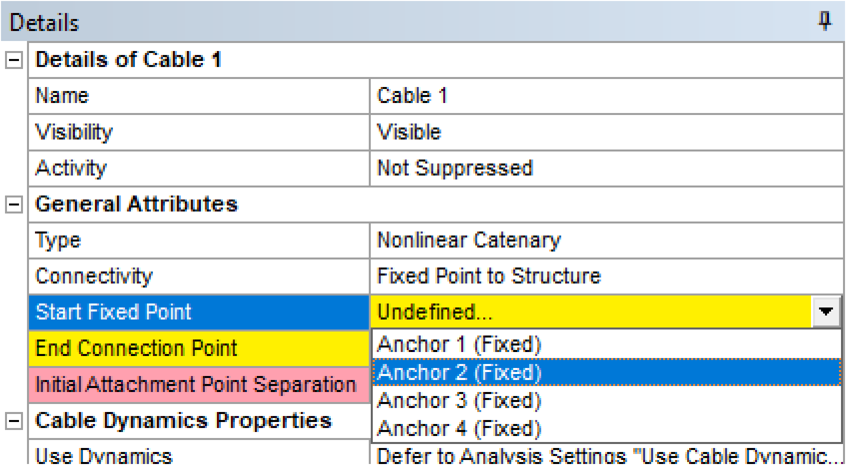
- To add a line select Connections > Insert Connection > Cable
- Ensure mooring line is named Cable 1
- Choose Non-Linear Catenary for Type and Fixed Point & Structure for Connectivity

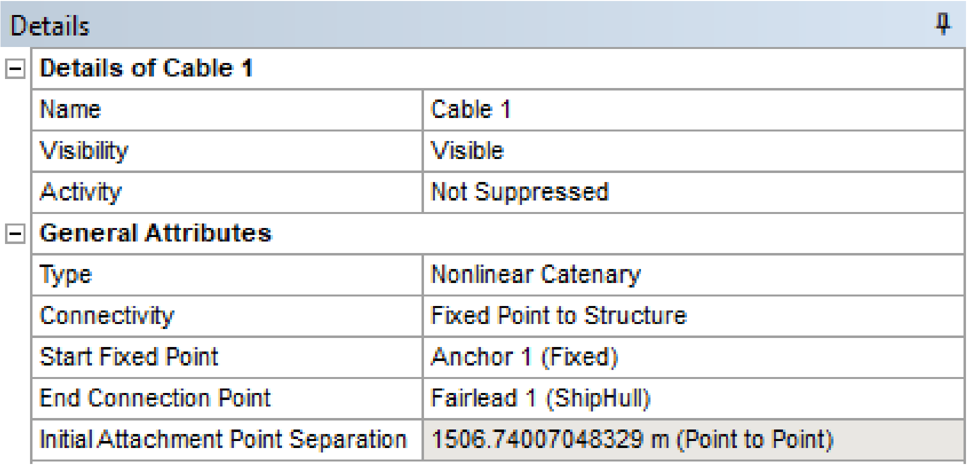


# Mooring Line Connections

- We need to define end connection points and the segments along the line, plus some data defining the possible dZ range of the fairlead (will be explained in a later lecture).
- Click on cell adjacent to Fixed Point and select Anchor 1 (Fixed) from drop down menu.
- Click on cell adjacent to End Connection Point and select Fairlead 1 (ShipHull) from drop down menu.



# Mooring Line Configuration

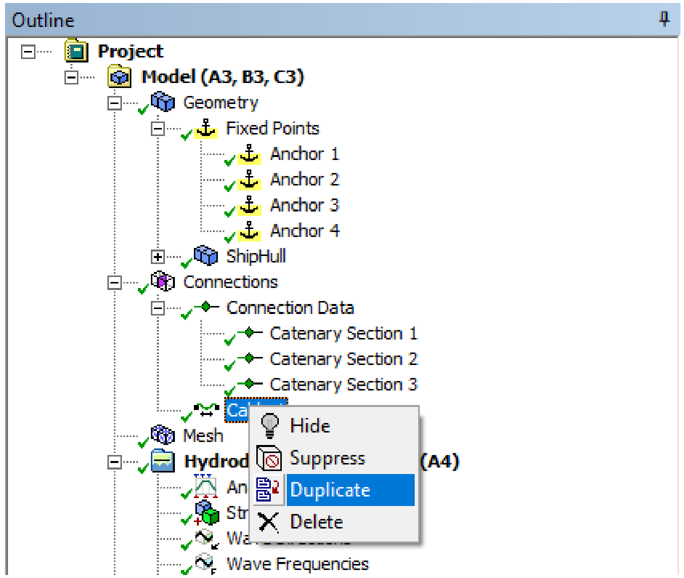
- The composition of the line is now defined
  - Section allocation
  - Line length
- Sections are defined from the *anchor* location up to the *fairlead*
- Section 1 type should be set to Catenary Section 1, length 400 m
- Repeat for sections 2 and 3 as shown
- Finally set the dZ Ranges under Cable Properties to 10m

Details

| | |
|---|---|
| **Details of Cable 1** | |
| Name | Cable 1 |
| Visibility | Visible |
| Activity | Not Suppressed |
| **General Attributes** | |
| Type | Nonlinear Catenary |
| Connectivity | Fixed Point to Structure |
| Start Fixed Point | Anchor 1 (Fixed) |
| End Connection Point | Fairlead 1 (ShipHull) |
| Initial Attachment Point Separation | 1506.74007048329 m (Point to Point) |
| **Cable Dynamics Properties** | |
| Use Dynamics | Defer to Analysis Settings "Use Cable Dynamic... |
| Number of Elements | 100 |
| **Catenary Section Selection** | |
| Section 1: Type | Catenary Section 1 |
| Section 1: Length | 400 m |
| Joint 1/2: Mass/Buoyancy | None |
| Section 2: Type | Catenary Section 2 |
| Section 2: Length | 500 m |
| Joint 2/3: Mass/Buoyancy | None |
| Section 3: Type | Catenary Section 3 |
| Section 3: Length | 700 m |
| Section 4: Type | None |
| **Cable Properties** | |
| Negative dZ Range of Expect... | 10 m |
| Positive dZ Range of Expecte... | 10 m |
| Number of Vertical Partitions | 15 |
| Number of X Coordinates | 40 |
| **Initial Cable Data** | |
| Initial Cable Tension at Start | 1005838.438 N |
| Initial Cable Tension at End | 1771235.25 N |

# Mooring Line Configuration

- Set up the remaining 3 mooring lines with identical properties but different connection points
- Use the Duplicate option to reduce data repetition




Details

| Details of Cable 2 | |
|---|---|
| Name | Cable 2 |
| Visibility | Visible |
| Activity | Not Suppressed |
| **General Attributes** | |
| Type | Nonlinear Catenary |
| Connectivity | Fixed Point to Structure |
| Start Fixed Point | Anchor 2 (Fixed) |
| End Connection Point | Fairlead 2 (ShipHull) |
| Initial Attachment Point Separation | 1506.74007048329 m (Point t |

Details

| Details of Cable 3 | |
|---|---|
| Name | Cable 3 |
| Visibility | Visible |
| Activity | Not Suppressed |
| **General Attributes** | |
| Type | Nonlinear Catenary |
| Connectivity | Fixed Point to Structure |
| Start Fixed Point | Anchor 3 (Fixed) |
| End Connection Point | Fairlead 3 (ShipHull) |
| Initial Attachment Point Separation | 1377.9404921839 m (Point to Poir |

Details

| Details of Cable 4 | |
|---|---|
| Name | Cable 4 |
| Visibility | Visible |
| Activity | Not Suppressed |
| **General Attributes** | |
| Type | Nonlinear Catenary |
| Connectivity | Fixed Point to Structure |
| Start Fixed Point | Anchor 4 (Fixed) |
| End Connection Point | Fairlead 4 (ShipHull) |
| Initial Attachment Point Separation | 1377.9404921839 m (Point to Point) |

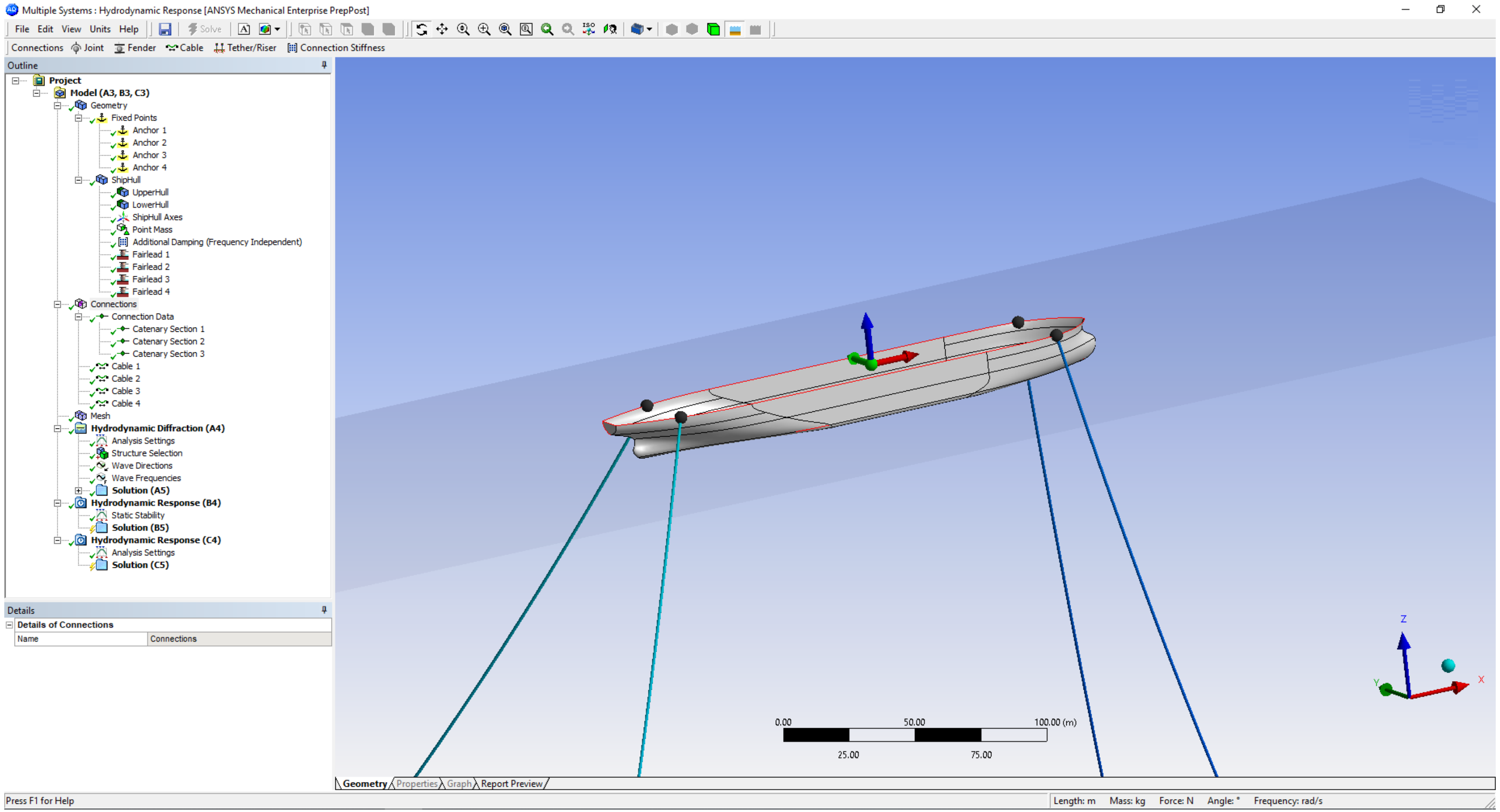
# Mooring Line Configuration

- Note that as you construct the line configuration the line properties are reported in a window called Catenary Cable Definition Data
- The Joint 1-2 etc refers to connection data such as clump weights or buoys

| Catenary Cable Definition Data | | | | | |
|---|---|---|---|---|---|
| | Section 1 | Joint 1-2 | Section 2 | Joint 2-3 | Section 3 |
| Type | Catenary Section 1 | None | Catenary Section 2 | None | Catenary Section 3 |
| Section Length (m) | 400 | - | 500 | - | 700 |
| Mass / Unit Length (kg/m) | 150 | - | 120 | - | 170 |
| Equivalent CSA ($m^2$) | 0.01 | - | 0.01 | - | 0.01 |
| Stiffness, EA (N) | 600000000 | - | 900000000 | - | 600000000 |
| Maximum Tension (N) | 7500000 | - | 7500000 | - | 7500000 |
| Bending Stiffness, EI ($N.m^2$) | 0.0 | - | 0.0 | - | 0.0 |
| Axial Stiffness Coefficient k1 (N) | 0.0 | - | 0.0 | - | 0.0 |
| Axial Stiffness Coefficient k2 (N) | 0.0 | - | 0.0 | - | 0.0 |
| Axial Stiffness Coefficient k3 (N) | 0.0 | - | 0.0 | - | 0.0 |
| Added Mass Coefficient | 1 | - | 1 | - | 1 |
| Transverse Drag Coefficient | 1 | - | 1 | - | 1 |
| Equivalent Diameter (m) | 0.1 | - | 0.1 | - | 0.1 |
| Longitudinal Drag Coefficient | 0.025 | - | 0.025 | - | 0.025 |
| Structural Mass (kg) | - | - | - | - | - |
| Displaced Mass of Water (kg) | - | - | - | - | - |
| Added Mass (kg) | - | - | - | - | - |
| Coefficient of Drag * Area ($m^2$) | - | - | - | - | - |

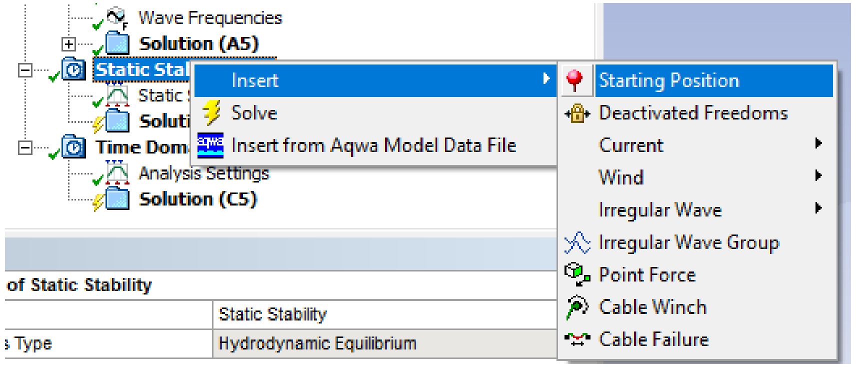
# Mooring Line Configuration

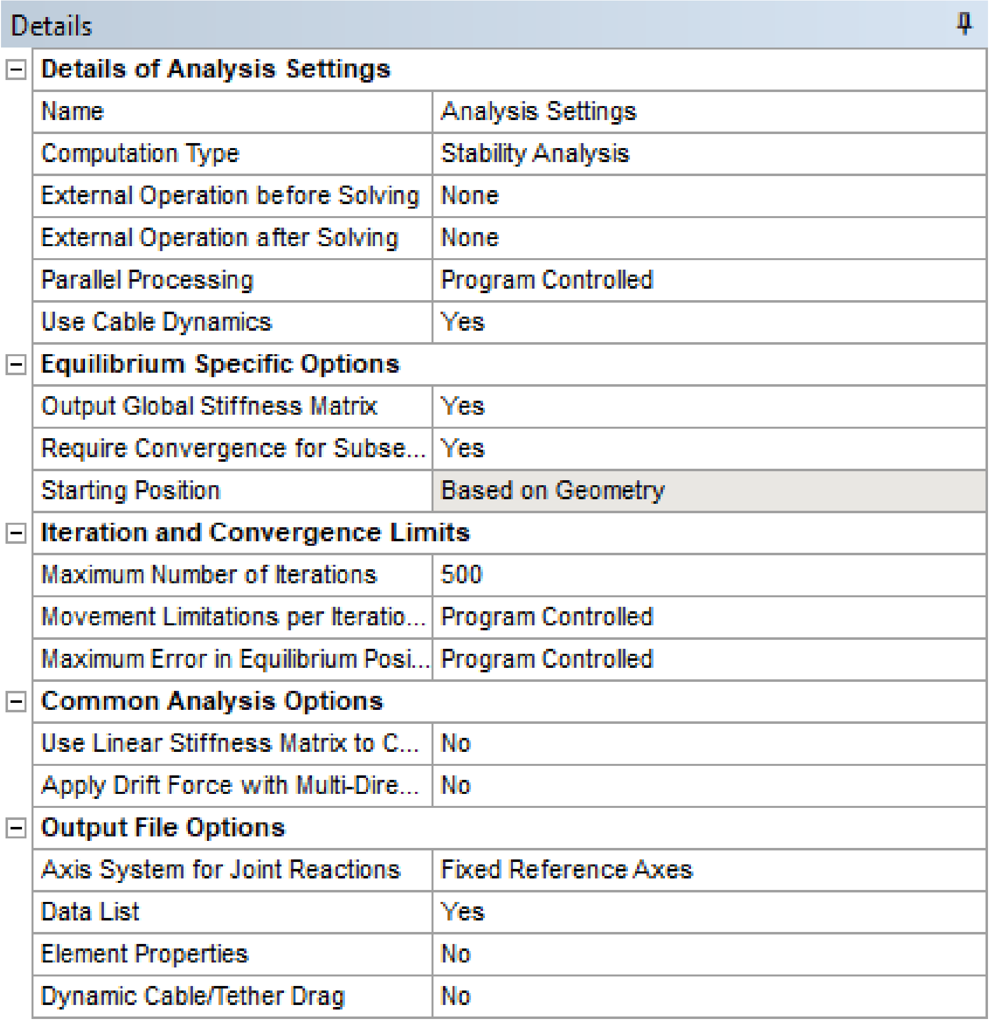
- Select Connections in the Outline to see the final mooring configuration



# Stability Analysis Options

- Utilize all default settings.
- Note that the initial position utilizes the definition based upon the defined geometry. If required this can be overridden by inserting a Starting Position under the Static Stability entry in the Outline. For this exercise leave the Starting Position as Based on Geometry.



Details

| | |
|---|---|
| **Details of Analysis Settings** | |
| Name | Analysis Settings |
| Computation Type | Stability Analysis |
| External Operation before Solving | None |
| External Operation after Solving | None |
| Parallel Processing | Program Controlled |
| Use Cable Dynamics | Yes |
| **Equilibrium Specific Options** | |
| Output Global Stiffness Matrix | Yes |
| Require Convergence for Subse... | Yes |
| Starting Position | Based on Geometry |
| **Iteration and Convergence Limits** | |
| Maximum Number of Iterations | 500 |
| Movement Limitations per Iteratio... | Program Controlled |
| Maximum Error in Equilibrium Posi... | Program Controlled |
| **Common Analysis Options** | |
| Use Linear Stiffness Matrix to C... | No |
| Apply Drift Force with Multi-Dire... | No |
| **Output File Options** | |
| Axis System for Joint Reactions | Fixed Reference Axes |
| Data List | Yes |
| Element Properties | No |
| Dynamic Cable/Tether Drag | No |

# Time Response Simulation Options

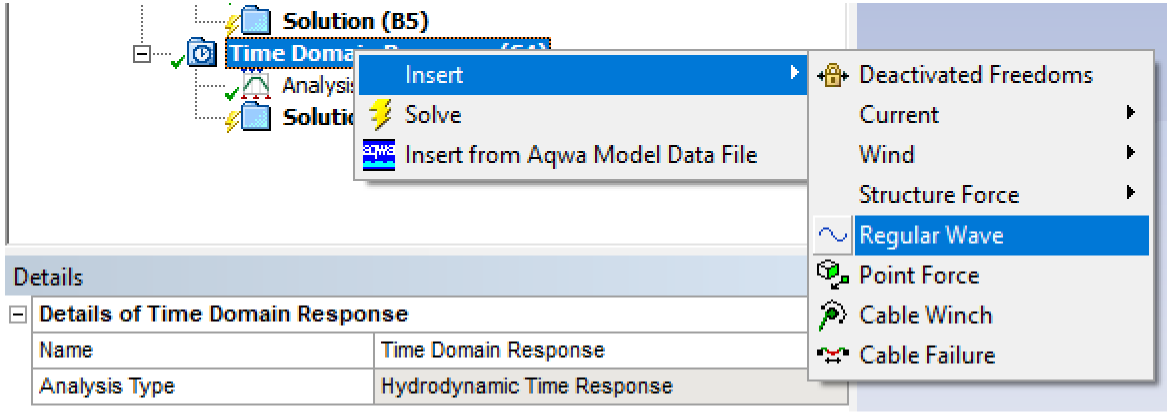
- Go to Time Response > Analysis Settings
- Set Analysis Type to Regular Wave Response
- Set Use Cable Dynamics to Yes
- Set Convolution to Yes
- Set Duration to 180 seconds

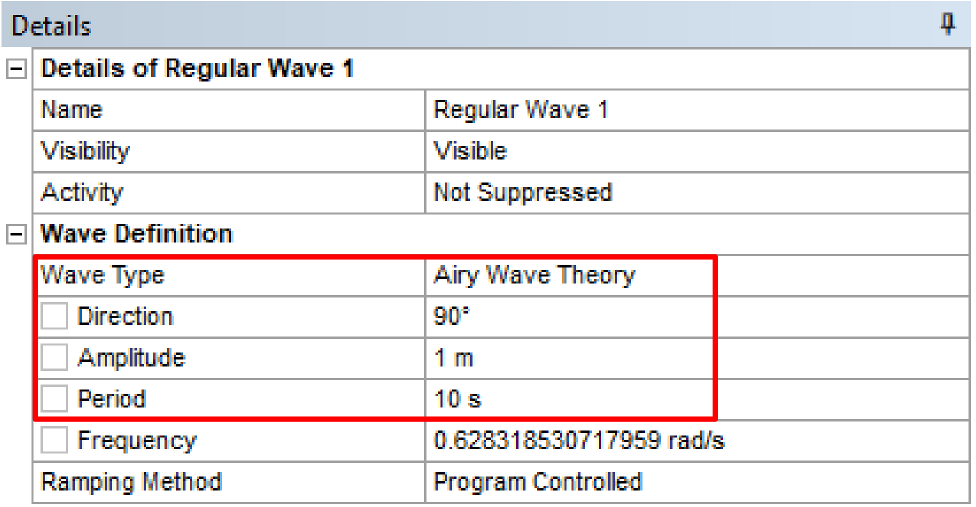
| Details | |
|---|---|
| **Details of Analysis Settings** | |
| Name | Analysis Settings |
| Computation Type | Time Response Analysis |
| External Operation before Solving | None |
| External Operation after Solving | None |
| Parallel Processing | Program Controlled |
| Use Cable Dynamics | Yes |
| **Time Response Specific Options** | |
| Analysis Type | Regular Wave Response |
| Start Time | 0.0 s |
| Time Step | 0.1 s |
| Output Step | 0.1 s |
| Duration | 180 s |
| Number of Steps | 1801 |
| Finish Time | 180 s |
| Starting Position | Determined by Upstream System |
| X-Position for Wave Surface Ele... | 0.0 m |
| Y-Position for Wave Surface Ele... | 0.0 m |
| **Common Analysis Options** | |
| Convolution | Yes |
| Call Routine "user_force" | No |
| Connect to Server for External "... | No |
| Use Linear Starting Conditions | No |
| Use Linear Stiffness Matrix to C... | No |
| Account for Current Phase Shift | Yes |
| Use Wheeler Stretching | Yes |
| **Output File Options** | |
| Axis System for Joint Reactions | Fixed Reference Axes |
| Data List | Yes |
| Element Properties | No |
| Dynamic Cable/Tether Drag | No |

# Define the Environment

- RMB on Time Domain Response > Insert > Regular Wave
- Set Wave Type to Airy Wave Theory
- Set Direction to 90 degrees
- Set Amplitude to 1 m
- Set Period to 10 s
- Run the analysis by RMB clicking on Solution and


**NOTE: Solving the Time Response analysis will automatically solve the upstream stability analysis to compute the correct starting condition.**



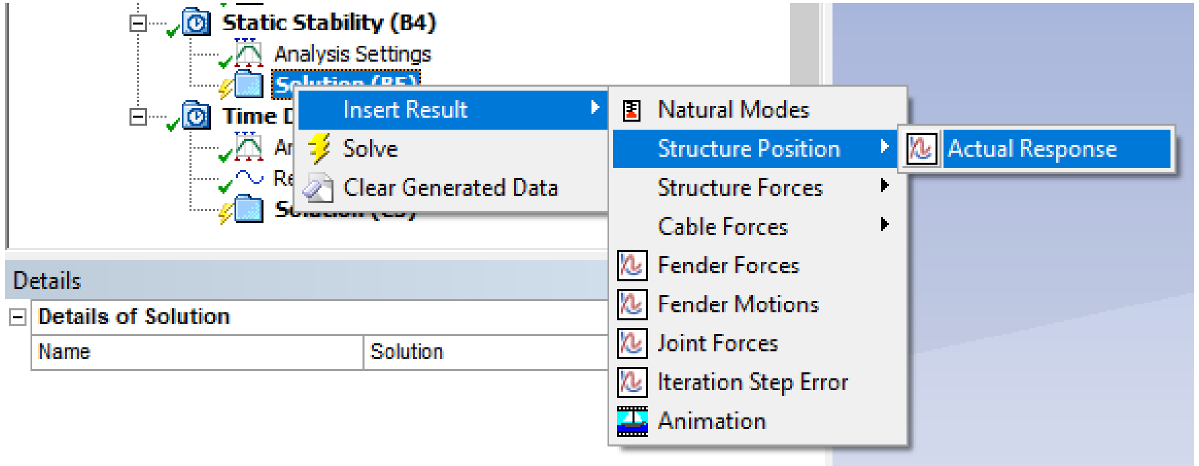



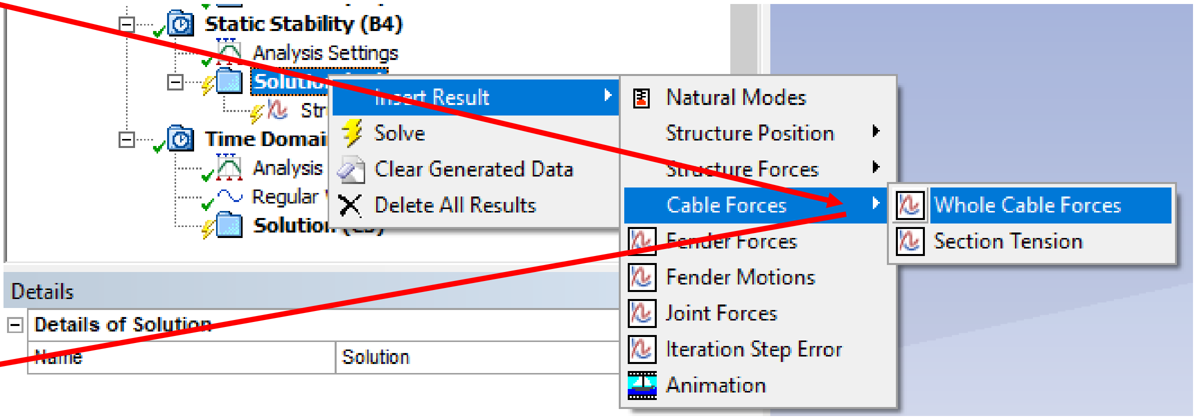



Ansys

# Check the Solution

- Go to Static Stability, RMB click on Solution then Insert Result >Structure Position >Actual Response.
- Scope the result to Global X, Y, Z motions.
- Following the same process insert  Whole Cable Forces, Tension.



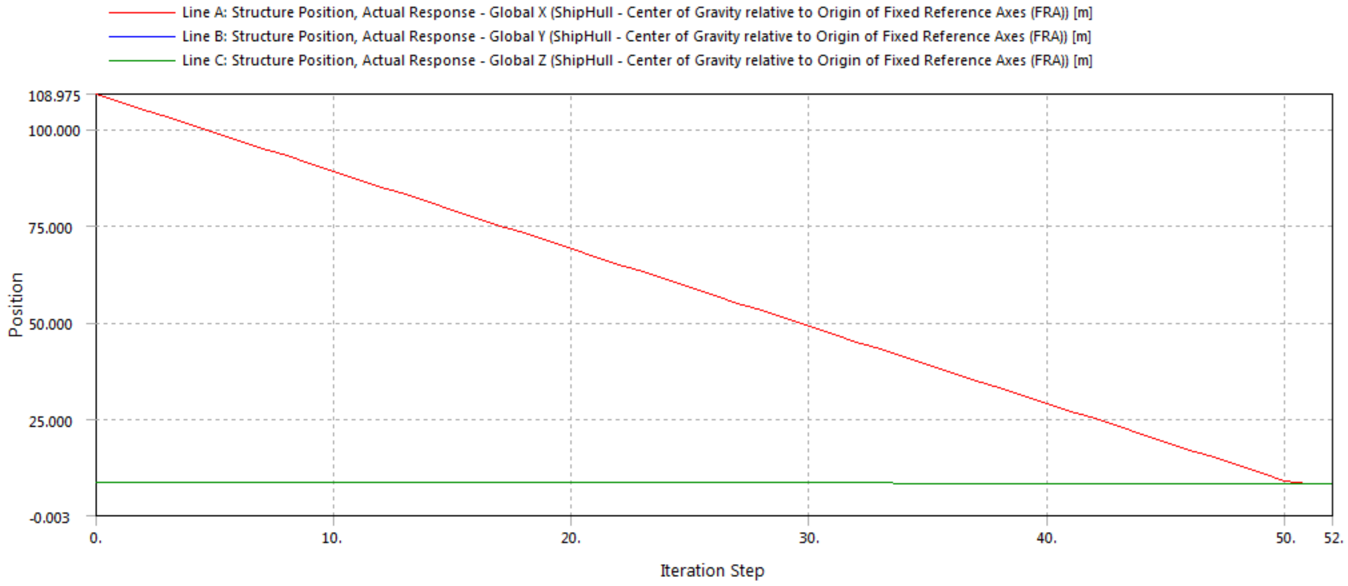




| Details | |
|---|---|
| **Details of Cable Forces, Whole Cable Forces** | |
| Name | Cable Forces, Whole Cable Forces |
| Presentation Method | Line |
| Axes Selection | Force/Moment vs Iteration Step |
| **Line A** | |
| Structure | ShipHull |
| Type | Cable Forces |
| SubType | Whole Cable Forces |
| Connection | Cable 1 |
| Component | Tension |

**Evaluate all results**

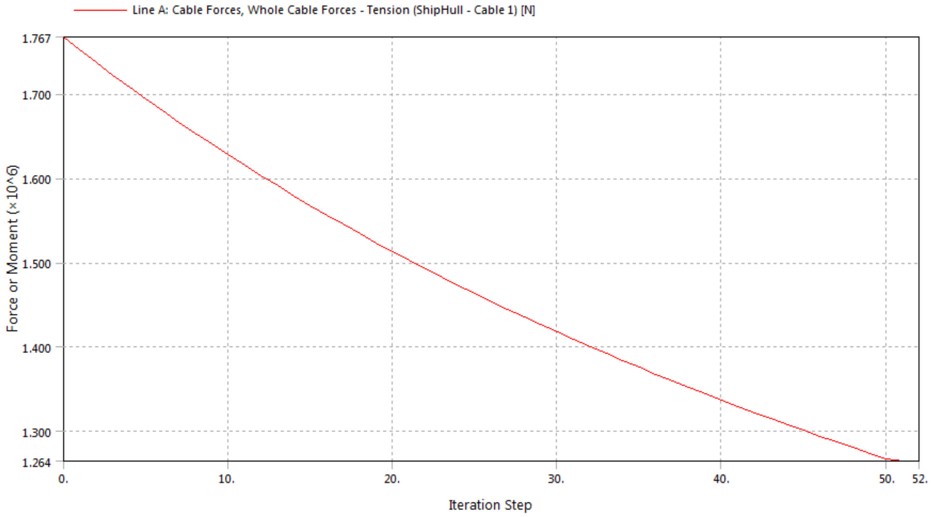
# Check the Stability Solution

- As you can see, the equilibrium position can be quite far away from the geometry position.



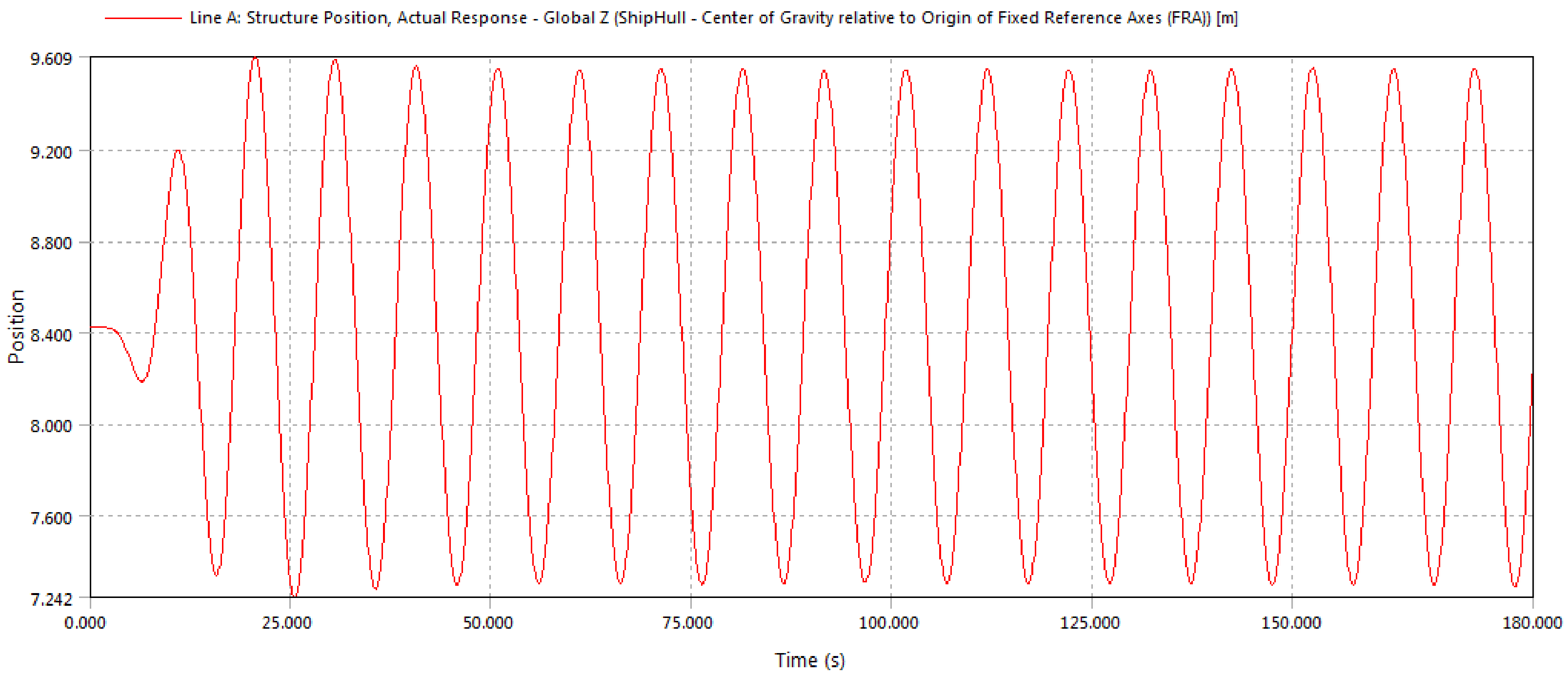

•IMPORTANT:

•The Stability Analysis in an important step of any dynamic analysis, and is <u>essential</u> when a Frequency Statistical Analysis is undertaken!



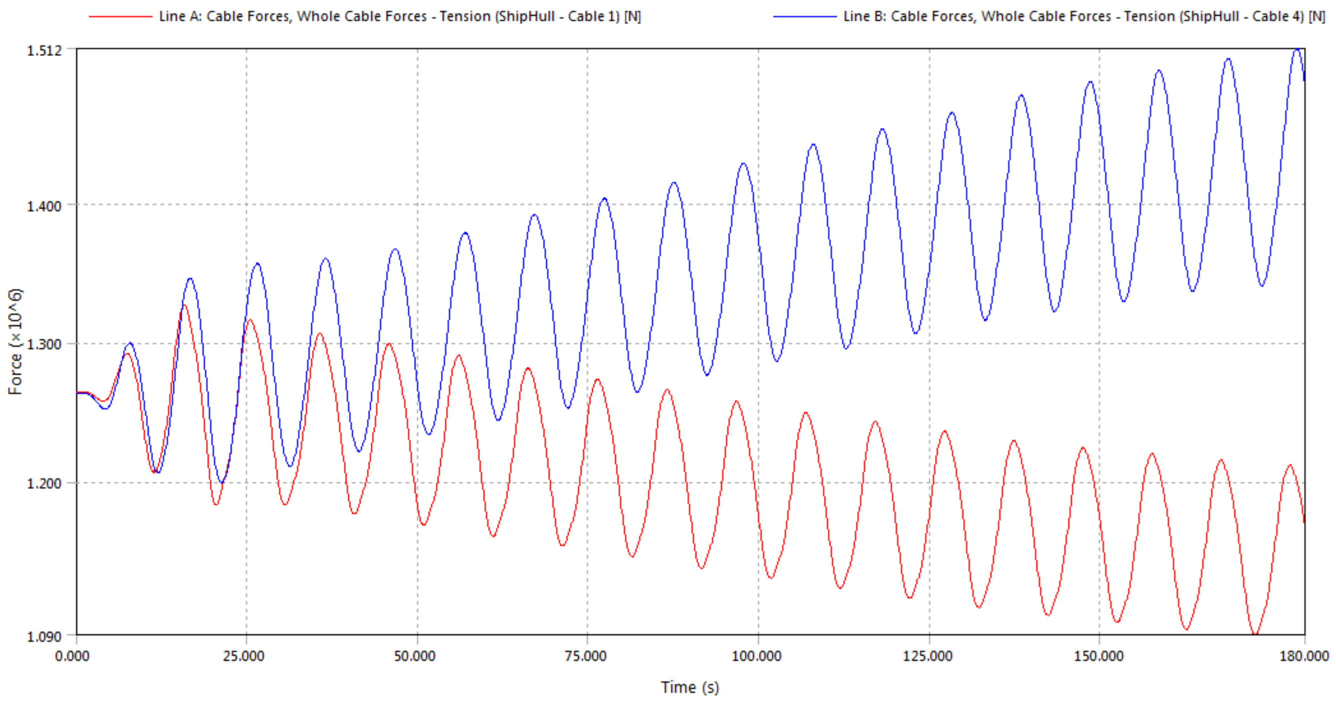
# Check the Time Response Solution

- Under the Time Response Analysis Solution add a graph item – Insert Result > Structure Position > Actual Response
- Select Global Z
- Evaluate All Results



# Check the Time Response Solution

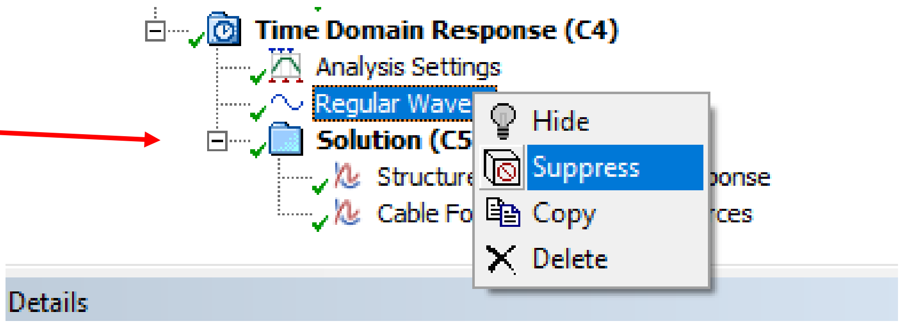
- Under the Time Response Analysis Solution add another graph item – Insert Result > Cable Forces > Whole Cable Forces
- Select Tension for Cables 1 and 4
- Evaluate All Results



# Irregular Wave Simulation

- RMB Regular Wave and Suppress

- Under Time Domain Response in the Analysis Settings panel change the analysis type to Irregular Wave Response



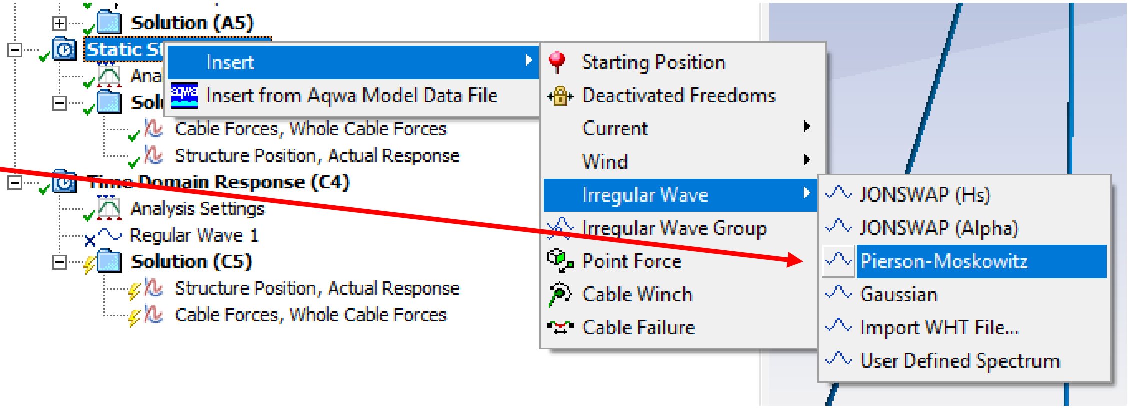

Details

| Details of Analysis Settings | |
|---|---|
| Name | Analysis Settings |
| Computation Type | Time Response Analysis |
| External Operation before Solving | None |
| External Operation after Solving | None |
| Parallel Processing | Program Controlled |
| Use Cable Dynamics | Yes |
| **Time Response Specific Options** | |
| Analysis Type | Irregular Wave Response |
| Start Time | 0.0 s |
| Time Step | 0.1 s |
| Output Step | 0.1 s |
| Duration | 180 s |
| Number of Steps | 1801 |
| Finish Time | 180 s |
| Starting Position | Determined by Upstream System |

# Set Irregular Wave Characteristics

- When simulating an irregular wave the stability analysis takes account of the mean drift forces to calculate the starting position to be used in time and frequency domain analysis.

**RMB on Static Stability and insert a Pierson-Moskowitz Wave Spectra**



# Set Irregular Wave Characteristics

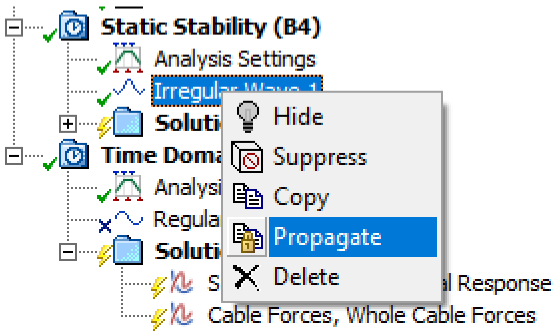
- Set Direction to 90 degrees
- Set Seed Definition to Manual Definition and Seed to 10001 (this is used to generate the random phase relationships of the wave components)
- Leave Start and Finish Frequency Definition as Program Controlled (they will depend upon the Significant Wave Height and Zero Crossing Period defined below)
- Set Significant Wave Height to 2.5 m
- Set Zero Crossing Period to 10 s
- Add Global X and Global Y to the existing Structure Position > Actual Response graph: right-click > Duplicate on the existing input and set the Component as necessary

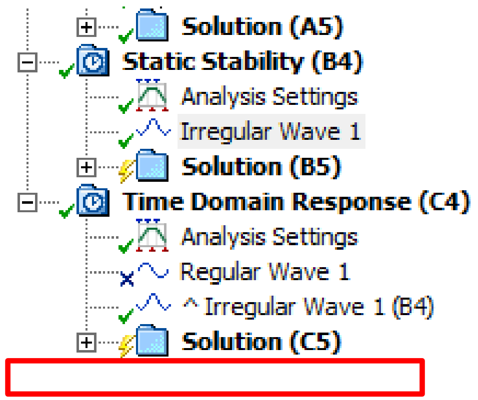
Details

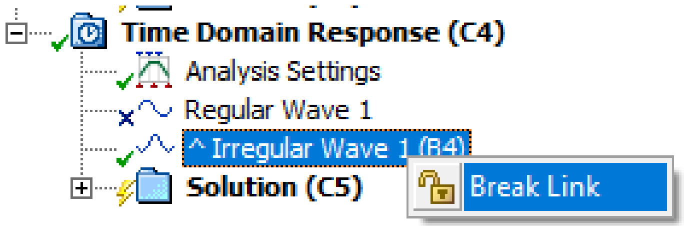
| Details of Irregular Wave 1 | |
|---|---|
| Name | Irregular Wave 1 |
| Visibility | Visible |
| Activity | Not Suppressed |
| Wave Range Defined By | Frequency |
| **Wave Spectrum Details** | |
| Wave Type | Pierson-Moskowitz |
| Direction of Spectrum | 90° |
| Wave Spreading | None (Long-Crested Waves) |
| Spectrum Presentation Method | 1D Graph |
| Seed Definition | Manual Definition |
| Seed | 10001 |
| Number of Spectral Lines Definiti... | Program Controlled |
| Omit Calculation of Drift Forces | No |
| Start and Finish Frequency Defin... | Program Controlled |
| Start Frequency | 0.25769 rad/s |
| Finish Frequency | 2.27036 rad/s |
| Significant Wave Height | 2.5 m |
| Zero Crossing Period | 10 s |
| Export CSV File | Select CSV File... |
| **Cross Swell Details** | |
| Wave Type | None |

# Propagate Ocean Environment

- Right-click on the Irregular Wave 1 and then Propagate.
- This command will automatically propagate the selected environmental condition to a downstream analysis.
- NOTE: this command creates a link between each of the generated environments, adjustments to the first definition will be propagated to any linked system.
- If needed the user can break the link with a simple click.

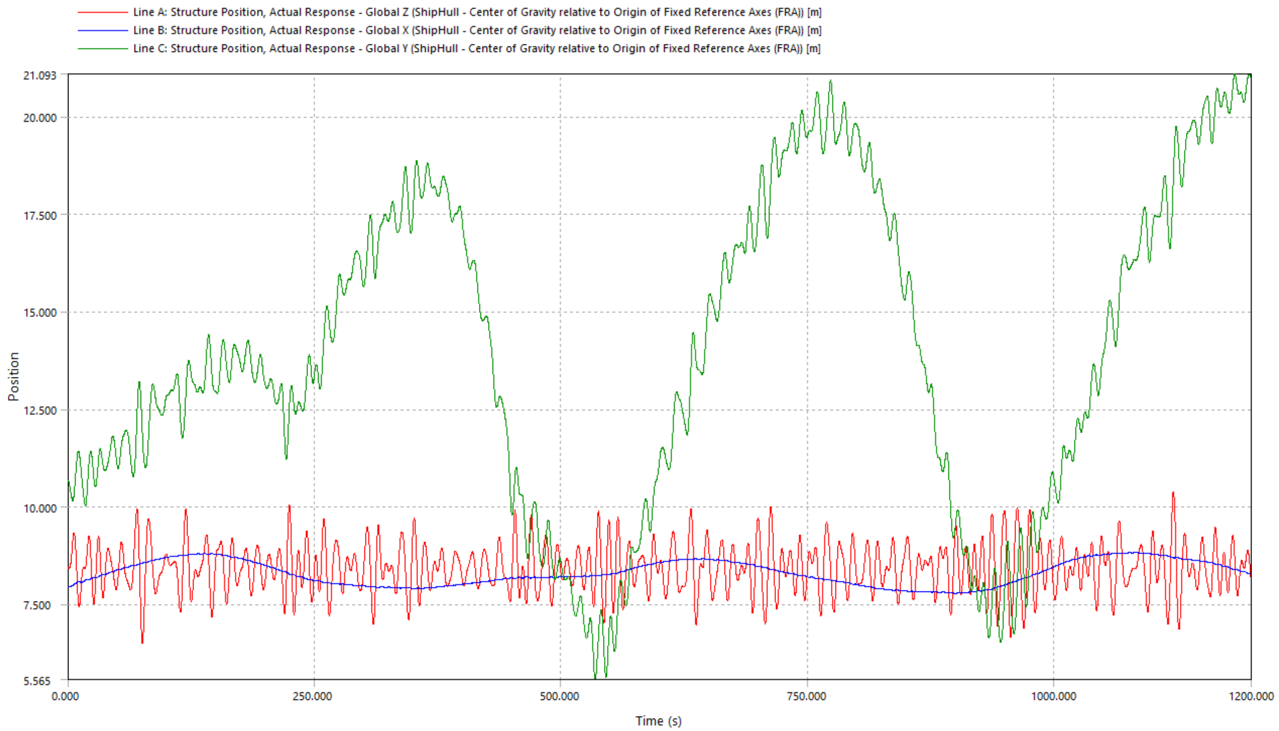


# Set Analysis Settings and Solve

- Set Duration to 1200 s
- Run the analysis by clicking on Solution and Solve
- After solving review the results

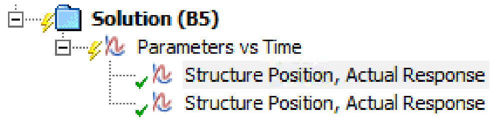



Details

| Details of Analysis Settings | |
|---|---|
| Name | Analysis Settings |
| Computation Type | Time Response Analysis |
| External Operation before Solving | None |
| External Operation after Solving | None |
| Parallel Processing | Program Controlled |
| Use Cable Dynamics | Yes |
| **Time Response Specific Options** | |
| Analysis Type | Irregular Wave Response |
| Start Time | 0.0 s |
| Time Step | 0.1 s |
| Output Step | 0.1 s |
| Duration | 1200 s |
| Number of Steps | 12001 |
| Finish Time | 1200 s |
| Starting Position | Determined by Upstream System |
| X-Position for Wave Surface Ele... | 0.0 m |
| Y-Position for Wave Surface Ele... | 0.0 m |
| **Common Analysis Options** | |
| Convolution | Yes |
| Call Routine "user_force" | No |
| Connect to Server for External "... | No |
| Use Linear Starting Conditions | No |
| Use Linear Stiffness Matrix to C... | No |
| Account for Current Phase Shift | Yes |
| Use Wheeler Stretching | With Linear Wave Theory |
| **Output File Options** | |
| Axis System for Joint Reactions | Fixed Reference Axes |
| Data List | Yes |
| Element Properties | No |
| Dynamic Cable/Tether Drag | No |

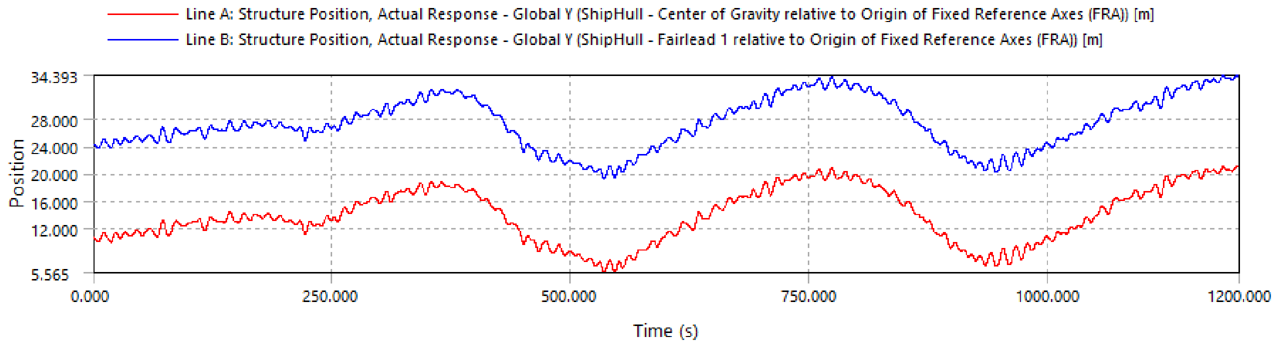
# Nodal Results

- The reference point for nodal results, such as position, velocity and acceleration, defaults to the centre of gravity for the structure.
- It is possible to also report results at other points defined as a connection point. Note that the connection point does not have to be associated with Connections data, it can be defined just to allow nodal results information.
- Insert a new Structure Position > Actual Response in the Time Response Analysis Solution, set the results selection as shown. Evaluate All Results.



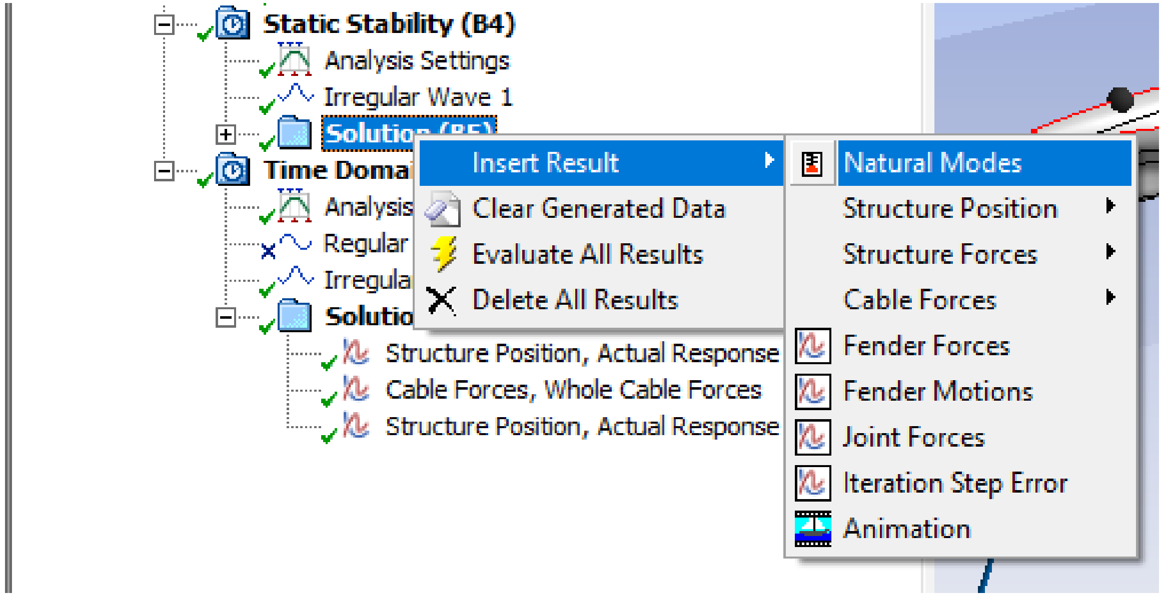

| Details | |
|---|---|
| **Details of Structure Position, Actual Response** | |
| Name | Structure Position, Actual Response |
| **Line Inputs** | |
| Structure | Ship |
| Type | Structure Position |
| SubType | Actual Response |
| Component | Global Y |
| Reference Point | Center of Gravity (Ship) |
| Motion Relative To | Origin of Fixed Reference Axes (FRA) |

| Details | |
|---|---|
| **Details of Structure Position, Actual Response** | |
| Name | Structure Position, Actual Response |
| **Line Inputs** | |
| Structure | Ship |
| Type | Structure Position |
| SubType | Actual Response |
| Component | Global Y |
| Reference Point | Fairlead 1 |
| Motion Relative To | Origin of Fixed Reference Axes (FRA) |



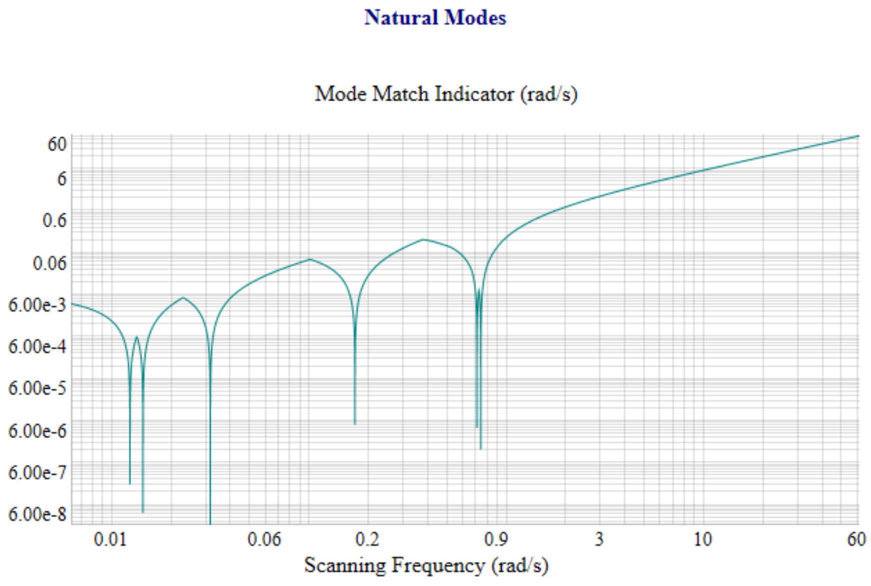
# Modal Analysis

- We can also check the natural modes for the floating system, including the effects of any external connections
- Under the Static Stability Solution Insert Result > Natural Modes, then select Evaluate All Results



# Natural Modes

- As Natural Modes depend on frequency-dependent Added Mass and Damping, a direct calculation of the eigenvalues and the eigenmodes of a single matrix is not possible. Aqwa scans a range of input frequencies and solves the eigenvalue problem based on Added Mass and Damping values for the current scanning frequency until the input frequency equals the response frequency.
- The Mode Match Indicator graphically shows the difference between the scanning frequency and the response frequency. The local minimums correspond to the natural frequencies of the system.



# Natural Modes

- Information related to a given mode is reported, together with the level of critical damping associated with that mode. The associated modal amplitudes provide an indicator of the modal direction, and can be animated by clicking on the Animate button. The primary mode is in sway.

**Found 6 modes**

**Mode 1 of 6 Modes found** **Stable**

**Frequency: 0.01250 rad/s**

**Period: 502.68639 seconds**

**Damping: 4.36647 % of Critical Damping**

**Structure: ShipHull**

Scroll back to graph

Animate

| | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Amplitudes | **X**: | 0.43872 m | **Y**: | 9.98008 m | **Z**: | 0.00028 m | **RX**: | 0.25696 ° | **RY**: | 0.00196 ° | **RZ**: | 0.37353 ° |
| Phases | **X**: | -12.49110 ° | **Y**: | 0.0000 ° | **Z**: | -177.20189 ° | **RX**: | 2.29344 ° | **RY**: | 1.88642 ° | **RZ**: | 1.56282 ° |

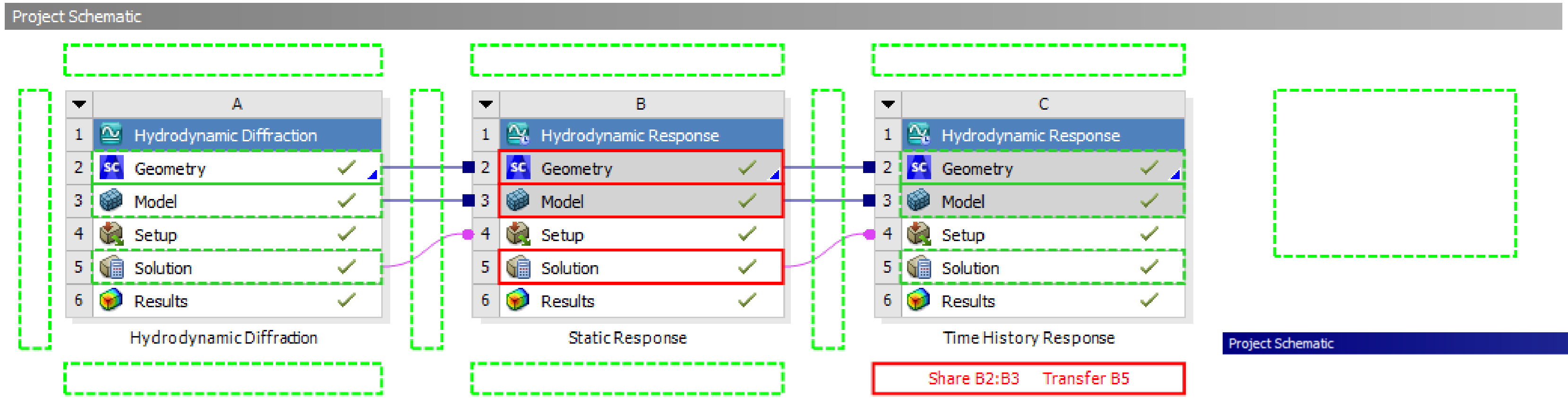




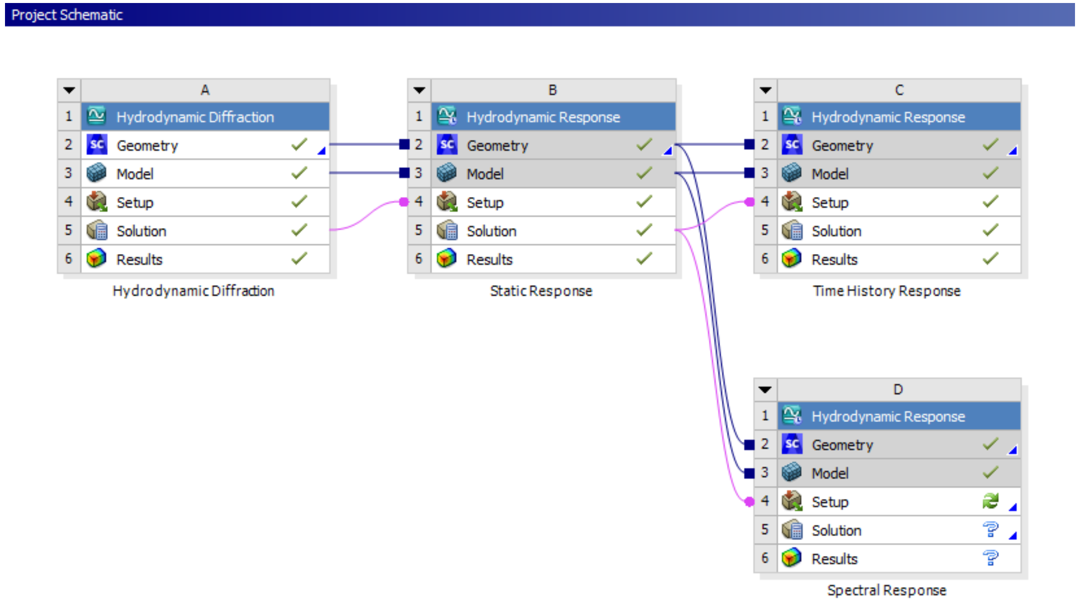
Ansys

# Frequency Domain Analysis

- Go back to the WB project page and add a new Hydrodynamic Response analysis system downstream  of the Stability (Static Response) system.




- Rename the new system as Spectral Response



# Frequency Domain Analysis Setting

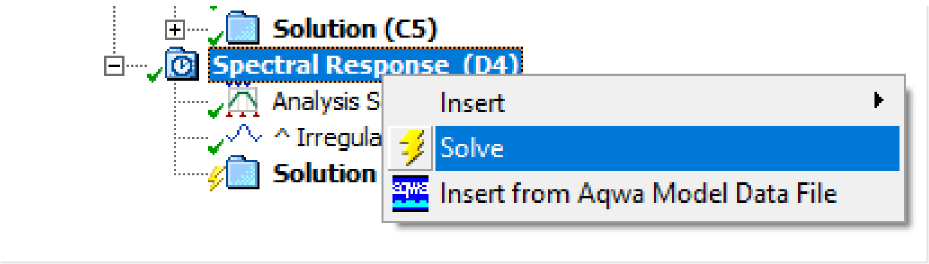
- In Aqwa rename the new Hydrodynamic Response to Spectral Response.
- Go to Analysis Settings Panel and set the following analysis options:
- Computation Type: Frequency Statistical Analysis
- Analysis Type: Wave and Drift Frequencies
- Note that the panel is showing an error (red highlight) for the Spectrum Sub-Direction of Output for RAOs. This is because at this point we do not have a wave spectrum defined.

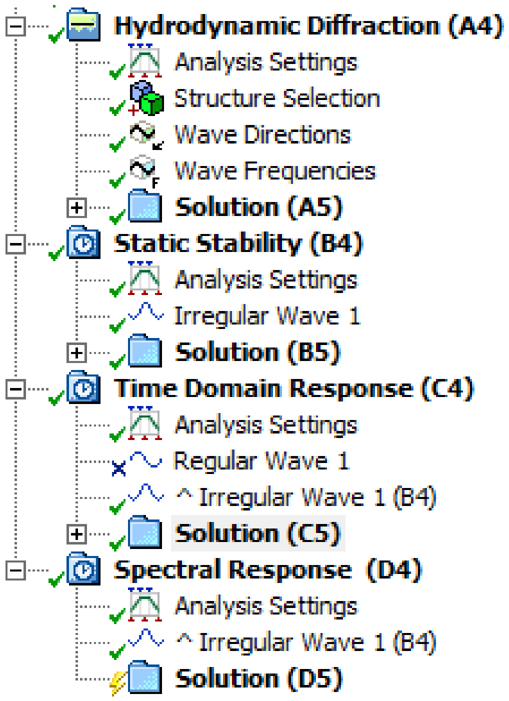
Details

| | |
|---|---|
| **Details of Analysis Settings** | |
| Name | Analysis Settings |
| Computation Type | Frequency Statistical Analysis |
| External Operation before Solving | None |
| External Operation after Solving | None |
| Parallel Processing | Program Controlled |
| Use Cable Dynamics | Yes |
| **Frequency Response Specific Options** | |
| Analysis Type | Wave and Drift Frequencies |
| Direction of Output for RAOs | Program Controlled |
| Spectrum Sub-Direction of Outp... | Not Available |
| Axis System for Motion Statistic... | Local Structure Axes |
| Starting Position | Determined by Upstream System |
| **Common Analysis Options** | |
| Calculate RAOs with Mooring Lin... | No |
| Apply Drift Force with Multi-Dire... | No |
| Linearized Morison Drag | No |
| **QTF Options** | |
| Use Full QTF Matrix | No |
| **Output File Options** | |
| Axis System for Joint Reactions | Fixed Reference Axes |
| Data List | Yes |
| Element Properties | No |

# Frequency Domain Environment definition

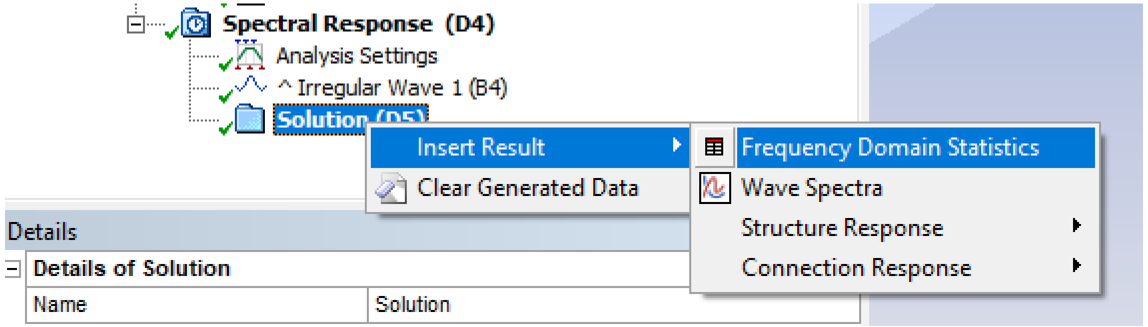
- From the Static Stability analysis propagate the Irregular Wave 1.
- RMB click on the Spectral Response and solve the model.

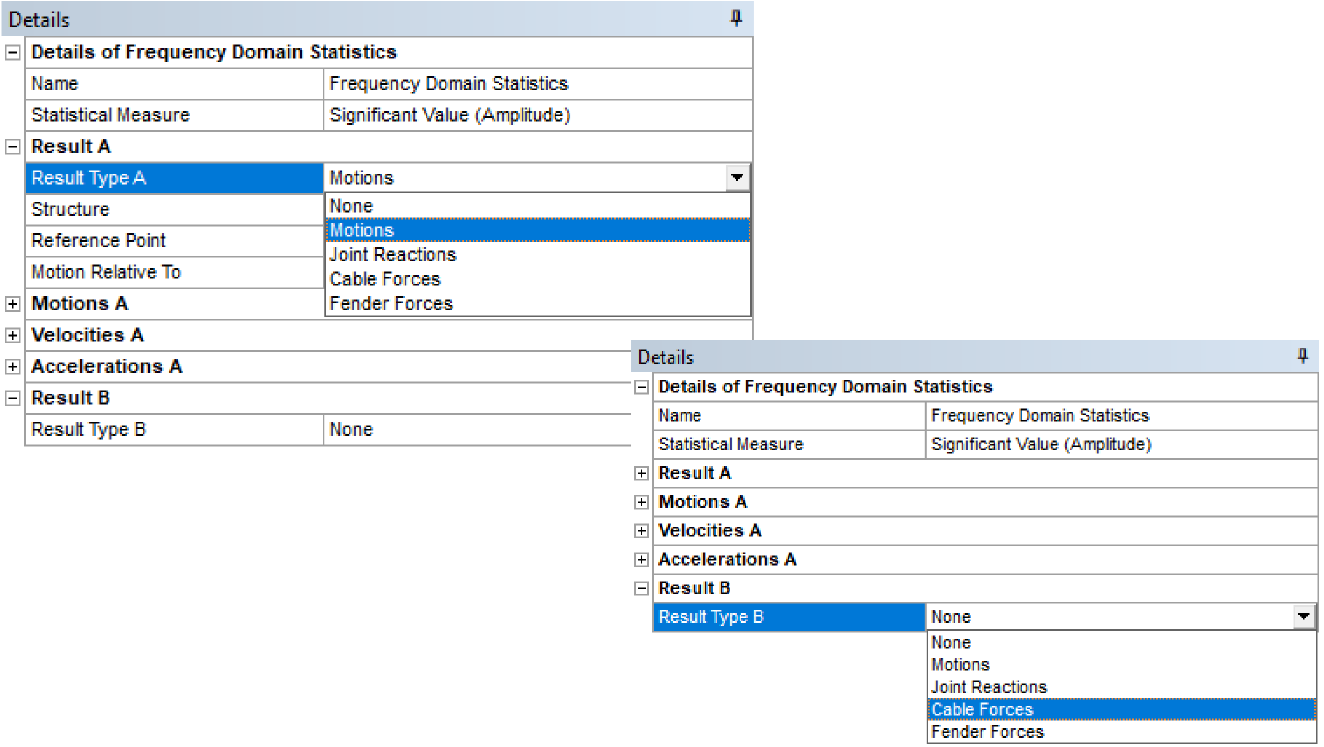


# Frequency Domain Results

- RMB click on Solution (D5) and Insert Result > Frequency Domain Statistics
- Specify the required results:
- Motions
- Cable Forces for cables 1 and 4
- Results are automatically updated



# Frequency Domain Results

**Frequency Domain Statistic Results**

Displaying Significant Value (Amplitude)

**Part Name: ShipHull**

| **Motions:** | *at Center of Gravity, in Local Structure Axes* | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Position | X: | 0.59641 m | Y: | 16.94637 m | Z: | 1.30368 m | RX: | 0.58872 ° | RY: | 0.08018 ° | RZ: | 0.71553 ° |
| Velocity | X: | 0.01132 m/s | Y: | 0.50281 m/s | Z: | 0.71169 m/s | RX: | 0.23651 °/s | RY: | 0.06070 °/s | RZ: | 0.01430 °/s |
| Acceleration | X: | 0.00596 m/s² | Y: | 0.26447 m/s² | Z: | 0.43228 m/s² | RX: | 0.17836 °/s² | RY: | 0.04791 °/s² | RZ: | 0.00418 °/s² |

| **Cable Tensions:** | | | | |
|---|---|---|---|---|
| Cable 1 | Fairlead 1 (ShipHull): | 80547.60156 N | Anchor 1 (Fixed): | 63395.20313 N |
| Cable 4 | Fairlead 4 (ShipHull): | 96992.70313 N | Anchor 4 (Fixed): | 80489.97656 N |