Workshop 2: Aqwa Basics – Hydrodynamic Response

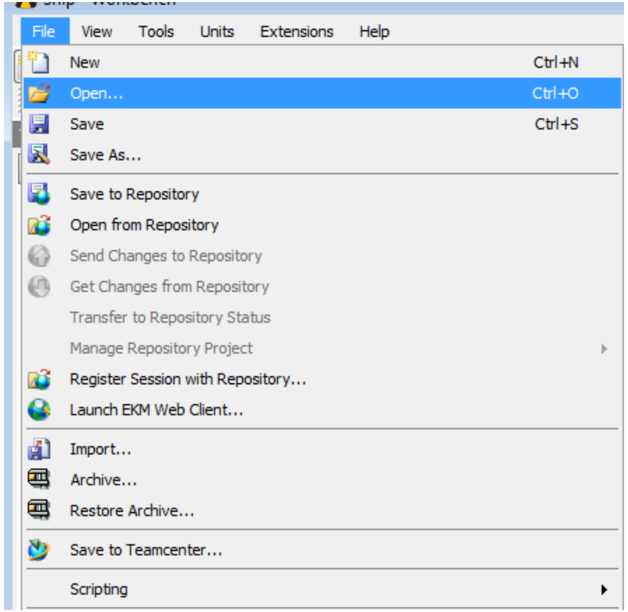
Introduction to Hydrodynamic Analysis with ANSYS Aqwa

ANSYS Release 19.2

Hydrodynamic Response (HR) Simulation

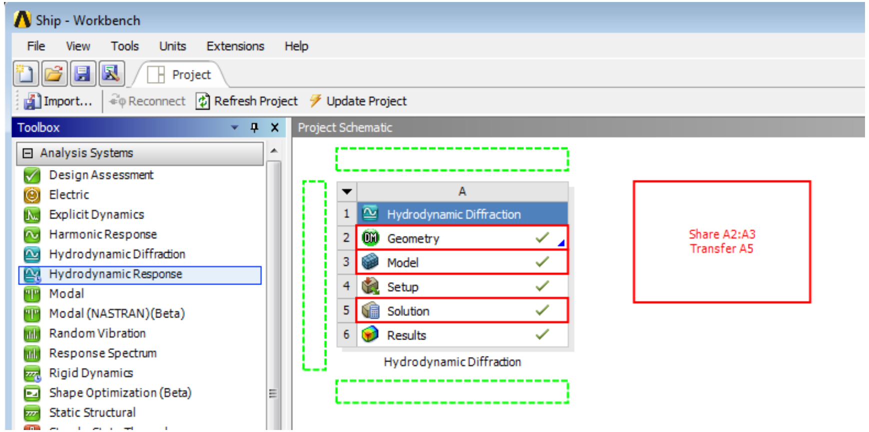
The goal of this workshop is to create a mooring system for a ship and run a series of static and dynamic response analyses

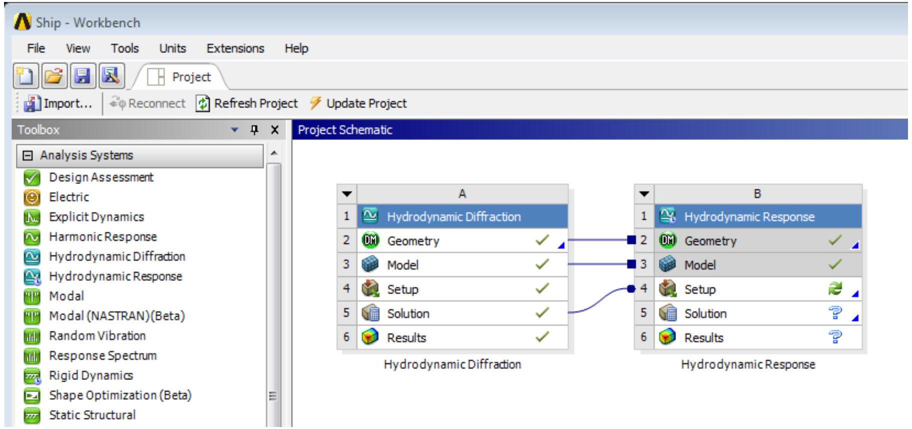
Open the Ship project previously created in Workshop 1



Adding the Static HR System

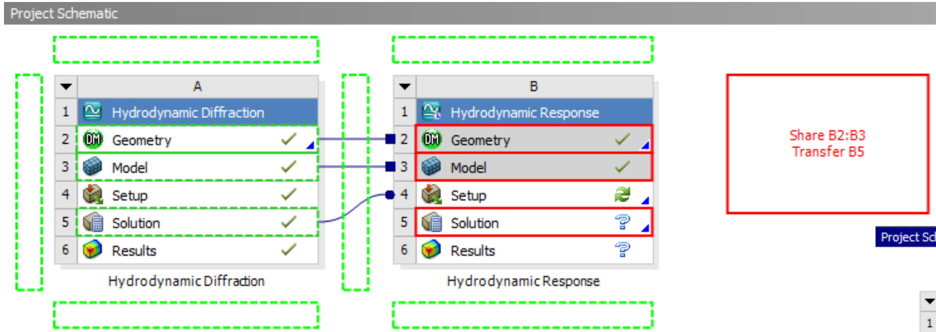
Drag and drop the Hydrodynamic Response to the WB Project Schematic while sharing the solution from Hydrodynamic Diffraction. This will become the static stability solution to be undertaken before running dynamic simulations

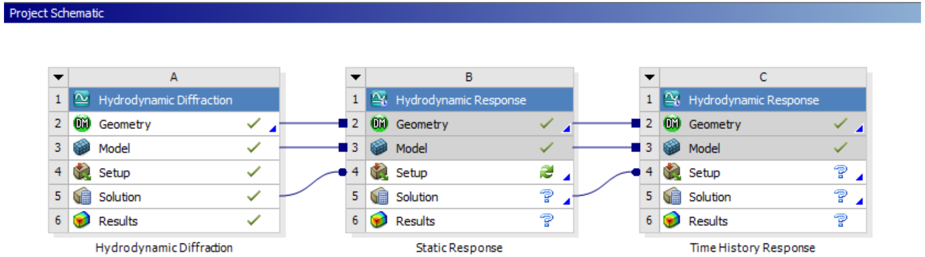




Adding a Dynamic HR System

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



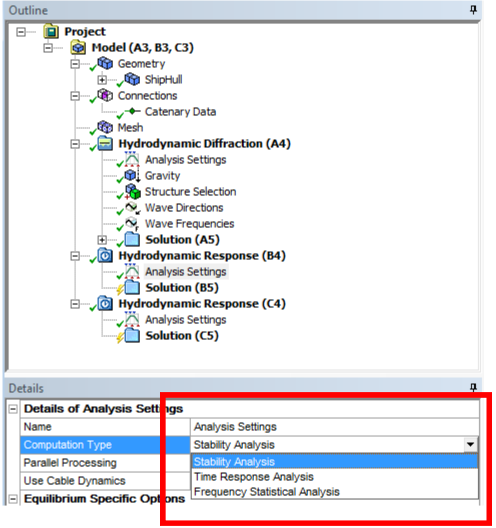


Drag and drop the new HR system on the Solution cell of the existing HR system, and double click on the Setup cell. Note that we can re-name the systems by RMB clicking on the system header and selecting Rename

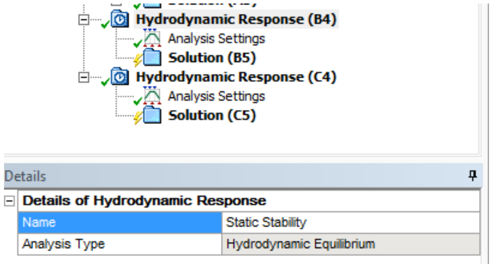
Set up Static Stability Response System

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.

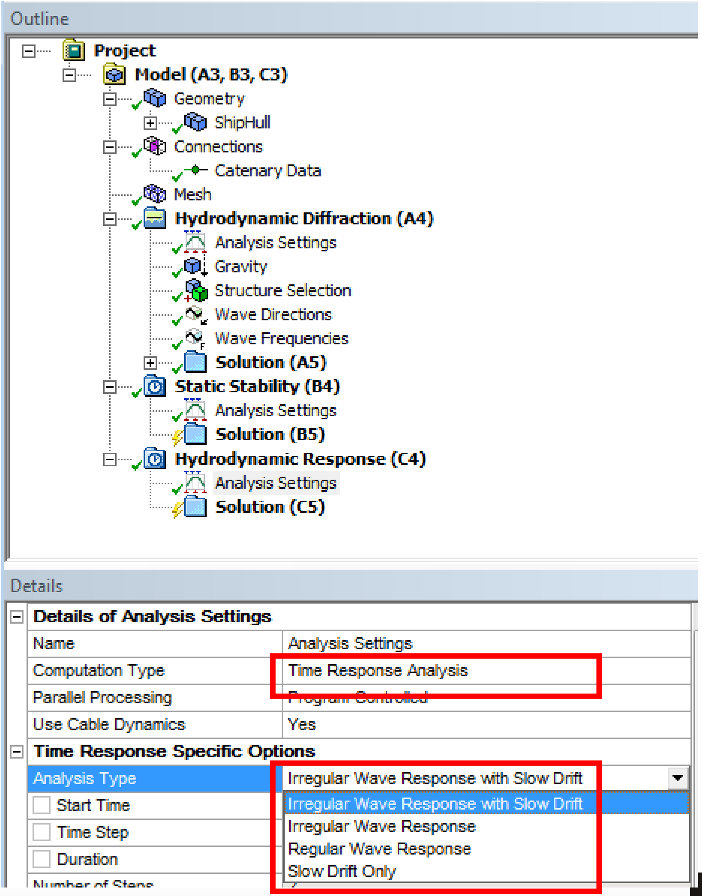


Set up Time Response System

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

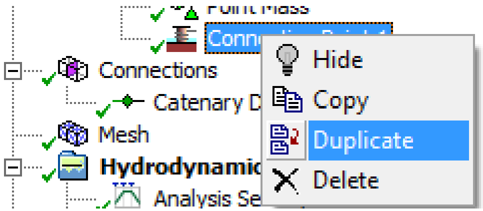
Add four fairlead locations for the mooring system

Select Geometry > ShipHull > Add > Connection Point

Enter X,Y,Z coordinates in details panel as below (in global system!)

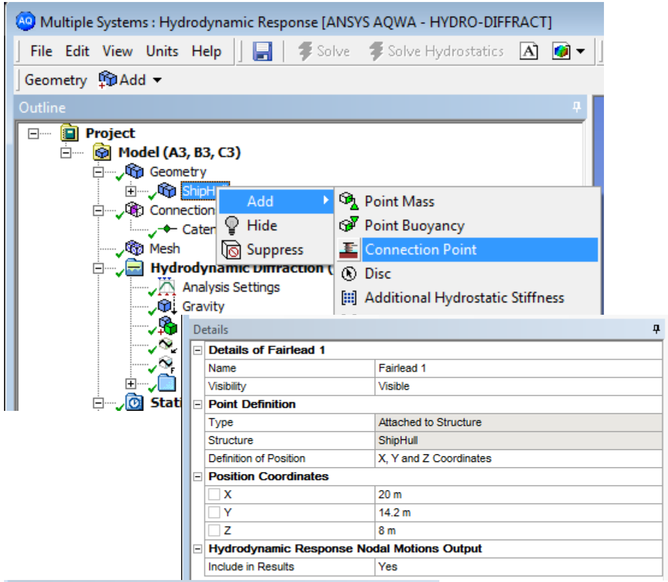
Add 3 other connection points (can use Duplicate as an alternative)

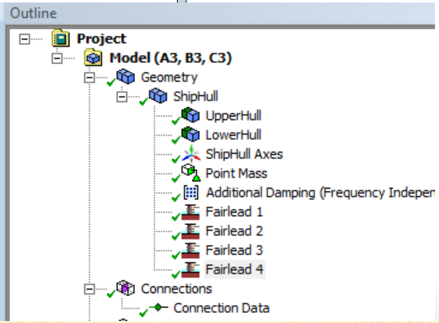
- Point 2 20.0, -14.2, 8.0
- Point 3 180.0, 16.0, 8.0
- Point 4 180.0, -16.0, 8.0



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

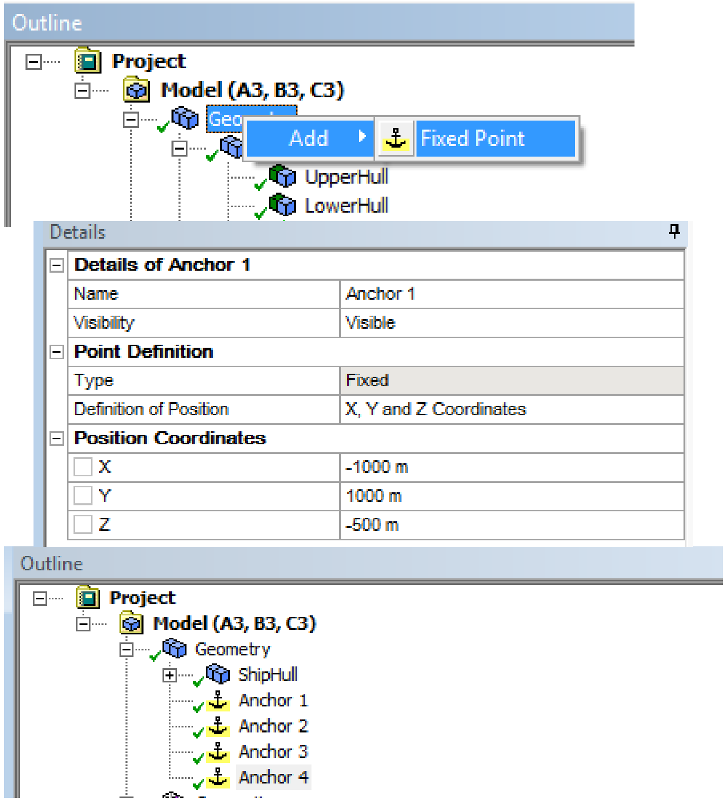
Select Model > Geometry > Add > Fixed Point

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

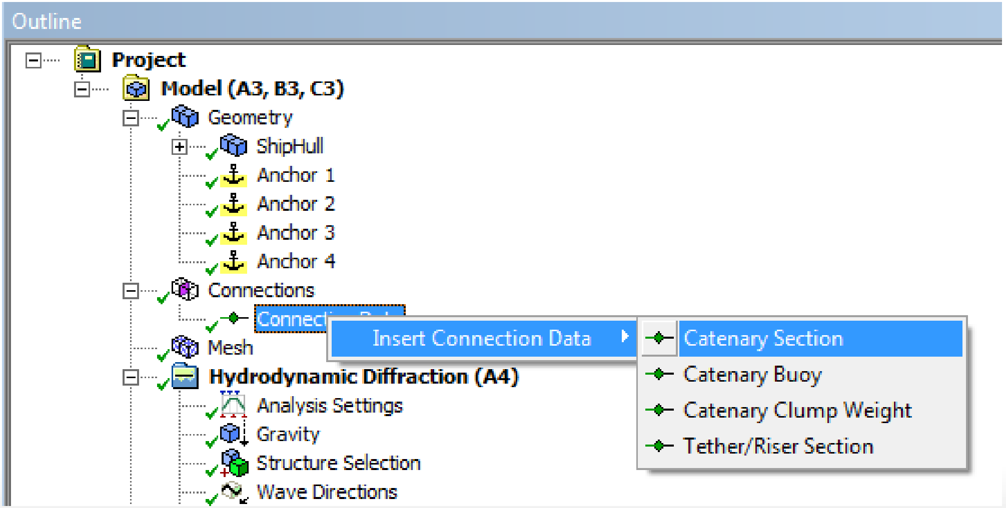
Rename Fixed Points as Anchor 1 through 4



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

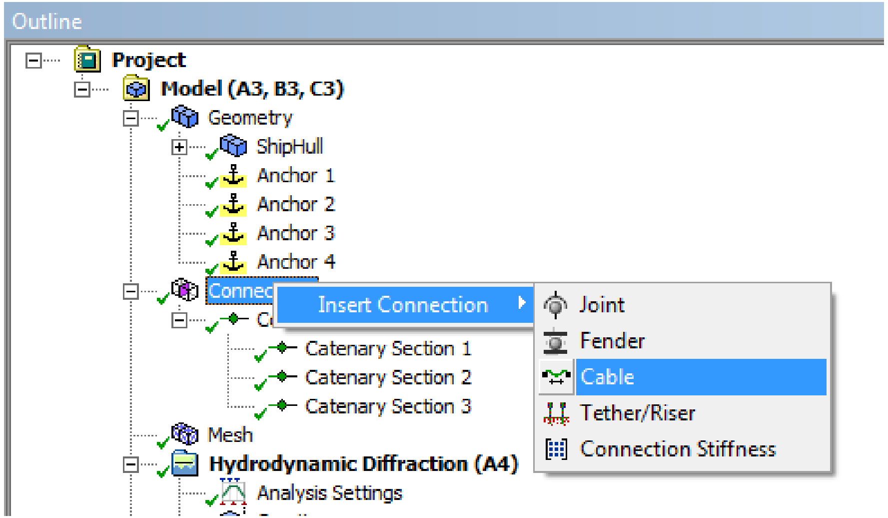
Details of Catenary Section 3	
Name	Catenary Section 3
Section Properties	
Mass/Unit Length	170 kg/m
Equivalent Cross-Sectional A...	0.01 m²
Stiffness, EA	600000000 N
Maximum Tension	7500000 N
Bending Stiffness, EI	0.0 N.m²
Axial Stiffness Coefficient k1	0.0 N
Axial Stiffness Coefficient k2	0.0 N
Axial 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

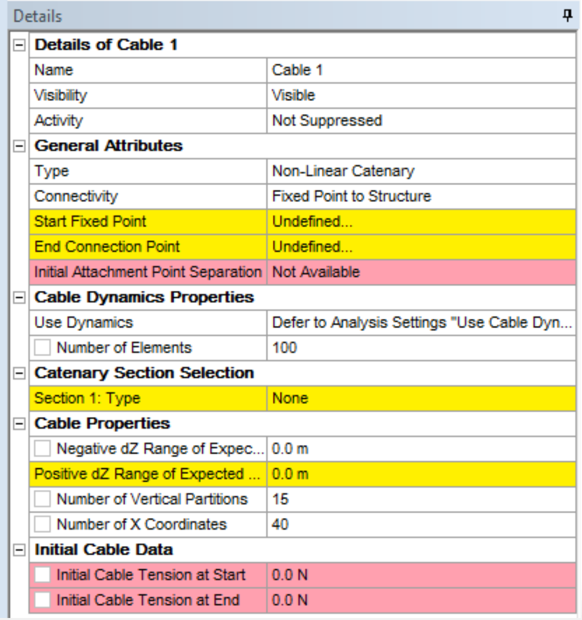
Mooring Line Connections

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



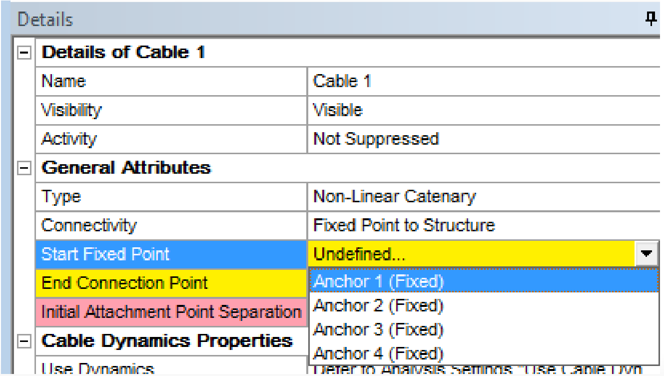


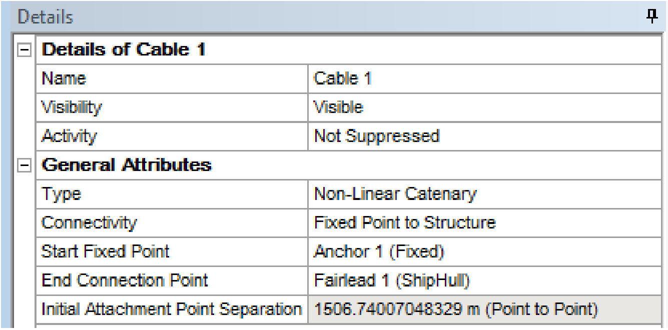
Mooring Line Connections

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

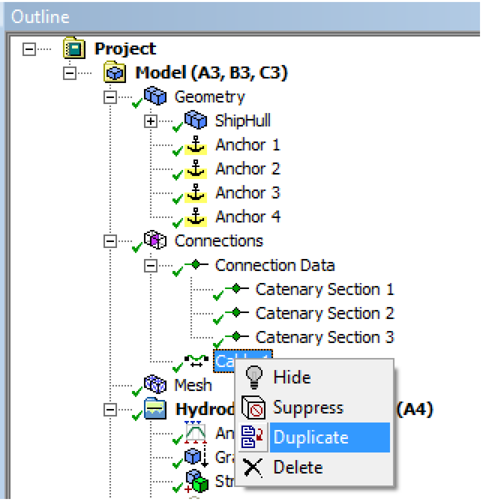
Details of Cable 1	
Name	Cable 1
Visibility	Visible
Activity	Not Suppressed
General Attributes	
Type	Non-Linear Catenary
Connectivity	Fixed Point to Structure
Start Fixed Point	Anchor 1 (Fixed)
End Connection Point	Fairlead 1 (ShipHull)
Initial Attachment Point Separation	1506.74007048329 m (Point to Point)
Cable Dynamics Properties	
Use Dynamics	Defer to Analysis Settings "Use Cable Dyn...
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 Expec...	10 m
Positive dZ Range of Expect...	10 m
Number of Vertical Partitions	15
Number of X Coordinates	40
Initial Cable Data	
Initial Cable Tension at Start	1005830.813 N
Initial Cable Tension at End	1771234.75 N

ANSYS

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	Non-Linear 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 (Po

Details

Details of Cable 3	
Name	Cable 3
Visibility	Visible
Activity	Not Suppressed
General Attributes	
Type	Non-Linear 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 Po

Details

Details of Cable 4	
Name	Cable 4
Visibility	Visible
Activity	Not Suppressed
General Attributes	
Type	Non-Linear 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

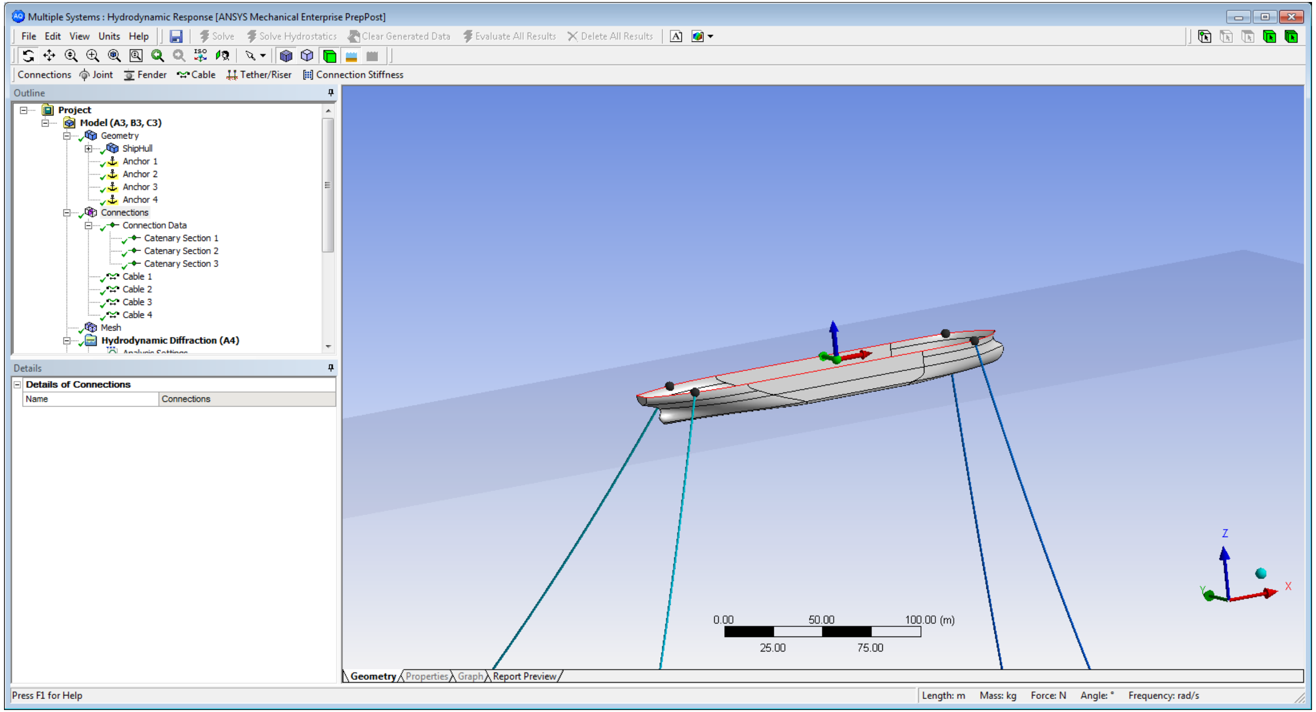
The Joint 1-2 etc refers to connection data such as clump weights or buoys

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

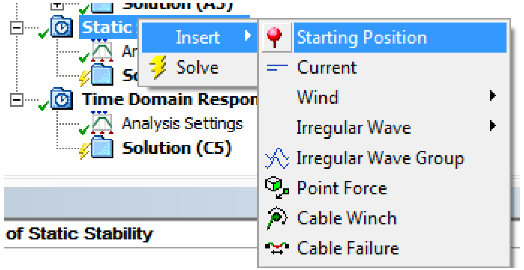
Select Connections in the Outline to see the final mooring configuration

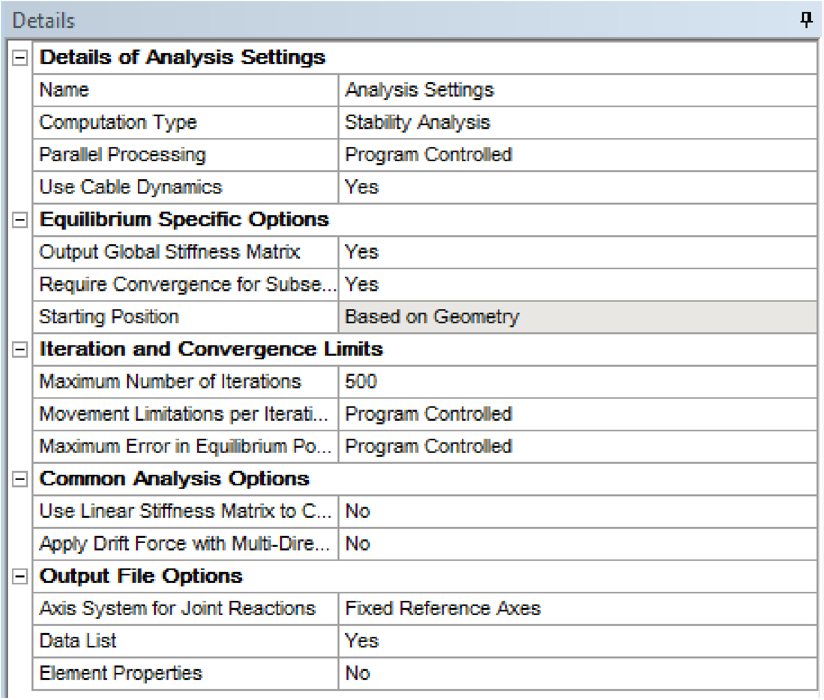


Stability Analysis Options

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
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 Iterati...	Program Controlled
Maximum Error in Equilibrium Po...	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

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
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
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
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

Define the Environment

RMB on Time Domain Response > Insert > Regular Wave

Set Wave Type to Airy Wave Theory

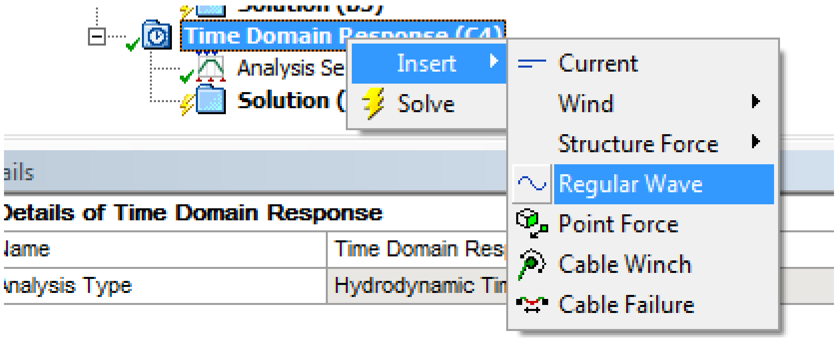
Set Direction to 90 degrees

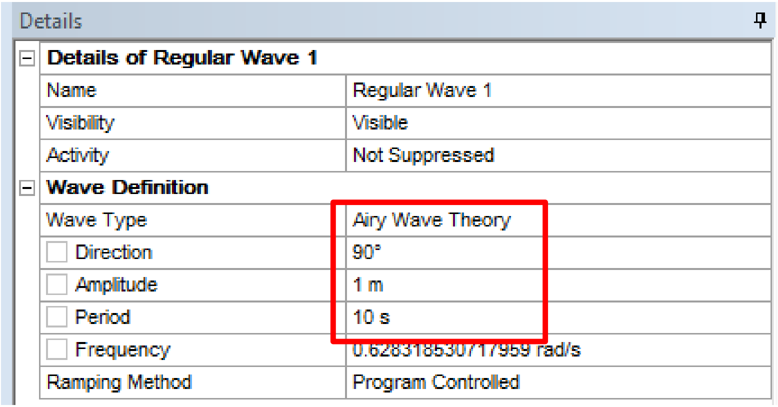
Set Amplitude to 1 m

Set Period to 10 s

Run the analysis by RMB clicking on Solution and Solve

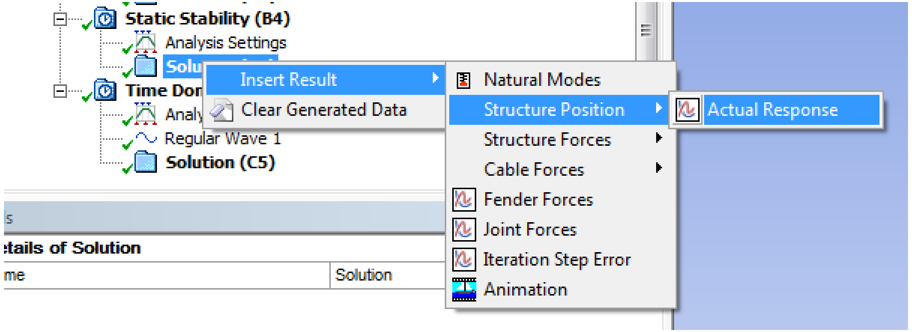
NOTE: Solving the Time Response analysis will automatically solve the upstream stability analysis to compute the correct starting condition.





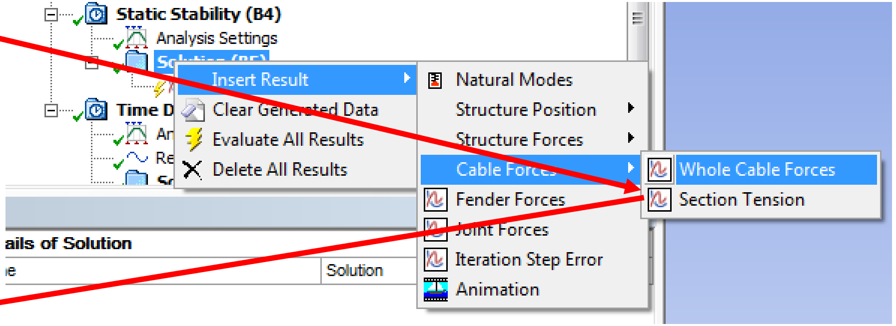
Check the Solution

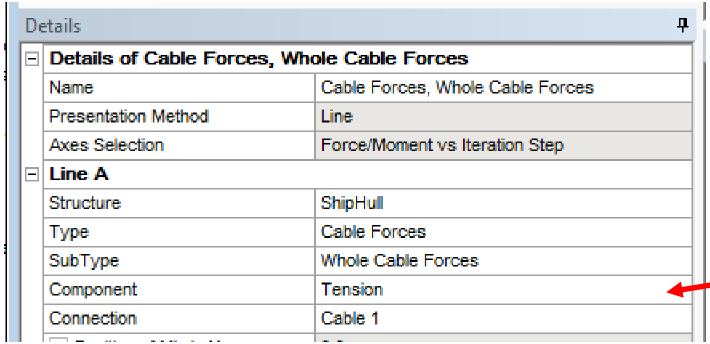
Go to Static Stability, RMB click on Solution then Insert Result >Structure Position >Actual Response.



Scope the result to Global X, Y, Z motions.

Following the same process insert Whole Cable Forces, Tension.





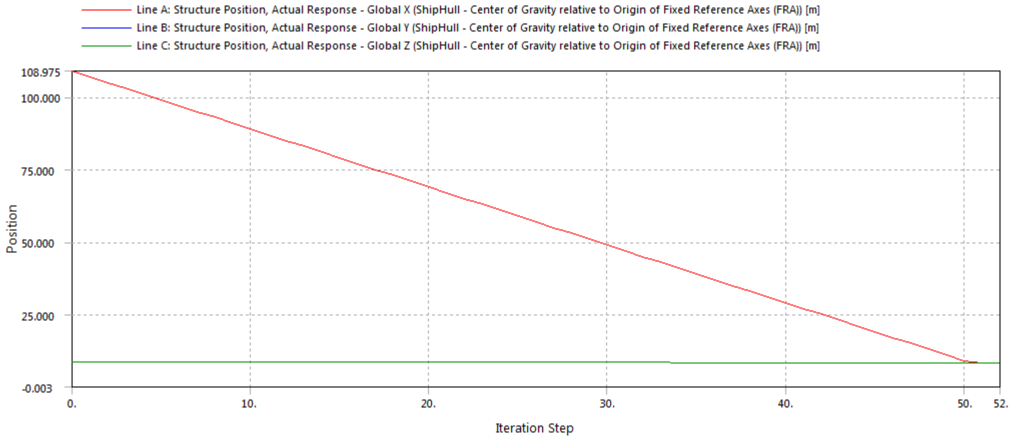
Details

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
Component	Tension
Connection	Cable 1

Evaluate all results

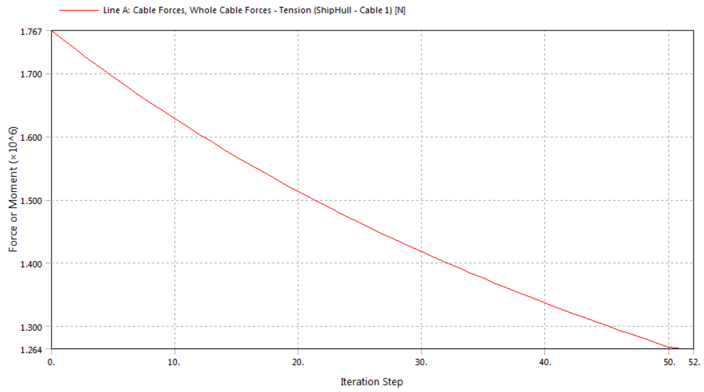
Check the Stability Solution

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 a dynamic analysis and becomes a <u>MUST</u> when a Frequency statistical analysis is undertaken!!!

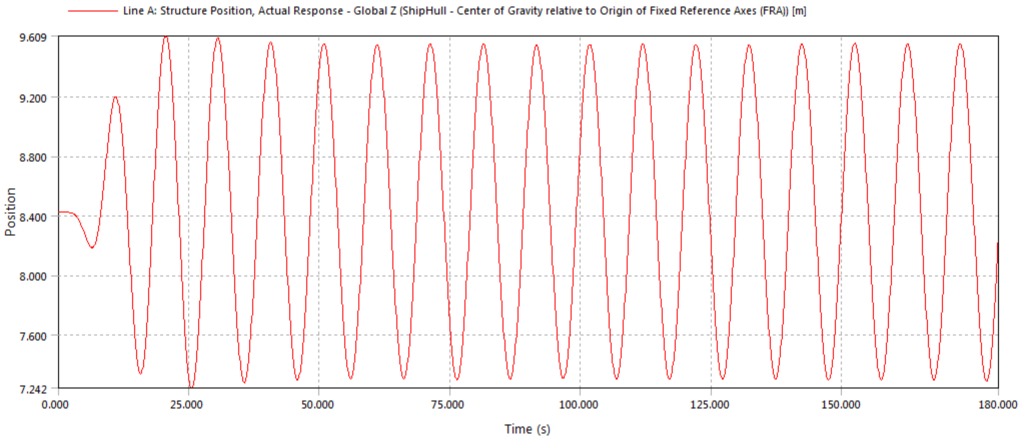


Check the Time Response Solution

Under the Time Response Analysis Solution add a graph item - Insert Result > Structure Position > Actual Response

Select Global Z

Evaluate All Results



Details

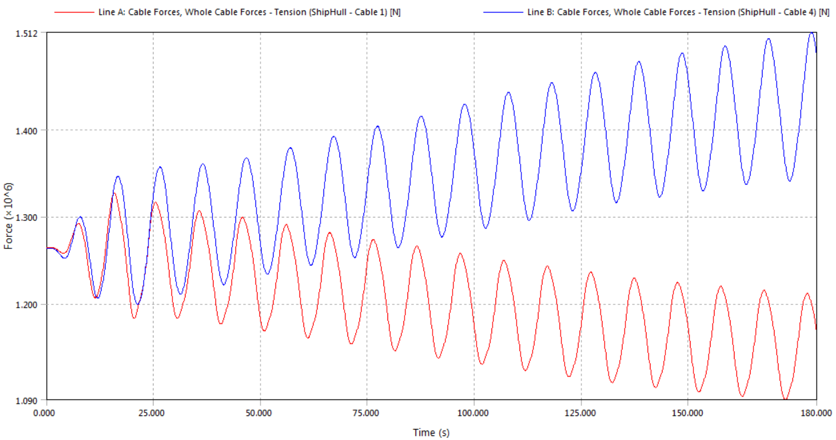
Details of Structure Position, Actual Response	
Name	Structure Position, Actual Response
Presentation Method	Line
Axes Selection	Distance/Rotation vs Time
Line A	
Structure	ShipHull
Type	Structure Position
SubType	Actual Response
Component	Global Z
Reference Point	Center of Gravity (ShipHull)
Motion Relative To	Origin of Fixed Reference Axes (FRA)
Abscissa Position of Minimum	0.0 s
Abscissa Position of Maximum	0.0 s
Minimum Value	0.0 m
Maximum Value	0.0 m
Line B	
Structure	Undefined...

Check the Time Response Solution

Under the Time Response Solution add a second graph item - Insert Result > Cable Forces > Whole Cable Forces

Select Tension for Cables 1 and 4

Evaluate All Results



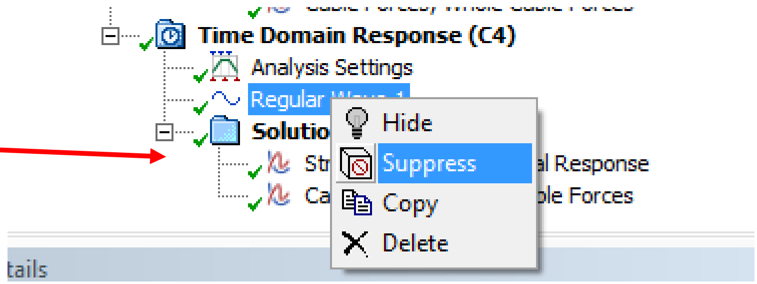
Details

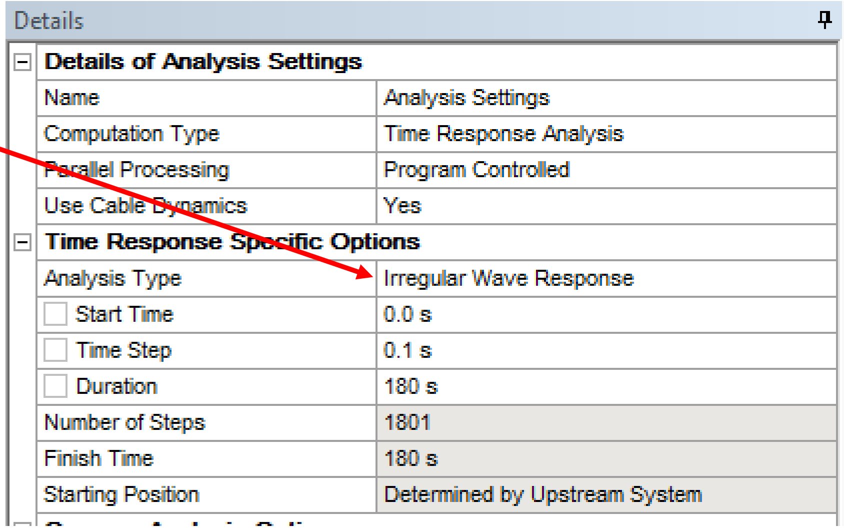
Details of Cable Forces, Whole Cable Forces	
Name	Cable Forces, Whole Cable Forces
Presentation Method	Line
Axes Selection	Force/Moment vs Time
Export CSV File	Select CSV File...
Line A	
Structure	ShipHull
Type	Cable Forces
SubType	Whole Cable Forces
Component	Tension
Connection	Cable 1
Abscissa Position of Minimum	172.7 s
Abscissa Position of Maximum	15.8 s
Minimum Value	1090140.625 N
Maximum Value	1327067.375 N
Line B	
Structure	ShipHull
Type	Cable Forces
SubType	Whole Cable Forces
Component	Tension
Connection	Cable 4
Abscissa Position of Minimum	21.2 s
Abscissa Position of Maximum	178.8 s
Minimum Value	1199565.5 N
Maximum Value	1511682.625 N
Line C	
Structure	Undefined...

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

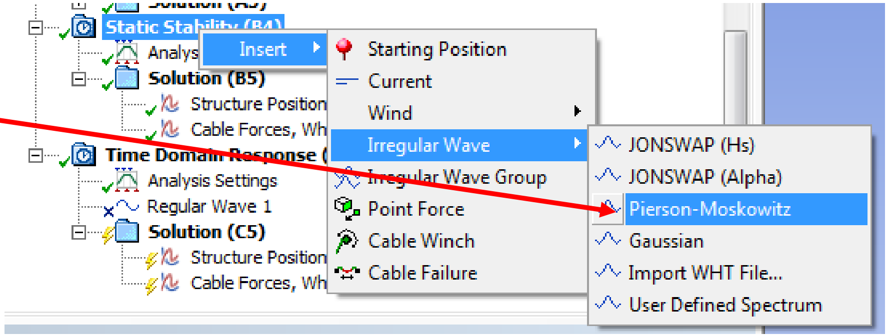




Set Irregular Wave Characteristics

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



ANSYS

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

Details

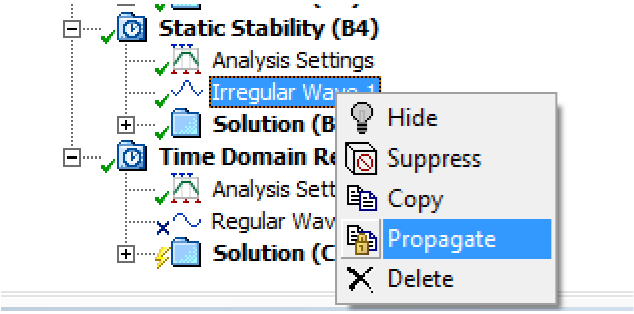
Details of Irregular Wave 1	
Name	Irregular Wave 1
Visibility	Visible
Activity	Not Suppressed
Wave Range Defined By	Frequency
Wave Spectrum Details	
Wave Type	Pierson-Moskowitz
Direction of Spectrum	90°
Wave Spreading	None (Long-Crested Waves)
Spectrum Presentation Method	1D Graph
Seed Definition	Manual Definition
Seed	10001
Number of Spectral Lines Definition	Program Controlled
Omit Calculation of Drift Forces	No
Start and Finish Frequency Definition	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

Details

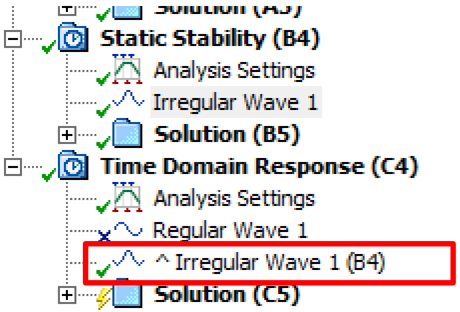
Details of Structure Position, Actual Response	
Name	Structure Position, Actual Response
Presentation Method	Line
Axes Selection	Distance/Rotation vs Time
Line A	
Structure	ShipHull
Type	Structure Position
SubType	Actual Response
Component	Global Z
Reference Point	Center of Gravity (ShipHull)
Motion Relative To	Origin of Fixed Reference Axes (FRA)
Abscissa Position of Minimum	0.0 s
Abscissa Position of Maximum	0.0 s
Minimum Value	0.0 m
Maximum Value	0.0 m
Line B	
Structure	ShipHull
Type	Structure Position
SubType	Actual Response
Component	Global X
Reference Point	Center of Gravity (ShipHull)
Motion Relative To	Origin of Fixed Reference Axes (FRA)
Abscissa Position of Minimum	0.0 s
Abscissa Position of Maximum	0.0 s
Minimum Value	0.0 m
Maximum Value	0.0 m
Line C	
Structure	ShipHull
Type	Structure Position
SubType	Actual Response
Component	Global Y
Reference Point	Center of Gravity (ShipHull)
Motion Relative To	Origin of Fixed Reference Axes (FRA)

Propagate Ocean Environment

RMB on the Irregular Wave 1 and then Propagate.

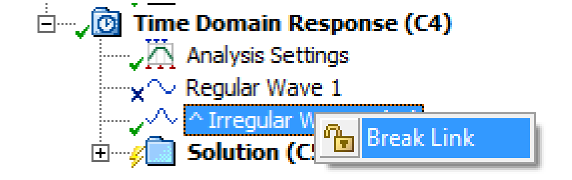


This command will automatically propagate the selected environmental condition to a downstream analysis.



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.

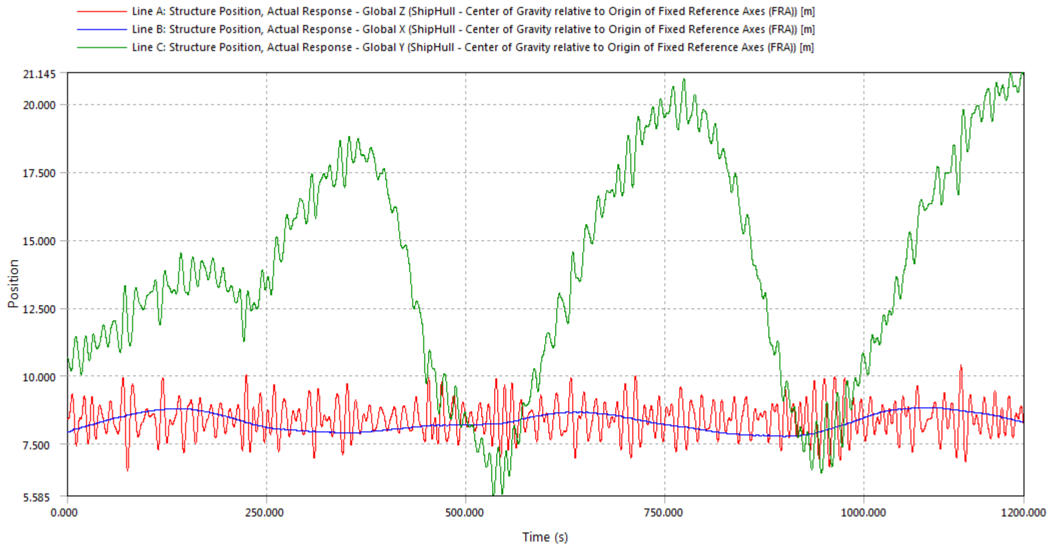


Set Analysis Settings and Solve

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
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
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
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

ANSYS

Nodal Results

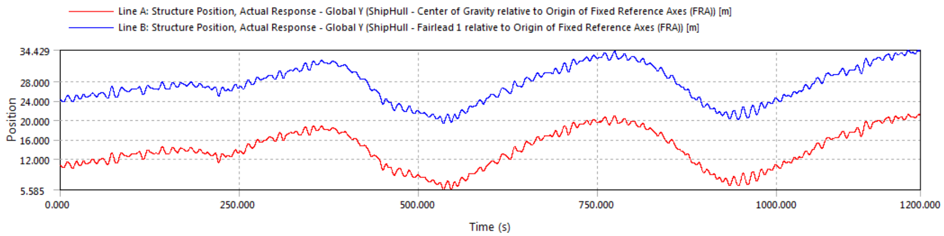
The reference point for nodal results, such as position, velocity and acceleration, defaults to the center of gravity for the structure.

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 Domain Response analysis Solution, set the results selection as shown. Evaluate All Results.

Details

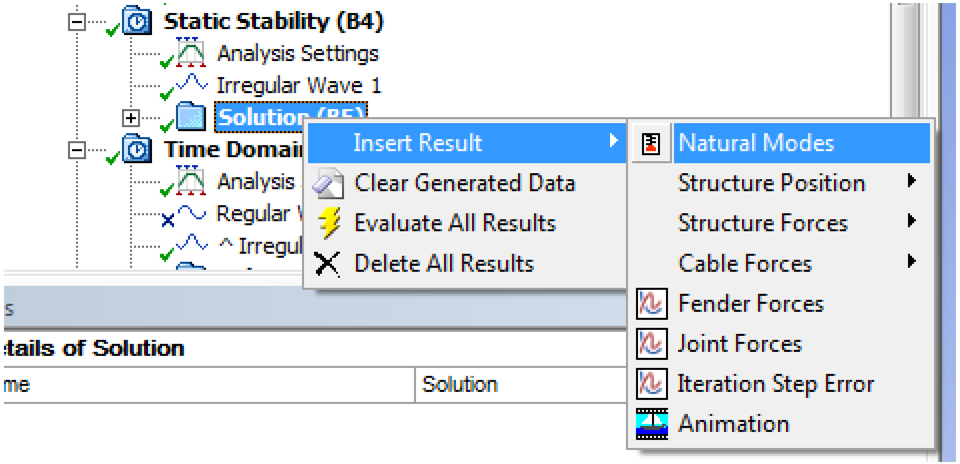
Details of Structure Position, Actual Response	
Name	Structure Position, Actual Response
Presentation Method	Line
Axes Selection	Distance/Rotation vs Time
Export CSV File	Select CSV File...
Line A	
Structure	ShipHull
Type	Structure Position
SubType	Actual Response
Component	Global Y
Reference Point	Center of Gravity (ShipHull)
Motion Relative To	Origin of Fixed Reference Axes (FRA)
Abscissa Position of Minimum	534.6 s
Abscissa Position of Maximum	1183.3 s
Minimum Value	5.585 m
Maximum Value	21.145 m
Line B	
Structure	ShipHull
Type	Structure Position
SubType	Actual Response
Component	Global Y
Reference Point	Fairlead 1
Motion Relative To	Origin of Fixed Reference Axes (FRA)
Abscissa Position of Minimum	534.5 s
Abscissa Position of Maximum	1183.2 s
Minimum Value	19.302 m
Maximum Value	34.429 m



Modal Analysis

We can also check the natural modes for the floating system, including the effects of any external connections

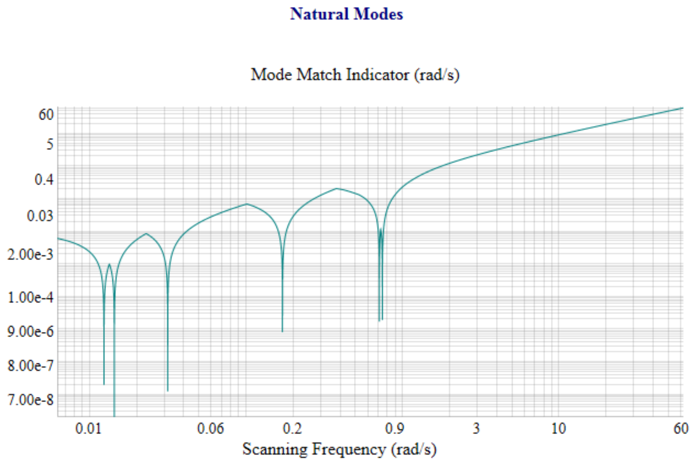
Under the Static Stability Solution Insert Result > Natural Modes, then select Evaluate All Results



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.



ANSYS

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

Damping : 4.36550 % of Critical Damping

Scroll back to graph

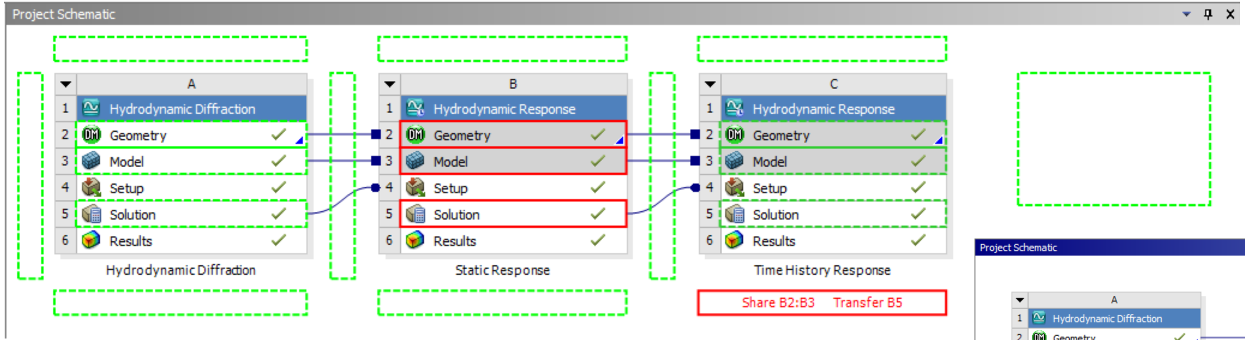
Animate

Structure : ShipHull

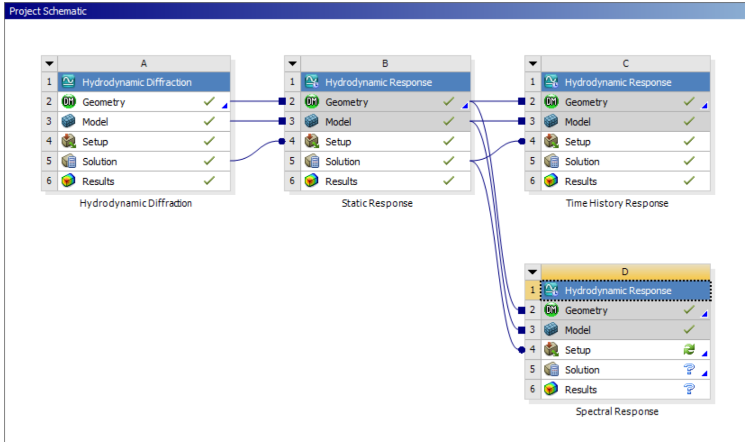
Amplitudes	**X** :	0.27280 m	**Y** :	9.98612 m	**Z** :	0.00028 m	**RX** :	0.25728 °	**RY** :	0.00190 °	**RZ** :	0.36996 °
Phases	**X** :	-11.14154 °	**Y** :	0.0000 °	**Z** :	-177.21461 °	**RX** :	2.23812 °	**RY** :	3.42615 °	**RZ** :	1.51425 °

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.

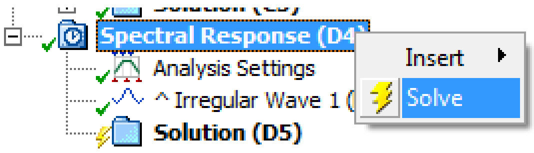
Details

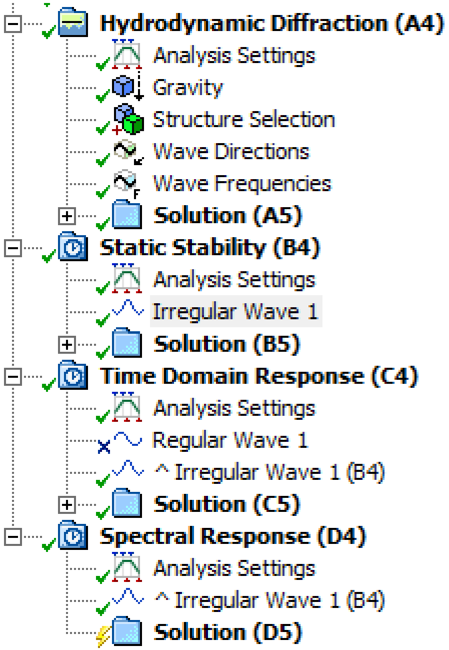
Details of Analysis Settings	
Name	Analysis Settings
Computation Type	Frequency Statistical Analysis
Parallel Processing	Program Controlled
Use Cable Dynamics	Yes
Frequency Response Specific Options	
Analysis Type	Wave and Drift Frequencies
Direction of Output for RAOs	Program Controlled
Spectrum Sub-Direction of Output for RAOs	Not Available
Axis System for Significant Motions and No...	Local Structure Axes
Starting Position	Determined by Upstream System
Common Analysis Options	
Calculate RAOs with Mooring Lines	No
Apply Drift Force with Multi-Directional Wav...	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

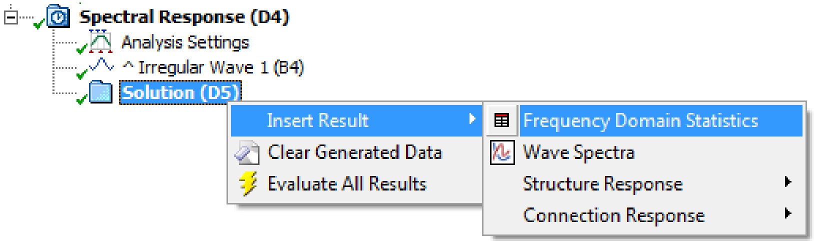
RMB click on Solution (D5) and Insert Result > Frequency Domain Statistics

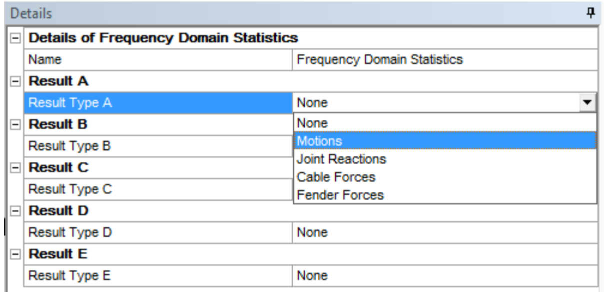
Specify the required results:

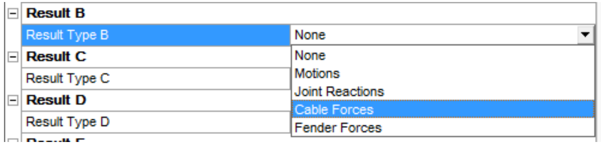
Motions

Cable Forces for cables 1 and 4

Results are automatically updated







Frequency Domain Results

Frequency Domain Statistic Results

Part Name: ShipHull

Motions:	*at Center of Gravity, in Local Structure Axes*											
Position	X:	0.60373 m	Y:	17.02285 m	Z:	1.30412 m	RX:	0.59084 °	RY:	0.08046 °	RZ:	0.72341 °
Velocity	X:	0.01141 m/s	Y:	0.50304 m/s	Z:	0.71186 m/s	RX:	0.23654 °/s	RY:	0.06089 °/s	RZ:	0.01459 °/s
Acceleration	X:	0.00597 m/s²	Y:	0.26426 m/s²	Z:	0.43230 m/s²	RX:	0.17819 °/s²	RY:	0.04804 °/s²	RZ:	0.00414 °/s²

Cable Tensions:

Cable 1	Fairlead 1 (ShipHull):	80688.65625 N	Anchor 1 (Fixed):	63596.10547 N
Cable 4	Fairlead 4 (ShipHull):	97212.17188 N	Anchor 4 (Fixed):	80769.79688 N