MIKE 21 Maritime

Mooring Analysis

Step-by-step training guide

DHI A/S
Agern Allé 5
DK-2970 Hørsholm
Denmark

+45 4516 9200 Telephone
mike@dhigroup.com
www.mikepoweredbydhi.com

Company Registration No.: DK36466871

CONTENTS

MIKE 21 Maritime
Mooring Analysis
Step-by-step training guide

1 MIKE 21 Mooring Analysis Step-by-Step Training Guide

This training guide includes the following four step by step examples:

The first example is a simple domain-independent example relying on 0D (time series) wave forcing used on a vessel in a user defined local domain, which on a basic level showcases the combined usage of Frequency Response Calculator and MIKE 21 Mooring Analysis (MA) in order to perform a simple mooring analysis.

The second example is more advanced and demonstrates the powerful combined workflow between MIKE 21 HD, Frequency Response Calculator and MIKE 21 MA in order to calculate the moored vessel motions of a tanker induced by a passing (transiting) vessel within the Port of Brisbane navigation channel. This example provides guidance on the entire simulation, from setting up and running the hydrodynamic model, to creating a moving pressure field and running a dynamic mooring analysis.

The third example details how to set up MIKE 21 MA for multibody simulations. Two bulk carriers, moored side by side, are subject to waves.

The fourth example demonstrates how to set up a Trelleborg AutoMoor automatic mooring unit in a MIKE 21 Mooring Analysis simulation through text edits to the *.m21ma input file.

The fifth example demonstrates how to apply a Trelleborg DynaMoor automatic mooring unit in a MIKE 21 Mooring Analysis simulation through text edits to the *.m21ma input file.

In addition, another example is provided in the example folder. The Simple Port folder contains the setups to show additional ways for including input data, e.g. from MIKE 21 Boussinesq Wave (BW) and MIKE 3 Wave FM. The description of this is included in the User Guide.

Note: You need to install the example files into a folder to which you have write access.

Note: If you have executed previous models from MIKE Zero project mode, where the output data is going to be used as input data in a subsequent MIKE 21 Mooring Analysis execution, please be aware that all the result files will be gathered in a common Result folder next to the (.mzp) file.

This means that if you are running the examples on basis of MZ project mode, then the paths to these files needs to be updated, as the default state of the examples are based on non-project mode, where all output data is stored in a Result Files folder next to each individual Setup file.

2 Example: 0D Wave Forcing on a Vessel

The step-by-step example in this Chapter demonstrates how to set up a MIKE 21 Mooring Analysis simulation, including usage of the Frequency Response Calculator, running the model and doing a preliminary analysis of the results. The example calculates vessel motions in 6-DoF (Six Degrees of Freedom) of a moored tanker subjected to a time series of incident wave conditions (0D wave forcing).

2.1 Step 1 – Launch the Frequency Response Calculator

The first step in any mooring analysis is to create a Frequency Response Calculator (FRC) setup, in order to obtain the frequency response data for the vessel in question. From the MIKE Zero page the (.fresponse) editor is launched by clicking:

'File' → 'New' → 'File…'

The (.fresponse) editor can be found under the Maritime subgroup of MIKE 21.
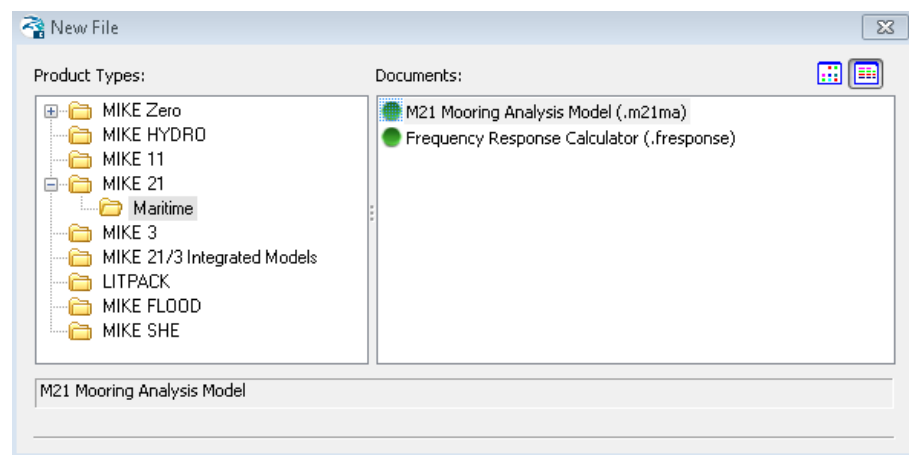
Figure 2-1   Location of Frequency Response Calculator setup editor

2.2 Step 2 - Create a Frequency Response Calculator setup

The set of steps below outlines how to create a new FRC setup.

1. Create a new instance of the (.fresponse) editor as shown in Figure 2-1 by clicking **OK**. The FRC main page (Figure 2-2) should appear on the screen.
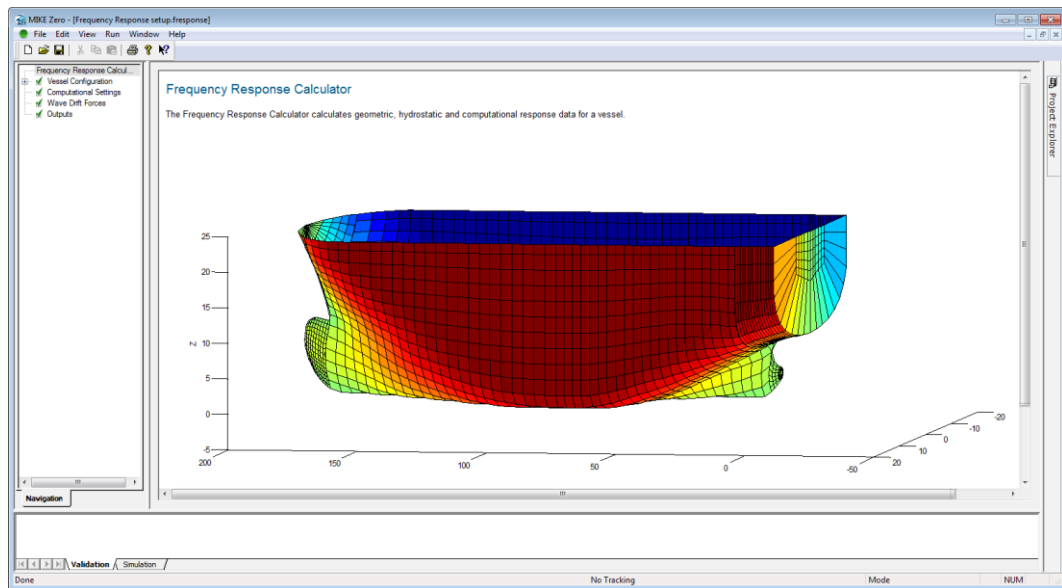
Figure 2-2   FRC main page

2. Save the file: Select **'File' → 'Save'** and save the filename in the root folder:
    (<example installation folder>\MIKE Zero
    Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Setups**Frequency
    Response.fresponse**).

2.3   Step 3 - Populate the FRC Setup with Data

Below all the sequential steps for all FRC GUI dialogs are described in descending order:
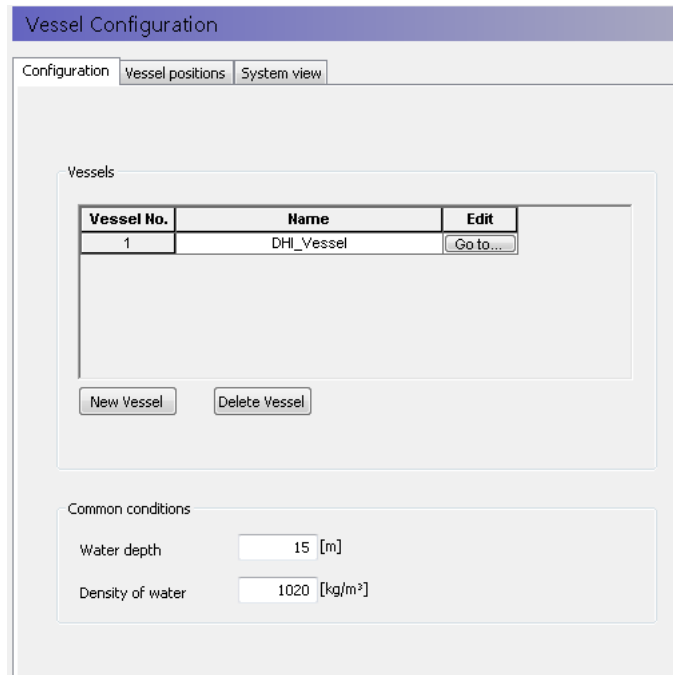
1. Dialog: Vessel Configuration
    This is the first dialog to consider when setting up FRC.

    Under the '**Configuration**' tab, select '**New Vessel**' and provide the vessel the name
    'DHI_Vessel'. Under common conditions use the following values: Set **Water depth**
    to 15 m and **Density of water** to 1020 kg/m³.

    Under the '**Vessel positions**' tab, select **Amidship in domain**. Leave the default
    values for the vessel positions at 0 m for both the x- and y- axis.

Figure 2-3   Vessel configuration dialog

> Finally, clicking on the '**Go to …**' button under the '**Configuration**' tab will take you to the **Vessel Instance** dialog 'DHI_Vessel'.

2. Dialog: Vessel Instance
    In this dialog you are to specify the vessel grid representing the 3D spatial vessel hull and the fundamental conditions which apply.

    This dialog is divided into three tabs: **Vessel Characteristics**, where the fundamental data for the vessel instance are to be provided, **Freedom of motion** where users are to define whether the vessel (or body) is freely floating or locked in motion (e.g. port structures) and **Vessel View**, where the deck plane and waterline contours can be inspected spatially in an XY coordinate system encapsulating the vessel.

    On the **Vessel Characteristics tab** start loading in the vessel grid:
    Leave the delimiter to be Space, and select the following vessel grid file:
    < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\VesselGrid**Tanker.grd**

    When the vessel grid is loaded, the remaining parameters on the **Vessel Characteristics** will be automatically updated with default values derived from the vessel hull itself.

    In this example, the values shown in Table 2.1 should be applied:

Table 2.1   Moored vessel parameters

Vessel Scaling (X-Dim)	1
Vessel Scaling (Y-Dim)	1
Vessel Scaling (Z-Dim)	1
r_{xx}	8.95 [m]
r_{yy}	55.95 [m]
r_{zz}	55.95 [m]
Vertical Center of Gravity (ZG)	10.8 [m]
Draft	9.8 [m]
Deck plane height	17.8 [m]

By setting these parameters as such, the FRC editor will automatically calculate the **Vessel Statistics** (non-editable) under the given conditions as a service for the user. These values will be:

Under the **Freedom of Motion** tab, select freely floating for the freedom of motion.

Finally, you can inspect the relative position of the waterline and deck plane contours by switching to the **Vessel View tab**, cf. Figure 2-4. The deck plane height can be changed using the slider or a specific value in meters can be provided (distance from the keel). By activating the check box "Display isolines", the user obtains the view on the vertical isolines to get an improved impression of the vessel hull shape.
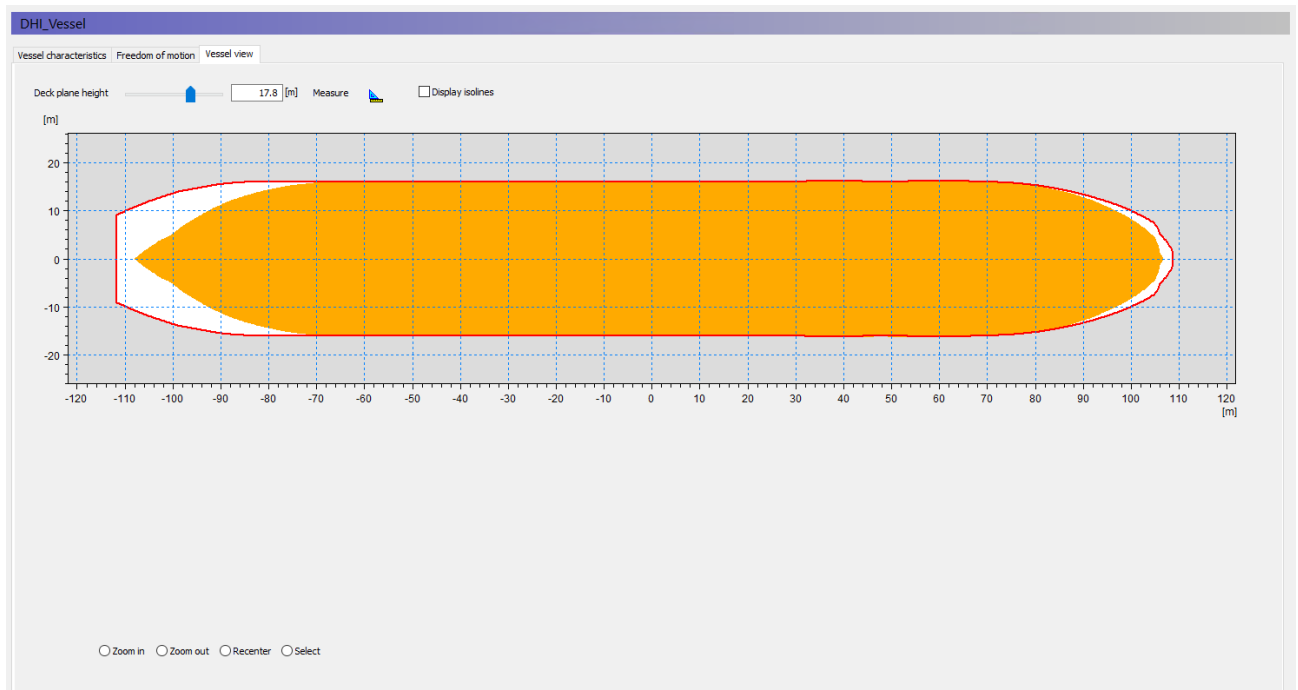
Figure 2-4   The Vessel View tab

3. Dialog: Computational Settings
In this dialog, leave the settings as default: **Simulation mode** = M21 Mooring Analysis compatible mode, and **Number of frequencies to solve** = 256.

Figure 2-5   Computational Settings dialog

4. Dialog: Wave Drift Forces
Do not include 2nd order Wave drift forces.
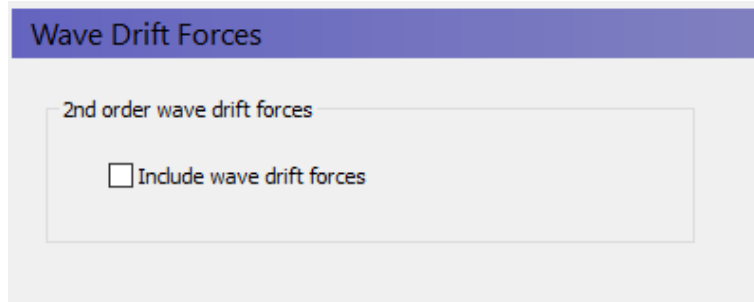
Figure 2-6   Wave Drift Forces dialog

5.   <u>Dialog: Outputs</u>
      All output data and file names are pre-populated with default settings. Do not change
      any settings here. The **Vessel response** output file will be the basis for all
      subsequent analysis.
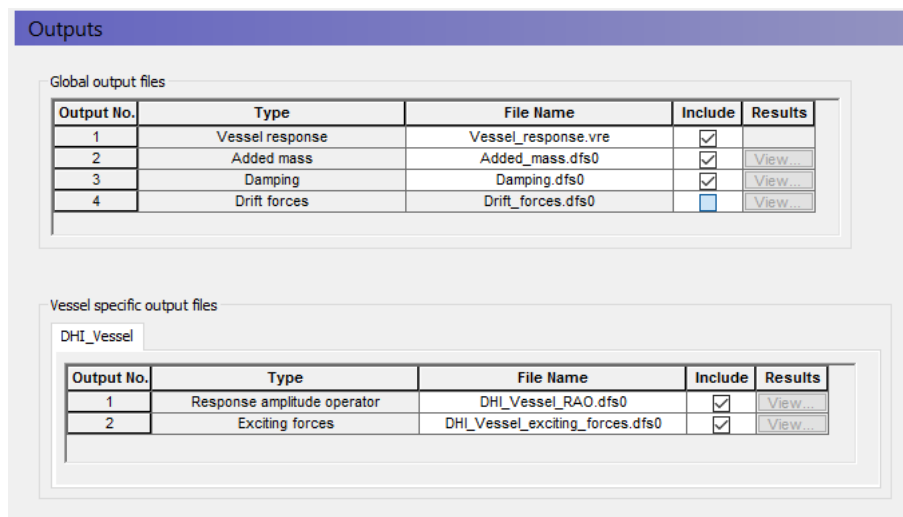
Figure 2-7   Outputs dialog

2.4   Step 4 – Run the FRC Simulation

Save the (.fresponse) editor and launch the FRC simulation.
The **Launch settings** pop-up is launched by clicking **Run → Start Simulation …**

Prior to the FRC execution, you can on the **Parallelization** tab in the **Launch settings**
pop-up window control how many cores the FRC simulation should utilise (among the
number of cores available on the hardware). Use the maximum number of cores (default).

Then click **OK**, and the FRC simulation will begin.

Once the simulation is completed (takes approx. 1 hours on a 4 core PC) please verify
that the following output file exists:

< example installation folder >\\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Setups\Frequency
Response - 0D wave forcing.fresponse - Result Files**Vessel_Response.vre**

2.5 Step 5 – Launch the MIKE 21 Mooring Analysis

Similar to the Frequency Response Calculator (.fresponse) editor, the MIKE 21 Mooring Analysis (.m21ma) editor is also found under the Maritime subgroup of MIKE 21.
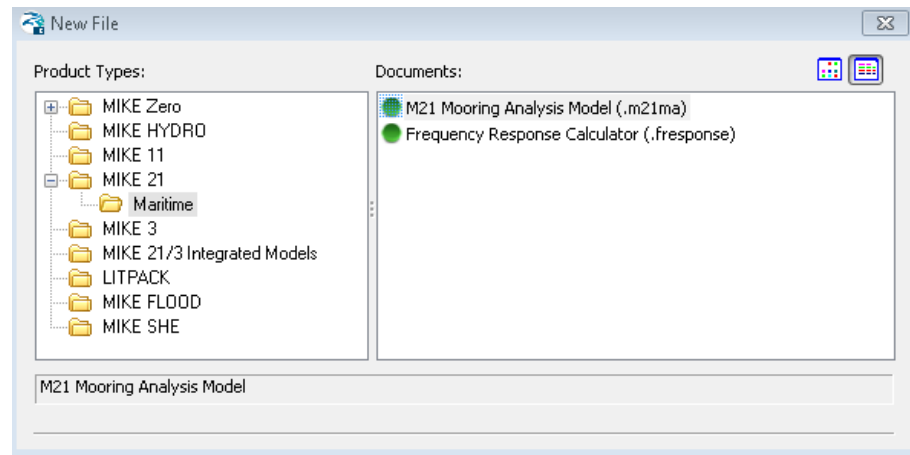
Figure 2-8   Location of MIKE 21 Mooring Analysis setup editor

2.6 Step 6 - Create a MIKE 21 Mooring Analysis Setup

The set of steps below outline how to create a new MIKE 21 Mooring Analysis (M21 MA) setup.

1. Open a new instance of the (.m21ma) editor as shown in Figure 2-8 by clicking **OK**. The M21 MA main page (Figure 2-9) should appear on the screen.
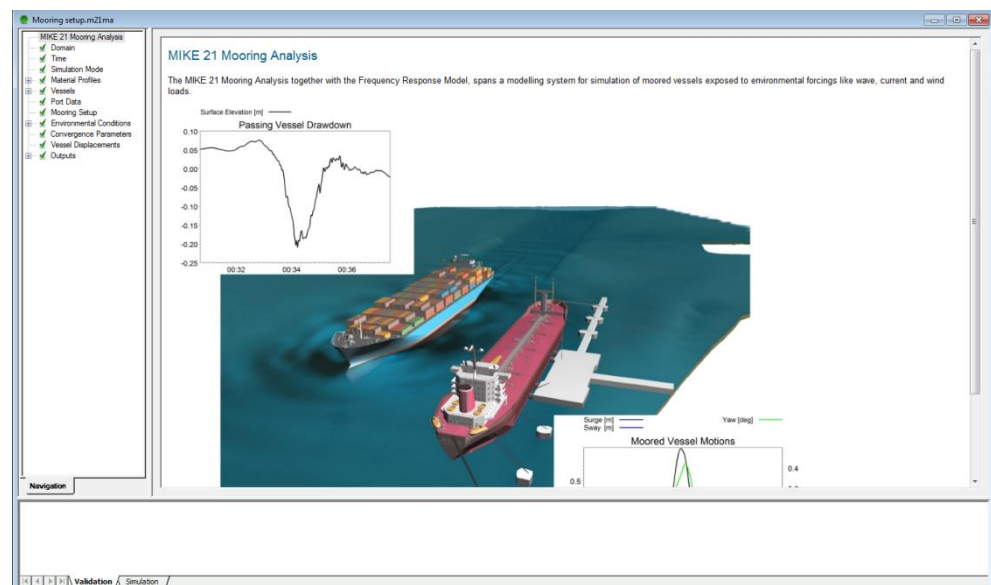
Figure 2-9   M21 MA main page

2. Start saving the file: Select '**File**' → '**Save**' and save the filename in the root folder: (<example installation folder>\MIKE Zero

Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Setups**Mooring setup.m21ma**).

2.7   Step 7 - Populate the MIKE 21 Mooring Analysis setup with data

Below all the sequential steps for all required M21 MA GUI dialogs are described in descending order:

1. Dialog: Domain (and creation of a Domain file)
    The first step in creation of a M21 MA model begins with selecting a **Domain file**.

    Precise domain files displaying the exact positions of berths and breakwater structures are essential when utilizing 2D environmental forcing in a mooring study. However, for this example we will utilise 0D (time-series) wave data only. We will produce a local **Domain file** with sufficient spatial extent to encapsulate the full mooring system.

    For illustrative purposes we have included bathymetry values to the domain file, even though they are not used in the calculations for 0D wave data.

    Create a domain file that can contain the extent of the vessel and mooring system by following these steps:

    a.   **File → New File → Grid Series**, and click **OK**
    b.   Select 'Blank Grid', and click OK
    c.   Select Type of file = 2 dimensional grid, and click Next
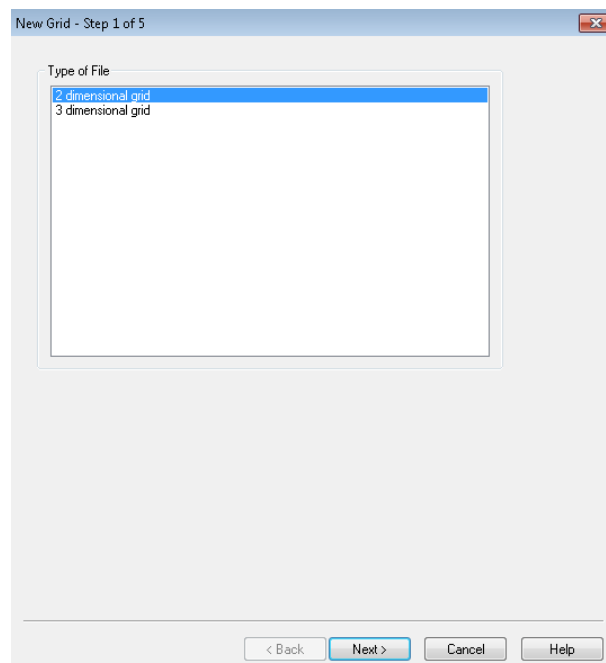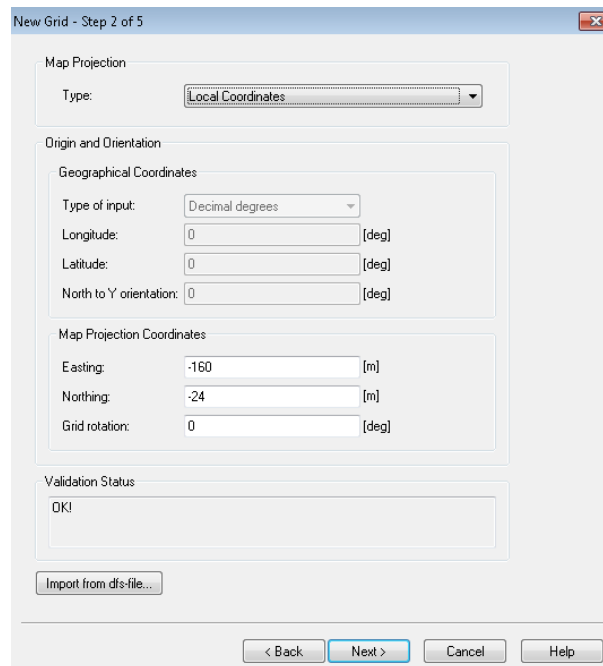
Figure 2-10   New Grid dialog – Step 1 of 5

    d.   Select Map projection: Type = Local Coordinates, and set Easting to -160 [m], Northing to -24 [m] and Grid Rotation to 0 [deg], and click Next
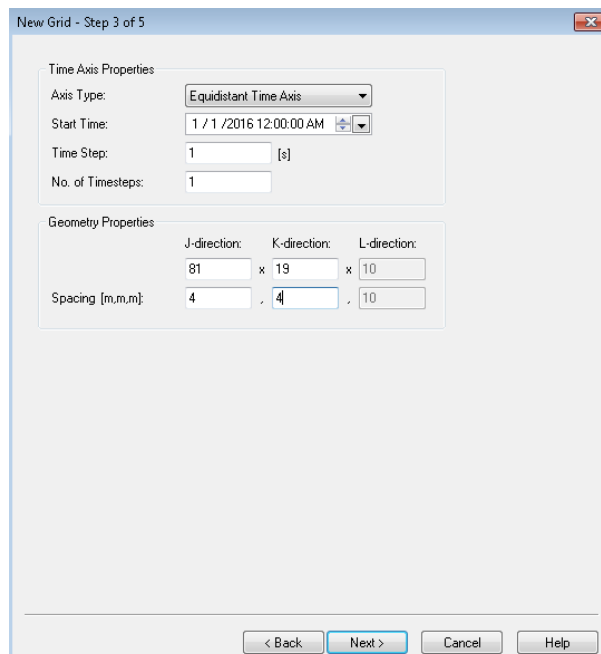
Figure 2-11   New Grid dialog – Step 2 of 5

e.   Set: Axis Type to Equidistant Time Axis, Start time to 01/01/2016 00:00:00, Time Step to 1 [s], No. of Timesteps to 1, Number of grid points (J-dir) to 81, Number of grid points (K-dir) to 19, Spacing (J-dir) to 4 [m], Spacing (K-dir) to 4 [m], and click Next

Figure 2-12   New Grid dialog – Step 3 of 5

f.   With respect to item record 1: type name = 'Bathymetry', Select Type = Bathymetry, set the Land value to 2 [m] and click Next
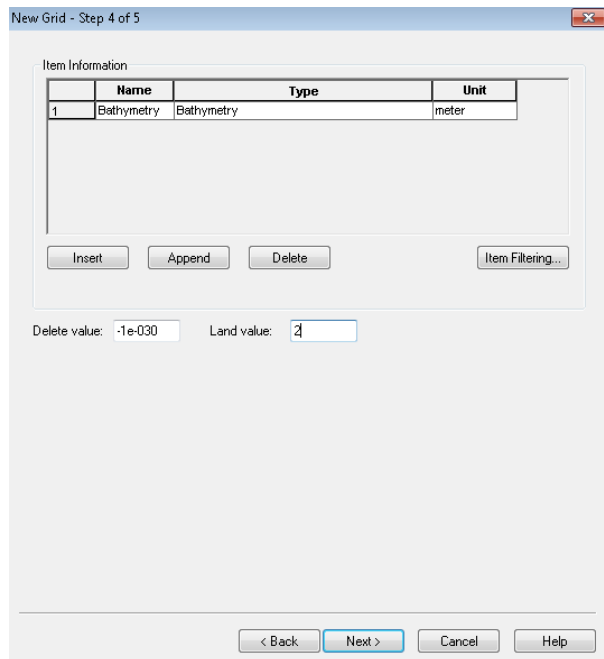
Figure 2-13   New Grid dialog – Step 4 of 5

g.   Click Finish
h.   Add grid values to the file: Set all values in row 0-9 to -15 [m], row 10-17 to 2 [m] and row 18 to 5 [m].

Save the file as: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Domain**Domain.dfs2**

Finally, select this Domain file in the Domain dialog as shown in Figure 2-14:
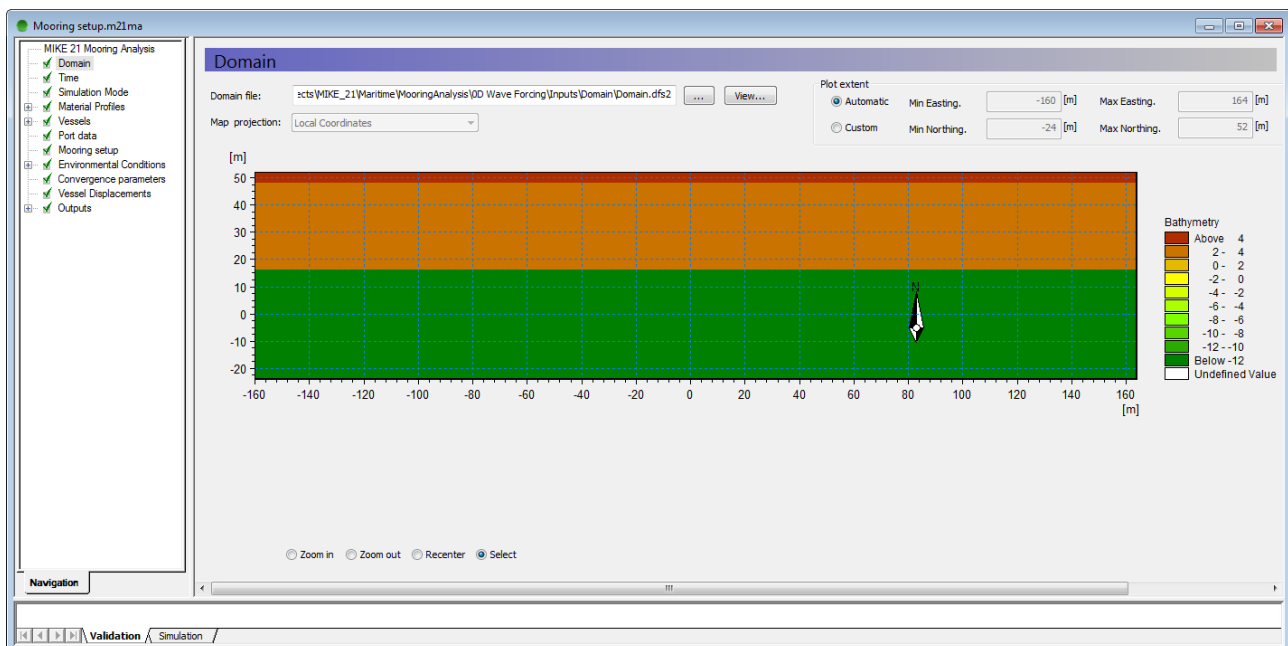
Figure 2-14   Selection of Domain file

2. <u>Dialog: Time</u>
Here you specify the temporal conditions applying for the time-domain computation (simulation mode = final computation mode).

Make the following:
a. Set the No. of timesteps to 500
b. Set the timestep interval to 0.25 [s]
c. Set the simulation start date to 01-01-2016 00:00:00
d. Set the No. of warm up time steps to 0
e. Set the No. of warm down time steps to 0

3. <u>Dialog: Simulation mode</u>
Set this to be Convergence mode.

Using convergence mode means that you prior to running the final time-domain simulation first will attempt to find the initial vessel displacement leading to a uniform tension distribution among all mooring lines.

Subsequently, determining a proper initial vessel displacement is an important pre-processing step to ensure a subsequent stable time-domain analysis.

4. <u>Dialog: Material Profiles</u>
Expand the node and go to Line profiles.

5. <u>Dialog: Line profiles</u>
Insert two Line profiles. Denote the first line 'Nylon_rope' and the second line 'Steel_wire'.

6. <u>Dialogs: Line profile instances</u>
On the **Nylon_rope** instance specify the following:
a. Total breaking strength = 74 [t]
b. Failure load = 45 [%]
c. Linear damping coefficient = 0 [kg/s]
d. Quadratic damping coefficient = 0 [kg/m]
Select this line profile: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Material Profiles**Polyamide_Double-Braided.dfs0**

On the **Steel_wire** instance specify the following:
a. Total breaking strength = 121 [t]
b. Failure load = 55 [%]
c. Linear damping coefficient = 0 [kg/s]
d. Quadratic damping coefficient = 0 [kg/m]
Select this line profile: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Material Profiles**SteelLine42mm.dfs0**

After this go to the Fender Profiles dialog.

7. <u>Dialog: Fender profiles</u>
Insert one Fender profile. Call it 'SCN1300'

8. <u>Dialog: Fender profile instance</u>
Specify this for the 'SCN1300' instance:
a. Max reaction force = 1357 [kN]
b. Max deflection = 1.3 [m]

    c.   Failure deflection = 75 [%]
    d.   Fender friction coefficient = 0.3
    e.   Linear damping coefficient = 0 [kg/s]
    f.   Fender thickness = 1.495 [m]

Select this fender profile: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Material Profiles**SuperCone.dfs0**

9.   Dialog: Bollard Profiles
    Specify on this instance:
    a.   Max. horizontal bollard pull force = 2000 [kN]
    b.   Max. vertical bollard pull force = 1000 [kN]
    c.   Max. total pull force = 2500 [kN]
    d.   Max. vertical line angle = 60 [deg]
    e.   Safety threshold = 75 [%]

10.   Dialog: AutoMoor profile
    We will skip usage of AutoMoor for now in this setup, so no AutoMoor profiles will be specified.
    A modified model setup including AutoMoor is described in section 5.

11.   Dialog: DynaMoor profile
    We will skip usage of DynaMoor for now in this setup, so no DynaMoor profiles will be specified.
    A modified model setup including DynaMoor is described in section 6.

12.   Dialog: Port_fairlead Profiles
    We will skip usage of DynaMoor for now in this setup, so no Port_Fairlead profiles will be specified.
    A modified model setup including DynaMoor is described in section 6.

13.   Dialog: Chain Profiles
    We will skip usage of chains in this setup, so no chains will be specified.

14.   Dialog: Vessels
    In this dialog you must select the vessel response file generated from the previously executed FRC setup:
    < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Setups\Frequency Response - 0D wave forcing.fresponse - Result Files**Vessel_Response.vre**

    Once the (.vre) file has been successfully loaded into M21 MA it will look like as shown in Figure 2-15.

    Clicking 'Go to …' on the vessel instance 'DHI_Vessel' will take you to the vessel instance dialog.
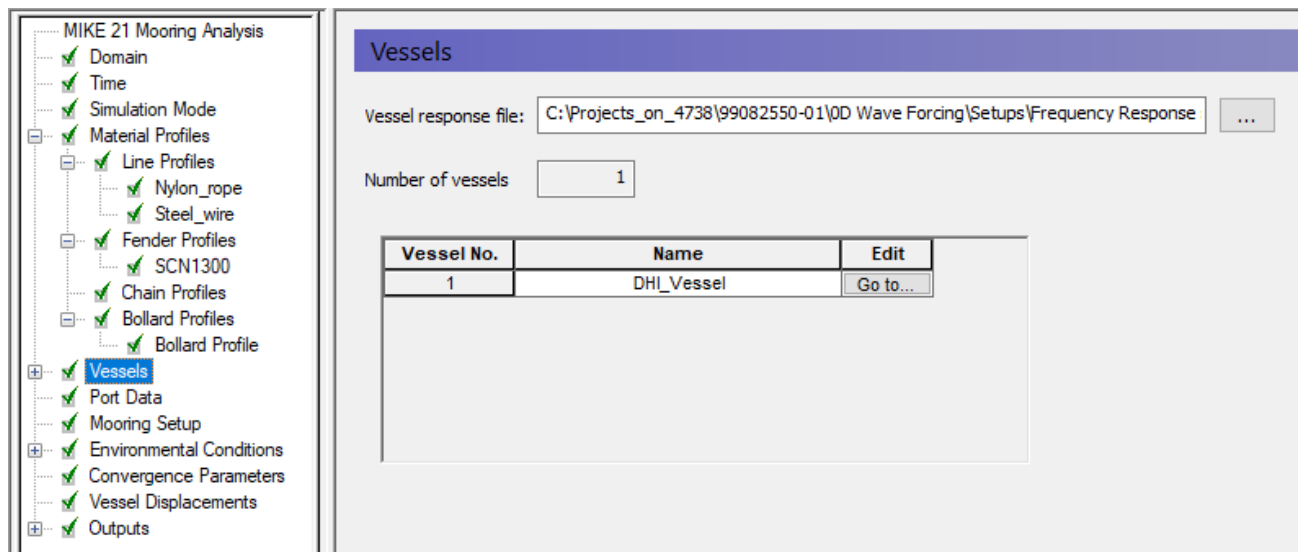
Figure 2-15   Vessels dialog after successful loading of (.vre) file

15. Dialog: Vessel instance
On the **Vessel data tab**, specify the data as shown in Table 2.2:

Table 2.2   Vessel data

Longitudinal Area (Wind)	2000 [m^2]
Transverse Area (Wind)	500 [m^2]
Lpp[1]	200 [m]
Loading Condition	90 [%]
Vessel Class	VLCC Conventional

[1] *Lpp refers to the length perpendicular of the vessel*

Apart from this you will also see a summary of the vertical attributes from the (.vre) file.

On the **Fairlead data tab**, load in data for the full set of fairlead positions (vessel fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Vessel Spatial Attributes**fairleads.xyz**

On the **Winch data tab**, load in data for the full set of winch positions (vessel fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Vessel Spatial Attributes**winches.xyz**

These values are as the ones shown in Table 2.3 and Table 2.4.The X and Y values are relative to amidship of the vessel and the Z values are relative to the keel.

Table 2.3   Fairlead positions relative to amidship at the vessel

Name	X [m]	Y [m]	Z [m]
Fairlead 1	110.455	-1	19.8
Fairlead 2	110.455	1	19.8
Fairlead 3	92.455	16	19.3
Fairlead 4	91.455	16	19.3
Fairlead 5	76.455	16	17.8
Fairlead 6	78.455	16	17.8
Fairlead 7	-83.545	16	17.8
Fairlead 8	-85.545	16	17.8
Fairlead 9	-96.545	14	19.3
Fairlead 10	-98.545	14	19.3
Fairlead 11	-111.545	1	19.8
Fairlead 12	-111.545	-1	19.8

Table 2.4   Winch positions relative to amidship at the vessel

Name	X [m]	Y [m]	Z [m]
Winch 1	101.455	-2	19.8
Winch 2	101.455	2	19.8
Winch 3	92.455	0	19.3
Winch 4	91.455	0	19.3
Winch 5	76.455	0	17.8
Winch 6	78.455	0	17.8
Winch 7	-83.545	0	17.8
Winch 8	-85.545	0	17.8
Winch 9	-96.545	5	19.3
Winch 10	-98.545	5	19.3
Winch 11	-101.545	1	19.8
Winch 12	-101.545	-1	19.8

On the **Vessel View tab**, you can see where the full set of fairleads and winches are positioned relative to the vessel. Leave the deck plane height at 17.8m. It should look as seen in Figure 2-16.
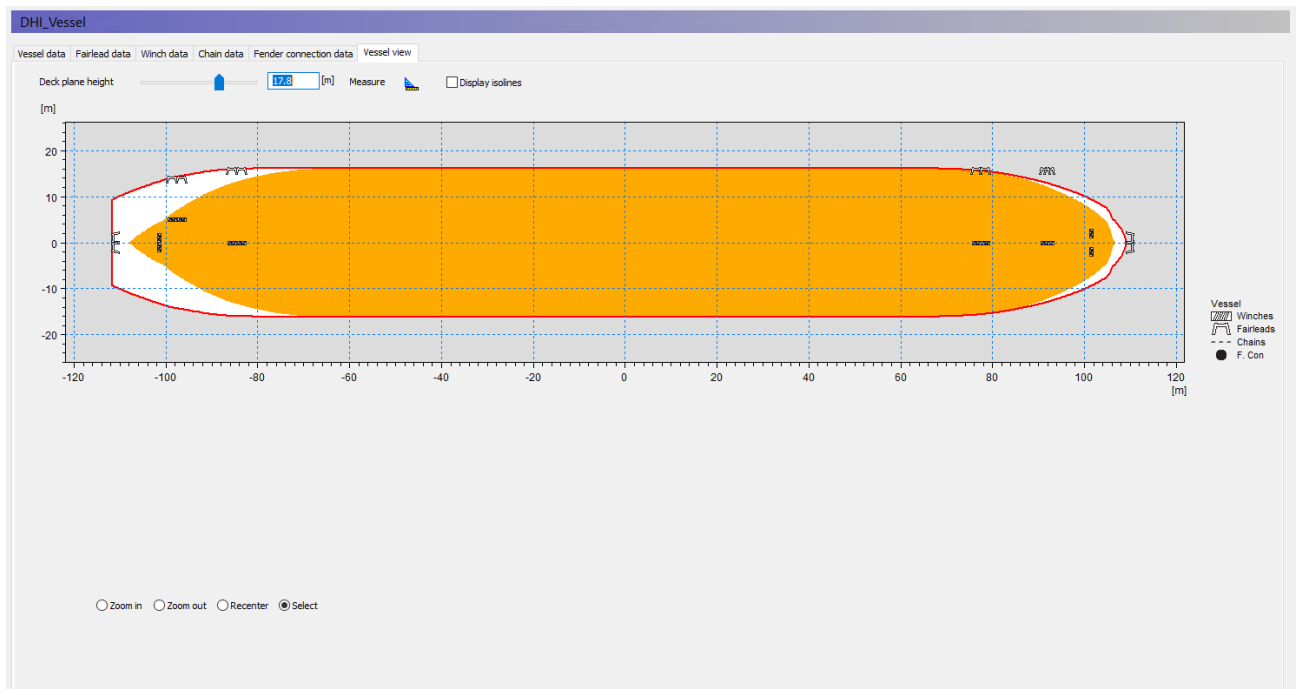
Figure 2-16   Vessel view after Fairlead and Winch positions have been specified

16.   Dialog: Port data

On the **Bollard data tab**, load in data for the full set of bollard positions (port/land fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Port Data**bollards.xyz**

On the **Fender data tab**, load in data for the full set of fender positions (port/land fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Port Data**fenders.xyz**

The bollard and fender positions in the custom made local dfs2 domain are as the ones shown in Table 2.5 and Table 2.6. The Z values are relative to the water surface.

Table 2.5   Bollard positions in domain

Name	Easting [m]	Northing [m]	Z [m]	Bollard profile
Bollard 1	151.4556	40	3	Bollard Profile
Bollard 2	121.4556	40	3	Bollard Profile

Bollard 3	91.4556	40	3	Bollard Profile
Bollard 4	46.4556	20	3	Bollard Profile
Bollard 5	-33.5444	20	3	Bollard Profile
Bollard 6	-78.5444	40	3	Bollard Profile
Bollard 7	-98.5444	40	3	Bollard Profile
Bollard 8	-138.5444	40	3	Bollard Profile

Table 2.6   Fender positions in domain

Fender Name	X [m]	Y [m]	Z [m]	Type
Fender 1	46.4556	16	1	SCN1300
Fender 2	-33.5444	16	1	SCN1300

There is only one material fender profile declared (SCN1300), so both fenders will have this profile associated. Set both fender force directions to be 270 [deg] (meaning: fender forces goes straight south (eg. straight against the vessel as we shall see). The fender force must point towards the vessel and its direction is defined as counter-clockwise positive from Easting axis (going to).

On the **Port View tab**, you can see where the full set of bollards and fenders are positioned in the domain. It should look as seen in Figure 2-17.
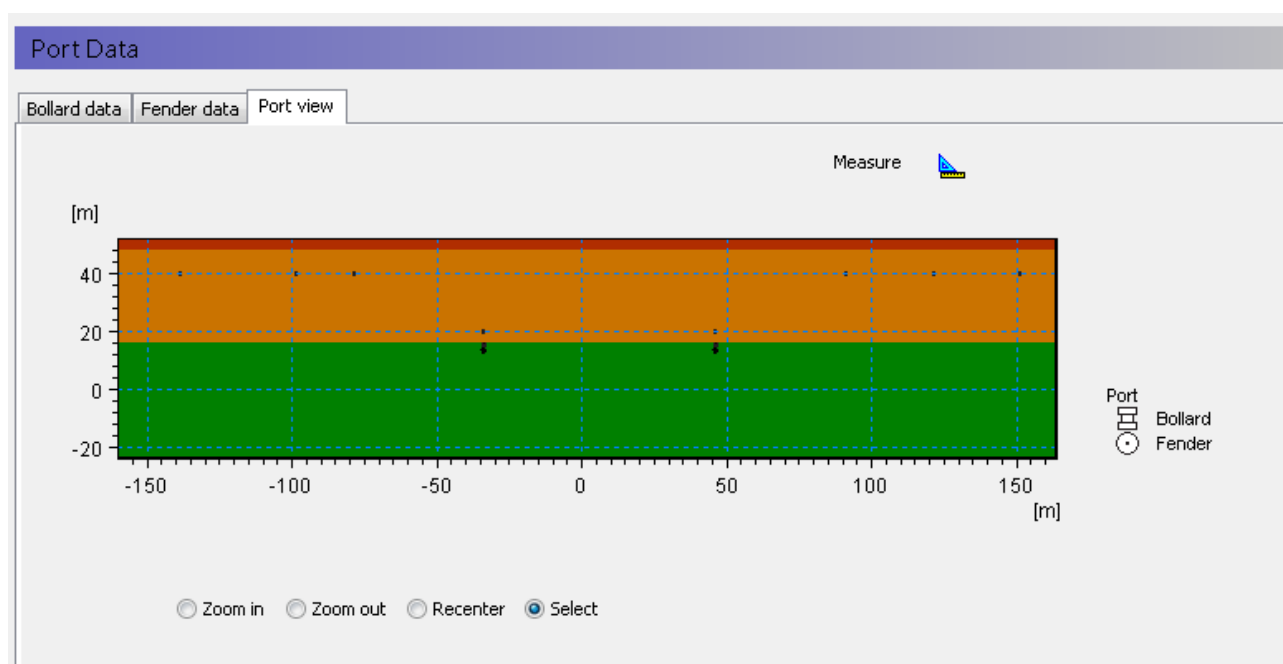
Figure 2-17   Bollard and Fender positions in the domain

17. Dialog: Mooring setup
There are two steps in defining the mooring setup and they should be performed in this order
- Position the vessel in the domain, reasonably
- Specify the mooring lines, by connecting (winch, fairlead, bollard) pairs

On the **Vessel position and mooring lines tab**, specify the vessel position to be at (easting, northing) = (0, -1.495) [m] and a vessel rotation of 0 [deg], and click Apply.

You will now see the vessel and its vessel-fixed attributes relative to the port-fixed attributes in a default extent as defined by the domain file, see Figure 2-18:
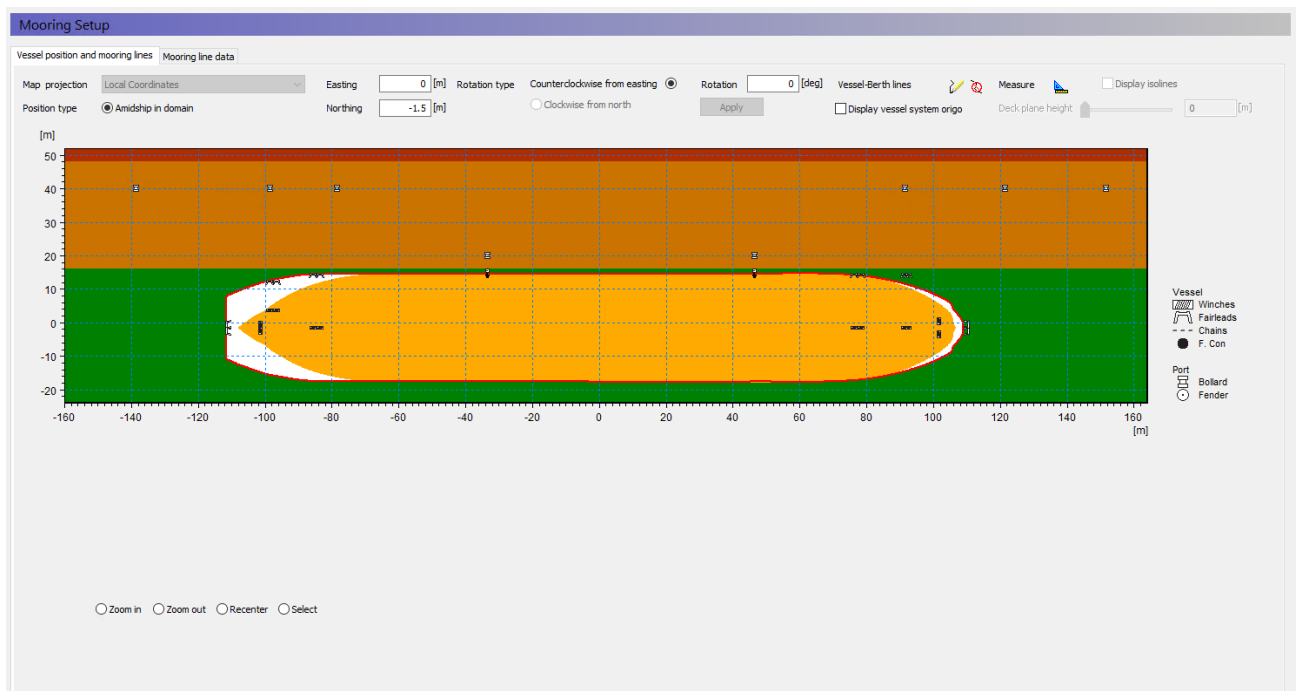
Figure 2-18   Vessel inserted into domain

Now that the vessel has been appropriately positioned, the next step is to define the mooring lines. The mooring system we would like to obtain is as given in Table 2.7 .

Each mooring line can be drawn/defined interactively by:
a. Visual identification of the relevant winch-fairlead-bollard pair, to span the mooring line

b. Initiate the mooring line insertion by: clicking the Vessel-Berth lines icon
c. Perform the mooring line insertion by: clicking the relevant winch icon, move the cursor to the relevant fairlead icon, clicking on the fairlead icon, continue the line to the bollard, and finally clicking on the bollard icon.

If you make an error by connecting some incorrect attributes, then delete the line

again by selecting the delete line icon   and subsequently click on the given line.

After insertion of all mooring lines, the Mooring setup should look like the one shown in Figure 2-19.
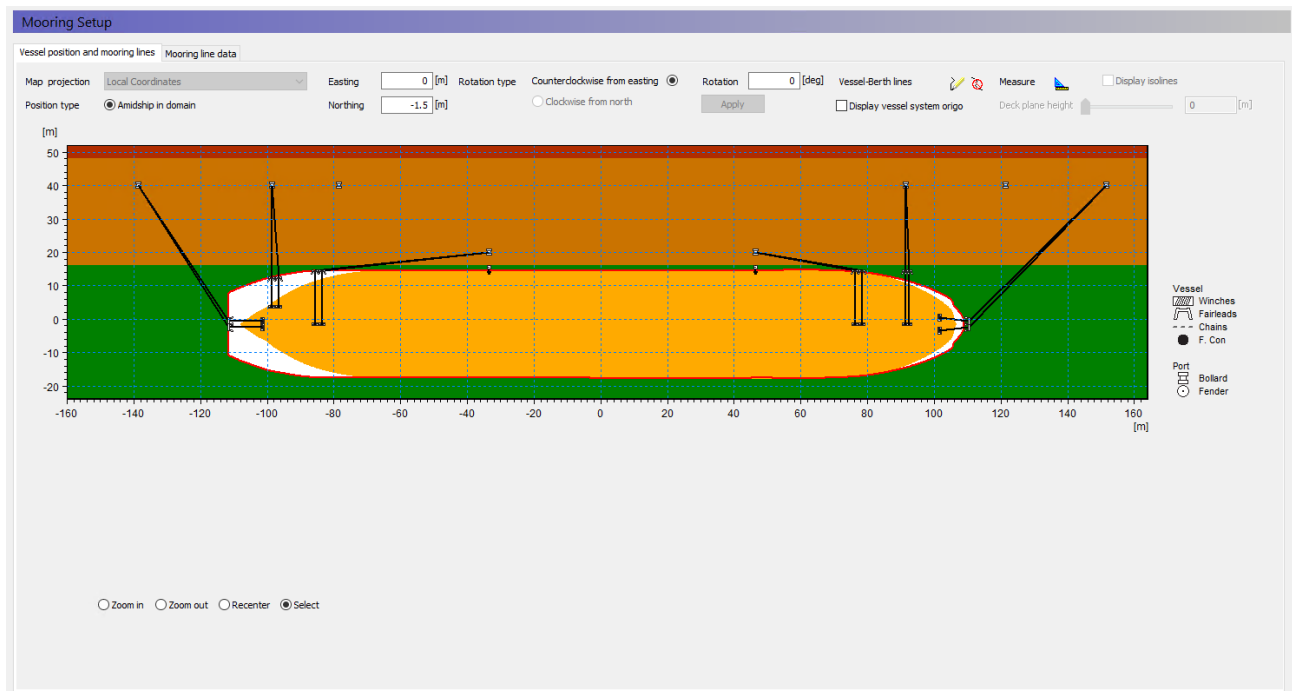
Figure 2-19   Vessel View after definition of all mooring lines

Table 2.7   The full Mooring Line definitions

Name	Bollard	Fairlead	Winch	Line profile	Line pretension (t)	Tail profile	Tail length (m)
Line 1	Bollard 1	Fairlead 1	Winch 1	Steel_wire	5	Nylon_rope	11
Line 2	Bollard 1	Fairlead 2	Winch 2	Steel_wire	5	Nylon_rope	11
Line 3	Bollard 3	Fairlead 3	Winch 3	Steel_wire	5	Nylon_rope	11
Line 4	Bollard 3	Fairlead 4	Winch 4	Steel_wire	5	Nylon_rope	11
Line 5	Bollard 4	Fairlead 5	Winch 5	Steel_wire	5	Nylon_rope	11
Line 6	Bollard 4	Fairlead 6	Winch 6	Steel_wire	5	Nylon_rope	11
Line 7	Bollard 5	Fairlead 7	Winch 7	Steel_wire	5	Nylon_rope	11
Line 8	Bollard 5	Fairlead 8	Winch 8	Steel_wire	5	Nylon_rope	11
Line 9	Bollard 7	Fairlead 9	Winch 9	Steel_wire	5	Nylon_rope	11
Line 10	Bollard 7	Fairlead 10	Winch 10	Steel_wire	5	Nylon_rope	11
Line 11	Bollard 8	Fairlead 11	Winch 11	Steel_wire	5	Nylon_rope	11
Line 12	Bollard 8	Fairlead 12	Winch 12	Steel_wire	5	Nylon_rope	11

To complete the mooring line definitions, go to the **Mooring line data tab** and set the Line profile, Line pretension, Tail profile and Tail length values for all the Mooring lines (Line 1 – Line 12) according to the values shown in Table 2.7.

18. Dialog: Physical Parameters
In this dialog, you should use the default values:
   - Density of water = 1020 [kg/m^3]
   - Density of air   = 1.28 [kg/m^3]
   - Viscosity (water) = 10^{-6} [m^2/s]
   - Viscosity (air)   = 1.48 • 10^{-5} [m^2/s]

19. Dialog: Drift Forces
Drift forces are not included.

20. Dialog: Current
Do not include a current field.

21. Dialog: Wind
Do not include a wind field.

22. Dialog: Convergence Parameters
In this dialog, you should specify the following values:
   - Max no. of iterations = 1000
   - Line convergence threshold = 0.1 %

23. Dialog: Vessel Displacements
Apply default parameters.

24. Dialog: Convergence Output
Apply the default output file names. All output files except 'External forces' and 'Chain forces' are included.

Ensure that all data in the (.m21ma) setup are saved.

2.8   Step 8 – Run the simulation in Convergence Mode and inspect results

To execute the setup in Convergence Mode, click:

Run → **Start simulation** … → Click **OK** in the Launch settings' pop-up window

As you will see, the simulation converges within the specified number of iterations.

Open the output file 'Convergence_line_pretension_deviation.dfs0'. As you can see, the percentage deviations between the line tensions and the line-pretension are all less than the specified line convergence threshold of 0.1 % for all lines, at the final iteration.

This means that the spatial vessel state (Surge, Sway, Heave, Roll, Pitch, Yaw) at the final iteration in file 'Convergence_DHI_Vessel_motions.dfs0' is a reasonable initial vessel displacement, within the chosen line convergence threshold of 0.1 %, to launch the final time-domain computation with.

With this initial vessel displacement known, we can move on with the final time-domain computation.

2.9 Step 9 – Create a Wave File

Before continuing with final time-domain simulation in M21 MA, we will have to create the time series wave file to be used as the primary forcing in this example.

1. Go to MIKE Zero and Click on '**New File**'.

2. Under '**MIKE 21'** select '**MIKE 21 Toolbox (.21t)**'.
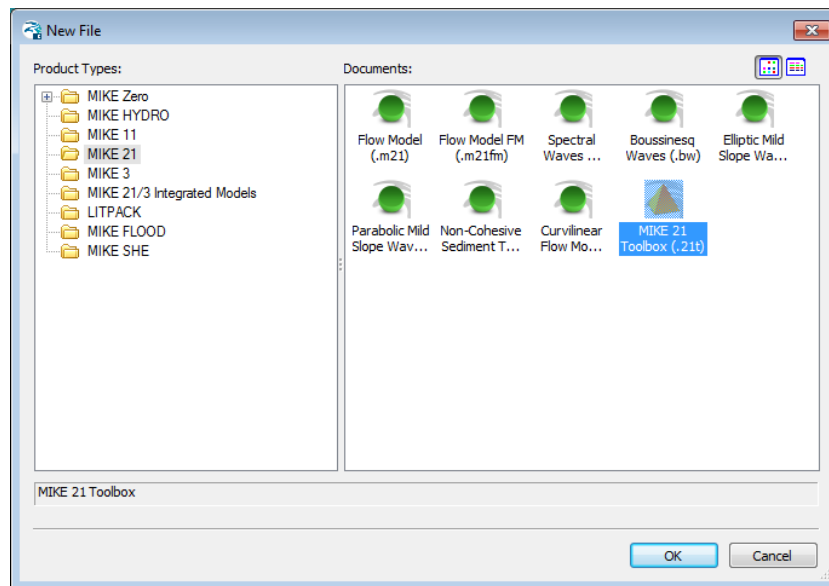
Figure 2-20   Location of MIKE 21 Toolbox

3. Expand the '**Waves**' Selection and double click on '**Random Wave Generation**'.
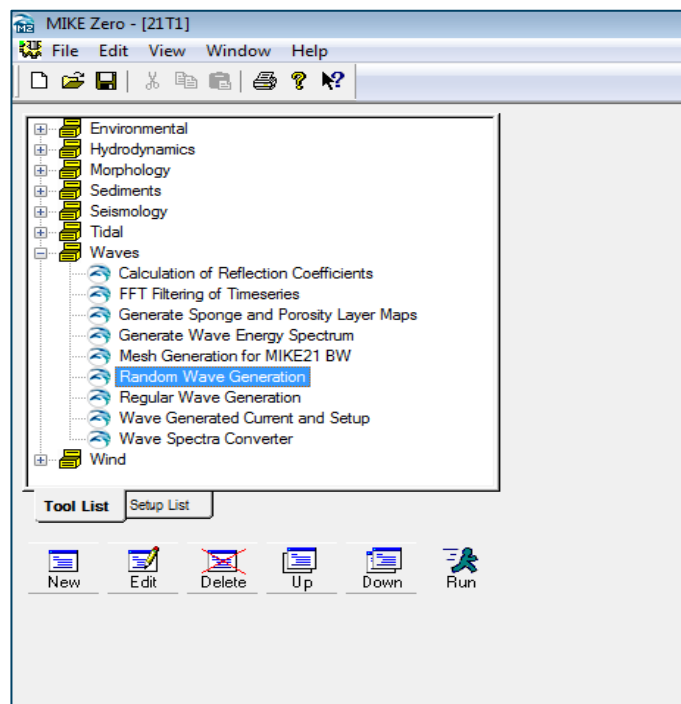
Figure 2-21   MIKE 21 Toolbox: Random Wave Generation

4.   Create a new Setup called 'M21 MA – 0D Wave'.

5.   Select the JONSWAP spectrum with the following parameters significant wave height (Hm0) 1m and peak wave period (Tp) 10 s.

6.   Set the water depth to 15 m.

7.   Set the smallest wave period in series to 1 s.

8.   Select 'Enable Second Order Correction'.

9.   Specify the Start date as: 01/01/2016 00:00:00 and 'Number of time steps' as 4500 and interval as 1 s.

10.   Save the file at the following location:
<example installation folder>\ MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing \Inputs\Metocean_timeseries**Hm0_1m_tp10s.dfs0**

11.   Click on 'Execute' to run.

12.   At successful completion, a pop up box saying 'Status: OK': will show up. Click 'OK' and 'Finish' and Save the setup as 'M21 MA -0D Wave.21t'.

13.   Open the .dfs0 that was created entitled Hm0_1m_tp10s.dfs0 and delete all the items except the Surface Elevation item.

14.   Create a zero warmup period of roughly 15 minutes:
    a.   Go to time steps 0 and 875 and set the surface elevation values to 0.
    b.   Delete all values between time steps from (and including) 1 – 874
    c.   Select all values from (and including) 0 and time step 875.
    d.   Click on 'Tools' →'Interpolation'. Click 'OK'

The final surface elevation time series are shown in Figure 2-23.
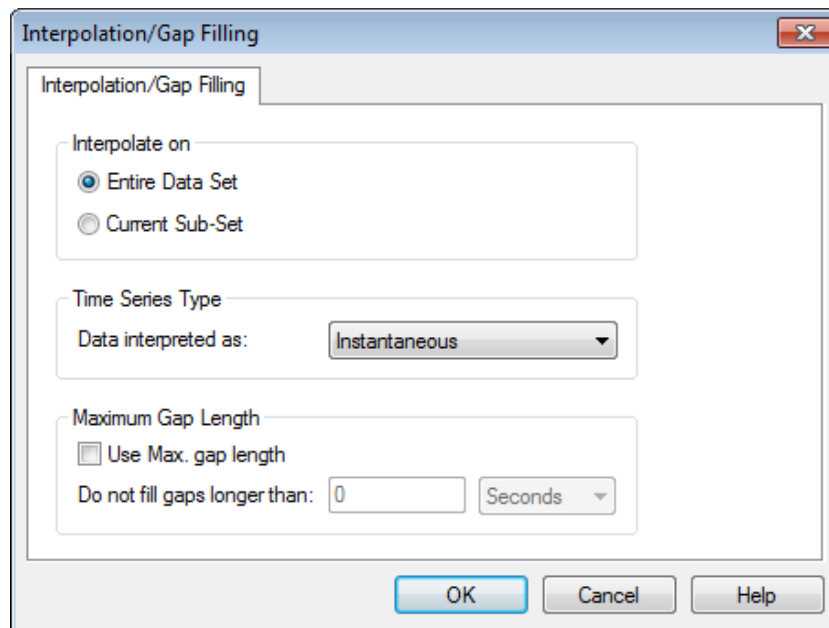
Figure 2-22   Interpolation feature from the time series (.dfs0) editor
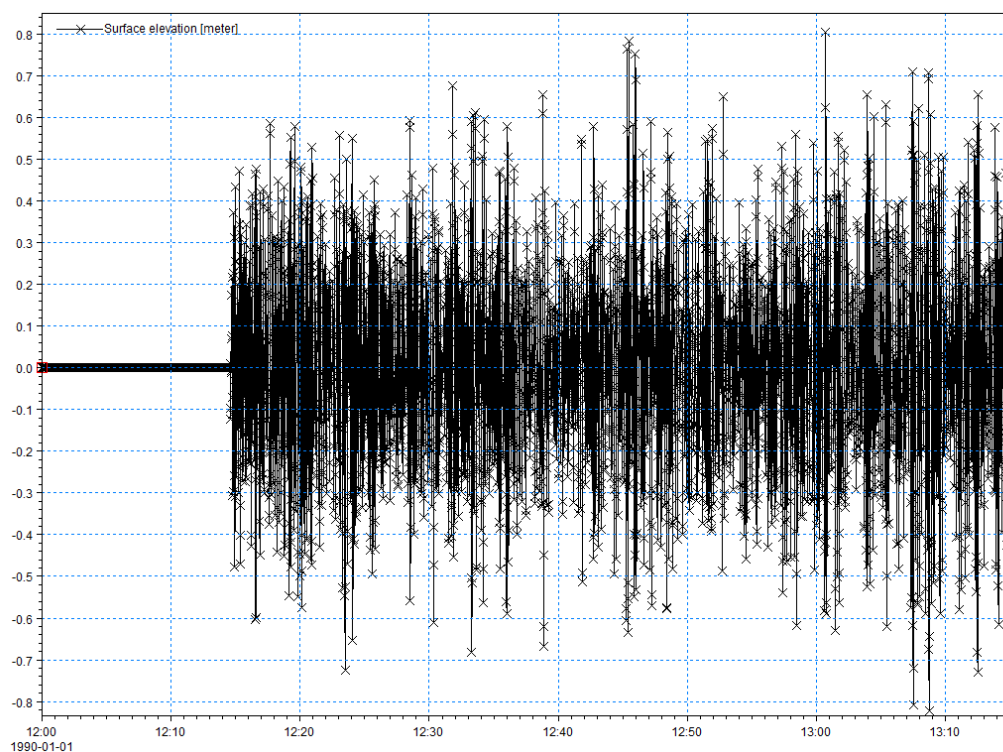
Figure 2-23   Water Surface Elevation time series with a zero warmup period

15. Next, insert a new wave direction item in front of the existing Surface Elevation item by following these steps:
    a.   Click Edit → Properties …

    b.   Mark the Surface Elevation item record by left-clicking on the record 1 field, so the whole record is highlighted (black)

    c.   Click Insert to open a new first item and give this new item the name 'Wave direction'

    d.   Select Type = Wave direction and set Unit to degree.

    e.   Click OK.

16. The wave direction in a time series wave file to be used in M21 MA must be provided constant per individual (wave direction, surface elevation) wave pair. Set the wave direction uniformly to 120 [deg] (eg. wave is coming from 120 [deg] True North) by following these steps:

    a.   Put a value of 120 [deg] at first time step (0) for the wave direction item

    b.   Put a value of 120 [deg] at last time step (4499) for the wave direction item

    c.   Highlight the whole wave direction column and click on 'Tools' →'Interpolation'. Click 'OK'

The wave direction is now uniformly set to 120 [deg] for the single wave pair (wave dir., surface elevation) used in this example.

2.10   Step 10 – Populate the M21 MA Setup with Final Execution Mode Data

1.   Dialog: Time
Here you specify the temporal conditions applying for the time-domain computation (simulation mode = final computation mode).

Make the following:
a.   Set the No. of timesteps to 17995
b.   Set the timestep interval to 0.25 [s]
c.   Set the simulation start date to 01-01-2016 00:00:00
d.   Set the No. of warm up time steps to 0

2.   Set the No. of warm down time steps to 0Dialog: Simulation Mode
In the setup 'Mooring setup .m21ma', go to the Simulation mode dialog and switch from Convergence Mode to Final Computation Mode. Check the number of timesteps in the Time instance as you might have used fewer timesteps in the convergence mode than required in the final computation mode.

This will open the Wave dialog, which is specific for this mode:

3.   Dialog: Wave
In this dialog you should do the following:
a.   Select Wave type = Wave field
b.   Uncheck 'Include warm up/down'
c.   Select Format = Varying in time, constant in domain
d.   Set Number of waves to 1
Set Power (N) to 12 (leading to FFT size = 4096)

Finally, select the wave file, which was created earlier:
<example installation folder>\ MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Inputs\Metocean_timeseries**Hm0_1m_tp10s.dfs0**

4.   Dialog: Vessel Displacements
In this dialog you should by now see an enabled Load button.

Click on the Load button. This will populate the 6 Initial Vessel displacement fields (Surge, Sway, Heave, Roll, Pitch, Yaw) with the data from the final iteration of the output file 'Convergence_DHI_Vessel_motions.dfs0' we previously produced from the convergence mode execution.

This action ensures the final time-domain computation starts from a vessel state where there is a sufficient state of tensional equilibrium in all mooring lines.

5.   Dialog: Final output
Leave the output file settings as default. All output files except 'External forces' and 'Chain forces' are included.

Set the vessel motion type to 'Relative to displaced position'.

2.11   Step 11 – Run the Final Time-domain Simulation and Inspect Results

To execute the (.m21ma) setup in Final Computation Mode, click:

Run → Start simulation … → Click OK in the Launch settings' pop-up window.

The simulation will be fast and produce the relevant output files for the time-domain analysis.

First thing to consider is the Operability Summaries for Lines and Fenders at the end of the simulation frame as shown in Figure 2-24. Note that you can find the information at the end of the (.log) file produced by the engine as well. For this simulation the critical bollard loads, line loads and critical compressions were never exceeded, indicating the mooring system dynamics is within the provided safety limits.

Figure 2-24   Operability Summaries for Bollards from the engines .log file. Scroll up for more information on Lines and Fenders.

The following steps will outline how to plot the results of three vessel motions (surge, sway and roll) acting on the vessel.

1. Go to the result files folder below:
    (<example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Setups**Mooring setup.m21ma - Result Files**).
    Open the file '**DHI_Vessel_motions.dfs0'**.

2. Create a new plot composer (.plc) file. Open a new MIKE Zero file. Click on 'New' → Plot Composer (.plc).

3. Create a new .plc file with three subplots,

4. Go to 'Plot' → 'Insert New Plot Object'. Select 'Multiple Plots Tiled' and set nx to 1 and ny to 3. Select 'Time Series Plot' and click OK.

5. For the first plot window: Mark the window, right click, click Properties…

6. Click on 'New Item', cf. Figure 2-25.
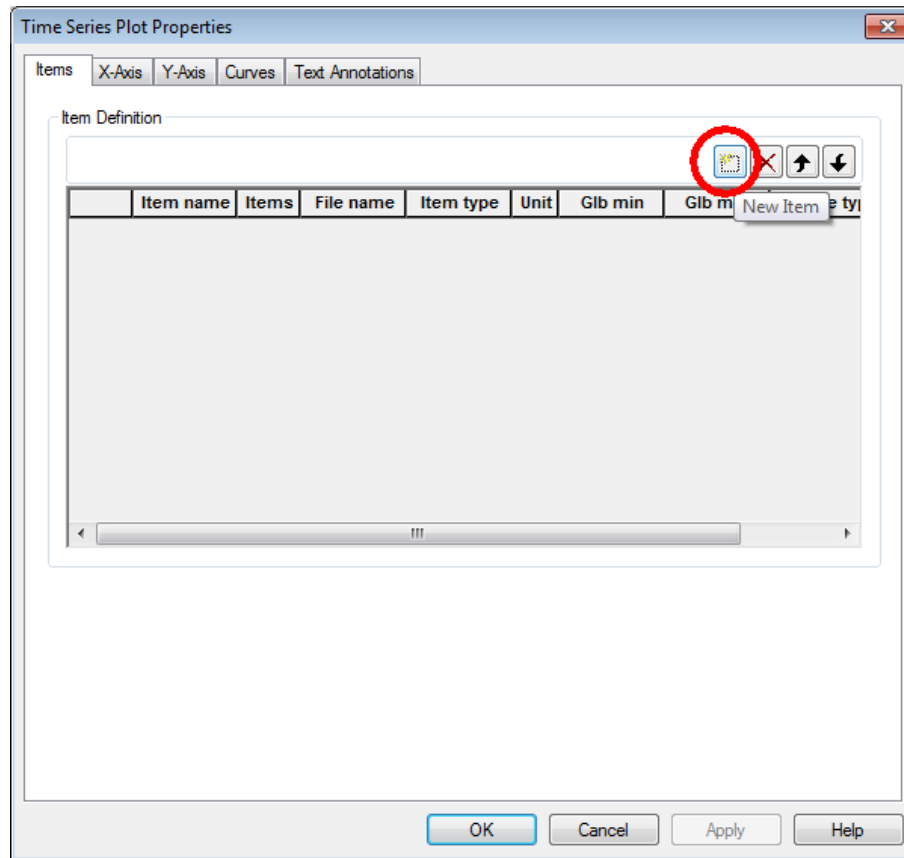
Figure 2-25   Add new time series to plc plot

7. Link to the time series file:
   <example installation folder>\MIKE Zero
   Projects\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing\Setups\Mooring
   setup - 0D wave forcing.m21ma - Result Files\ **DHI_Vessel_motions.dfs0'**

8. Select item 1 – 'Surge'

9. A plot of surge will appear in the first window.

10. Repeat the steps 6-9 for the remaining 2 windows and plot items: sway and roll (from
    output file: DHI_Vessel_motions.dfs0). The final plots are shown in Figure 2-26.
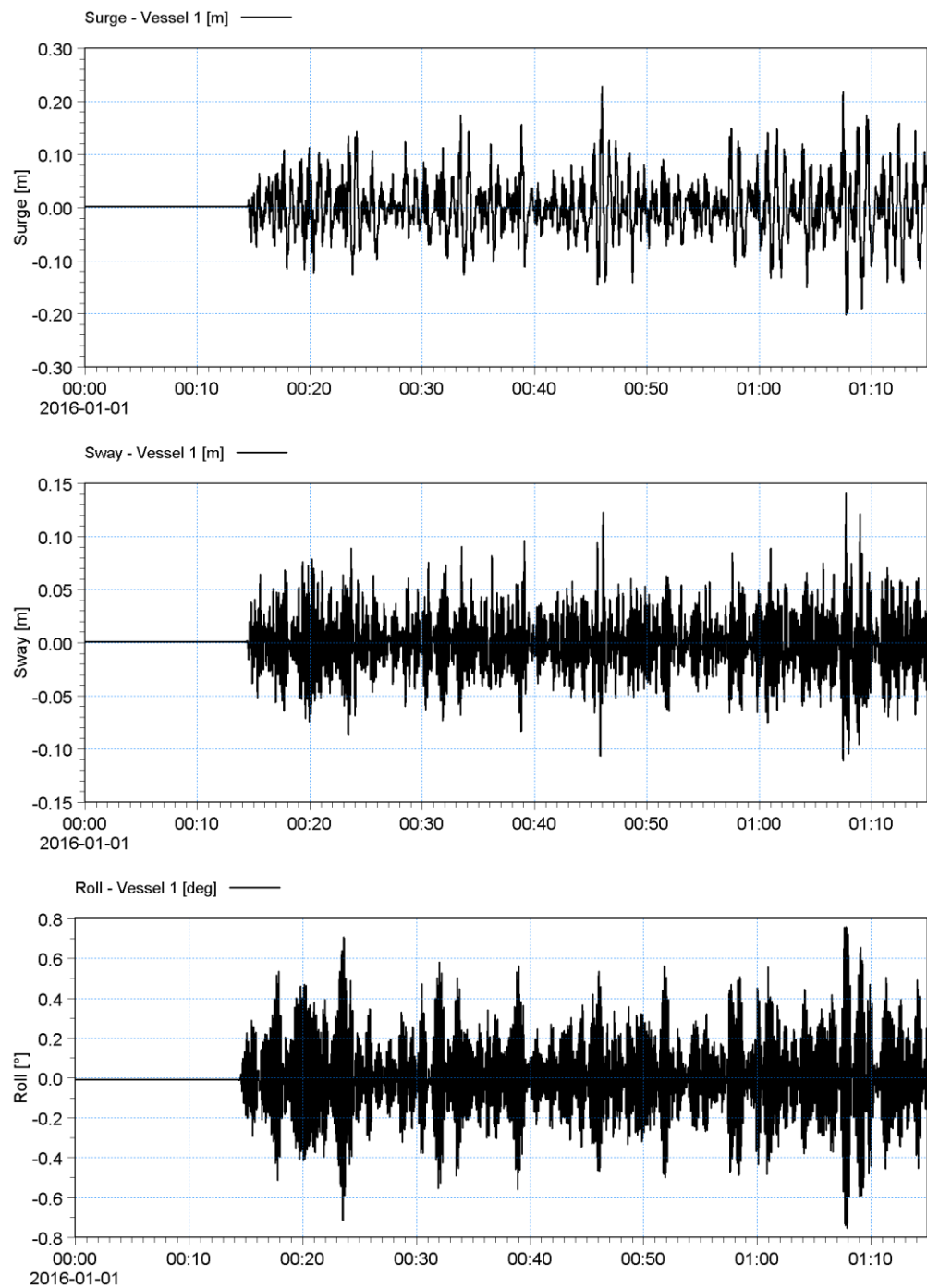
Figure 2-26   Time series of vessel motions

2.12   Step 12 – Visualise Vessel Movements in 3D

It is possible to visualise the vessel movements by using MIKE Animator Plus.
MIKE Animator Plus is a digital video production studio which enables you to turn e.g.
MIKE 21 model results into 3D video presentations.

The results from the MIKE 21 Mooring Analysis module include a time series file with
vessel motions. By using information from this together with a 3D solid representing the
vessel, and data files with bathymetry and time-varying water level it is possible to use
MIKE Animator Plus to create an informative animation of the vessel movements for the
given mooring situation.

The following sections describe how to setup MIKE Animator Plus to visualise the findings
from the MIKE 21 Mooring Analysis model.
Note that you can still visualise the data if you do not have a license for MIKE Animator
Plus, however in that case a DHI icon will be displayed in front in the graphics window.

All references to file locations in the following are given with reference to the top folder
.\MIKE_21\Maritime\MooringAnalysis\0D Wave Forcing.

2.12.1   Generate input files for MIKE Animator Plus

Before opening MIKE Animator Plus you must ensure the input data files fits the purpose.
This example only displays the results for 10 minutes from the end of the simulation.

Bathymetry

The bathymetry is already defined in the file .\Inputs\Domain**Domain.dfs2.**
For better overview in the MIKE Animator Plus setup, copy the file to the folder dedicated
the MIKE Animator Plus setup: .\MAPlus**Domain.dfs2.**

Surface elevation

The surface elevation in the mooring analysis simulation was specified as a dfs0 file,
however this information needs to be represented by a dfs2 file in MIKE Animator Plus.
In order to reduce the size of the dfs2 file holding the water levels over time first a coarse
grid version of the bathymetry is created:

1.   Go to MIKE Zero and Click on 'New File'

2.   Under 'MIKE Zero' select 'MIKE Zero Toolbox'

3.   Expand the 'Transformation' section and double click on 'Rotate Grid'
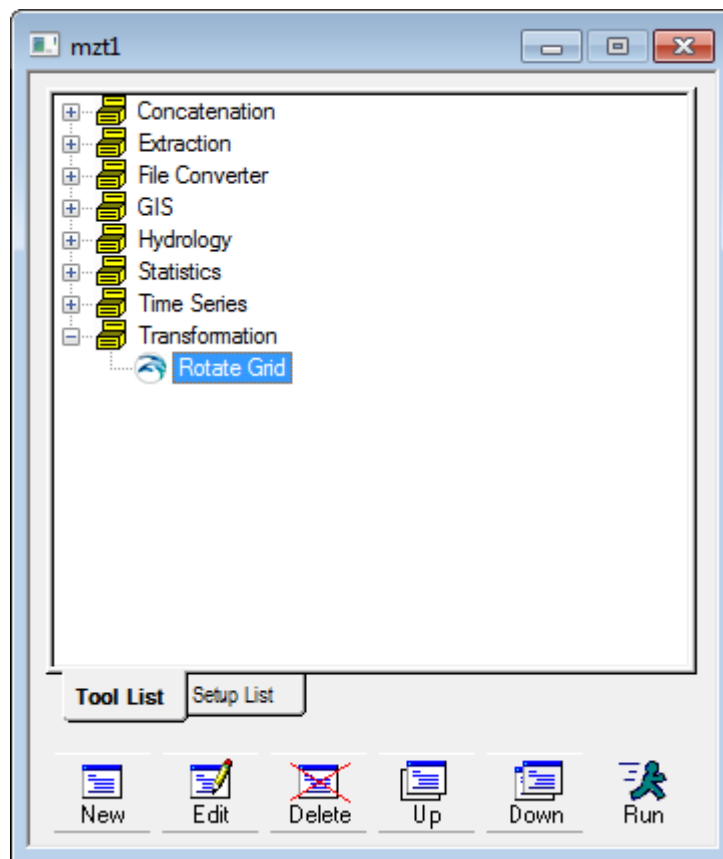
Figure 2-27   MIKE Zero Toolbox: Rotate Grid

4.   Create a new setup called 'Coarse Bathy'

5.   Select the input bathymetry file **Domain.dfs2** mentioned above

6.   Accept the subseries selection

7.   Select item as 'Scalar'

8.   Define a new grid with 9 grid points in the X direction and 2 grid points in the Y direction, both with grid spacing 40 m. The relative origin is set to (0,0) cells and the rotation angle to 0 deg.

9.   Define Land-water interpolation option to 'Data is bathymetry data' and set minimum land value to 2 m.

10. Save the resulting dfs2 file in the following location: \MAPlus**CoarseDomain.dfs2**

11. Click on 'Execute' to run.

12. At successful completion, a pop up box saying 'Status OK!' will show up. Click 'OK* and 'Finish' and save the setup as 'GenerateTSData.mzt'.

Next the water levels from the input dfs0 file must be populated into a coarse grid file.

1.   In MIKE Zero open the MIKE Zero Toolbox file 'GenerateTSData.mzt' (if closed)

2.   Expand the 'Time Series' section and double click on 'Preprocessing Temporal Data'
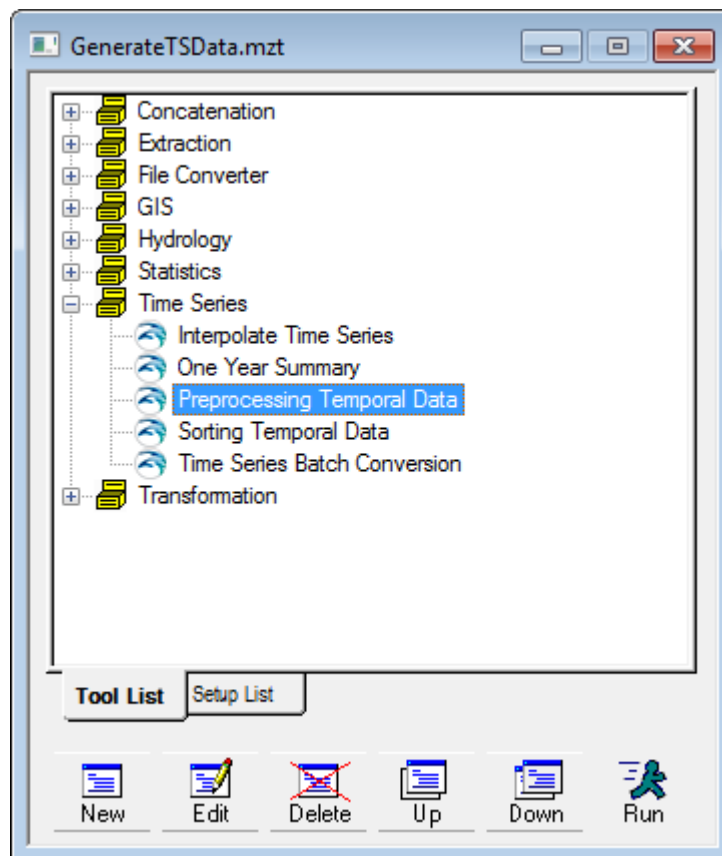
Figure 2-28   MIKE Zero Toolbox: Preprocessing Temporal Data

3.   Create a new setup called 'Waterlevel 2D'

4.   Select to import station location 'From x,y,name file' and select the existing file
      .\Inputs\Port Data**Fenders.xyz.**

5.   Set the Projection to 'Local Coordinates' and press 'Import stations'.

6.   Highlight the second row and press the 'Delete' button (only one point in the Domain
      is needed)

7.   Press the '…' button in the remaining row and select the second item ('Surface
      elevation) from the previously generated time series file
      .\Inputs\Metocean_timeseries**Hm0_1m_tp10s.dfs0**

8.   Press Next and accept default calculation method. Press Next.

9.   For Model output select 'Spatially and temporally distributed data'. Define the static
      model area by 'dfs2 file' and select the previously generated coarse bathymetry file
      .\MAPlus**CoarseDomain.dfs2**

10.   Save the output file name at the following location: .\MAPlus**Waterlevel_2D.dfs2**.
      Define the 'Item name' as 'Water level'

11.   Define 'Start time' as **01/01/2016 01:00:00** and 'End time' as **01/01/2016 01:10:00**.
      Set the 'Time step in output' as 1 second.

12.   The Overview will show the position of the station in the domain

13. Click on 'Execute' to run.

14. At successful completion, a pop up box saying 'Finished successfully!' will show up. Click 'OK* and 'Finish' and save the setup.

Vessel movements

In order to visualise the movements of the vessel in MIKE Animator Plus a 3Dsolid (.3ds) can be linked to a trajectory file. In this example the 3Dsolid Container.3ds is selected as a representative for the vessel.
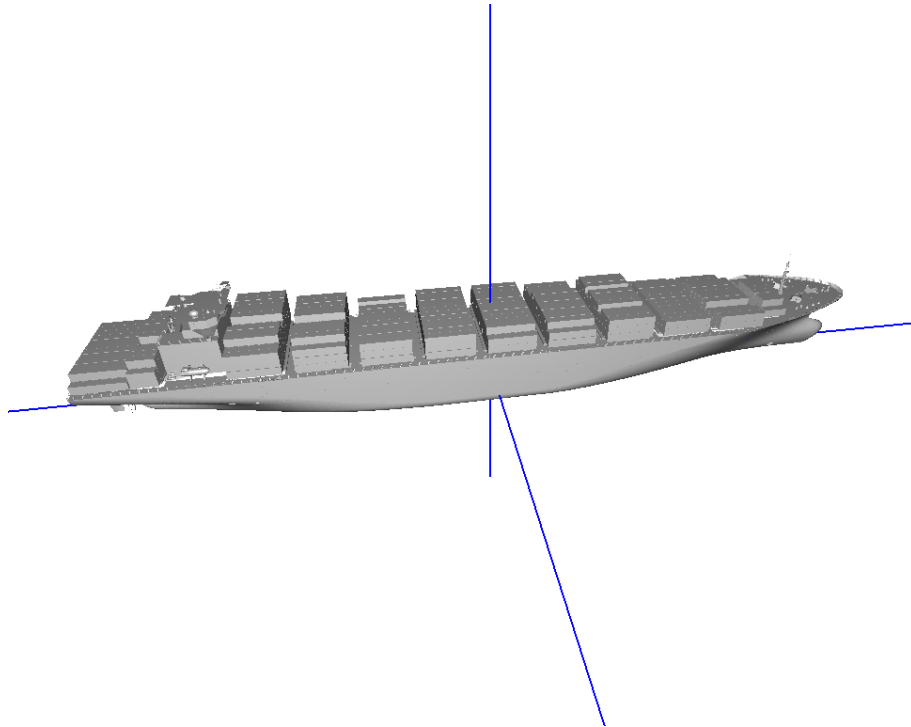
Figure 2-29   Container.3ds as displayed in MIKE Animator Plus.
The cross shows the origin of the 3D solid in its local coordinate system

The MIKE 21 Mooring Analysis simulation creates an output file with resulting vessel movements relative to the origin (0,0). As the selected 3D solid is defined with a local Z-level origin at the bottom of the ship, it is necessary to modify the vessel motion values to position the vessel correctly with the water level. Also, the item descriptions need to be modified in order to meet the standard for trajectory files in MIKE Animator Plus. For simplicity the time step in the file is modified to match the water level, i.e. 1 second.

1. In MIKE Zero open the MIKE Zero Toolbox file 'GenerateTSData.mzt' (if closed)

2. Expand the 'Time Series' section and double click on 'Interpolate Time Series'

Figure 2-30   MIKE Zero Toolbox: Interpolating Time Series

3.   Create a new setup called 'MAPlus Trajectory file'

4.   Select the output file .\Setups\Final Mooring setup - 0D wave forcing.m21ma - Result Files**DHI_Vessel_motions.dfs0'** and select all six items in the file. Press OK and Next.

5.   Define 'Start time' as **01/01/2016 01:00:00** and 'End time' as **01/01/2016 01:10:00**. Set the time Interval to 1 second. Press Next

6.   Specify the new output file as .\MAPlus**DHI_Vessel_motions_MAPlus.dfs0**

7.   Press next and click on 'Execute' to run.

8.   At successful completion, a pop up box saying 'Interpolation succeeded' will show up. Click 'OK* and 'Finish' and save the setup.

Now it is necessary to modify the dfs0 file to match the requirements for this MIKE Animator Plus setup.

1.   Open **DHI_Vessel_motions_MAPlus.dfs0** in Time Series Editor

2.   Subtract 9.8 m from the third item (Heave) to account for the Draft of the vessel (as defined in Table 2.1), see Figure 2-31. Press OK and save and close the file 'GenerateTSData.mzt'.
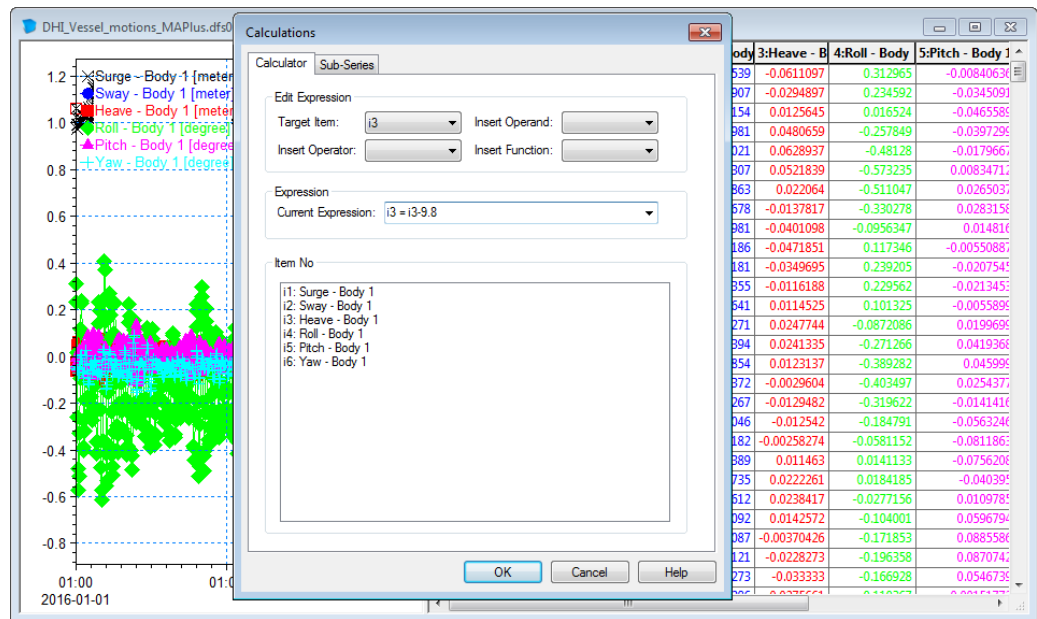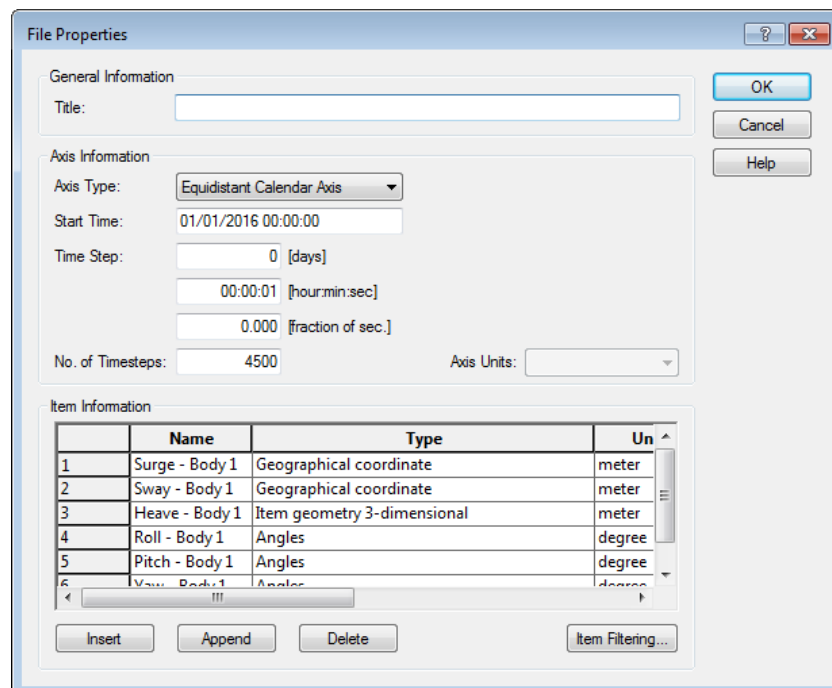
Figure 2-31   Modifying position of vessel to accommodate local coordinate system of 3Dsolid.

3.   Change the item descriptions to meet MIKE Animator Plus requirements as follows:
   a.   Click Edit → Properties …
   b.   Change item type for item 1 (Surge) to be 'Geographical coordinate'
   c.   Change item type for item 2 (Sway) to be 'Geographical coordinate'
   d.   Change item type for item 3 (Heave) to be 'Item geometry 3-dimensional'
   e.   Leave item types for item 4 to item 6 as is (Angles)
   f.   Click OK and save file.

Figure 2-32   Valid item descriptions in trajectory file for MIKE Animator Plus

2.12.2 Create a MIKE Animator Plus setup

The set of steps below outline how to create a new MIKE Animator Plus setup.
The first task is to create a setup to display the initial conditions. After that the MIKE Animator Plus setup can be extended to make an animation by including time varying data.

Initial conditions

1. Open the MIKE Animator Plus editor. It can be located by searching for it in the Windows start menu.

2. Click 'File' → 'Insert' → 'Scene' and select the domain bathymetry file .\MAPlus**Domain.dfs2**

3. Highlight 'MIKE 21 file – Domain.dfs2' in the property tree and double-click 'Bathymetry' to select to show bathymetry
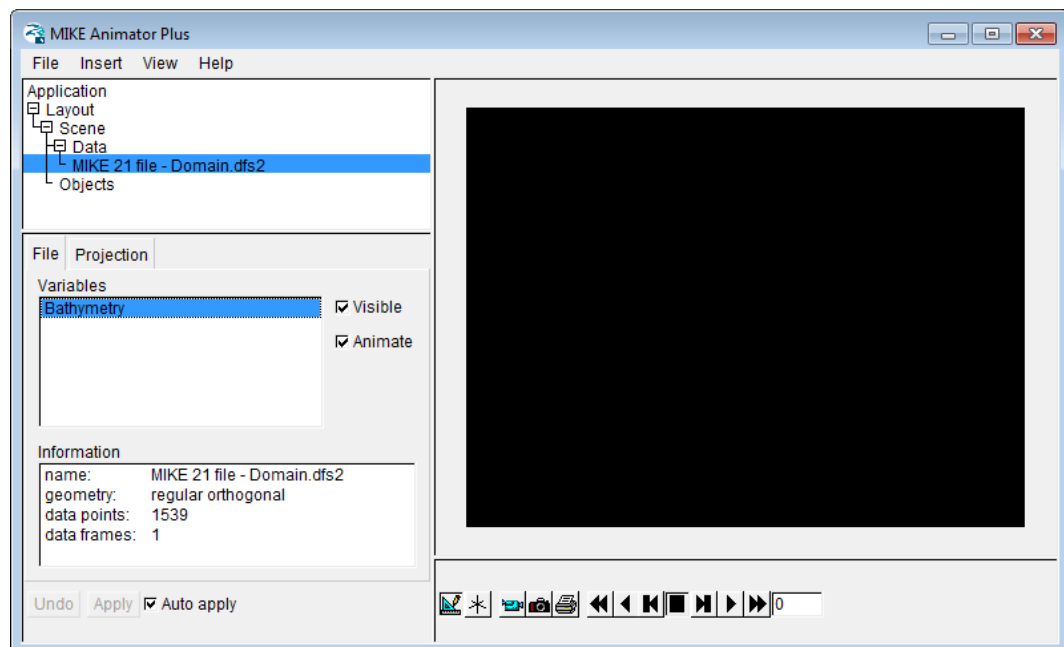
Figure 2-33   Inserting and selecting bathymetry data in MIKE Animator Plus

4. Click on 'Scene' and select the 'Fill' tab. Click on 'Fountain fill' to change the colour of the background.

5. 'Select Scene mode' in the toolbar as shown in Figure 2-34 and change the viewpoint by click-and-drag in the graphics window.

.
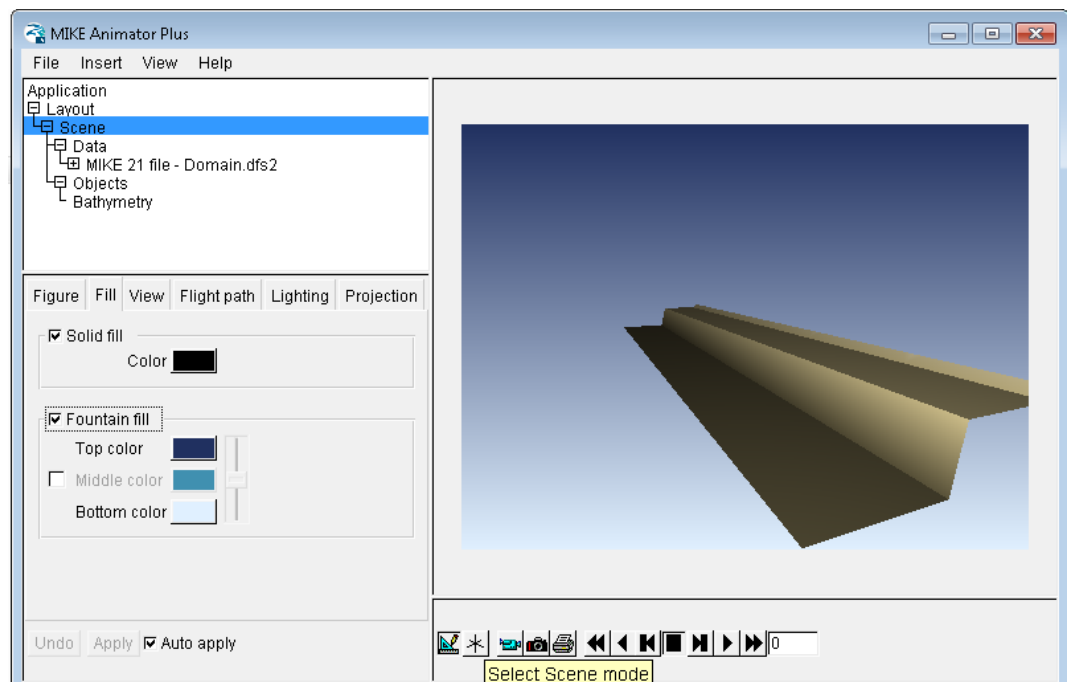
Figure 2-34   View of domain bathymetry after changing the fill colour and viewpoint

6.   Click 'File' → 'Save Layout' and save the setup to the new file
.\MAPlus**Vessel_motions.lyt**

7.   Right-click 'Scene' and select 'Load data..'
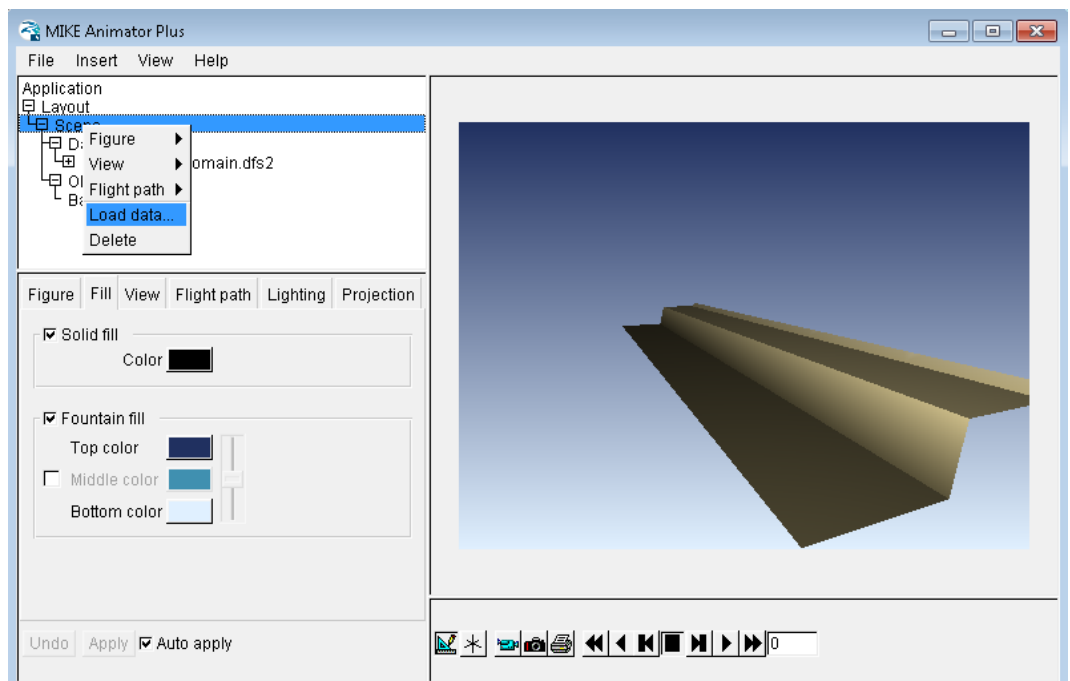
Figure 2-35   Adding data to existing scene

8. Select the file .\MAPlus**Waterlevel_2D.dfs2** and double-click the 'Water level' variable to display the item data in the graphics window

9. In the 'Surface' tab uncheck 'Mulit-color' and select 'Smooth' surface

10. Finally click the 'Color' button and change the Alpha value from 255 to 155 to include transparency of the water surface
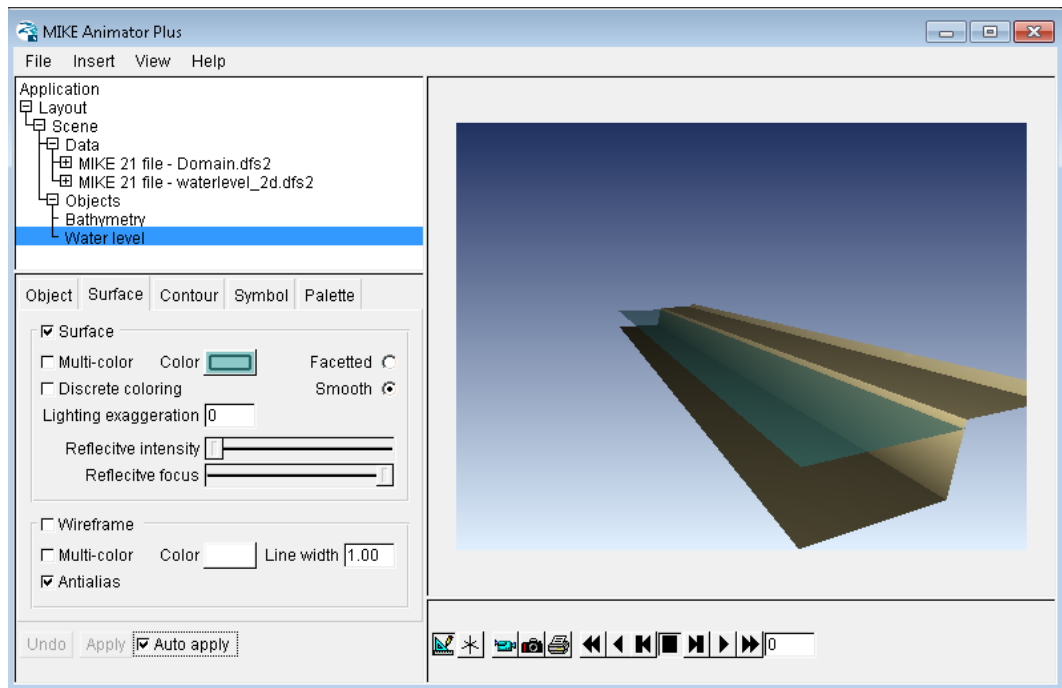
Figure 2-36   Transparent water level in the graphics window

11. Click 'File' → 'Save Layout' and save the setup

12. Right-click 'Scene' and select 'Load data..'

13. Select the 3Dsolids file .\MAPlus**Container.3ds** and double-click the 'Solids Model' variable to display the solids file in the graphics window

14. Now scale the Solids Model to resemble the outline of the vessel used in the Mooring Analysis simulations. Set the variables as follows:
    a. Uncheck 'Uniform Scaling'
    b. Set X extent to 223.8 m
    c. Set Y extent to 32.4 m
    Set Z extent to 35.6 m (twice the deck height)

15. To position the Solids Model correctly in the vertical for the initial time step, go to the 'Solids' tab and change the translation in the Z-domain to -9.8 m.

16. Click 'Objects' in the tree and change the Drawing order so the Water level is the last item drawn.
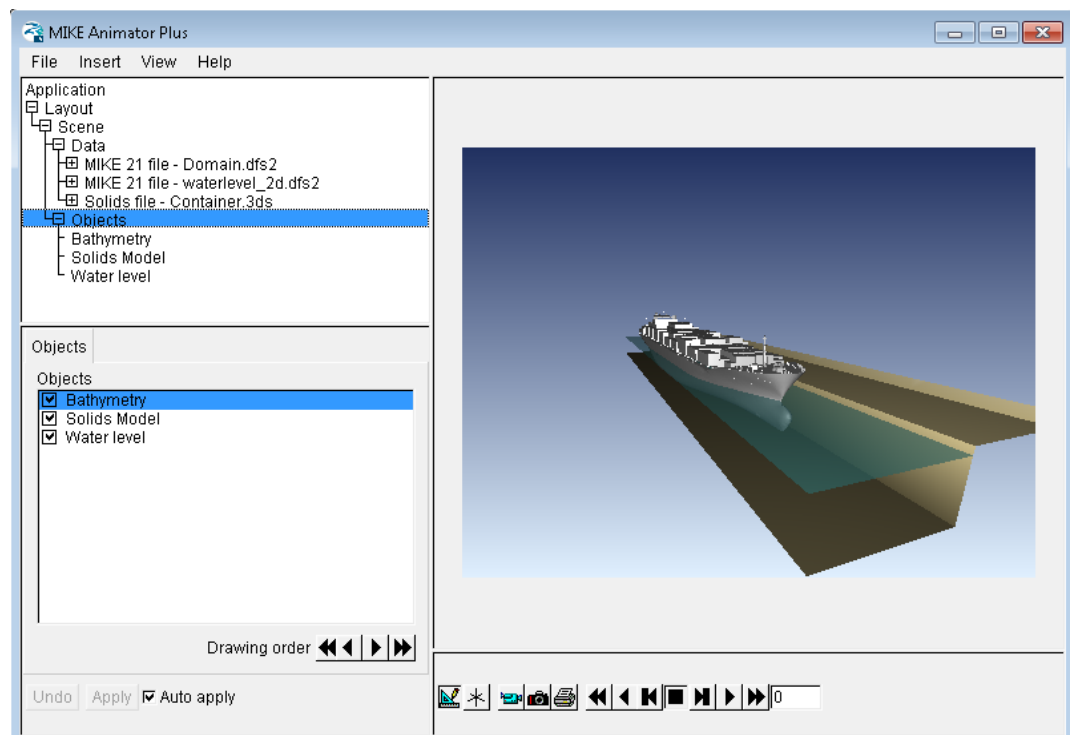
Figure 2-37   View of Solids Model after scaling and repositioning from default values

17. Click 'File' → 'Save Layout' and save the setup

Animation

1.   Click 'Insert' → 'Clock' to add a clock to the graphics window

2.   Click the 'Select Layout mode' icon in the toolbar
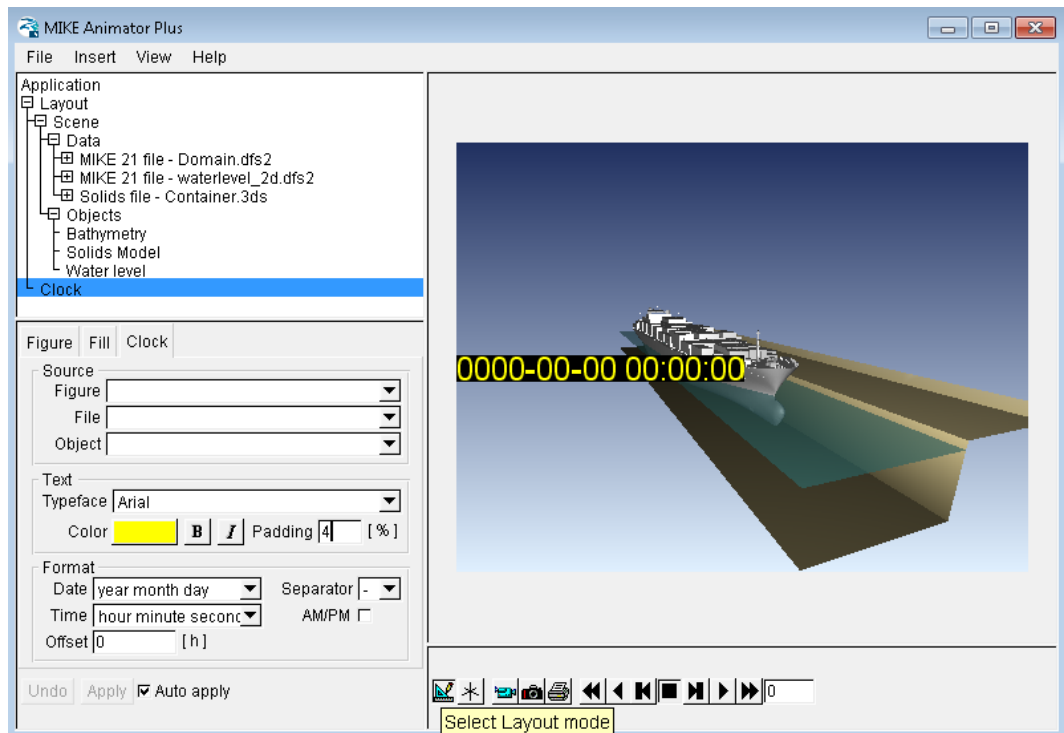
Figure 2-38   Initial view of Clock object in graphics view

3.   In the 'Fill' tab un-check 'Solid fill'

4.   Click-and-drag the Clock object in the graphics-window to a position in the upper left corner

5.   In the 'Clock' tab select the Source Figure to be 'Figure 1 Scene' and the Source File to be 'File 2 – MIKE 21 file – Waterlevel_2D.dfs2'
     (Now the clock will indicate the time stamp in the file while the animation is running)

6.   Click the 'Play' button to watch the water level change in time
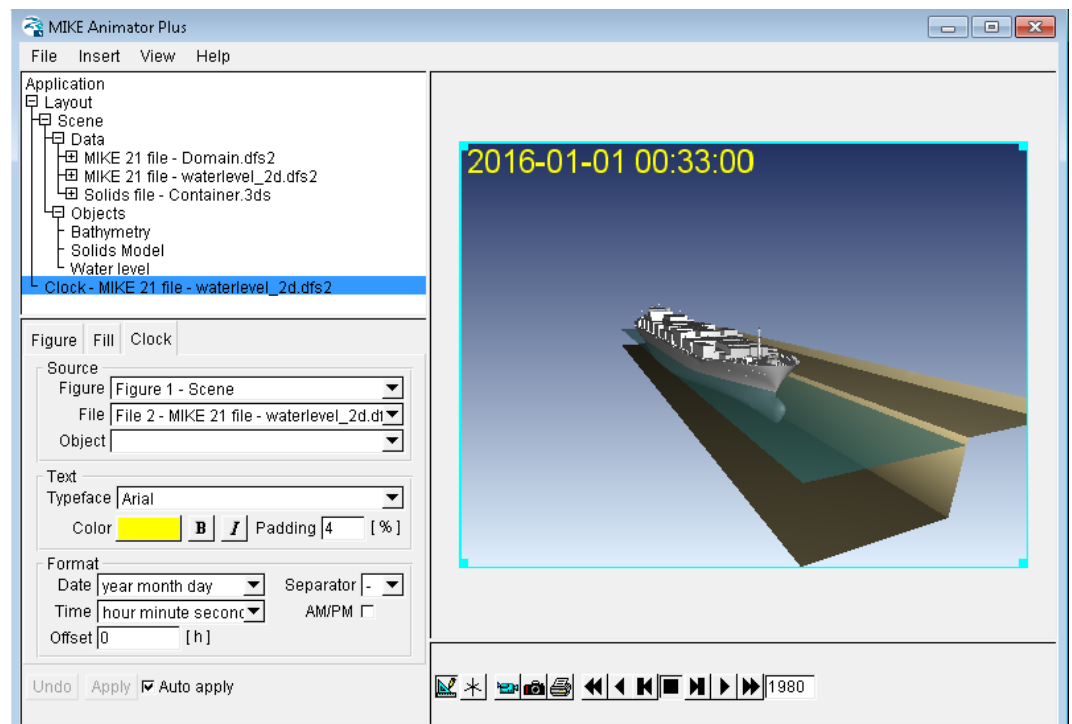
Figure 2-39   Animating time-varying water level with clock showing instant time step

7.   Next click the Solids Model Object and select the 'Trajectory' tab.

8.   Define the Movement to '6 parameters' and select the Source file
       .\MAPlus**DHI_Vessel_motions_MAPlus.dfs0**
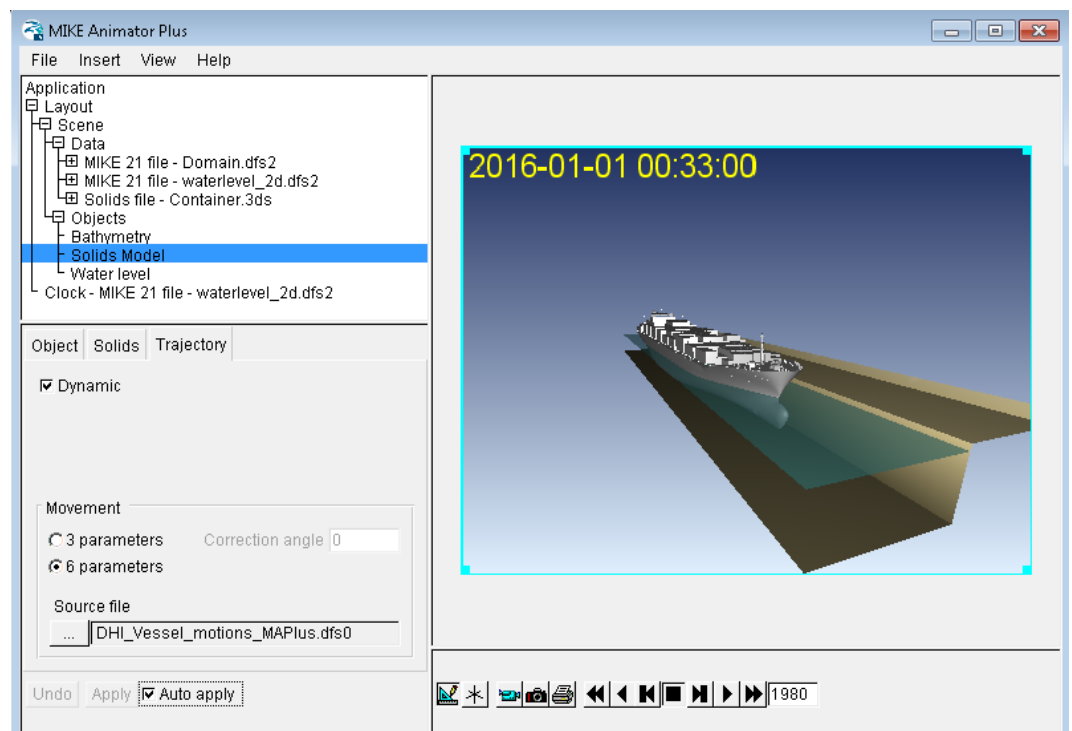
Figure 2-40   Specifying data file for vessel movement

9. Click the 'Play' button to watch the vessel move and water level change in time

10. Click 'File' → 'Save Layout' and save the setup

11. Click Scene mode to rotate and position the domain to get the optimal view for you

12. Save the animation by clicking the 'Animate' button in the toolbar below the graphics window.

13. Save the file as e.g. **Vessel_Motions.avi**.

14. While the model records the animation the graphics window will be greyed out, and a progress bar appear in a new window.
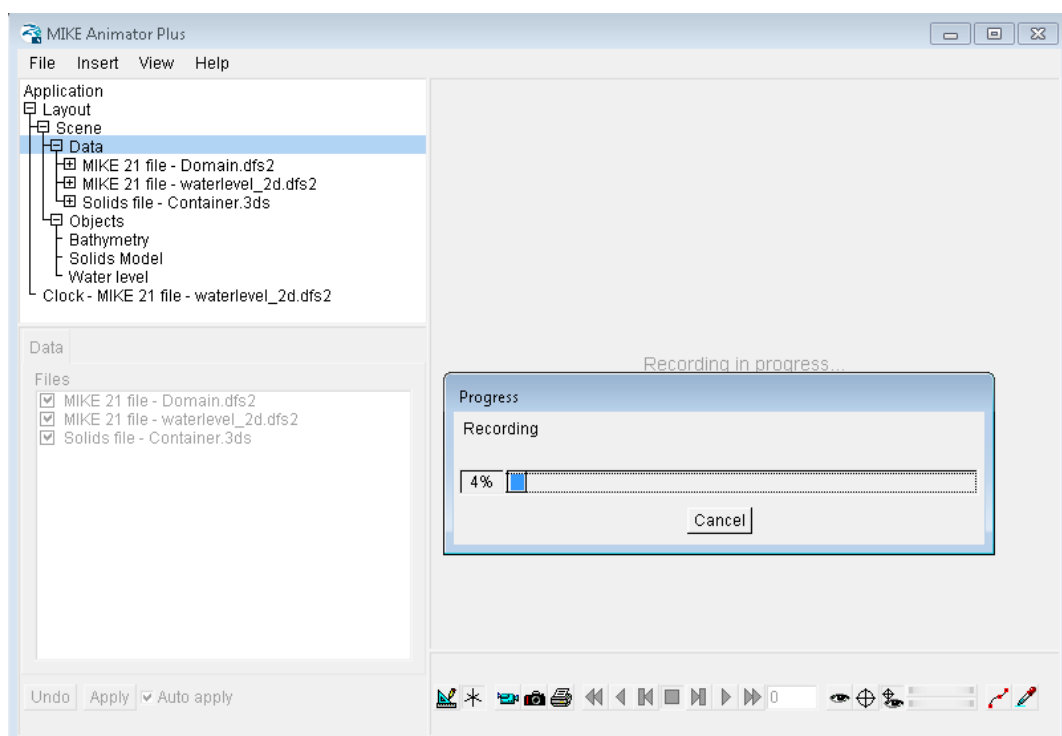
Figure 2-41   Recording animation of vessel movement

15. When the recording is done click 'Close'.

16. The resulting animation file **Vessel_Motions.avi** can now be shown in e.g. Windows Media Player as shown in Figure 2-42.
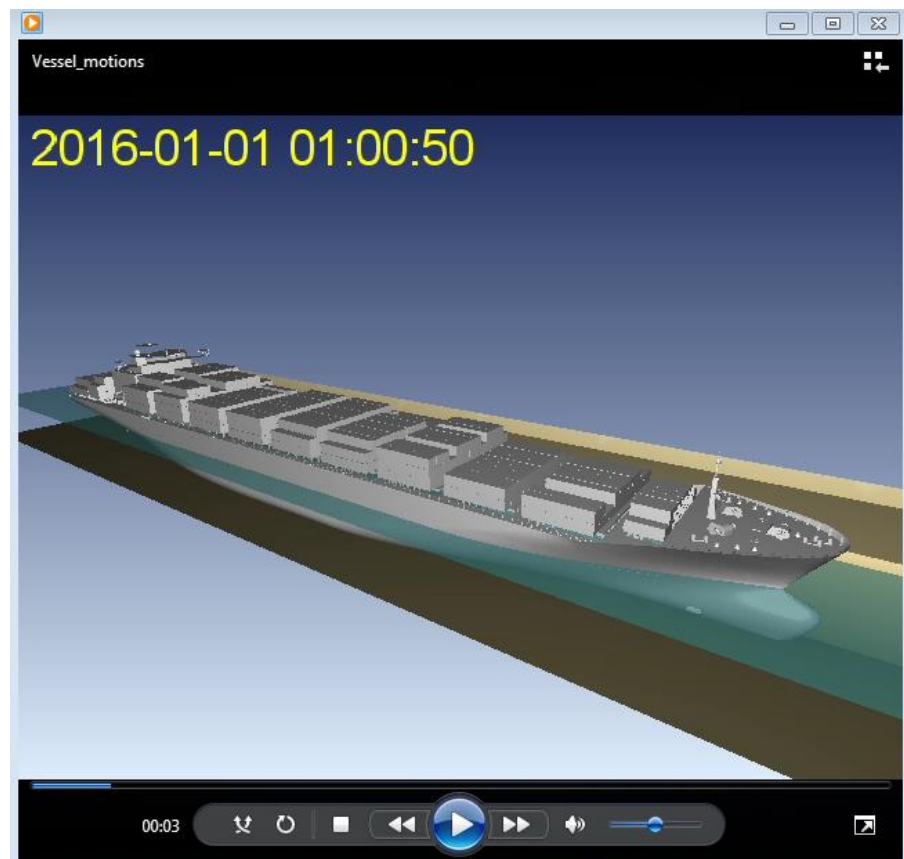
Figure 2-42   Recorded animation as shown in Windows Media Player

3 Example: 2D Wave Forcing (Port of Brisbane, Passing Vessel Induced Moored Vessel Motions)

This example is based on a scenario from the 2009 validation of physical model tests of passing vessel induced moored vessel interaction carried out for the Port of Brisbane, Australia presented in Mortensen et al (2009). The example files present a simplified version of the numerical validation.

The example consists of a passing vessel scenario where moored ship motions induced by the passing of a 47,000 m^3 tanker were investigated at the Shell Berth in the Upper Lytton Reach of the Brisbane River.

This tutorial consists of three major sections:

1. Creating a passing vessel induced pressure field

2. Running a MIKE 21 FM simulation to calculate passing vessel induced displacement waves

3. Utilisation of the calculated displacement wave in M21 MA in order to calculate moored vessel motions, line and fender forces as a result of the passing vessel induced displacement waves.

3.1 Launch Frequency Response Calculator

As per Section 2.1, the first step in this analysis is to create an FRC setup to calculate the frequency determined response of the vessel.

1. Create a new instance of the (.fresponse) editor or open the file **DHI_Tanker.fresponse** in the folder <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\FRC\Setup

2. In the Vessel Configuration Dialog select the grid **1-3DHI.grd** located here:

   <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\FRC\Inputs\VesselGrid\1-3DHI.grd

3. Set the water depth to 13.2m and Density of water to 1024.61 [kg/m³]

4. Click on the 'plus' symbol ⊞ to open the *Vessel 1* tab. Under Vessel Characteristics, enter the following parameter values

Table 3.1   Vessel Characteristics - Vessel 1

Parameter	Value	Unit	
Vessel Scaling (X-Dim)	1	dimensionless	
Vessel Scaling (Y-Dim)	1	dimensionless	
Vessel Scaling (Z-Dim)	1	dimensionless	
rxx	7.38	[m]	
ryy	46	[m]	
rzz	46	[m]	
Vertical Center of Gravity (ZG)	10	[m]	**Relative to vessel keel**
Draft	10	[m]	**Relative to vessel keel**
Deck plane height	14	[m]	**Relative to vessel keel**

5.   Under Freedom of motion, set the freedom of motion to freely floating.

Dialog: Computational Settings

6.   Set *Simulation Mode to : M21 Mooring Analysis compatible mode*

7.   Set Number of frequencies to solve to 512

Dialog: Wave Drift Forces

8.   Ensure that Wave drift forces box is unticked. This is not necessary for this example
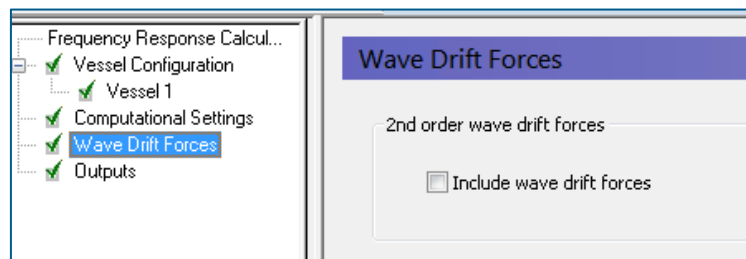
Figure 3-1   Wave drift forces checkbox

Dialog: Outputs

9.   Ensure that the Vessel Response checkbox is ticked. This will output the *.vre* file required for the M21 MA simulation.

10.   You may also wish to check the other boxes however, they are not essential for this example.

11.   Run the simulation.

3.2 Prerequisite Tasks
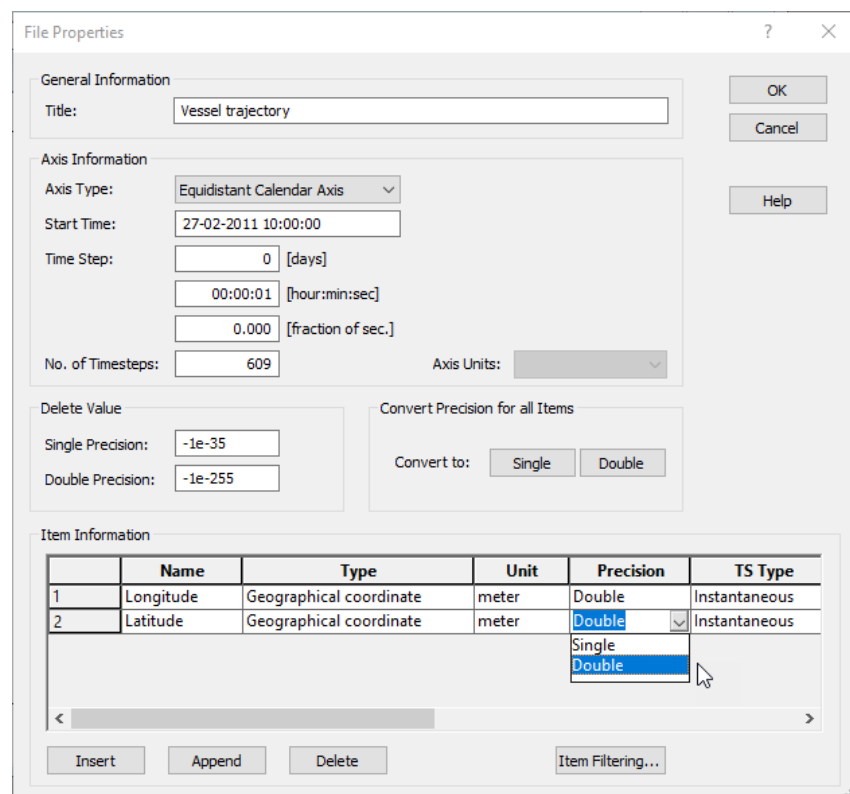
Prerequisite Task 1 – Create vessel track

The first prerequisite task is to create the trajectory of the passing vessel and associated track boundaries for inclusion into the HD mesh. The track should comprise a sequence of coordinates of the centre of the vessel at each point in time and should be in a dfs0 format (equidistant calendar axis).

Note: The model engine to create the moving pressure field requires that the input coordinates are specified in double precision.

The general steps for putting a sail route into a dfs0 file in double precision are provided below:

1. Create a new dfs0 file, with the standard time series (.dfs) editor
2. Specify the relevant start date, time step and number of time steps
3. Include 2 items, name them 'Easting' and 'Northing'. Set them both to item type 'Geographical coordinate' and with precision 'Double'.
4. Save the file as TrackCentre_8knots.dfs0.
5. Finally populate the Easting and Northing columns with the data values representing the sail route and save the file.

Figure 3-2   Time Series Editor – Change precision

You now have a file **TrackCentre_8knots.dfs0** with the Easting and Northing coordinates stored in double precision.

For the purposes of this example, a version of the specific vessel track file is provided here:

<example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\HD\PassingVessel**TrackCentre_8knots.dfs0**

Prerequisite Task 2 – Create HD mesh

Prior to creating a moving pressure field of the passing vessel, it is essential to first create a mesh on which the vessel will be positioned at each time step. For the purpose of this step-by-step guide, a mesh file comprising the berth pocket and adjacent passing vessel trajectory has been provided here:

<example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\HD\Bathy**2DExample_Mesh.mesh**

The mesh was developed using a simplified, constant bathymetry, however, the complexity of the mesh layout was retained to provide an example of typical mesh requirements for such a passing vessel analysis.

The mesh is made up of three primary element types:

1. Coarse resolution triangular elements – *away from the moored vessel*
2. High resolution triangular elements – *in the vicinity of the moored vessel*
3. Quadrilateral mesh – *along the length of the vessel track*

In order to prevent shock waves occurring from the introduction of the passing vessel into the model domain, it is recommended that the vessel track is created such that it extends well past the vessel, on either end. The quad mesh must be long enough to cover the entire vessel track. This is due to the fact that the engine for creation of the moving pressure field of the passing vessel, only supports a sail route which goes through quad mesh regions.
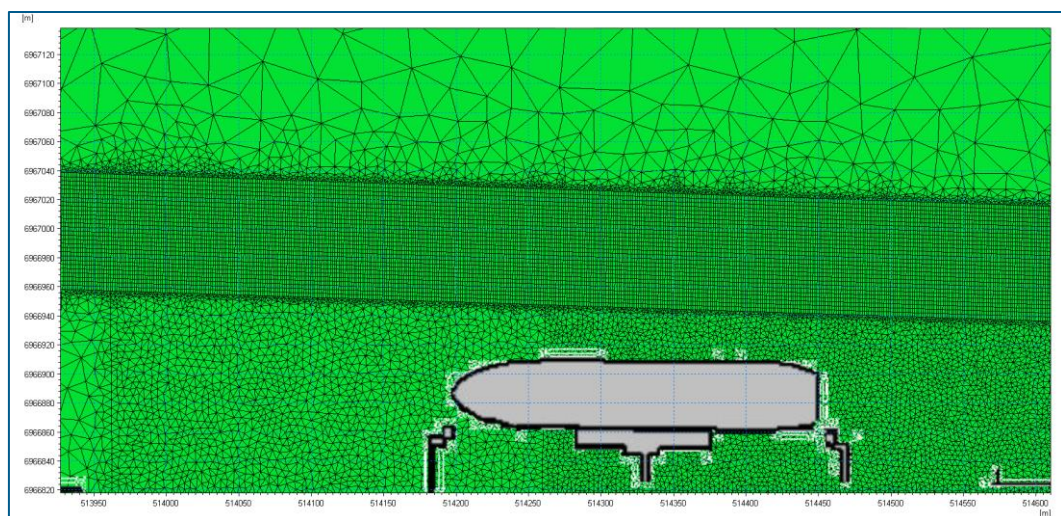
Figure 3-3   Extract from Port of Brisbane mesh showing the coarse mesh resolution further afield, high resolution in the vicinity of the moored vessel (visualised with graphics at the bottom of the plot), and quadrilateral mesh along the trajectory of the passing vessel

Instructions for mesh creation

This step by step guide does not provide detailed information on mesh creation, however, this is covered in the MIKE 21 HD FM manual.

To access this document go to:
- Start Menu -> Search for 'MIKE Zero Documentation Index'. The document will open in a web browser.

- From here, select MIKE 21 Documentation → **MIKE 21 Flow Model FM** →MIKE 21 & MIKE 3 Flow Model FM, Hydrodynamic Module, Step-by-Step, Training Guide

3.3 Create the Moving Pressure Field of the Passing Vessel

With the time series of the vessel position (vessel track) and mesh created, the next step is to create the moving pressure field.

For the purpose of modelling moving vessel induced displacement waves in MIKE 21 FM, the moving vessel can be represented by either a moving pressure field or a moving block of ice. In this tutorial the moving vessel will be represented by a moving pressure field. The moving pressure field for the passing vessel will be obtained through usage of a numerical engine in the MIKE installation PassingVessel.exe which as input requires a sail route trajectory in (.dfs0) format, a domain mesh (.mesh) and a panelised 3D vessel hull (.grd) file.

The first stage of the processing in the engine **PassingVessel.exe** will interpolate the 3D vessel hull onto a 3D stencil as illustrated in Figure 3-4. It should be noted that although the overall 3D shape of the vessel is well preserved, features such as the submerged part of the stern or bulb keel may be only partly represented.

A spatially and temporally varying pressure field is used to simulate the displacement of water caused by the moving vessel. The pressure field is generated from the vessel grid file containing the hull 3D geometry data, which is then interpolated to a 2D stencil in a dfsu format. Subsequently a pressure field representing the moving stencil is interpolated onto the mesh, with the vessel position's changing according to the pre-defined vessel track (the dfs0 based sail route trajectory).
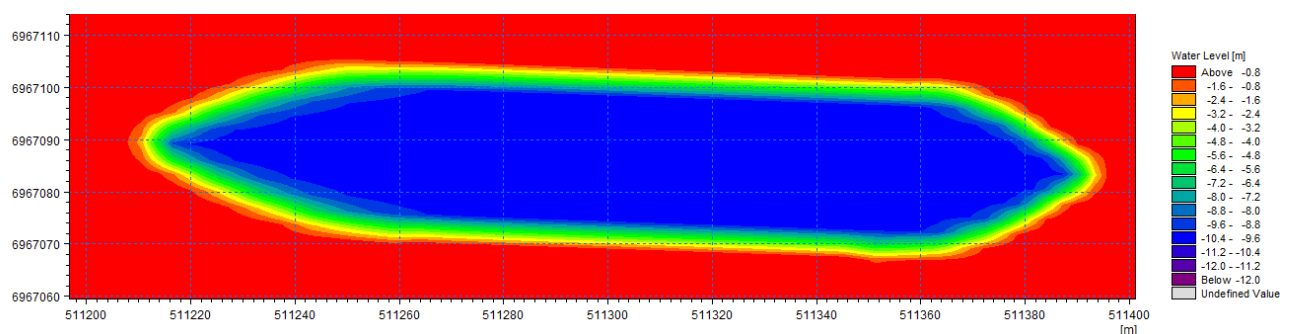
Figure 3-4   WSE around the passing vessel. **(Note**: The vessel track is at a slight angle in this example)

In its current form, the Passing Vessel engine is a stand-alone engine without a user interface. So the engine can currently only be executed from the command line (batch

mode). The steps below show how to run the Passing Vessel engine against the compatible input file (.m21pv) in batch mode by using a (.bat) file.

Follow the steps below to create the moving pressure field of the passing vessel:

1. Go to the folder

   <example installation folder>\MIKE Zero
   Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\HD\PassingVessel

2. Open the file **TrackCentre_8knots_1-3DHI.m21pv** with a text editor such as Notepad or Textpad. You will see that there are 7 main PFS sections in this file:
   - [DOMAIN]
   - [VESSEL]
   - [SIMULATION_TIME]
   - [VESSEL_TRAJECTORY_DATA]
   - [PHYSICAL_DATA]
   - [COMPUTATIONAL_SETTINGS]
   - [OUTPUT]

Dialog: Domain
3. Link to the file **2DExample_Mesh.mesh** as below

```
[DOMAIN]
   file_name = |..\Bathy\2DExample_Mesh.mesh|
EndSect  // DOMAIN
```

Dialog: Vessel
4. Link to the file **1-3DHI.grd** and set the scaling parameters as below

```
[VESSEL]
   vessel_grid = |.\1-3DHI.grd|
   draft = 10
   [VESSEL_SCALING]
      x = 0.999
      y = 1.038
      z = 1
EndSect  // VESSEL_SCALING
```

Dialog: Simulation Time
5. Set the *start_time* to 2011, 2, 27, 10, 0, 0
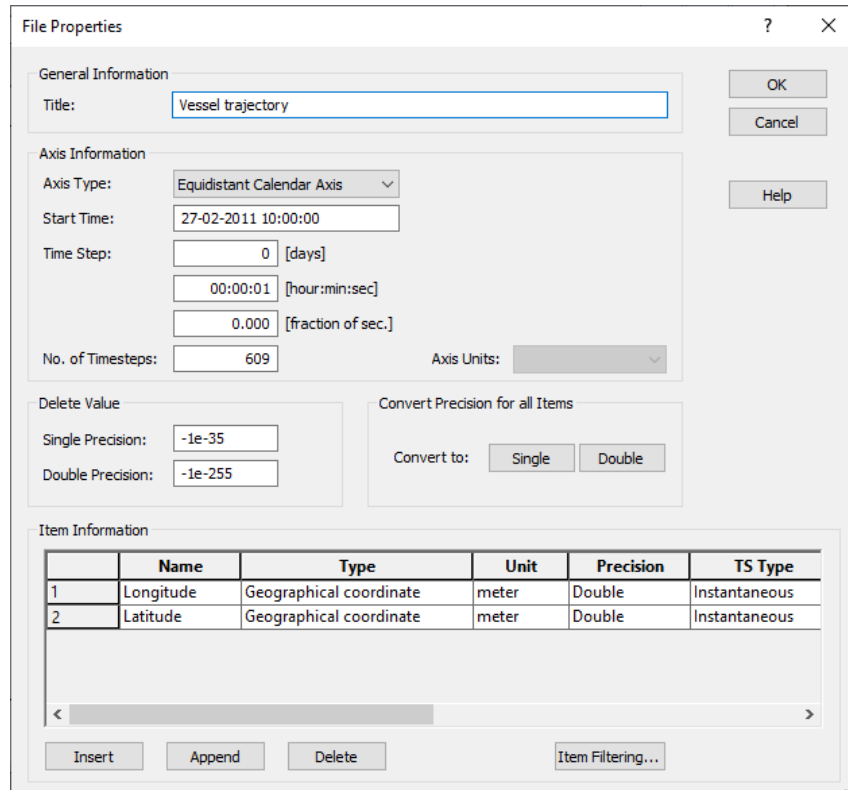
6. Set the *number_of_timesteps* to 540

7. Set the *time_step_size* to 1

```
[SIMULATION_TIME]
   start_time = 2011, 2, 27, 10, 0, 0
   number_of_time_steps = 540
   time_step_size = 1
EndSect // SIMULATION_TIME
```

Dialog: Vessel Trajectory Data
8. Set *type* to 0. This is currently the only mode supported in the passing vessel engine and is based on a dfs0 file (Equidistant calendar axis) containing 2 items which

represents the sail route trajectory of the centre point of the vessel at each time step, with equidistant time steps. To check that the dfs0 has the correct format, open the dfs0 file and go to *File →Edit →Properties* and check that settings are as shown below.

Figure 3-5   Properties of input vessel track dfs0 file

9.   Ensure that the projection is set to MGA-56.

10.   Link the *vessel_ file* to the time series of vessel position - *TrackCentre_8knots.dfs0*

11.   Set *items = 1, 2*

```
[VESSEL_TRAJECTORY_DATA]
   type = 0
   projection = 'PROJCS["MGA-56",GEOGCS["Unused",DATUM["Australian
   Datum",SPHEROID["Geodetic Reference System
   1980",6378137,298.257222101]],PRIMEM["Greenwich",0],UNIT["Degree",0.01
   74532925199433]],PROJECTION["Transverse_Mercator"],PARAMETER["False_Ea
   sting",500000],PARAMETER["False_Northing",10000000],PARAMETER["Central
   _Meridian",153],PARAMETER["Scale_Factor",0.9996],PARAMETER["Latitude_O
   f_Origin",0],UNIT["Meter",1]]'
   vessel_file = |.\TrackCentre_8knots.dfs0|
   items = 1, 2
EndSect  // VESSEL_TRAJECTORY_DATA
```

Note: The map projection string above must be obtained manually. The general and easiest way to obtain map projection strings is as follows:
•   Open up a MIKE Zero window
•   Under New File, select Mesh Generator (mdf)
•   Under the dropdown list that appears, select the map projection that you would like to use and save the file

- Load the newly created *.mdf* file in a text editor
- Scroll to the section [*MESH_DATA*]. Under the item *UTMZone* you will see the projection string required for the passing vessel input file, which can now be copy-pasted.

Dialog: Physical Data

12. Set gravity_constant to 9.81
13. Set ambient_pressure to 1013.0
14. Set density_type to 1 - *Set to 1 to define a density value or 2 to derive from the temperature & salinity values*
15. Set rho to 1024.61

If *density_type is set to 2 then desired temperature and salinity values must be specified*

```
[PHYSICAL_DATA]
   gravity_constant = 9.81
   ambient_pressure = 1013.0
   density_type = 1
   rho = 1024.61
   temperature = 10.0
   salinity = 32.0
EndSect // PHYSICAL_DATA
```

Dialog: Computational Settings

16. Set mesh_x_resolution to 0.1 [m]
17. Set mesh_y_resolution to 0.1 [m]
18. Set interpolation_threshold to 0.05
19. Set include_max_pressure_difference to 0
20. Set include_soft_start_duration to 0

```
[COMPUTATIONAL_SETTINGS]
   mesh_x_resolution = 0.1
   mesh_y_resolution = 0.1
   interpolation_threshold = 0.05
   include_max_pressure_difference = 0
   max_pressure_difference = 3
   include_soft_start_duration = 0
   soft_start_duration = 30.0
EndSect // COMPUTATIONAL_SETTINGS
```

Dialog :Output

The following default outputs are specified:

- **Vessel_pressure_field.dfsu** – this file represents the moving pressure field of the passing vessel and contains three items: Wind X, Wind Y and Pressure
- **Global_initial_WL.dfsu** - water level file of the initial position of the moving vessel (used as initial condition in MIKE 21)
- **Vessel_orientation.dfs0** – The engine produces a timeseries (dfs0) file showing a time-series of the first order time derivative of the vessel azimuth. The curve is a quantification of the magnitude of the rotation of the vessel in each time step. The value should be less than at least 10^{-3} in order to avoid shockwaves in the MIKE 21 simulations.
- **Vessel_log.dfs0**

- **Local_Vessel_WL.dfs**2

```
[OUTPUT]
   [MOVING_PRESSURE_FIELD]
      file_name = 'Vessel_pressure_field.dfsu'
   EndSect // MOVING_PRESSURE_FIELD
   [INITIAL_WATER_LEVEL]
      file_name = 'Global_initial_WL.dfsu'
   EndSect // INITIAL_WATER_LEVEL
   [VESSEL_ORIENTATION]
      include = 1
      file_name = 'Vessel_orientation.dfs0'
   EndSect // VESSEL_ORIENTATION
   [VESSEL_LOG]
      file_name = 'Vessel_log.dfs0'
   EndSect // VESSEL_LOG
   [LOCAL_VESSEL_WATER_LEVEL]
      include = 1
      file_name = 'Local_Vessel_WL.dfs2'
   EndSect // LOCAL_VESSEL_WATER_LEVEL
   [SOFT_START]
      include = 1
      file_name = 'Vessel_pressure_field_soft_start.dfsu'
   EndSect // SOFT_START
EndSect // OUTPUT
```
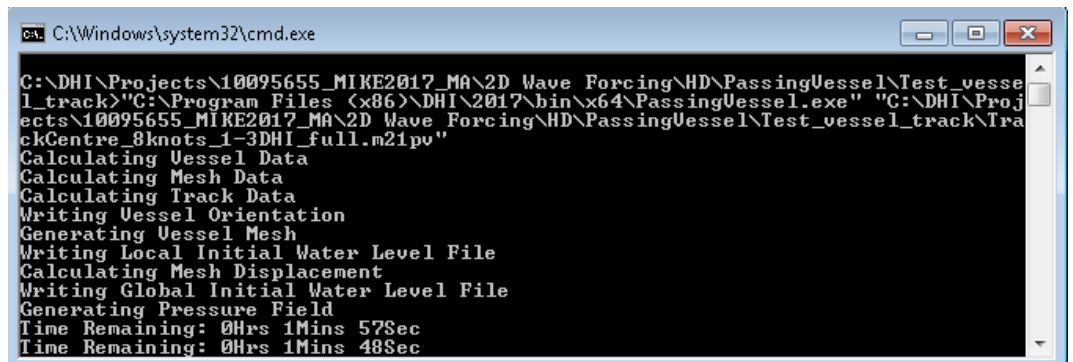
The passing vessel input file is now complete and the passing vessel simulation is ready to be run.

Steps (continued)

21. Double click on the .bat file **Run Passing Vessel.bat**, located here:

   <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\HD\PassingVessel**Run Passing Vessel.bat**

   The following window will pop up, dynamically updating the remaining time for completion of the simulation. Once the simulation is completed, a log file will be generated, with details of the simulation. The output files listed above will be located in the results folder.

Figure 3-6   Passing Vessel engine (Passingvessel.exe) - working messages

22. Check that the pressure field generated has three items (Wind X, Wind Y, Pressure). The 2 Wind composant items will be 0, whereas the Pressure item will show the

vessel moving through the pressure field as seen in Figure 3-7 below. This dfsu file will be used in the ensuing HD simulation.
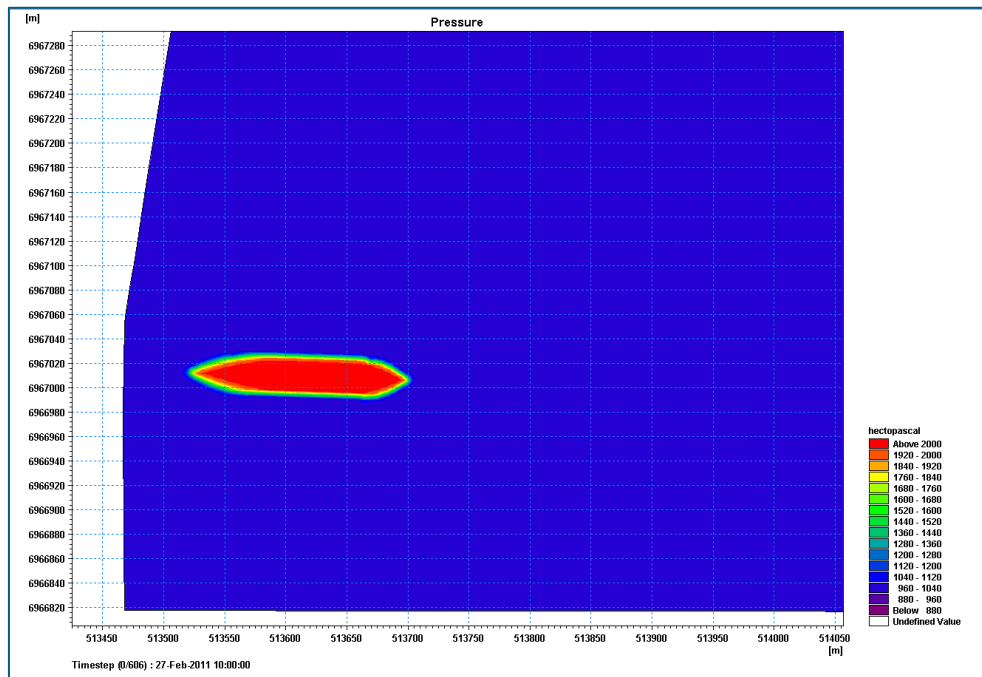
Figure 3-7   Vessel Pressure field dfsu - Pressure item

23.   Open the Vessel_orientation.dfs0 file and ensure that the values are very small.
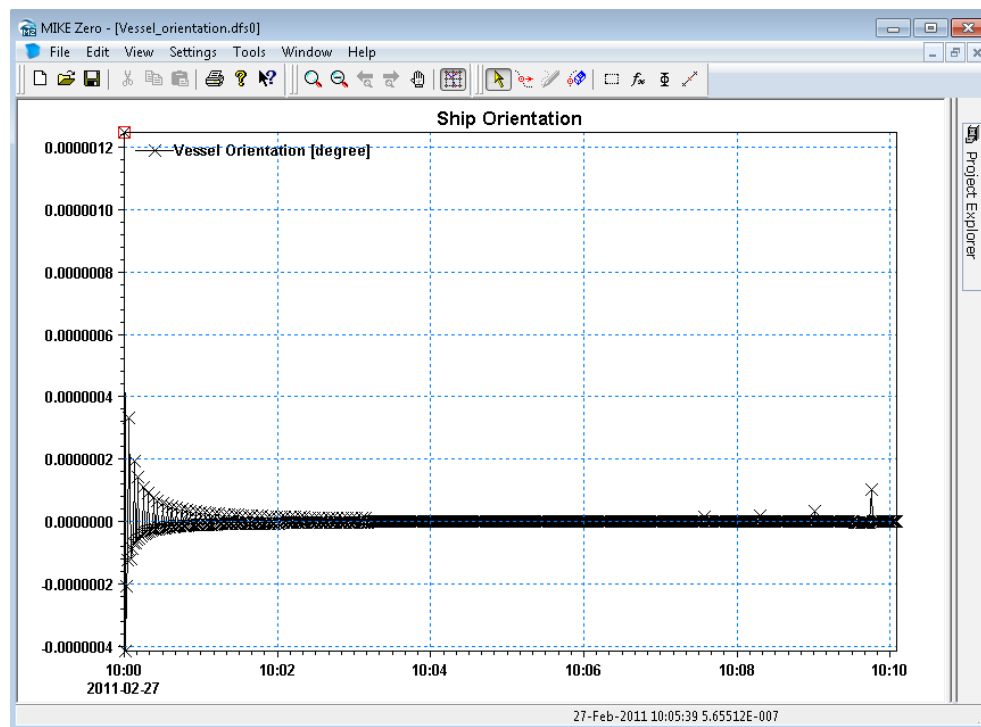
Figure 3-8   Vessel Orientation file. Values should be less than 10^{-3} to avoid shock waves.

3.4 HD Simulation – Create the Displacement Wave

With the moving pressure field now created, a hydrodynamic (HD) simulation will be undertaken to create the displacement wave caused by the passing vessel, which then propagates to the adjacent berth pocket and impacts the moored vessel. For the purpose of this step-by-step guide, the MIKE 21 FM simulation file **PassingVessel_Example.m21fm** has been provided here:

<example installation folder>\MIKE Zero Projects\MIKE_21_Maritime\MooringAnalysis\2D Wave Forcing\HD\Setup**PassingVessel_Example.m21fm**

Alternatively, create a new MIKE 21 FM simulation file by opening MIKE Zero -> Click on 'New' and select 'Flow Model FM (m21fm)', and save the file as PassingVessel_Example_*Username*.m21fm in the folder.

Edit the file according to the following steps.
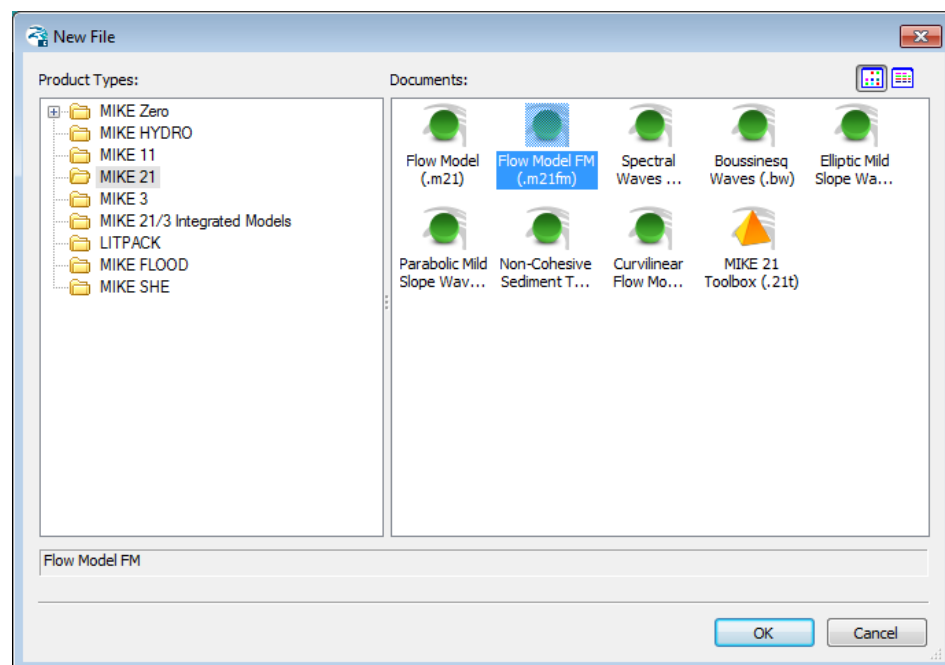
Figure 3-9   Creating a new MIKE 21 FM model

1. In the 'Domain' window, select the mesh file created earlier.

2. In the 'Time' window set the '*No. of time steps'* to be equal to the number of time steps in the .dfs2 file created by the moving pressure field tool.
   - Set the '*No. of time steps'* to 539.
   - Set the '*Time step interval*' to 1 second.
   - Set the '*Start date*' to 27-02-2011 10:00:00

3. Save the setup.

4. Select '*Higher Order*' scheme for both Time and Space in '*Solution Technique*'. Reduce the maximum time step to 1 second.

5. Deactivate Flood & Dry.

6. Select Wind Forcing to be 'varying in time and domain' and select the newly created moving vessel pressure field dfsu file:

   <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\HD\PassingVessel\TrackCentre_8knots_1-3DHI.m21pv - Result Files**Vessel_pressure_field.dfsu**

7. Set initial condition as '*Spatial varying surface elevation*' and select the created initial water level file.

   <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\HD\PassingVessel\TrackCentre_8knots_1-3DHI.m21pv - Result Files\ **Global_initial_WL.dfsu**

8. Define Boundaries:
   - Go to 'Boundary Conditions'.
   - Under Land boundary ensure the Type is set to 'Land (zero normal velocity)'.

9. Create a new output called 'Vessel_Induced_Flux', which should include a 2D extraction of *Total water depth, P flux and Q flux* for the entire model domain. Under the tab *Output Specification,* Set *Field Type* to 2D (horizontal) and *Output format to* Area series.

10. Generate an output called '*Wave Gauges*'. The output section should include a point output of surface elevation for the two following points (MGA-56):

Easting	Northing	Name
514327.8	6966882	Wave Gauge 1
514475.7	6966858	Moored Vessel

11. Click 'Run' and start the simulation.

12. The simulation will generate an output dfsu file called **Vessel_Induced_Flux.dfsu** containing the three selected flux variables. In order to input this data into M21 MA, the dfsu file must be converted into a dfs2 file (structured grid file). Go to MIKE Zero homepage. Click on '*New*' → select '*Grid Series (.dfs3,dfs2)*'.
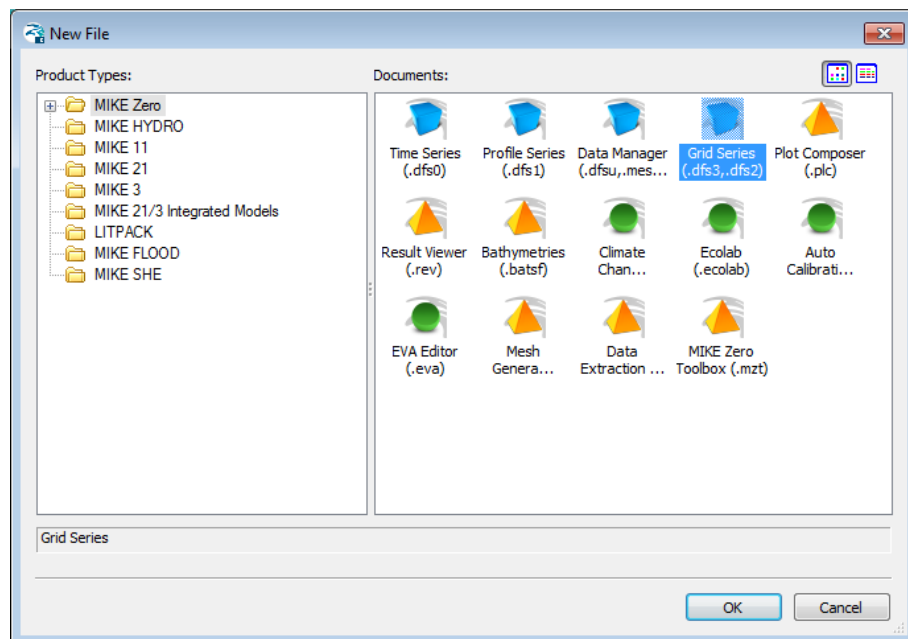
Figure 3-10   Convert dfsu file to dfs2 grid – step 1

13. Double click, select '*From Dfsu File*' and link to the dfsu created by the MIKE 21 FM simulation.

14. When prompted, input the following parameters.

Projection	MGA-56
Easting Origin	514339
Northing Origin	6966809
Rotation	0.067
Spacing in J-dir. and K-dir	2 m
No. of grid points J-dir	136
No. of grid points K-dir	45

15. Click Next until Step 5 then click *Finish*. This concludes the mapping of data from the original (.dfsu) file to a (.dfs2) file with the desired properties. This (dfs2) file now contains the flux due to a passing vessel induced wave.

16. M21 MA uses FFT when calculating the incident wave forcing. In order to capture the entire simulation we will use 4096 time steps in the FFT analysis. This is done as follows:
    - Make a copy of the newly created dfs2 file.
    - Select *Edit->Time Steps* and select the first time step.
    - Click the *copy* radio button and select *before*.
    - Then type in the number of time steps required to reach *4096* (eg. 3556) and press *Insert*.
    - Save the dfs2 file as '**Grid1_Extended.dfs2**'.

3.5 Overview of MIKE 21 Mooring Analysis (MA) Steps

In order to run the MIKE 21 MA simulation with 2D forcing successfully, a number of steps must be followed. These are outlined below and the step by step guide is structured to provide the user with an understanding of the sequence and method for undertaking the more complex M21 MA simulations, coupled to the usage of other MIKE models.

The ensuing guide is structured as follows:
1. Prerequisite Task – Access to mooring line and fender profiles
2. Launch MIKE 21 MA
3. Specify mooring line and fender profiles
4. Setup the Mooring System
5. Specify Environmental Conditions
6. Run simulation in Convergence mode
7. Run simulation in Final computation mode

3.6 Prerequisite Task – Access to Mooring Line and Fender Profiles

For this example mooring line and fender profiles are given, but for general usage the M21 MA includes access to a comprehensive selection of standard mooring lines and fenders which the user can select. Mooring line stiffness curves have been digitised from OCIMF guidelines (OCIMF, 2008) and fender compression curves were obtained from Trelleborg manuals.

Non-standard mooring lines and fenders can if needed be custom-made and included in the model as time series (dfs0 – relative item axis) of percentage elongation (mooring lines) and compression (fender) as x-axis and percentage of reaction force as item 1.

Figure 3-11 and Figure 3-12 show examples of the line and fender curves supported in M21 MA.
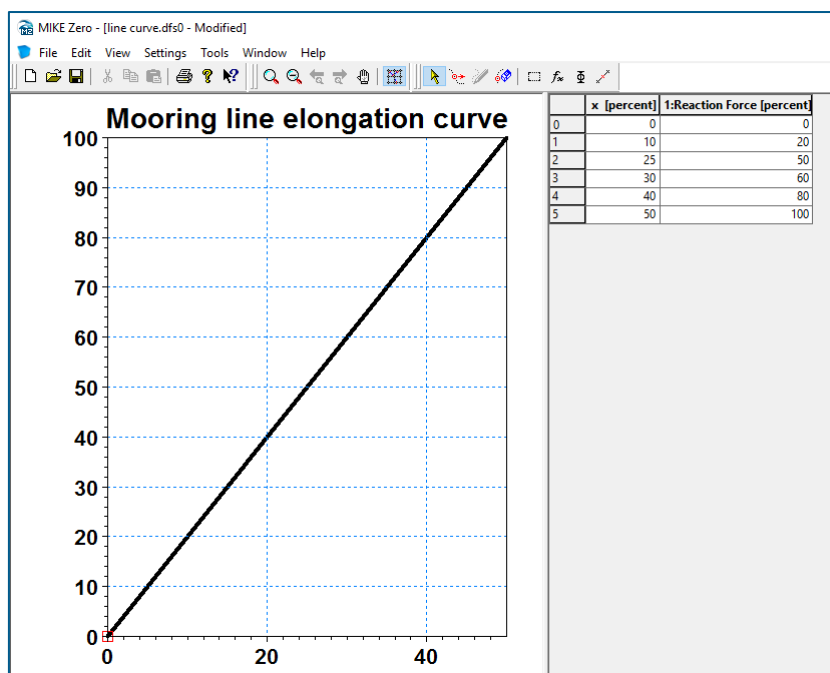
Figure 3-11   Linear Mooring line elongation curve – dfs0 for input into M21 MA
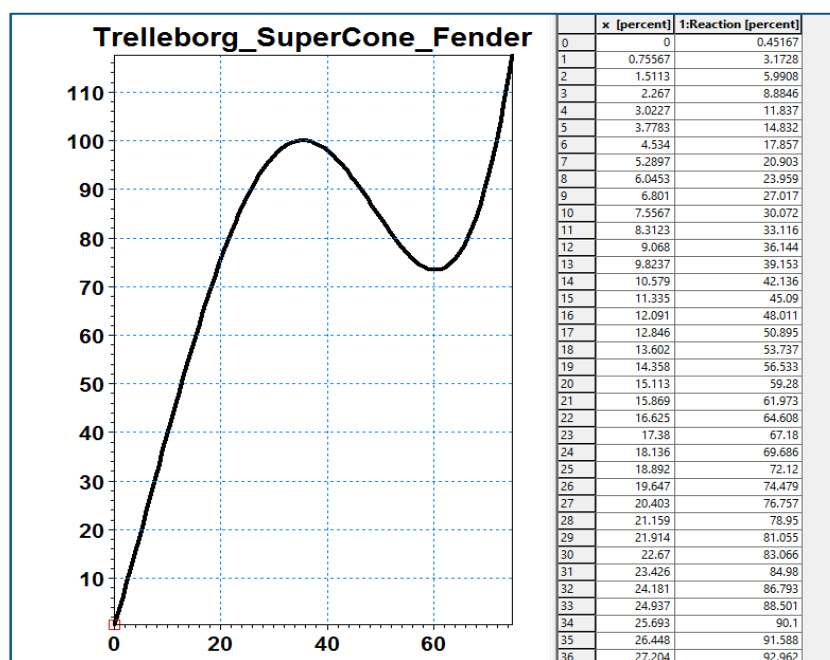
Figure 3-12   Trelleborg Supercone Fencer compression curve – dfs0 file for input into M21 MA

3.7   Launch MIKE 21 MA

Similar to the Frequency Response Calculator editor (.fresponse), the MIKE 21 Mooring Analysis editor (.m21ma) is also found under the Maritime subgroup of MIKE 21.
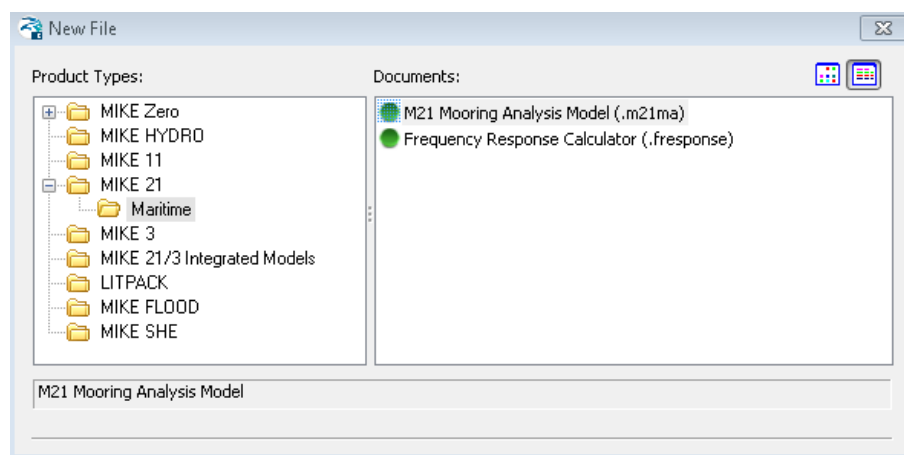
Figure 3-13   Location of MIKE 21 Mooring Analysis setup editor

The set of steps below outline how to create a new MIKE 21 Mooring Analysis (M21 MA) setup.

1.   Open a new instance of the (.m21ma) editor as shown in Figure 2-8, by clicking OK. The M21 MA main page (Figure 3-14) should appear on the screen.
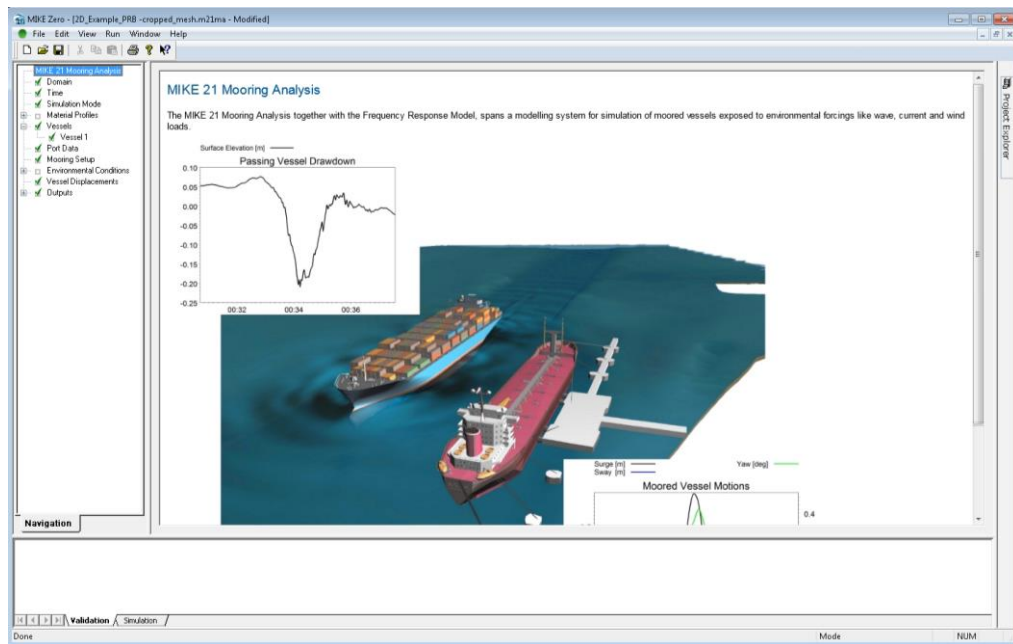
Figure 3-14   M21 MA main page

2. Load the provided tutorial file:

    \<example installation folder>\MIKE Zero
    Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave
    Forcing\M21MA\Setup**2D_Example.m21ma**

Alternatively
Start saving the file: Select 'File' -> 'Save' and save the filename in the root folder

\<example installation folder>\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave
Forcing\M21MA\Setup\2D_Example_*username*.m21ma

Take care not to overwrite the tutorial file

Dialog: Domain
In the 2D mode, M21MA displays the vessel within the 2D grid domain so that the user
can clearly see the position of the ship in the domain where the spatial wave file extent
applies. The first step is that a (dfs2) grid file comprising the bathymetry of the port/berth
pocket must be loaded into the 'Domain' dialog. This domain file can be created by
importing the (.mesh) file from the MIKE 21 FM setup into the mesh generator and
exporting it as a (.dfsu) file with a Bathymetry item. The resulting (.dfsu) file can then be
mapped into the (.dfs2) format, with the exact same settings as those we applied when
mapping the wave file from (.dfsu) to (.dfs2) in Section 3.4.

This domain file will dictate the coordinate projection of the wave (dfs2) file, as in all other
structured grid based M21 models. The user may then use the 'Plot extent' function to plot
the area in the immediate vicinity of the vessel.

1. Load the file and apply the custom settings seen in Figure 3-15

   <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\M21MA\Inputs**Domain\Domain.dfs2**
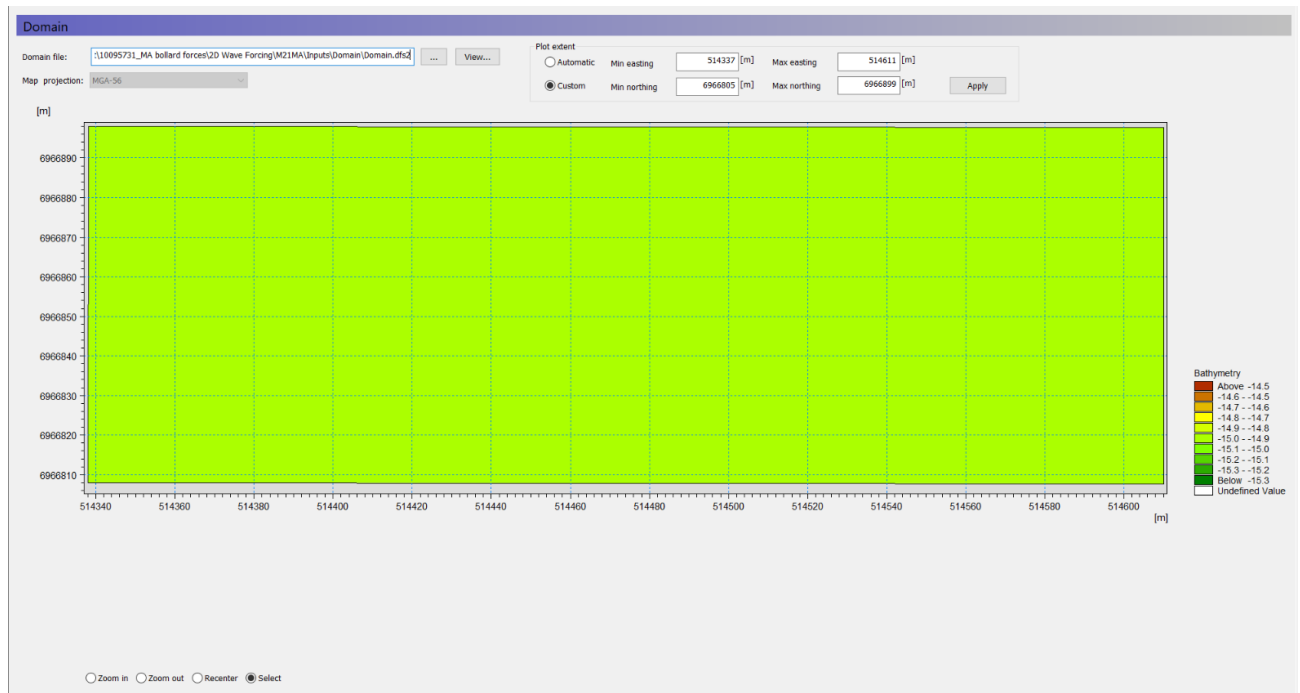
Figure 3-15   Domain window

We will skip the 'Time and Simulation mode' dialogs and return to them once the mooring system has been set up.

3.8   Specify Mooring Line and Fender Profiles

Dialog: Material Profiles

1. Click on the tab *Line Profiles, click new profile* and then click *Go to* button on the table that record appears
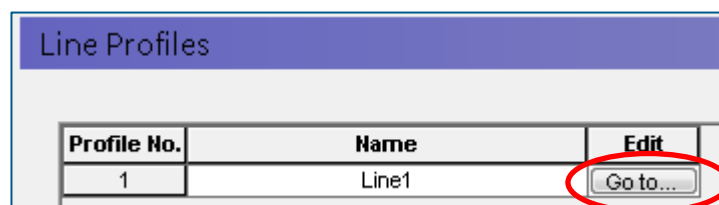
2. This will open up a separate tab for *Line 1* where these properties of the mooring line profile should be set.
   - Set the Total Breaking Strength to 148 [t]
   - Set the *Failure Load* to 50 [%]
   - Set the Linear damping coefficient to 0 [kg/s]
   - Set the Quadratic damping coefficient to 0 [kg/m]

- Click on the *Select* button and load the file **line curve.dfs0** here:
  <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\M21MA\Inputs\Material Profiles**line curve.dfs0**

3. Click on *Fender Profiles, click new profile* and then click on the *Go to…* button

4. This will open up a separate tab where these properties of the fender profile should be set:
   - Set the Max reaction force to 4900 [kN]
   - Set the Max deflection to 1 [m]
   - Set the *failure deflection* to 100 [%]
   - Fender friction coefficient to 0.45
   - Linear damping coefficient to 0 [kg/s]
   - Fender thickness = 1.5 [m]
   - Click on the *Select* button and link to the file **fender curve.dfs0** here:
     <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\M21MA\Inputs\Material Profiles**fender curve.dfs0**

For the purposes of this example, linear mooring line and fender curves have been used.

5. Click on *Bollard Profiles*, click *New profile* and then click on the *Go to…* button

6. This will open a separate tab where these properties of the bollard profile should be set:
   - Max. horizontal bollard pull force to 2000 [kN]
   - Max. vertical bollard pull force to 1000 [kN]
   - Max. total pull force to 2500 [kN]
   - Max. vertical line angle to 60 [deg]
   - Safety threshold 75 [%]

Be sure to activate the check boxes to include all these settings in this example.

Dialog: Vessels

The Vessels dialog requires the user to select the vessel response file generated with FRC and it will subsequently show the involved vessels in the table records.

1. Click on the *Vessels* dialog

2. Under *vessel response file*, click on the button and select the (*.vre*) file created from the earlier FRC run. This should be located in the results subfolder here

   <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing\FRC\Setup

   From here, click on *Go to*

3. This will load the *Vessel data* tab representing the properties of the (to be) moored vessel. From here:
   - Set Longitudinal area (wind) to 1200 [m²]
   - Set Transverse area (wind) to195 [m²]
   - Set *Lpp* to 189.6 [m]
   - Set the Loading condition to 0 [%]
   - Select the vessel class '*VLCC conventional*' from the following available vessel classes in the drop-down list shown below

Figure 3-16   Vessel data tab showing available vessel class selection

4.   Go to the *Fairlead data* tab. Copy and paste the following values into the fairleads table. **Note**: This data can also be loaded into the table if available in xyz format. The *symbol orientation* value allows the user to rotate the symbol of the connection point in the next tab *Vessel View*.

Table 3.2   Fairlead coordinates

Name	X [m]	Y [m]	Z [m]	Symbol orientation
Fairlead 1	91	5.8	20	0
Fairlead 2	-97	9.7	18.5	0
Fairlead 3	77	14	19.2	0
Fairlead 4	-82	13.4	18.5	0
Fairlead 5	48	15.1	18.5	0
Fairlead 6	-53	15.2	18.5	0

5. Go to the *Winch data* tab. Similarly, copy and paste the following values into the winches table.

Table 3.3   Winch coordinates

Name	X [m]	Y [m]	Z [m]	Symbol orientation
Winch 1	86	5.8	20	0
Winch 2	-79	9.7	18.5	0
Winch 3	78	-1	19.2	0
Winch 4	-82	4.4	18.5	0
Winch 5	48	4.3	18.5	0
Winch 6	-53	5.5	18.5	0

6. Leave the *Chain data* and *Fender connection data* tabs as default.
7. The final *Vessel view* tab shows the position of the fairleads and winches with respect to the mean deck level (defined by the deck plane height value) (red line) and water plane (defined by draft) of the vessel (orange). Set the *Deck plane height* to 15 [m]. Note that only the mean deck level has been shown here and that simultaneous visualisation of multiple deck levels is not supported. Often, the far most fore/aft fairleads are located on higher decks. In this case, they will thus appear as located outside the red line.
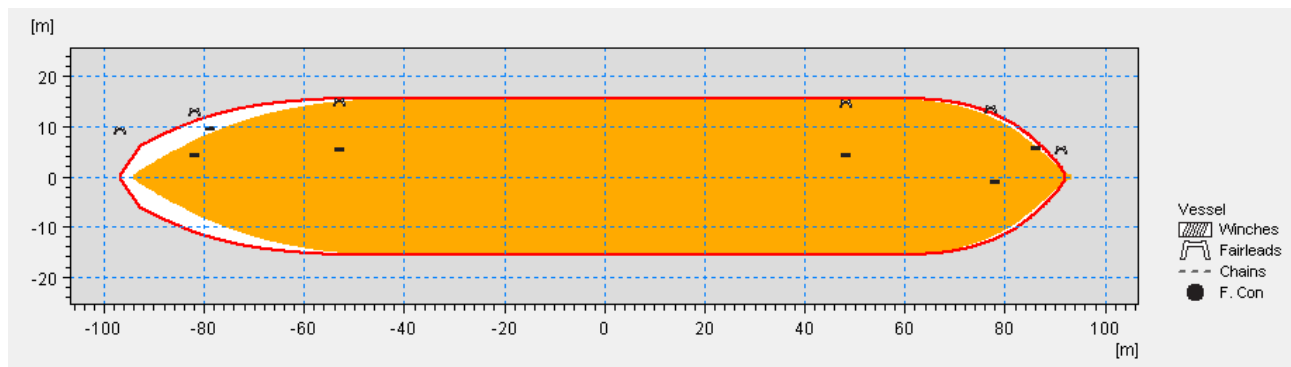
Figure 3-17   Vessel view window showing waterplane area (orange), mean deck level (red) and position of fairleads and winches

Dialog: Port Data
The Port data tab allows the user to position and visualise the bollards and fenders within the model domain.

1. Click on the *Bollard data* tab. Set the *Map Projection* to MGA 56. Copy and paste the following data into the bollard table.

Table 3.4   Bollard coordinates and Symbol orientation

Name	X [m]	Y [m]	Z [m]	Bollard profile
Bollard 1	514385.53	6966835.65	4	Bollard Profile
Bollard 2	514594.93	6966835.10	4	Bollard Profile
Bollard 3	514411.20	6966807.42	1.9	Bollard Profile
Bollard 4	514570.40	6966807.13	1.9	Bollard Profile
Bollard 5	514476.63	6966836.64	4	Bollard Profile
Bollard 6	514505.43	6966836.31	4	Bollard Profile

2. Click on the *Fender data* tab. Set the *Map Projection* to MGA 56. Copy and paste the following data into the fender table.

Table 3.5   Fender coordinates, profile and Symbol orientation

Name	X [m]	Y [m]	Z [m]	Force dir. [deg]
Fender 1	514448.54	6966839.46	2	90
Fender 2	514533.54	6966839.43	2	90
Fender 3	514476.74	6966839.45	2	90
Fender 4	514505.44	6966839.44	2	90

Under *Fender profile*, select the Fender profile created in the Material Profiles. M21 MA allows the user to create several fender profiles from which to select. This also allows different fender types to be used within the same berth layout.

3.9   Setup Mooring System

Now that both vessel and shore based connection points have been specified, it is time to connect the entire mooring system together. The Mooring setup dialog allows the user to position the vessel in the domain correctly, create the mooring setup and visualise the vessel, connection points and bathymetry in the same window.

Dialog: Mooring Setup (Mooring line data tab)

1. Create the mooring arrangement as below

Table 3.6   Mooring setup for M21 MA 2D example

Name	Vessel	Bollard	Fairlead	Winch	Line profile	Line pretension [t]	Tail profile
Line 1	Vessel 1	Bollard 1	Fairlead 1	Winch 1	Line1	10	None
Line 2	Vessel 1	Bollard 2	Fairlead 2	Winch 2	Line1	10	None
Line 3	Vessel 1	Bollard 3	Fairlead 3	Winch 3	Line1	10	None
Line 4	Vessel 1	Bollard 4	Fairlead 4	Winch 4	Line1	10	None
Line 5	Vessel 1	Bollard 5	Fairlead 5	Winch 5	Line1	10	None
Line 6	Vessel 1	Bollard 6	Fairlead 6	Winch 6	Line1	10	None

2. Click on the *Vessel position and mooring lines* tab.

3. Set the map projection to MGA-56. Specify the following vessel coordinates:
   - Easting - 514488.9
   - Northing - 6966856.3

4. Set the vessel rotation to 180 deg.

5. The image shown in Figure 3-18 should show up on your screen showing the berth pocket depth (constant 15 m), vessel and shore based connection points and the vessel position.
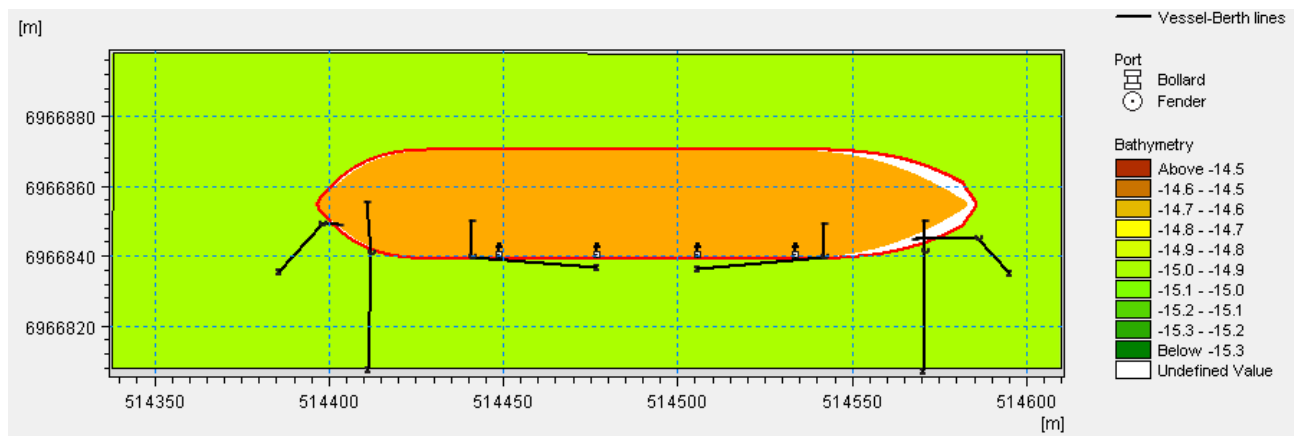
Figure 3-18   Moored vessel in port domain with mooring system connected

Note: The [icon] icon can be used to delete lines and, likewise, mooring lines can be attached using the [icon] icon.

3.10 Environmental Forcings

Dialog: Environmental conditions

This dialog defines the environmental forcing on the vessel i.e. wind, currents, first order waves and 2nd order wave drift forces.

1. Physical Parameters dialog:
   - Set *Density of water* to 1024.61 [kg/m³]
   - Set Density of air to 1.28 [kg/m³]
   - Set Viscosity (water) to 1e-006 [m²/s]
   - Set Viscosity (air) to 1.48e-005 [m²/s]

2. Leave the *Drift Forces* checkbox unticked

For the purposes of this example we will not be using current or wind files as inputs to the M21 MA simulation. Therefore, leave these two dialogs set with '*No current*' and '*No wind*', respectively.

Now that the mooring setup is ready and the environmental forcing has been included, the next step is to find the equilibrium position of the vessel under the proposed mooring system. This is outlined further in the following.

3.11 Vessel Convergence

In practice, to ensure an even load distribution amongst the mooring lines, the tension in the mooring lines is manually adjusted by winching in the ropes until the vessel is idle. In M21 MA, this process is done as an automated convergence process which finds the equilibrium position of the vessel under the proposed mooring system and forcings, except for the influence of first order waves.

To do so, we will setup a convergence simulation. This will have a separate runtime and simulation mode and results from this will be used as an initial condition to the final M21 MA computation. The M21 MA simulation must be run in *convergence mode* prior to running in *Final computation mode* to ensure a reasonable initial vessel displacement, which during the final computation mode leads to more accurate modelling of the moored vessel motions.
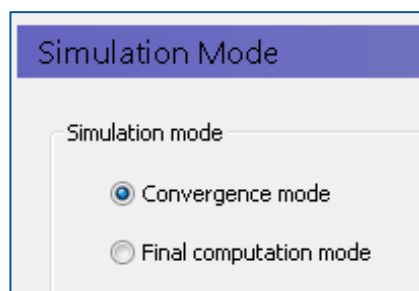
Figure 3-19   Simulation Mode - Convergence

Please follow the steps below to run the simulation in convergence mode:

Dialog: Time
1. Set the *No. of Timesteps* to 500
2. Set the *Time step interval* to 1

3. Set the *Simulation start date* to 27-Feb 2011 10:00:00. This must match the date in the HD simulation.
4. Set the *No. of warm up steps to* 0
5. Set the *No. of warm down steps to* 0

Dialog: Simulation Mode
6. Set the Simulation Mode to *Convergence mode* as shown in Figure 3-19.

Dialog: Convergence Parameters
7. Set the *Max no. iterations* to 100
8. Set the *Line convergence threshold* to 0.5 [%] This sets the simulation to converge when the dynamic line tensions in all lines are within 0.5% of the specified pretension values.

Dialog: Vessel Displacements
9. Apply default parameters

Dialog: Outputs
10. Select the following output files:
    - Convergence_line_pretension_deviation.dfs0
    - Convergence_lforces.dfs0
    - Convergence_Vessel 1_motions.dfs0

In the MIKE ribbon, Go to *Run → Start Simulation →OK*. The simulation will run and a summary will appear in the simulation window at the bottom of the MIKE page. A message *Successful Completion* will appear. Check that the final line forces (*Convergence_lforces.dfs0)* are within 0.5% of the specified value and that the convergence deviation values (*Convergence_line_pretension_deviation.dfs0*) are small.
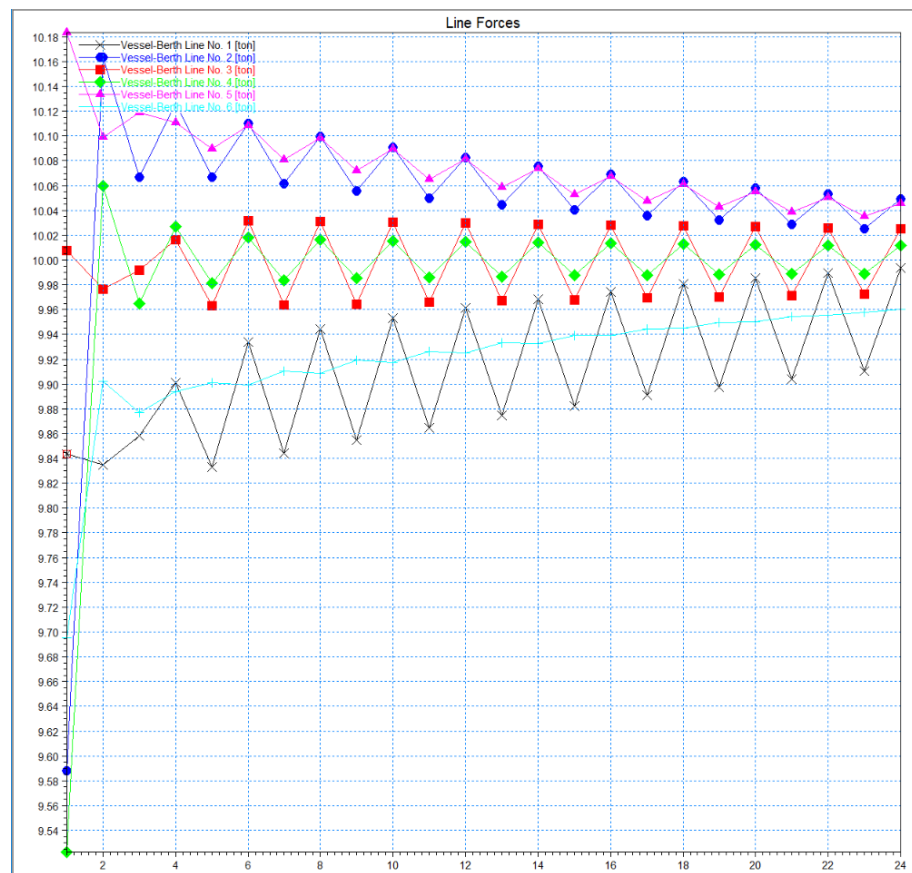
Figure 3-20   Final line pretensions after successful convergence shown in Convergence_lforces.dfs0 file
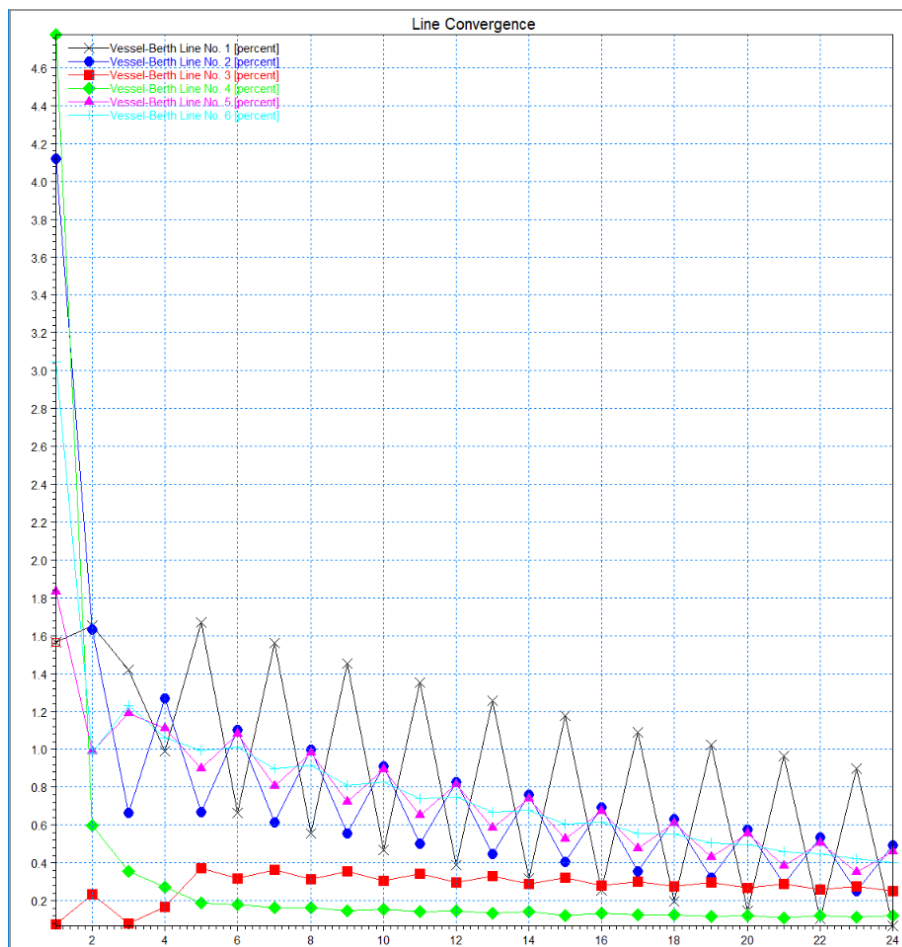
Figure 3-21   Convergence deviation values shown in Convergence_line_pretension_deviation.dfs0 file

Note: If a Convergence Mode run is not executed prior to the final time-domain computation (Final computation mode), the mooring system can cause the vessel to move. This will change the tension in each mooring line. For unbalanced mooring systems, the vessel response can be so significant that tension in some lines disappear completely. The subsequent vessel response in an incident wave/current/wind field will resultantly be significant different to specified design conditions and will most often be regarded as a serious flaw in the model setup.

3.12   Final M21 MA Simulation

Once the vessel has successfully converged, we are now ready to run the final M21 MA simulation. Please follow the steps below.

Dialog: Time
1. Continue using the 2D_Example.m21ma, or alternatively skip the following steps 2-11 and use the prepared setup final_2D_Example.m21ma
2. Set the *No. of Timesteps* to 16380
3. Set the *Time step interval* to 0.25
4. Set the *Simulation start date* to 27-Feb 2011 10:00:00. This must match the date in the HD simulation

5. Set the *No. of warm up steps to* 50
6. Set the *No. of warm down steps to* 0

Dialog: Simulation Mode
7. Set the *Simulation mode to Final computation mode*

Dialog: Wave
8. Go to the *Wave* tab dialog:
   - Click on the dropdown list for *Wave type* and select Wave field
   - Click on the box to include warm up/down:

   - Set Format to Varying in time and domain
   - Set Lower frequency to 0 [Hz]
   - Set Upper frequency to 0.5 [Hz]
   - Set Power (N) to 12. This sets the FFT size (2^N) to 4096

Under Data file and items, click on the *Select* button and select the newly created Grid file. Select the grid (.dfs2) file from Section 3.4 where the time steps were extended to 4096 steps.
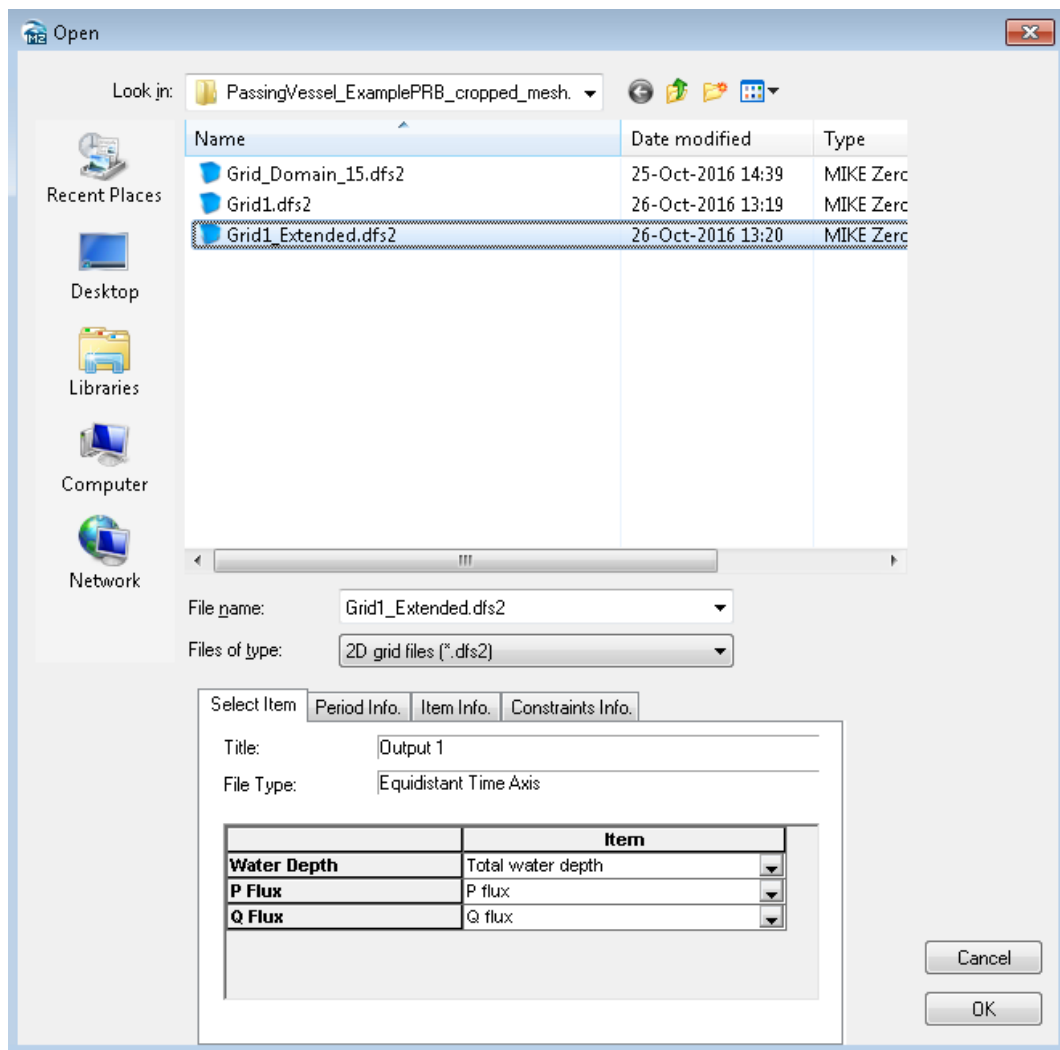
Figure 3-22   Selecting wave input file for M21 MA simulation

9.   Save the setup file.

If you have followed the steps shown in the guide, the wave file should have the correct time and items and a green tick symbol will appear in the GUI tree in the M21 MA editor. If this does not appear, go to the *Constraints Info* tab shown in Figure 3-22. This will show you exactly which item/setting is causing M21 MA to reject the input file.

Dialog: Vessel Displacements

Click on the 'load' button to load the initial (equilibrium) position of the vessel, obtained from the previous convergence mode simulation. You should see the numbers load automatically in the table as per Figure 3-23.

Vessel No.	Name	Surge [m]	Sway [m]	Heave [m]	Roll [deg]	Pitch [deg]	Yaw [deg]
1	Vessel 1	0.284232944	-0.0075321160	-0.0023951306	-0.1773176193	-9.8359225376	-0.0218567345

Figure 3-23   Initial displacement of moored vessel

Dialog: Outputs

10.   Select the following outputs:
- Lpositions.dfs0
- fforces.dfs0
- fforces.dfs0
- bforces.dfs0
- Vessel 1_motions.dfs0
- Vessel 1_mforces.dfs0
- Vessel 1_selev.dfs0
- Vessel 1_diff_forces.dfs0

11.   Set the vessel motion type to 'Relative to displaced position'.

12.   Go to *Run → Start Simulation* and click OK. A summary of the run will appear in the window and a 'Successful Completion' should appear.

13.   Check either the log file (created automatically in the .m21ma setup folder) or the Simulation window, to make sure that neither the fender compression nor mooring line tensions exceeded acceptable levels.

14.   Open the newly created *Vessel1_motions.dfs0* file and plot the *Surge, Sway and Yaw motions.* You should see that the vessel is stable until around 11:05 when the passing vessel generated displacement waves arrive at the berth and induce a large surge motion of the moored vessel. Motions in the other degrees of freedom are relatively small. Your plotted surge, sway and yaw motions should appear as shown in Figure 3-24.
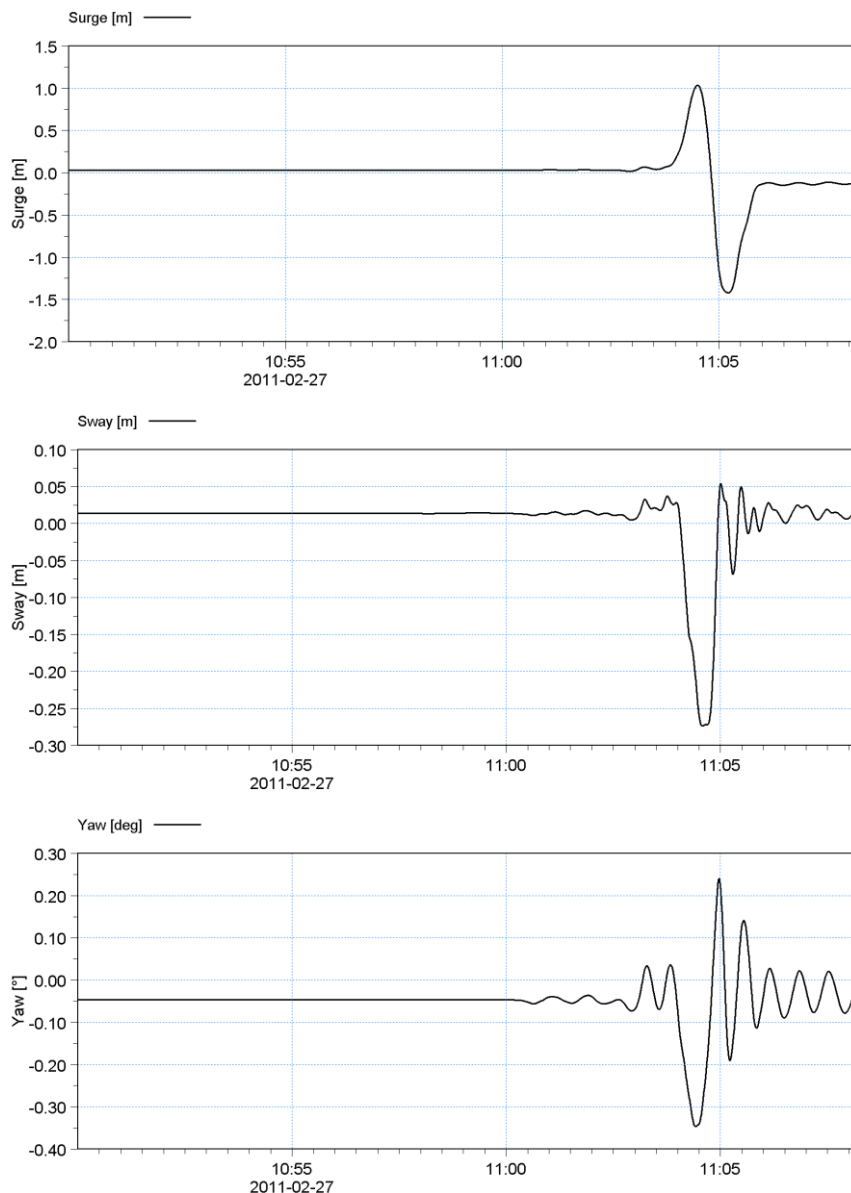
Figure 3-24   Passing vessel induced moored vessel motions - surge, sway and yaw

3.13   Visualise Vessel Movements in 3D

It is possible to visualise the vessel movements by using MIKE Animator Plus. MIKE Animator Plus is a digital video production studio that enables you to turn e.g. MIKE 21 model results into 3D video presentations.

The results from the MIKE 21 Mooring Analysis module include a time series file with vessel motions. By using information from this together with a 3D solid representing the vessel, and data files with bathymetry and time-varying water level it is possible to use MIKE Animator Plus to create an informative animation of the vessel movements for the given mooring situation.

The following sections describe how to setup MIKE Animator Plus to visualise the findings from the MIKE 21 Mooring Analysis model.

Note that you can still visualise the data if you do not have a license for MIKE Animator Plus, however in that case a DHI icon will be displayed in front in the graphics window.

All references to file locations in the following are given with reference to the top folder **.\MIKE_21\Maritime\MooringAnalysis\2D Wave Forcing**.

3.13.1 Generate input files for MIKE Animator Plus

Before opening MIKE Animator Plus you must ensure the input data files fits the purpose. This example displays the results for 9 minutes at the end of the simulation.
For better overview in the MIKE Animator Plus setup, new files are saved to the folder dedicated the MIKE Animator Plus setup: \MAPlus.

Domain area

The wave forcing simulation is carried out using the MGA-56 map projection, and the vessel location is defined as (514488.88 m, 6966854.82 m). However, the vessel motion output from the mooring simulation is always defined relative to the local amidship position of the vessel with an origo of (0,0). In order to show the vessel movements together with the 2D wave forcings, the two types of input must relate to the same coordinate system.

An easy way to do this is to define a local map projection, based on the MGA-56 coordinate system, but with a different origin corresponding to the ships local origo. This can be achieved by modifying the false easting and northing as follows:
New False Easting   = Old false Easting – Easting ship
New False Northing = Old False Northing – Northing Ship

The file **MGA56_Local.prj** reflects the local coordinate system described above and must be imported into MIKE Zero before further use:

1. Open MIKE Zero and select File → Options → Edit Map Projections …

2. Select Import Projection File…'

3. Select the file **MGA56_Local.prj** and press OK.

Now you can use this map projection in files for the MIKE Animator Plus setup.

Surface elevation

The surface elevation and bathymetry for the MIKE Animator Plus setup are derived from the previously calculated wave forcing:

1. Go to MIKE Zero and open the file 'Grid1_Extended.dfs2'.

2. Select Tools → Crop…

3. Select the item 'Total water depth' and set the start date as **2011/02/27 10:59.16**', corresponding to the last 540 time steps. Save as Waterlevel_Local.dfs2 and press OK.

Figure 3-25   Cropping data to relevant extent

4.   Select Edit → Time steps…

5.   Change the Start Time to **2011/02/27 10:00:00** and press OK. (note that this setting is shifted for the animation only).

6.   Select Edit → Items…

7.   Change the Item description and Item type to **Water level** and press OK.

8.   Select Tools → Calculator…

9.   Specify the expression **s = s-15** in order to derive the water surface. Go to the 'Sub-Set' tab and make sure to select all time steps. Then press OK.

10.   Select Edit → Geographical Information…

11.   Select the newly imported map projection **MGA56-Local**, press Open and OK.

12.   In the pop-up window, select to 'Keep "Geographical coordinates" and derive "Map projection coordinates". This will result in a new origin for the 2D data as (-149.88 m, -45.82 m) in the local coordinate system.
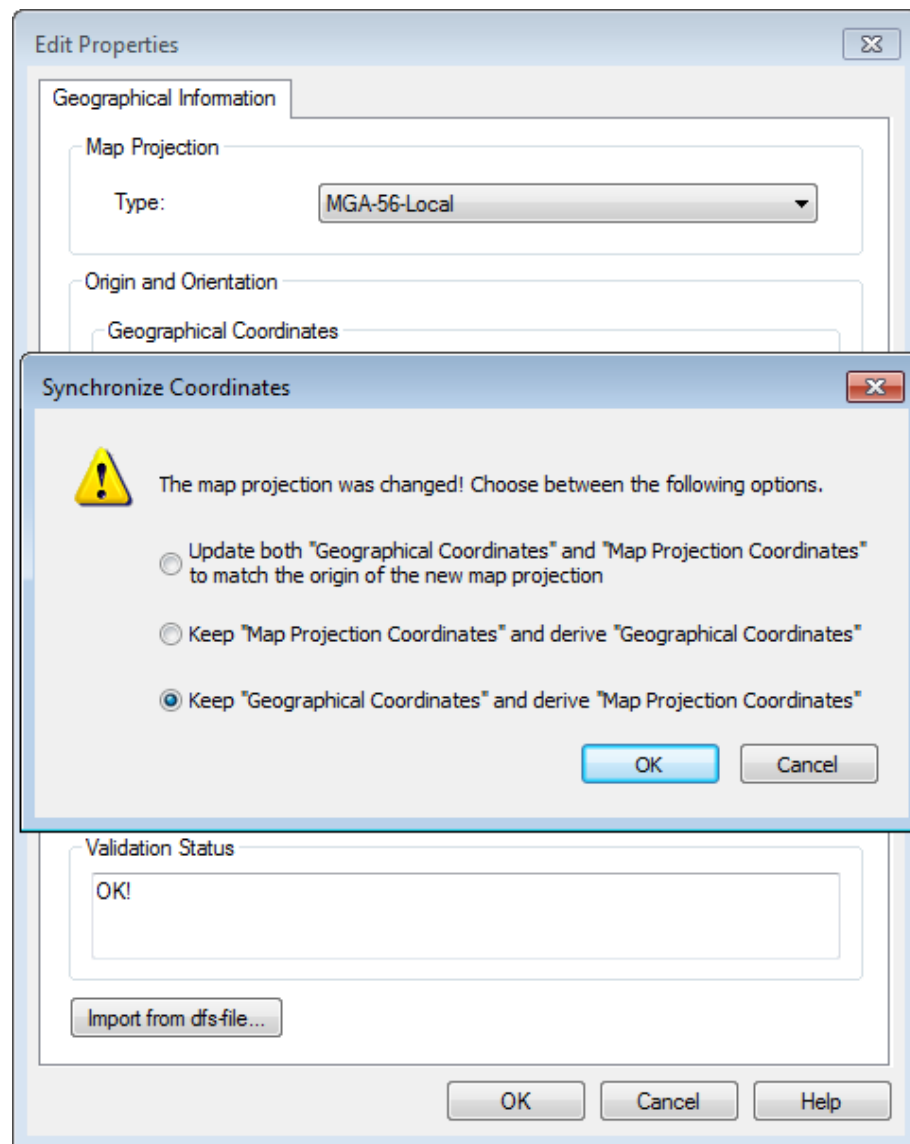
Figure 3-26   Changing map projection

13. Save the resulting dfs2 file in the following location: .\MAPlus**Waterlevel_Local.dfs2**

Note that the bathymetry is contained in the file as a static item.

Vessel Movements

In order to visualise the movements of the vessel in MIKE Animator Plus a 3Dsolid (.3ds) can be linked to a trajectory file. In this example the 3Dsolid **Container.3d**s is selected as a representative for the vessel.
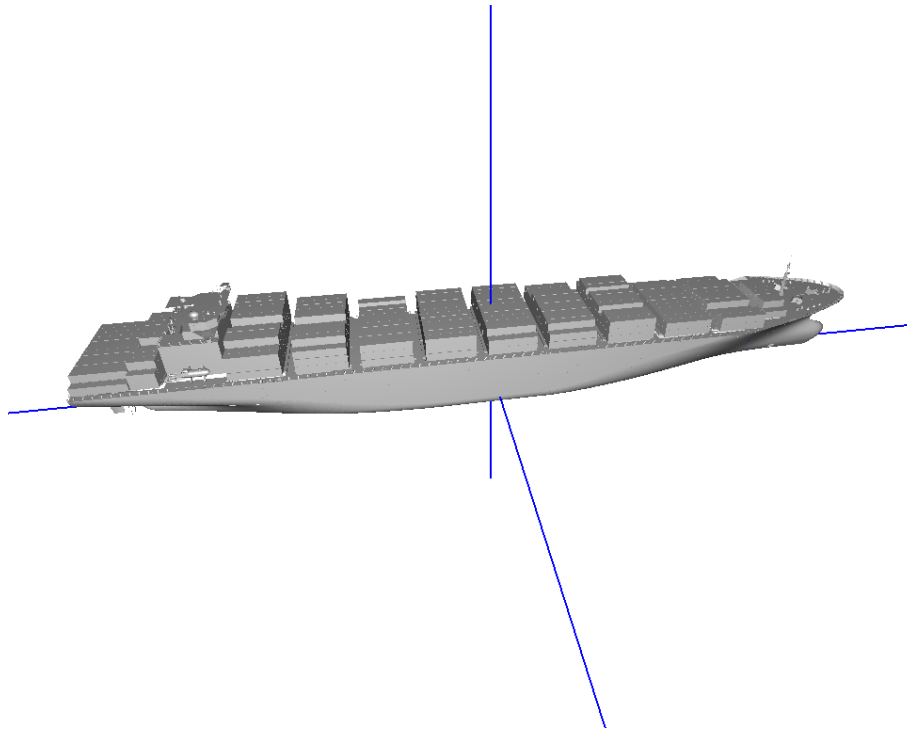
Figure 3-27   Container.3ds as displayed in MIKE Animator Plus.
               The cross shows the origin of the 3D solid in its local coordinate system

The MIKE 21 Mooring Analysis simulation creates an output file with resulting vessel movements relative to the origin (0,0). As the selected 3D solid is defined with a local Z-level origin at the bottom of the ship, it is necessary to modify the vessel motion values to position the vessel correctly with the water level. Also, the item descriptions need to be modified in order to meet the standard for trajectory files in MIKE Animator Plus. For simplicity the time step in the file is modified to match the water level, i.e. 1 second.

1.   In MIKE Zero create a new MIKE Zero Toolbox file and save it as 'GenerateTSData.mzt'.

2.   Expand the 'Time Series' section and double click on 'Interpolate Time Series'.
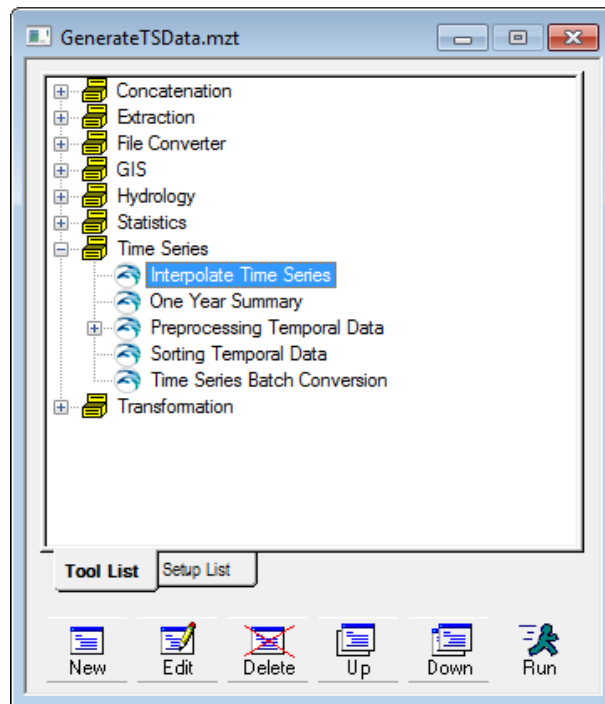
Figure 3-28   MIKE Zero Toolbox: Interpolating Time Series

3. Create a new setup called 'MAPlus Vessel Movements file'.

4. Select the output file .\Setups\Final 2D_Example.m21ma - Result Files**Vessel 1_motions.dfs0'** and select all six items in the file. Press OK and Next.

5. Define 'Start time' as **27/02/2011 10:59.16**' and 'End time' as **27/02/2011 11:08:15**. Set the time Interval to 1 second. Press Next.

6. Specify the new output file as .\MAPlus**Vessel1_motions_MAPlus.dfs0**

7. Press next and click on 'Execute' to run.

8. At successful completion, a pop up box saying 'Interpolation succeeded' will show up. Click 'OK' and 'Finish' and save the setup as 'GenerateTSData.mzt'.

Now it is necessary to modify the dfs0 file to match the requirements for a trajectory file for this MIKE Animator Plus setup.

9. Open **Vessel1_motions_MAPlus.dfs0** in Time Series Editor

10. Select Edit → Properties…

11. Modify the Start time to **27/02/2011 10:00:00** and press OK to match the arbitrary start time for the animation.

12. Subtract 10 m from the third item (Heave) to account for the Draft of the vessel, see Figure 3-29. Press OK and Save
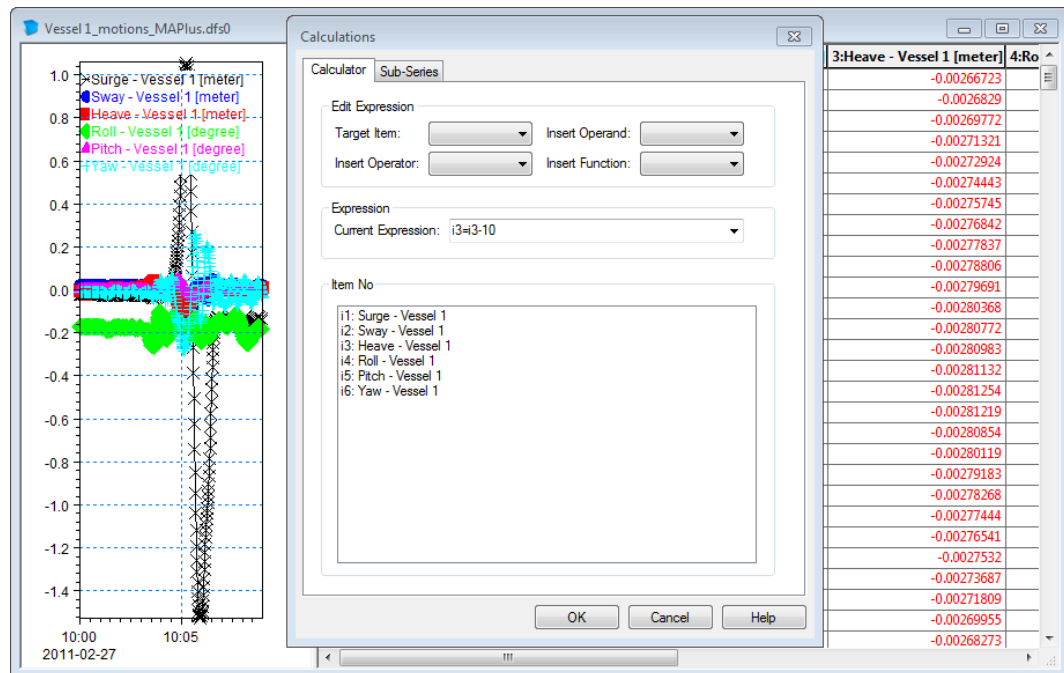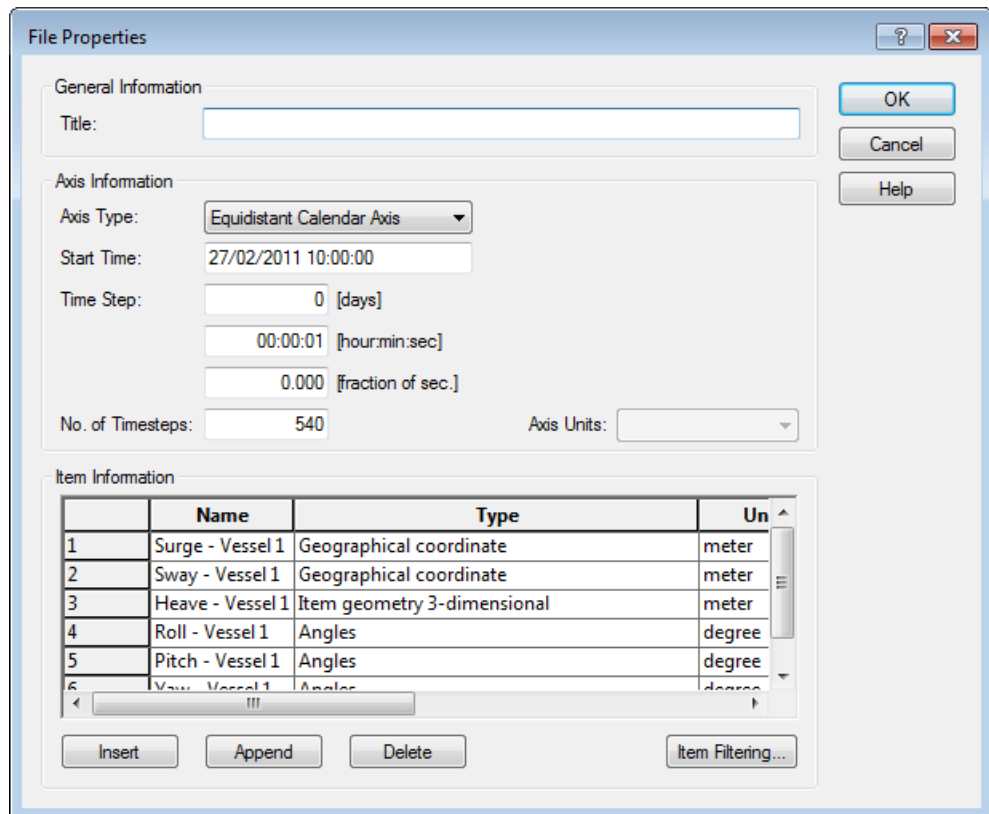
Figure 3-29   Modifying position of vessel to accommodate local coordinate system of 3Dsolid.

13. Change the item descriptions to meet MIKE Animator Plus requirements as follows:
    a. Click Edit → Properties …
    b. Change item type for item 1 (Surge) to be 'Geographical coordinate'
    c. Change item type for item 2 (Sway) to be 'Geographical coordinate'
    d. Change item type for item 3 (Heave) to be 'Item geometry 3-dimensional'
    e. Leave item types for item 4 to item 6 as is (Angles)
    Click OK and save the file.

Figure 3-30   Valid item descriptions in trajectory file for MIKE Animator Plus

3.13.2 Create a MIKE Animator Plus setup

The set of steps below outline how to create a new MIKE Animator Plus setup.

The first task is to create a setup to display the initial conditions. After that the MIKE Animator Plus setup can be extended to make an animation by including time varying data. Finally an additional scene is created to show the overall wave forcings.

Initial conditions

1. Open the MIKE Animator Plus editor. It can be located by searching for it in the Windows start menu.

2. Click 'File' → 'Insert' → 'Scene' and select the wave forcings file .\MAPlus**Waterlevel_Local.dfs2**

3. Highlight 'MIKE 21 file – Waterlevel_Local.dfs2' in the property tree and double-click variable 'Bathymetry' to select to show bathymetry.

4. Click on 'Scene' and select the 'Fill' tab. Click on 'Fountain fill' to change the colour of the background.

5. Select the 'View' tab and change the Focalpoint parameters to (0,0,0) and the Viewpoint parameters to (100,150,40) as shown in Figure 3-31.
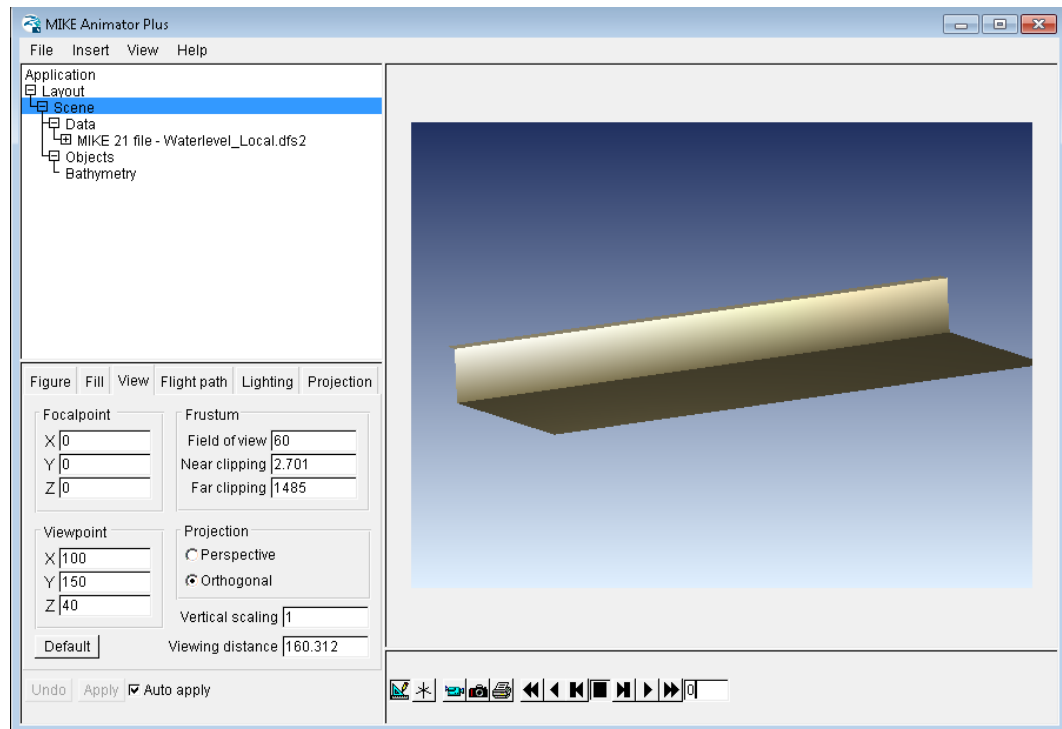
Figure 3-31   Defining View settings in MIKE Animator Plus

6.   Highlight 'MIKE 21 file – Waterlevel_Local.dfs2' in the property tree, right-click variable 'Waterlevel', and select 'Surface - Mapped to magnitude' to show water level.

7.   Select the 'Palette' tab and right-click 'Palette'.

8.   From the drop-down menu select 'Preset' and 'Water', see Figure 3-32.
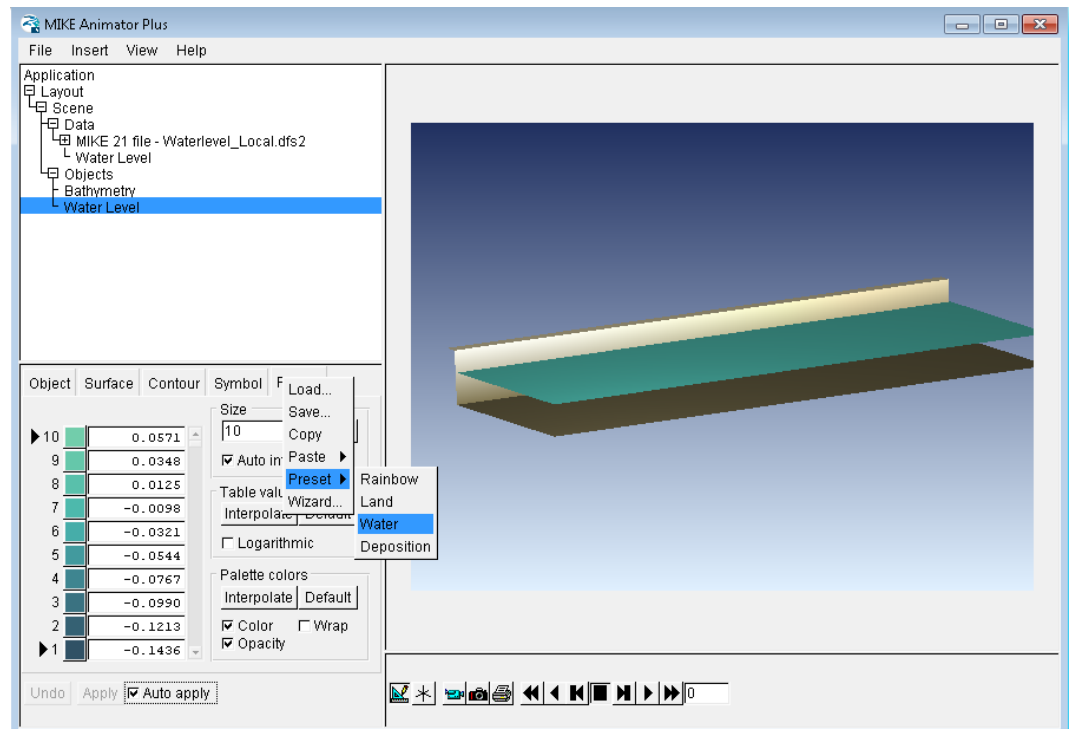
Figure 3-32   Adding data to existing scene

9.   'Click 'File' → 'Save Layout' and save the setup to the new file
       .\MAPlus**Vessel_motions.lyt**

10.   Right-click 'Scene' and select 'Load data...'

11.   Select the 3Dsolids file .\MAPlus**Container.3ds** and double-click the 'Solids Model'
       variable to display the solids file in the graphics window.

12.   Now scale the Solids Model to resemble the outline of the vessel used in the Mooring
       Analysis simulations. Set the variables as follows:
       a.   Uncheck 'Uniform Scaling'
       b.   Set X extent to 193.43 m
       c.   Set Y extent to 31.424 m
       Set Z extent to 28 m (twice the deck height).

13.   To position the Solids Model correctly in the vertical for the initial time step, go to the
       'Solids' tab and change the translation in the Z-domain to -10 m.

14.   Click 'File' → 'Save Layout' and save the setup.
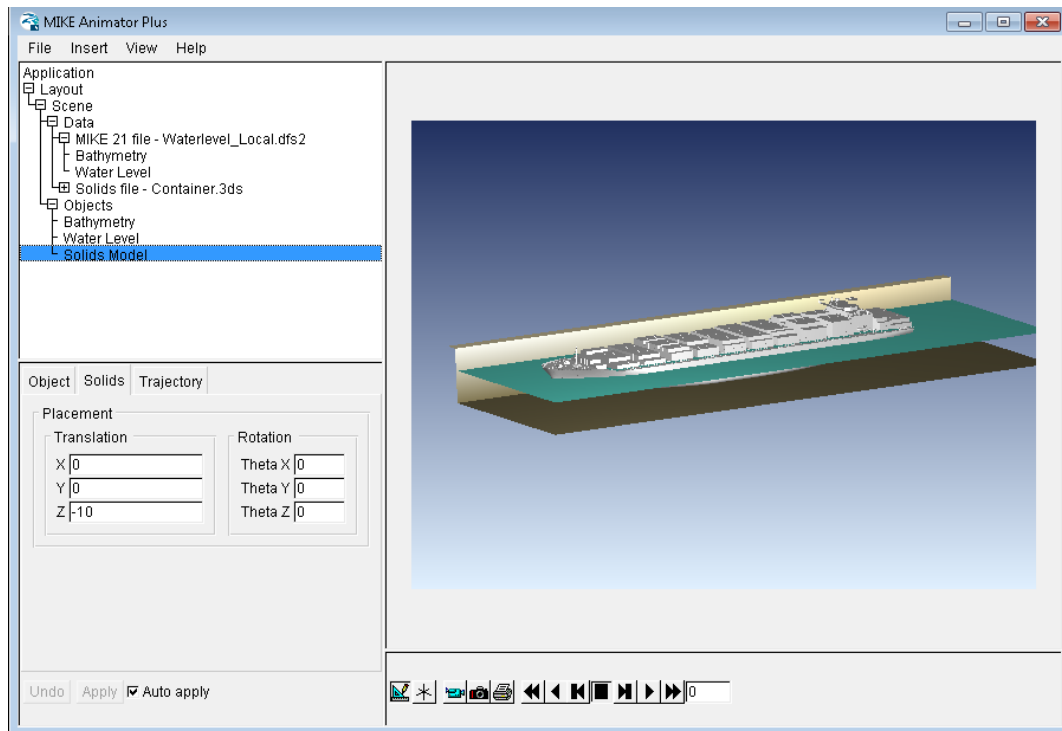
Figure 3-33   View of Initial conditions after scaling and repositioning 3DS solid

Animation

1. Click 'Insert' → 'Clock' to add a clock to the graphics window.

2. In the 'Fill' tab un-check 'Solid fill'.

3. Click-and-drag the Clock object in the graphics-window to a position in the upper left corner.

4. In the 'Clock' tab select the Source Figure to be 'Figure 1 Scene' and the Source File to be 'File 1 – MIKE 21 file – Waterlevel_Local.dfs2'

   (Now the clock will indicate the time stamp in the file while the animation is running).
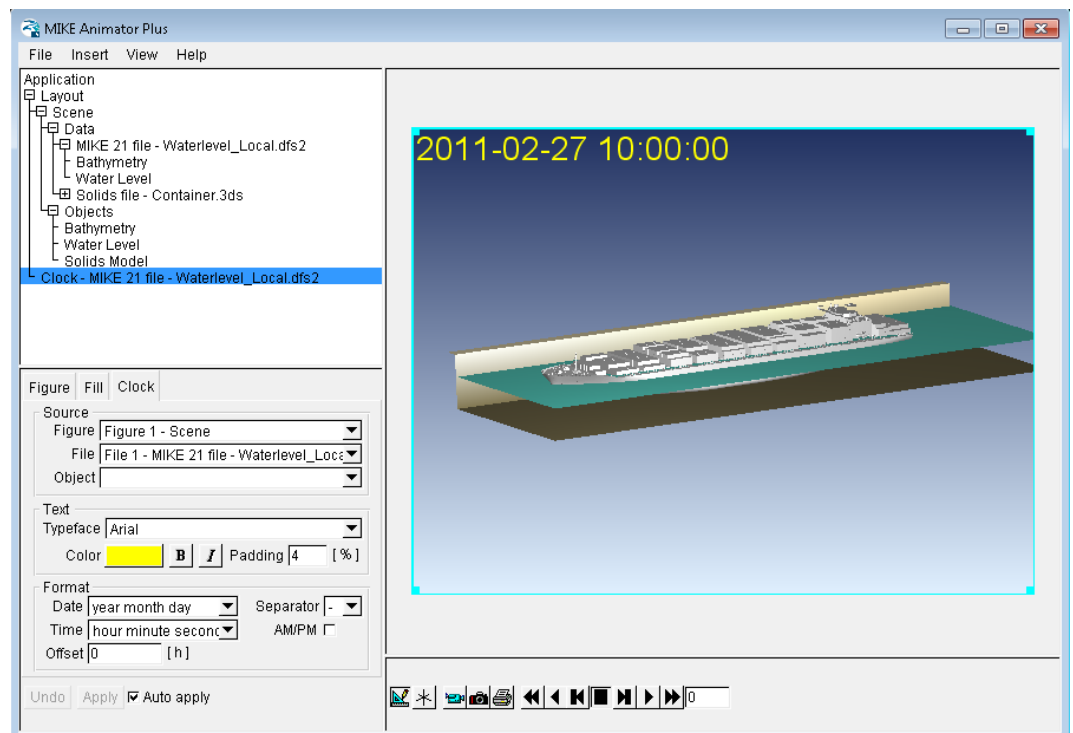
Figure 3-34   Clock showing instant time step in animation

5.   Next click the Solids Model Object and select the 'Trajectory' tab.

6.   Define the Movement to '6 parameters' and select the Source file
      .\MAPlus**Vessel1_motions_MAPlus.dfs0**
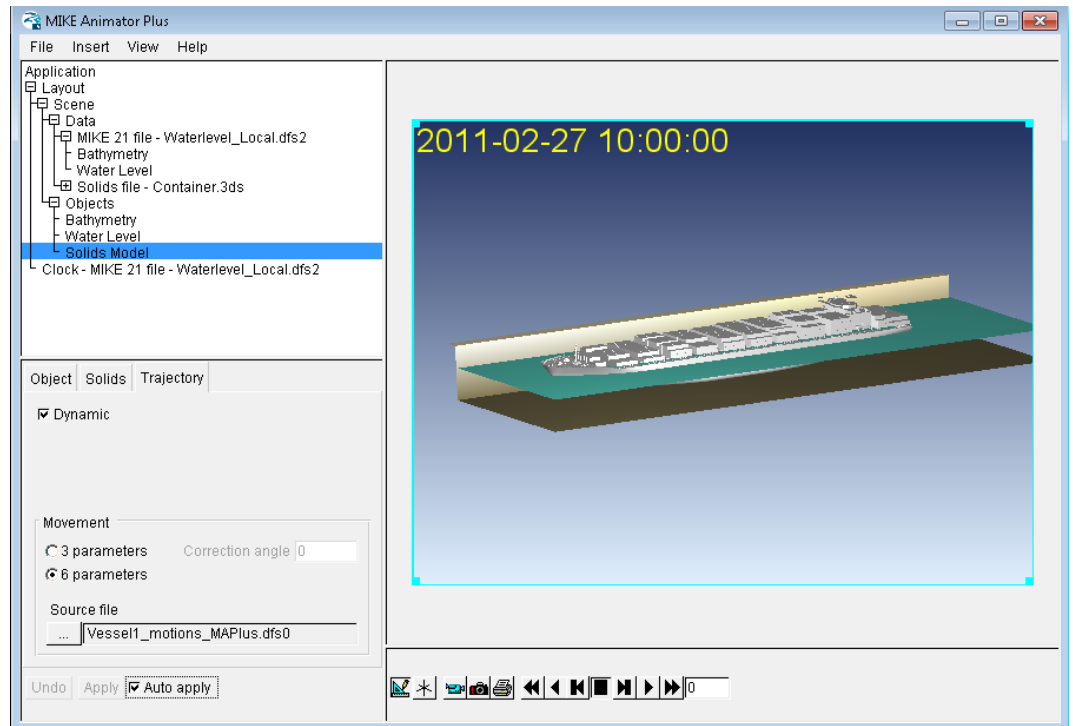
Figure 3-35   Specifying data file for vessel movement

7. Click 'File' → 'Save Layout' and save the setup (as **Vessel_Motions.lyt**)

8. Click the 'Play' button to watch the vessel move and water level change in time.

Combining Scenes in Animation

It is possible to visualise from where the wave forcing originate from by adding an additional scene to the MIKE Animator Plus view that shows the passing vessel in a larger domain.

1. Click 'File' → 'Insert' → 'Scene' and select the calculated wave forcings file **.**\HD\Setup\PassingVessel_Example.m21fm - Result Files\ **Vessel_Induced_Flux.dfsu**

2. Highlight 'MIKE 21/3 FM file – Vessel_Induced_Flux.dfsu' in the property tree and right-click variable 'Total water depth' to create a 'Surface - Mapped to magnitude'. (This will show the whole domain from above. Note the suppression of the water level indicating the passing vessel).
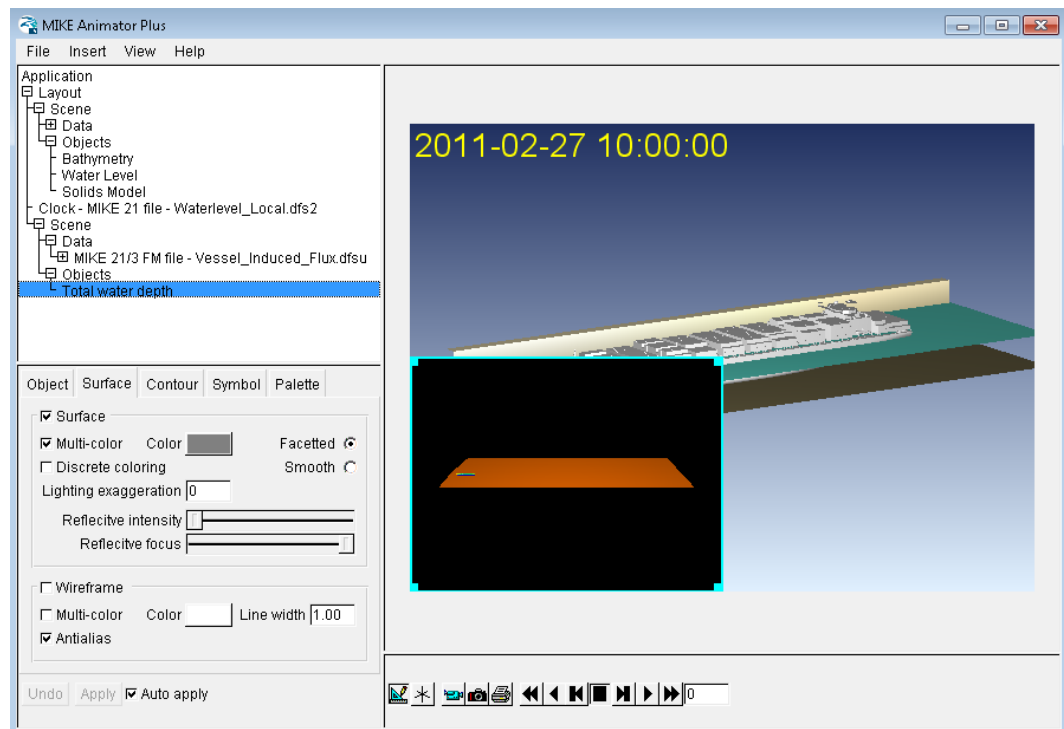
Figure 3-36   Added new scene, after selecting variable 'Total water depth'

3. Select 'Scene mode' to edit the layout of the new scene.

4. Select the new scene and click the 'Fill' tab to un-check 'Solid fill'.

5. Set the Focalpoint to the location of the moored vessel in the MGA-56 domain (514488.88, 6966854.82, 15).

6. Define the Viewpoint such that the Focalpoint indicator is shown in the top of the domain and the initial position of the passing vessel to the left.
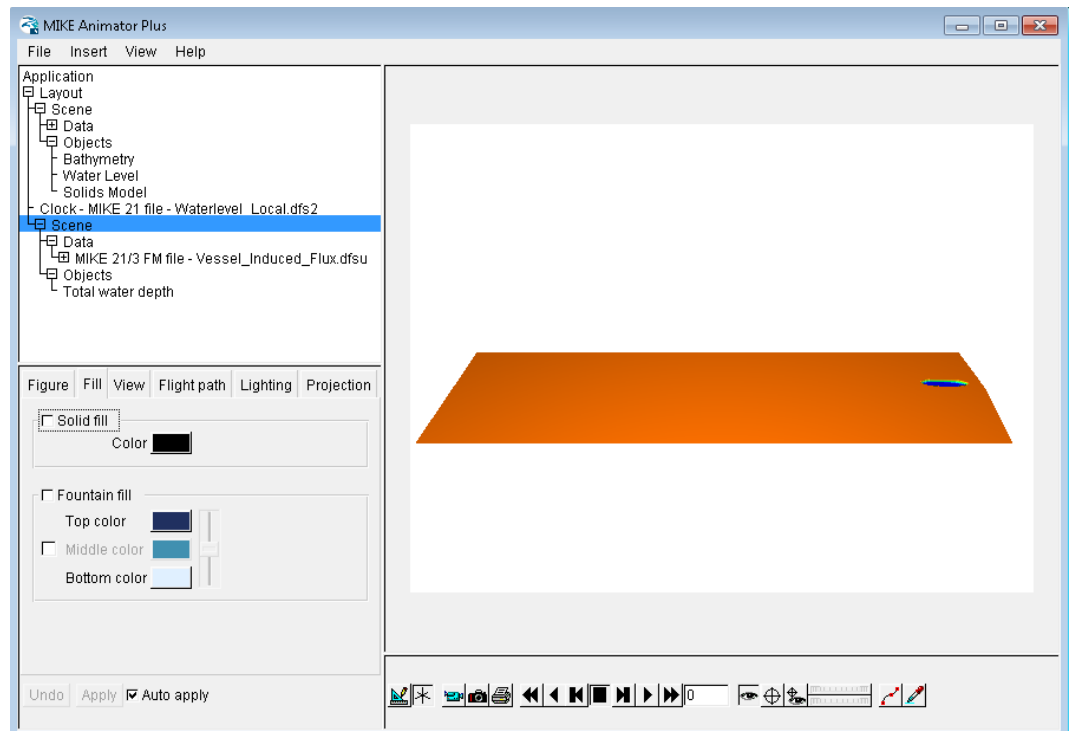
Figure 3-37   Defined Focal point and Added new scene, after selecting variable 'Total water depth'

7.   Select the variable 'Total water depth' and click on 'Palette'.

8.   Define the palette by blue colours to reflect the detailed variation around MWL and a darker value to show passing vessel. Save the palette file for future use.
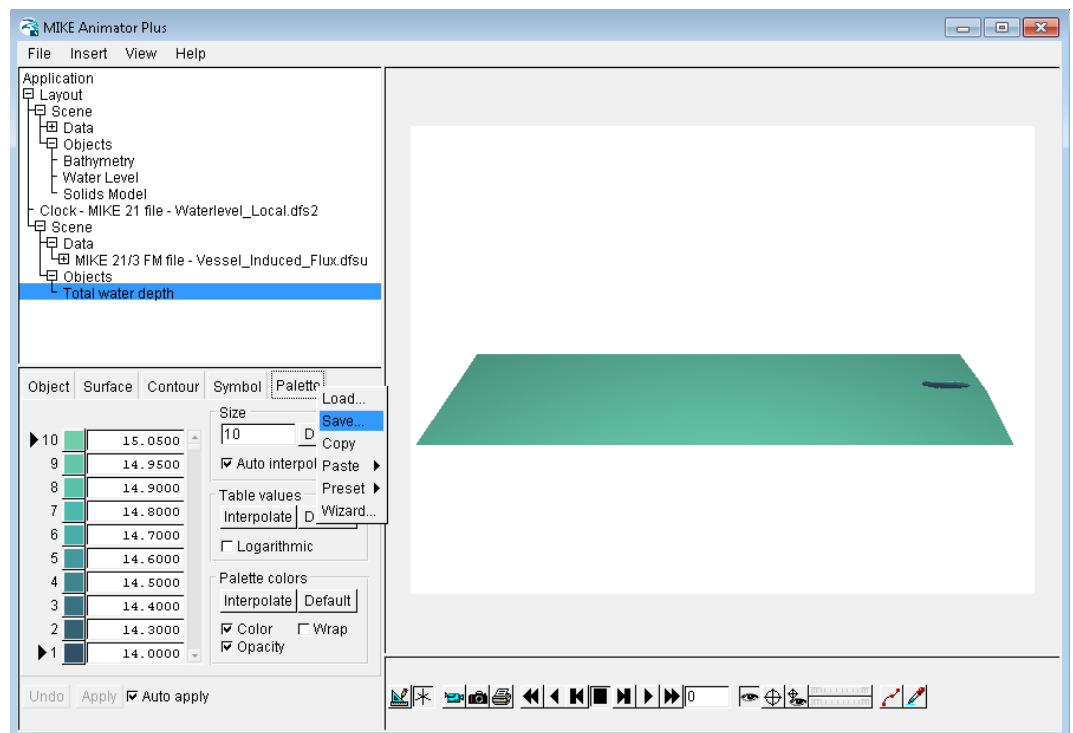
Figure 3-38   Defined colour scale to represent passing vessel and total water depth

9. Next insert the 3Dsolids file .\MAPlus**Container.3ds** into the scene the same way as described previously. Scale the vessel in similar way but set the location of the vessel to (514488.88, 6966854.82, 5) to represent the location in the MGA-56 domain.
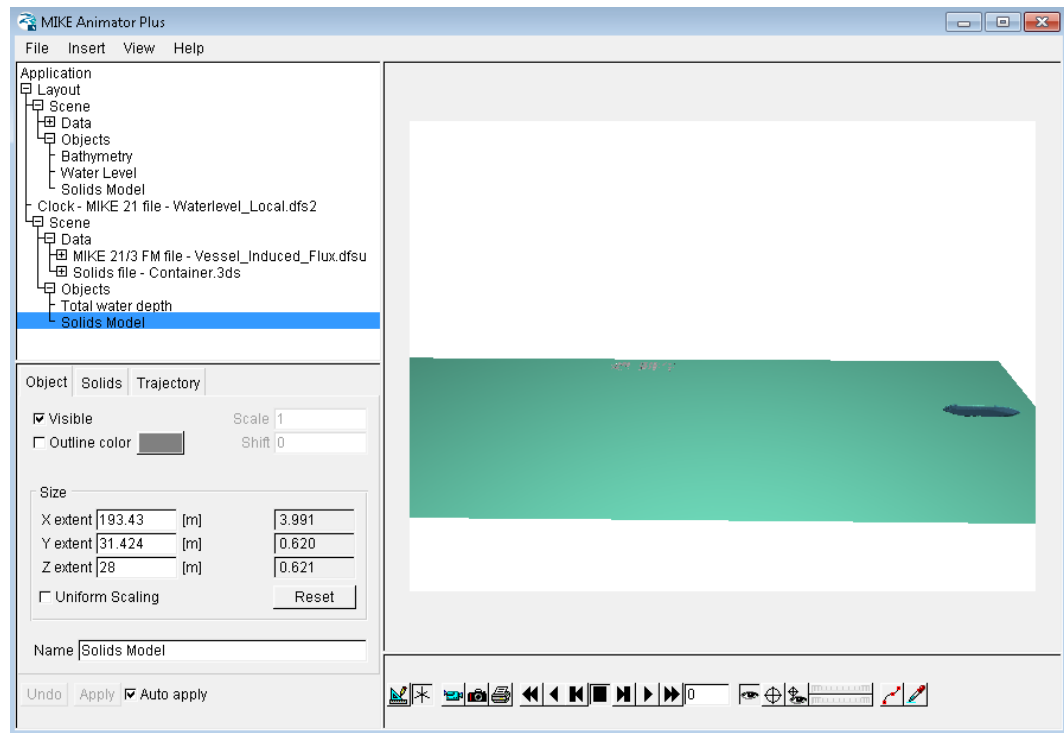
Figure 3-39   Location of moored vessel in overview scene

10. Click 'File' → 'Save Layout' and save the setup under a new name: **Vessel_Motions_CombinedScenes.lyt**

11. Click 'Layout mode' to view both scenes together along with the clock. Drag and rescale scenes with the mouse weel to place scenes as required. Change the Viewpoint by changing the Viewing distance to zoom into the respective domains.

12. To insert a Colour legend for the water surface click 'File' → 'Insert' → 'Legend'.

13. Select the Palette source as 'Scene 0 – File 0 – Waterlevel' and accept the default location of the palette bar to the left in the view.

14. To match the colour legend for the two scenes, click the 'Waterlevel' object and right-click to Load the saved water depth palette. Change the palette values from representing total water depth to represent surface elevation by subtracting the still water depth, 15 m, from the original values.
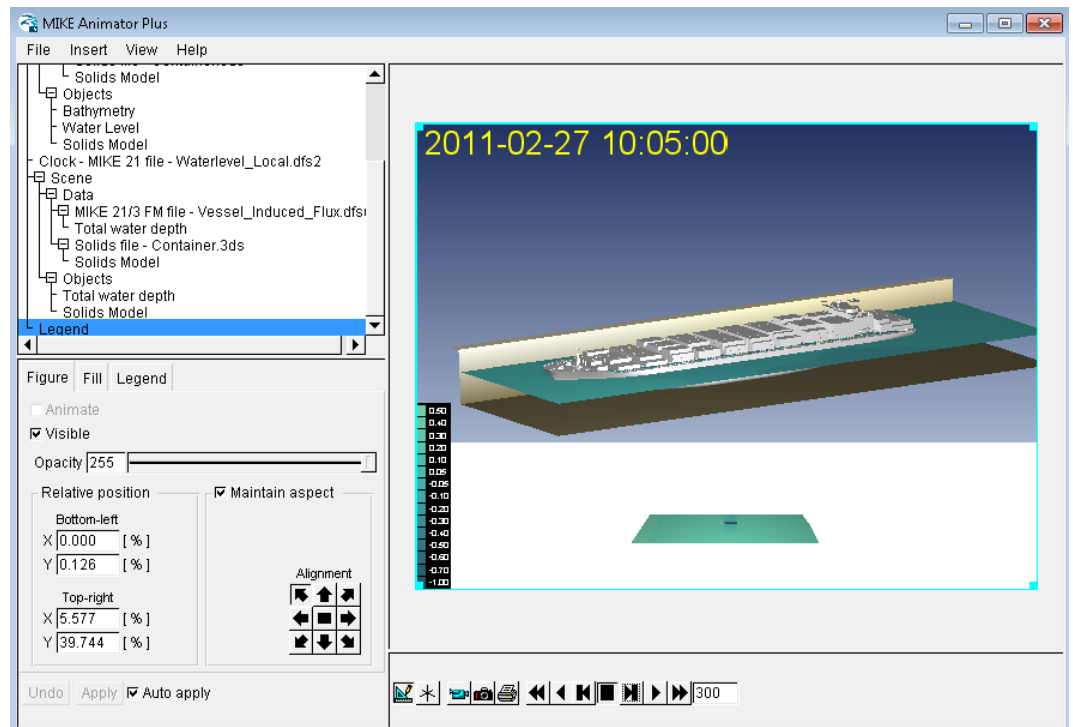
Figure 3-40   Combined overview of scenes, clock and Legend

15.   Click 'File' → 'Save Layout' and save the setup.

Click the 'Play' button to watch the vessel move and water level change in time. Because of the very small movements, the moored vessel is set to be static in the scene showing the overview.

4 Example: Multibody Application

The step-by-step example in this Chapter demonstrates how to set up a MIKE 21 Mooring Analysis simulation for multibody application. The example calculates vessel motions for two adjacent vessels that are moored to each other.

4.1 Step 1 - Launch Frequency Response Calculator

As per the previous sections, the first step in this analysis is to create an FRC setup to calculate the frequency determined response of the two respective vessels.

Create a new instance of the (.fresponse) editor or open the file **Multibody.fresponse** in the folder <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\ MooringAnalysis\MultiBody\FRC\Setup\

If you create a new (.fresponse) editor, save the file as 'Multibody.fresponse'.

4.2 Step 2 - Populate the FRC Setup with Data

The sequential steps for all FRC GUI dialogs are described below:

Dialog: Vessel Configuration

Select **New Vessel** twice and define the vessel names as 'Vessel 1' and 'Vessel 2' respectively. Under common conditions, use the following values: Set **Water depth** to 25.142 m and **Density of water** to 1020 kg/m^3.

Figure 4-1   Vessel configuration dialog

Leave the **Vessel positions** and **System view** tabs for the moment and proceed to the next step.

Dialog: Vessel Instance

For Vessel 1, on the **Vessel Characteristics** tab, load the vessel grid **ogv_v2.grd** that is located here: <example installation folder>\MIKE Zero Projects\MIKE_21\ Maritime\MooringAnalysis\MultiBody\FRC\Inputs\VesselGrid\Bulk_Carrier_1.grd

Enter the vessel parameter values as shown in Table 4.1.

Table 4.1   Moored vessel parameters for Vessel 1

Vessel Scaling (X-Dim)	0.872
Vessel Scaling (Y-Dim)	0.781
Vessel Scaling (Z-Dim)	1
r_{xx}	15.925 [m]
r_{yy}	72.725 [m]
r_{zz}	72.725 [m]
Vertical Center of Gravity (ZG)	11.54 [m]
Draft	9 [m]
Deck plane height	25 [m]

Leave the settings as default on the **Freedom of Motion** tab.

For Vessel 2, on the **Vessel Characteristics** tab, load the vessel grid **TSV_Fine.grd** that is located here: <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\MultiBody\FRC\Inputs\VesselGrid\ Bulk_Carrier_2.grd

Enter the vessel parameter values as shown in Table 4.2.

Table 4.2   Moored vessel parameters for Vessel 2

Vessel Scaling (X-Dim)	0.850
Vessel Scaling (Y-Dim)	0.947
Vessel Scaling (Z-Dim)	1
r_{xx}	15.480 [m]
r_{yy}	57.375 [m]
r_{zz}	57.375 [m]

Vertical Center of Gravity (ZG)	9.51 [m]
Draft	14 [m]
Deck plane height	20 [m]

Leave the settings as default on the **Freedom of Motion** tab.

Dialog: Vessel Configuration

Return to the **Vessel Configuration** dialog and go to the **Vessel positions** tab. Select the position type as **Amidship in domain** and define the respective vessel positions as following:

Vessel 1: X-Position = -8.483 m, Y-Position = -0.0376 m
Vessel 2: X-Position = 0.553 m, Y-Position = 43.0171 m

Click on the **System view** tab to inspect the relative position of the two vessels. If the two vessels are not in full view, the view extent can be adjusted.
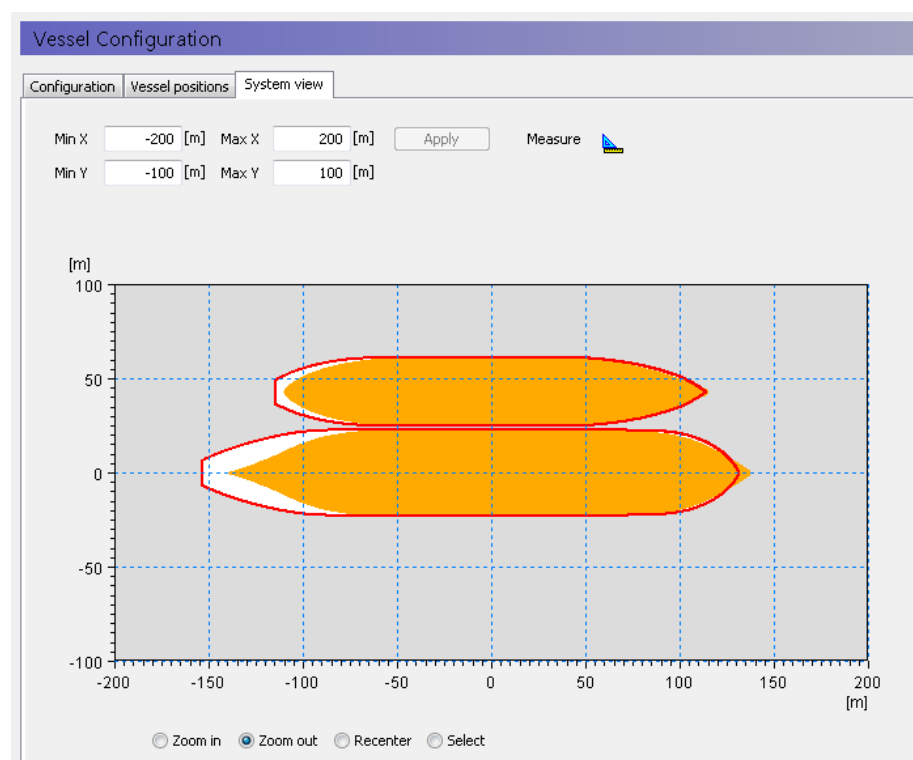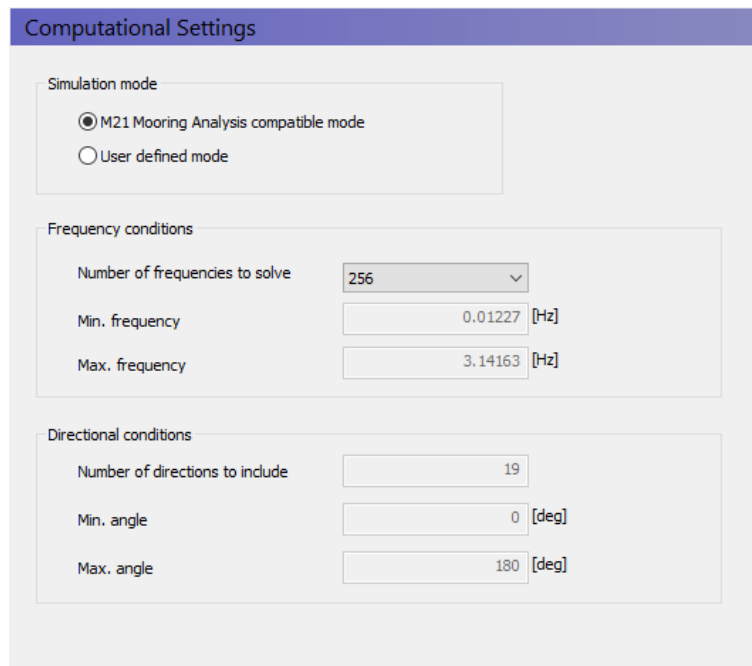
Figure 4-2   Vessel configuration dialog (System view tab)

Dialog: Computational Settings

In this dialog, leave the settings as default: **Simulation mode** = M21 Mooring Analysis compatible mode, and **Number of frequencies to solve** = 256.

Figure 4-3   Computational Settings dialog

Dialog: Wave Drift Forces

For this example, tick 'Include wave drift forces' in order to account for long period components in the wave field.

Figure 4-4   Wave Drift Forces dialog

Dialog: Outputs

All output data and file names are pre-populated with default settings. Do not change any settings here. The **Vessel response** output file will be the basis for all subsequent analysis.

Note that the Response Amplitude Operator (RAO) curves and exciting forces will be calculated and generated for each respective vessel.

Figure 4-5   Outputs dialog

4.3   Step 3 – Run the FRC Simulation

Save the (.fresponse) editor and launch the FRC simulation.

4.4   Step 4 - Launch the MIKE 21 Mooring Analysis

The next step is to create a M21 Mooring Analysis setup to compute the vessel motions for the two adjacent vessels that are moored to each other.

Create a new instance of the (.m21ma) editor or open the file **Multibody.m21ma** in the folder <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\ MooringAnalysis\MultiBody\M21MA\Setup\
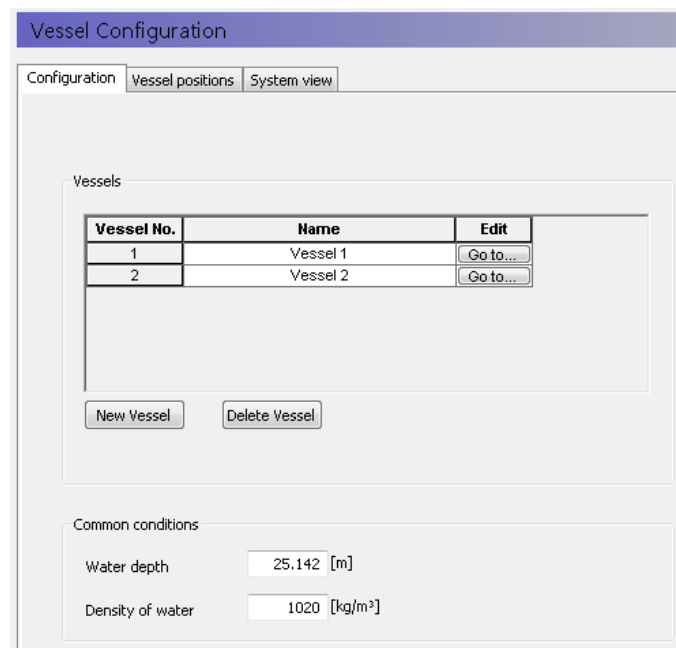
If you create a new (.m21ma) editor, save the file as 'Multibody.m21ma'.
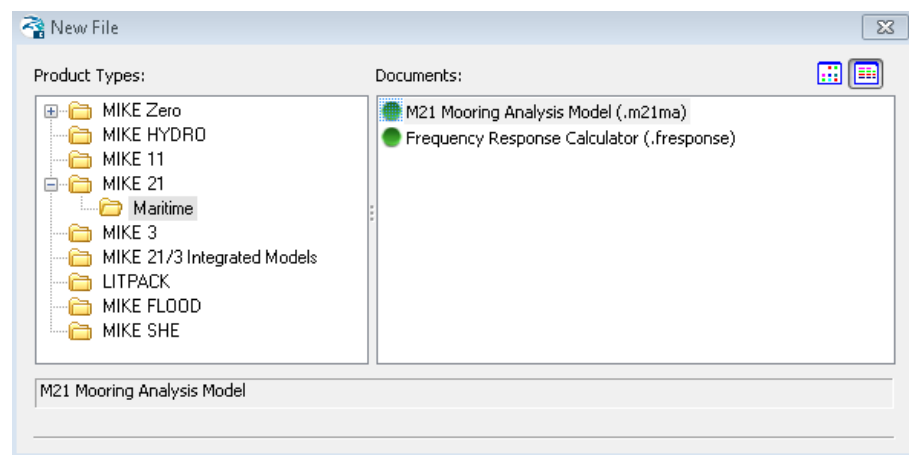
Figure 4-6   Location of MIKE 21 Mooring Analysis setup editor

4.5 Step 5 - Populate the MIKE 21 Mooring Analysis setup

Below all the sequential steps for all required M21 MA GUI dialogs are described in descending order:

Dialog: Domain

The first step in creation of a M21 MA model begins with selecting a **Domain file**. Load the domain file that is located here: <example installation folder>\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Bathy\ **bathy_cropped.dfs2**

Dialog: Time

Here you specify the temporal conditions applying for the time-domain computation (simulation mode = final computation mode).

Make the following:
a. Set the No. of timesteps to 7200
b. Set the timestep interval to 0.25 [s]
c. Set the simulation start date to 01-01-1990 12:30:00 PM
d. Set the No. of warm up time steps to 50
e. Set the No. of warm down time steps to 50

Dialog: Simulation mode

Set this to be Convergence mode.

Using convergence mode means that you prior to running the final time-domain simulation first will attempt to find the initial vessel displacement leading to a uniform tension distribution among all mooring lines.

Subsequently, determining a proper initial vessel displacement is an important pre-processing step to ensure a subsequent stable time-domain analysis.

Dialog: Material Profiles

Expand the node and go to Line profiles.

1. Dialog: Line Profiles
   Insert two Line profiles. Denote the first line 'HMPE' and the second line 'Polyester'.

2. Dialogs: Line Profile instances
   On the **HMPE** instance specify the following:
   a. Total breaking strength = 85 [t]
   b. Failure load = 50 [%]
   c. Linear damping coefficient = 0 [kg/s]
   d. Quadratic damping coefficient = 0 [kg/m]
   Select this line profile: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Lines**\ HMPE.dfs0**

   On the **Polyester** instance specify the following:
   a. Total breaking strength = 120 [t]
   b. Failure load = 50 [%]
   c. Linear damping coefficient = 0 [kg/s]
   d. Quadratic damping coefficient = 0 [kg/m]

Select this line profile: < example installation folder >\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Lines**\
OCIMF_Polyester_Polypropylene_or_mixed_Polyolefins_Broken-In.dfs0**

After this go to the Fender Profiles dialog.

3. Dialog: Fender Profiles
    Insert two Fender profiles. Leave their default names as 'SCN 2000 E1.5' and
    'Pneumatic Fender 3300x6500 0.5bar' respectively.

4. Dialog: Fender Profile instances
    On the **SCN 2000 E1.5** instance specify the following:
    a. Max reaction force = 3320 [kN]
    b. Max deflection = 2 [m]
    c. Failure deflection = 72 [%]
    d. Fender friction coefficient = 0.2
    e. Linear damping coefficient = 0 [kg/s]
    f. Fender thickness = 2.5 [m]
    Select this fender profile: < example installation folder >\MIKE Zero
        Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Fenders\
        Trelleborg_SuperCone_fender.dfs0

    On the **Pneumatic Fender** instance specify the following:
    a. Max reaction force = 3169 [kN]
    b. Max deflection = 3.3 [m]
    c. Failure deflection = 60 [%]
    d. Fender friction coefficient = 0.2
    e. Linear damping coefficient = 0 [kg/s]
    f. Fender thickness = 3.3 [m]
    Select this fender profile: < example installation folder >\MIKE Zero
        Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Fenders\
        Trelleborg_Floating_Pneumatic_Fender.dfs0

5. Dialog: Chain Profiles

    We will skip usage of chains in this setup, so no chains will be specified.

6. Dialog: Bollard Profiles
    Insert one *New profile* and *Go to…* it. Then, specify the following:
    a. Max. horizontal bollard pull force = 2000 [kN]
    b. Max. vertical bollard pull force = 1000 [kN]
    c. Max. total pull force = 2500 [kN]
    d. Max. vertical line angle = 60 [deg]
    e. Safety threshold = 75 [%]

    f. Dialogs: AutoMoor Profiles, DynaMoor Profiles and Port_Fairlead profiles

    We will skip usage of these types of profiles in this setup.

Dialog: Vessels

In this dialog you must select the vessel response file generated from the previously
executed FRC setup:
< example installation folder >\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\FRC\Setup\MultiBody.frespo
nse - Result Files**Vessel_Response.vre**

Once the (.vre) file has been successfully loaded into M21 MA, it will look like as shown in Figure 4-7
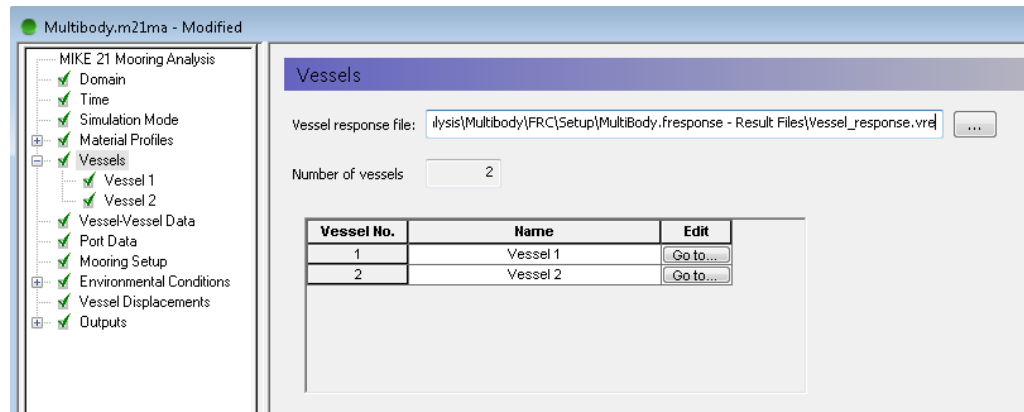
Figure 4-7   Vessels dialog after successful loading of (.vre) file

Clicking 'Go to …' on the vessel instance will take you to the vessel instance dialog.

1.   Dialog: Vessel 1 instance
      On the **Vessel data tab**, specify the data as shown in Table 4.3:

Table 4.3   Vessel 1 data

Longitudinal Area (Wind)	5426.43 [m^2]
Transverse Area (Wind)	863.96 [m^2]
Lpp	285.06 [m]
Loading Condition	0 [%]
Vessel Class	User defined

[1] Lpp refers to the length perpendicular of the vessel

Note that for user defined vessel class, you will have to import drag curves for the vessel:
a.   Select the wind drag curve: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Drag Curves\Wind**BulkCarrier.dfs0**
b.   Select the current drag curve: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Drag Curves\Current**BulkCarrier.dfs0**

Apart from this, you will also see a summary of the vertical attributes from the (.vre) file.

On the **Fairlead data tab**, load in data for the full set of fairlead positions (vessel fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Vessel Spatial Attributes**vessel1_fairleads.xyz**

On the **Winch data tab**, load in data for the full set of winch positions (vessel fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Vessel Spatial Attributes**vessel1_winches.xyz**

On the **Fender connection tab**, load in data for the full set of fenders (vessel fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Vessel Spatial Attributes**vessel1_fenderConnection.xyz**

These values are shown in Table 4.4 to Table 4.6. The X and Y values are relative to amidship of the vessel and the Z values are relative to the keel.

Table 4.4   Fairlead positions relative to amidship at Vessel 1

Name	X [m]	Y [m]	Z [m]
Fairlead 1	-145	7	25
Fairlead 2	-145	-7	25
Fairlead 3	-100	-20	25
Fairlead 4	-60	-22	25
Fairlead 5	60	-22	25
Fairlead 6	110	-20	25
Fairlead 7	130	-10	25
Fairlead 8	138	0	25
Fairlead 9 Multi	-125	15	25
Fairlead 10 Multi	-120	17	25
Fairlead 11 Multi	-115	18	25
Fairlead 12 Multi	-110	19	25
Fairlead 13 Multi	-23	23	25
Fairlead 14 Multi	-20	23	25
Fairlead 15 Multi	20	23	25
Fairlead 16 Multi	23	23	25
Fairlead 17 Multi	110	20.5	25
Fairlead 18 Multi	115	19	25
Fairlead 19 Multi	125	15	25
Fairlead 20 Multi	120	17	25

Table 4.5   Winch positions relative to amidship at Vessel 1

Name	X [m]	Y [m]	Z [m]
Winch 1	-130	7	25
Winch 2	-130	-7	25
Winch 3	-100	7	25
Winch 4	-60	7	25
Winch 5	60	7	25
Winch 6	110	7	25
Winch 7	115	-5	25
Winch 8	123	5	25
Winch 9	-130	9	25
Winch 10	-130	-9	25
Winch 11	-102	7	25
Winch 12	-62	7	25
Winch 13	62	7	25
Winch 14	112	7	25
Winch 15	115	-7	25
Winch 16	123	7	25
Winch 17 Multi	-125	0	25
Winch 18 Multi	-120	2	25
Winch 19 Multi	-115	3	25
Winch 20 Multi	-110	4	25
Winch 21 Multi	-23	8	25
Winch 22 Multi	-20	8	25
Winch 23 Multi	20	8	25
Winch 24 Multi	23	8	25
Winch 25 Multi	110	5.5	25
Winch 26 Multi	115	4	25
Winch 27 Multi	125	0	25
Winch 28 Multi	120	2	25

Table 4.6　　Fender connection positions relative to amidship at Vessel 1

Name	X [m]	Y [m]	Z [m]
Fender connection 1	-62.557	22.61	10.15
Fender connection 2	-22.557	22.62	10.15
Fender connection 3	17.443	22.62	10.15
Fender connection 4	77.443	22.18	10.15

On the **Vessel View tab**, you can see overview of where the full set of fairleads, winches and fender connections are positioned relative to the vessel. It should look as seen in Figure 4-8. Note that some of the vessel attributes may appear to be missing (or not visible) with the full extent view. It is recommended that you zoom in to properly inspect them.
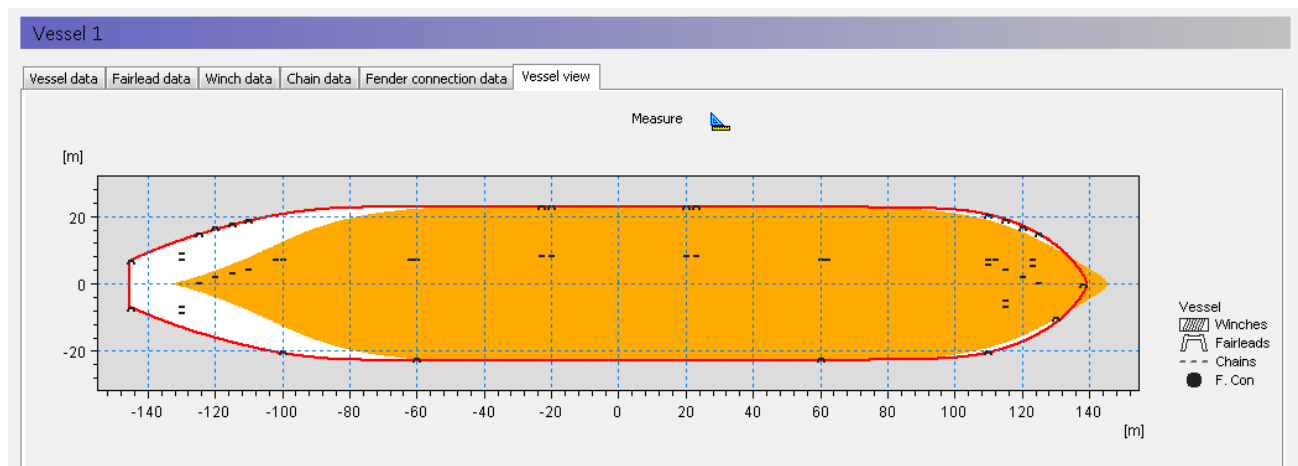
Figure 4-8　　Vessel view after fairlead, winch and fender connection positions have been specified for Vessel 1

2. Dialog: Vessel 2 instance
On the **Vessel data tab**, specify the data as shown in Table 4.7:

Table 4.7　　Vessel 2 data

Longitudinal Area (Wind)	2478.32 [m^2]
Transverse Area (Wind)	396.65 [m^2]
Lpp	224.89 [m]
Loading Condition	0 [%]
Vessel Class	User defined

Similar to Vessel 1, you will have to import drag curves for the vessel:
a. Select the wind drag curve: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Drag Curves\Wind**BulkCarrier.dfs0**

b. Select the current drag curve: < example installation folder >\MIKE Zero
    Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Drag
    Curves\Current**BulkCarrier.dfs0**

On the **Fairlead data tab**, load in data for the full set of fairlead positions (vessel
fixed) from the file: < example installation folder >\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Vessel Spatial
Attributes**vessel2_fairleads.xyz**

On the **Winch data tab**, load in data for the full set of winch positions (vessel fixed)
from the file: < example installation folder >\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Vessel Spatial
Attributes**vessel2_winches.xyz**

These values are as the ones shown in Table 4.8 to Table 4.9. The X and Y values
are relative to amidship of the vessel and the Z values are relative to the keel.

Table 4.8   Fairlead positions relative to amidship at Vessel 2

Name	X [m]	Y [m]	Z [m]
Fairlead 1	-97	-13	20
Fairlead 2	-90	-14.5	20
Fairlead 3	-85	-15.5	20
Fairlead 4	-80	-16.5	20
Fairlead 5	-65	-17.5	20
Fairlead 6	-60	-17.5	20
Fairlead 7	60	-16.5	20
Fairlead 8	65	-16	20
Fairlead 9	80	-13.5	20
Fairlead 10	85	-12.5	20
Fairlead 11	89	-11.5	20
Fairlead 12	93	-10	20

Table 4.9 — Winch positions relative to amidship at Vessel 2

Name	X [m]	Y [m]	Z [m]
Winch 1	-97	-5	20
Winch 2	-90	-6.5	20
Winch 3	-85	-6.5	20
Winch 4	-80	-6.5	20
Winch 5	-65	-6.5	20
Winch 6	-60	-6.5	20
Winch 7	60	-6.5	20
Winch 8	65	-6.5	20
Winch 9	80	-6.5	20
Winch 10	85	-6	20
Winch 11	89	-4.5	20
Winch 12	93	-2	20

On the **Vessel View tab**, you can see overview of where the full set of fairleads and winches are positioned relative to the vessel. It should look as seen in Figure 4-9Figure 4-8. Note that some of the vessel attributes may appear to be missing (or not visible) with the full extent view. It is recommended that you zoom in to properly inspect them.
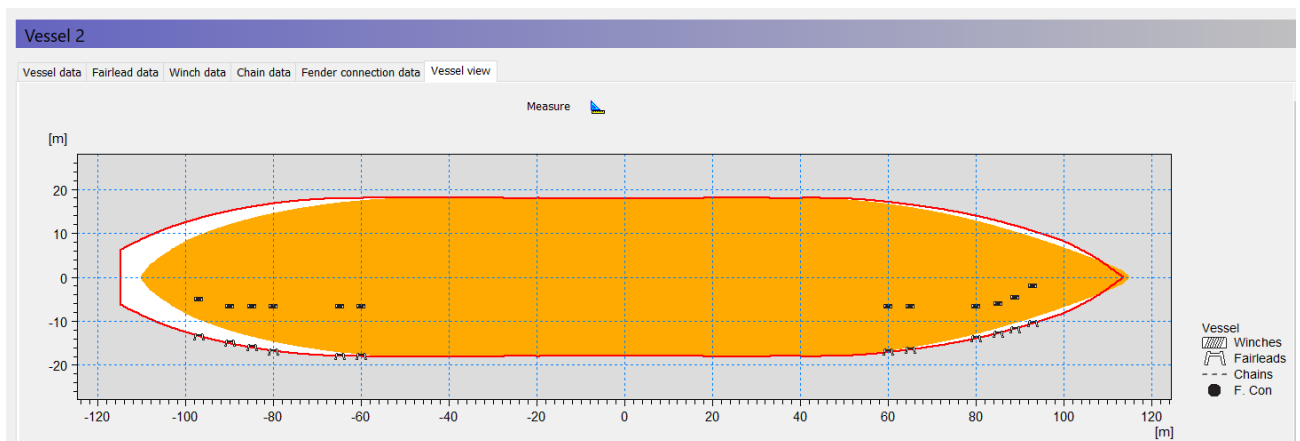
Figure 4-9 — Vessel view after fairlead and winch positions have been specified for Vessel 2

Dialog: Vessel-Vessel Data

Under this dialog, you will be able to see the two adjacent vessel and their respective vessel-fixed attributes, as shown in Figure 4-10. You are then able to define the mooring lines and fender connection between these two vessels.
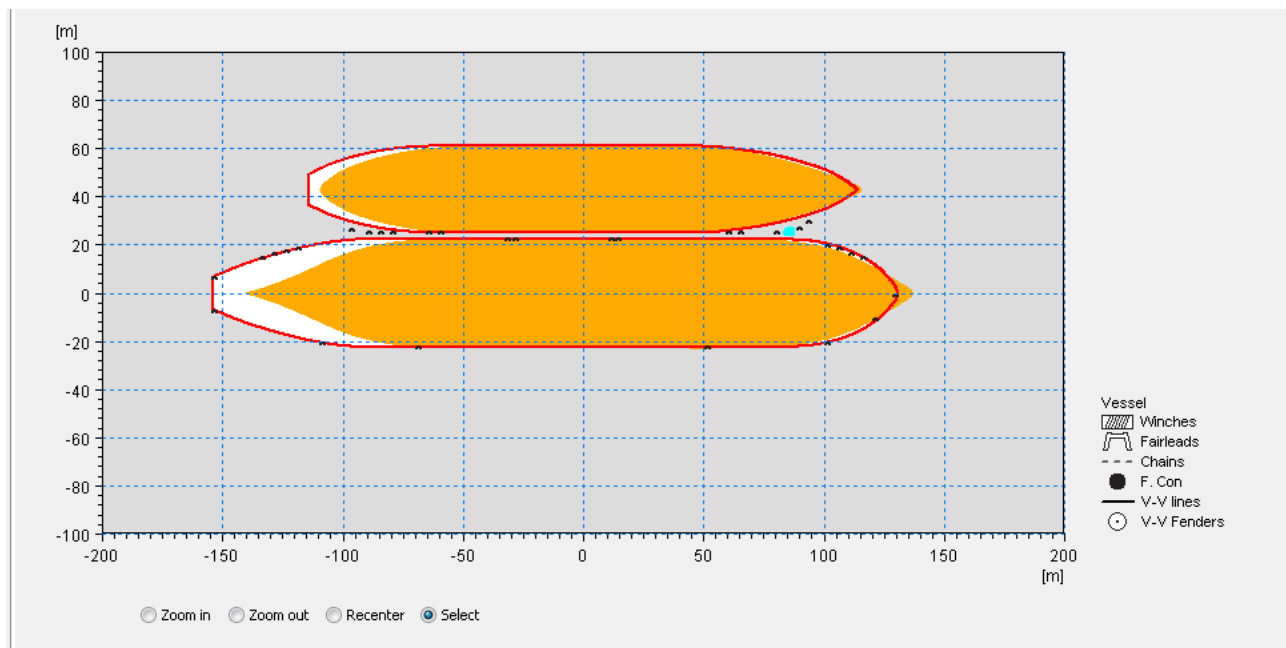
Figure 4-10   Vessel-vessel data view with their respective vessel-fixed attributes

First, we will define the mooring lines between them. The mooring system that we would like to obtain is as given in Table 4.10.

Each mooring line can be drawn/defined interactively by:
a.   Visual identification of the winch (first vessel) → fairlead (first vessel) → fairlead (second vessel) → winch (second vessel) pair, to span the mooring line

b.   Initiate the mooring line insertion by: clicking the **Vessel-Vessel lines** icon

c.   Perform the mooring line insertion by moving the cursor and clicking the following in sequence: Winch icon (*vessel from*) → Fairlead icon (*vessel from*) → Fairlead icon (*vessel to*) → Winch icon (*vessel to*)

If you make an error by connecting some incorrect attributes, then delete the line by selecting the delete line icon   and subsequently clicking on the given line.

After insertion of all mooring lines, the vessel-vessel mooring setup should look like the one shown in Figure 4-11.
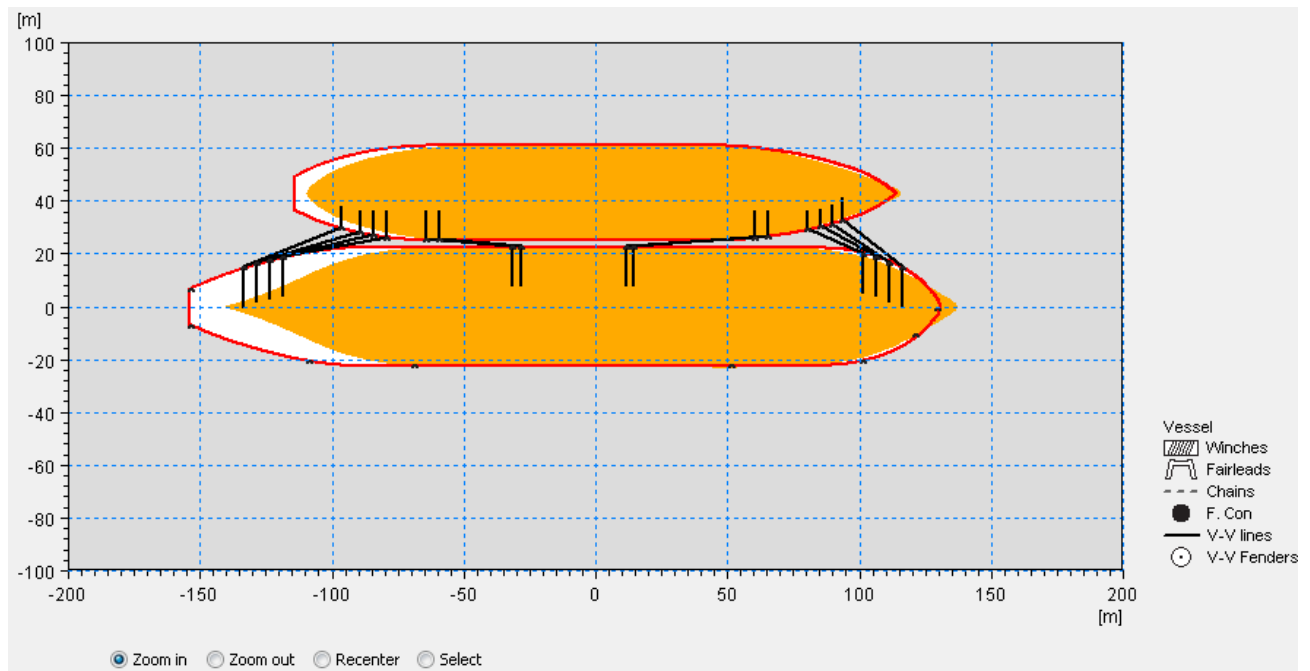
Figure 4-11   System view after definition of vessel-vessel mooring lines

To complete the mooring line definitions, go to the **Vessel-vessel line data** tab and set the line profile, line pretension, tail profile and tail length values for all the vessel-vessel mooring lines (V-V Line 1 – V-V Line 12) according to the values shown in Table 4.10.

Make sure that the vessel-vessel lines go from vessel 1 to vessel 2 and not vice versa!

Table 4.10   The full vessel-vessel lines definition

Name	Vessel (from)	Winch (from)	Fairlead (from)	Vessel (to)	Winch (to)	Fairlead (to)	Line profile	Line pre-tension (t)	Tail profile	Tail length
V-V Line 1	Vessel 1	Winch 17	Fairlead 9	Vessel 2	Winch 1	Fairlead 1	Polyester	10	None	0
V-V Line 2	Vessel 1	Winch 18	Fairlead 10	Vessel 2	Winch 2	Fairlead 2	Polyester	10	None	0
V-V Line 3	Vessel 1	Winch 19	Fairlead 11	Vessel 2	Winch 3	Fairlead 3	Polyester	10	None	0
V-V Line 4	Vessel 1	Winch 20	Fairlead 12	Vessel 2	Winch 4	Fairlead 4	Polyester	10	None	0
V-V Line 5	Vessel 1	Winch 21	Fairlead 13	Vessel 2	Winch 5	Fairlead 5	Polyester	10	None	0
V-V Line 6	Vessel 1	Winch 22	Fairlead 14	Vessel 2	Winch 6	Fairlead 6	Polyester	10	None	0

Name	Vessel (from)	Winch (from)	Fairlead (from)	Vessel (to)	Winch (to)	Fairlead (to)	Line profile	Line pre-tension (t)	Tail profile	Tail length
V-V Line 7	Vessel 1	Winch 23	Fairlead 15	Vessel 2	Winch 7	Fairlead 7	Polyester	10	None	0
V-V Line 8	Vessel 1	Winch 24	Fairlead 16	Vessel 2	Winch 8	Fairlead 8	Polyester	10	None	0
V-V Line 9	Vessel 1	Winch 25	Fairlead 17	Vessel 2	Winch 9	Fairlead 9	Polyester	10	None	0
V-V Line 10	Vessel 1	Winch 26	Fairlead 18	Vessel 2	Winch 10	Fairlead 10	Polyester	10	None	0
V-V Line 11	Vessel 1	Winch 28	Fairlead 20	Vessel 2	Winch 11	Fairlead 11	Polyester	10	None	0
V-V Line 12	Vessel 1	Winch 27	Fairlead 19	Vessel 2	Winch 12	Fairlead 12	Polyester	10	None	0

Next, we will define the fender connections between the two vessels. The vessel-vessel fender connections can be defined by following:

a. Initiate the insertion of fender connection by: clicking the **Vessel-Vessel fenders** icon

b. Moving the cursor and clicking on the F. Con icon (see Figure 4-12).

Figure 4-12   Example of a newly defined fender connection (dotted white circle with arrow icon showing the fender force direction)

To complete the fender connection definitions, go to the **Vessel-vessel fender data** tab. Set the fender force direction and fender profile for all the vessel-vessel fender connections (V-V Fender 1 – V-V Fender 4) with the values shown in Table 4.10.

A fender force direction of 90 [deg] means that fender forces goes towards north (i.e. towards Vessel 2).

Table 4.11   The full vessel-vessel fender definition

Name	Vessel (connected)	Fender connection	Vessel (affected)	Force dir. [deg]	Fender profile
V-V Fender 1	Vessel 1	Fender connection 1	Vessel 2	90	Pneumatic Fender
V-V Fender 2	Vessel 1	Fender connection 2	Vessel 2	90	Pneumatic Fender
V-V Fender 3	Vessel 1	Fender connection 3	Vessel 2	90	Pneumatic Fender
V-V Fender 4	Vessel 1	Fender connection 4	Vessel 2	90	Pneumatic Fender

Dialog: Port Data

On the **Bollard data tab**, set the map projection to UTM-30. Then load in data for the full set of bollard positions (port/land fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Port Data**Bollards.xyz**

Then, select the Bollard profile *Bollard Profile* by clicking into the cells of the corresponding column.

On the **Fender data tab**, set the map projection to UTM-30. Then load in data for the full set of fender positions (port/land fixed) from the file: < example installation folder >\MIKE Zero Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Port Data**Fenders.xyz**

The bollard and fender positions in the grid series (.dfs2) domain are shown in Table 4.12 and Table 4.13. The Z values are relative to the water surface.

Table 4.12   Bollard positions in local domain

Name	Easting [m]	Northing [m]	Z [m]	Bollard profile
Bollard 1	835283.7619	718.8273511	3	Bollard Profile
Bollard 2	835255.4776	690.5430798	3	Bollard Profile
Bollard 3	835205.9802	556.1927914	3	Bollard Profile
Bollard 4	835202.4446	552.6572575	3	Bollard Profile
Bollard 5	835071.6299	506.6953167	3	Bollard Profile
Bollard 6	835043.3456	478.4110455	3	Bollard Profile

Table 4.13   Fender positions in local domain

Name	Easting [m]	Northing [m]	Z [m]	Force dir. [deg]	Fender profile	Vessel (affected)
Fender 1	835263.1143862	612.1956485	2	315	SCN 2000 E1.5	Vessel 1
Fender 2	835220.6879793	569.7692416	2	315	SCN 2000 E1.5	Vessel 1
Fender 3	835192.4037081	541.4849704	2	315	SCN 2000 E1.5	Vessel 1
Fender 4	835135.8351656	484.9164279	2	315	SCN 2000 E1.5	Vessel 1

Set all fender profiles to Fender Profile 1 while the fenders force directions to 315 [deg] (meaning the fender forces goes towards south-east). The port-fixed fenders will be acting towards Vessel 1 (i.e. vessel affected).

On the **Port View tab**, you can see where the bollards and fenders are positioned in the domain. The full set of bollards and fenders should look like in Figure 4-13. Note that some of the port-fixed attributes may appear to be missing (or not visible) with the full extent view. It is recommended that you zoom in to properly inspect them.
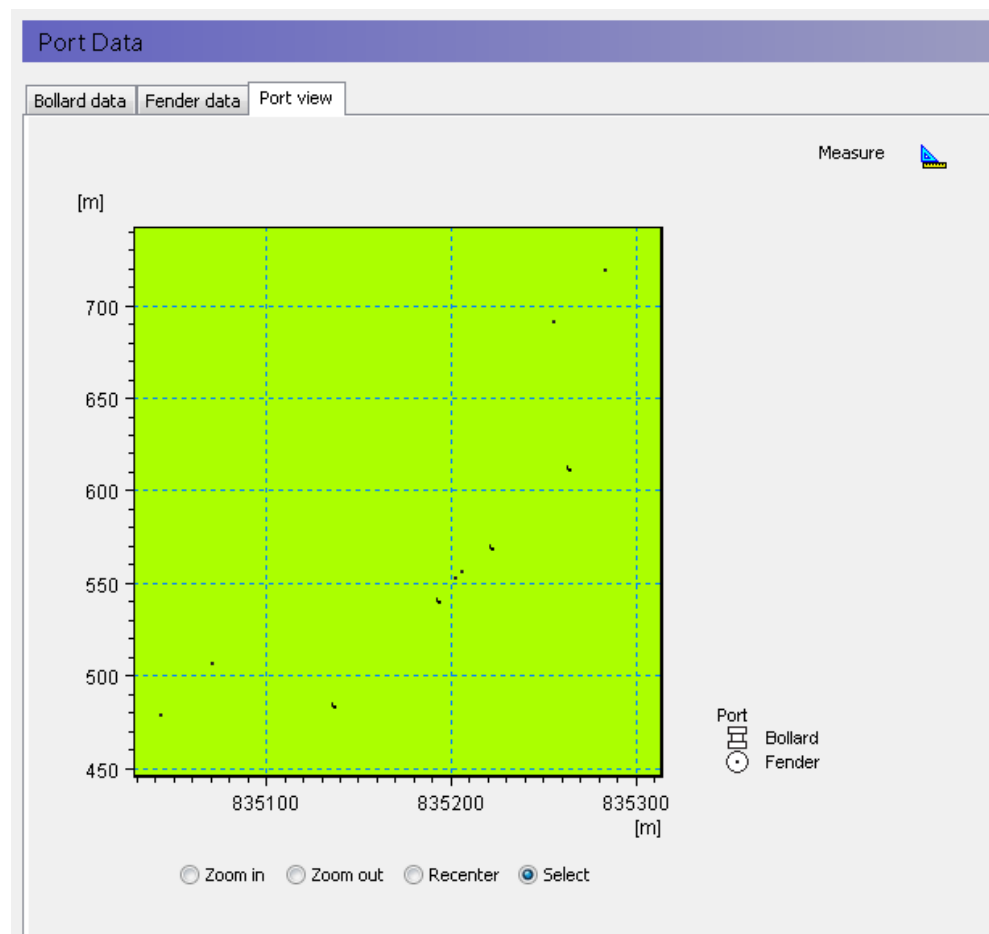
Figure 4-13   Bollard and fender positions

Dialog: Mooring Setup

On the **Vessel position and mooring lines tab**, set the map projection to UTM-30. Specify the vessel position to be at (easting, northing) = (835222, 537) [m] and a vessel rotation of 225 [deg], and click Apply.

You will now see the position of the two moored vessel relative to the port-fixed attributes, as shown in Figure 4-14.
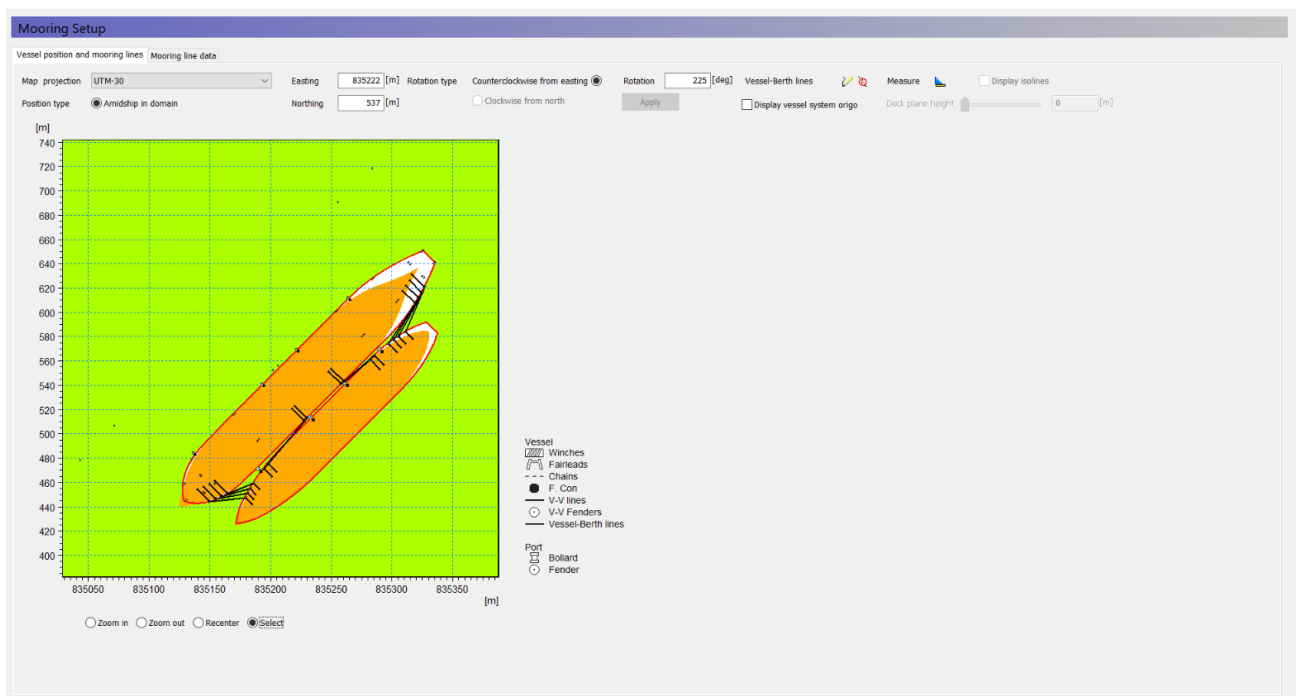
Figure 4-14   Vessel inserted into domain

The next step is to define the mooring lines between Vessel 1 and the port-fixed bollards. The mooring system we would like to obtain is as given in Table 4.14.

Each mooring line can be drawn/defined interactively by:
a. Visual identification of the relevant winch-fairlead-bollard pair, to span the mooring line

b. Initiate the mooring line insertion by: clicking the Vessel-Berth lines icon
c. Perform the mooring line insertion by: clicking the relevant winch icon, move the cursor to the relevant fairlead icon, clicking on the fairlead icon, continue the line to the bollard, and finally clicking on the bollard icon.

After insertion of all mooring lines, the mooring setup should look like the one shown in Figure 4-15.

To complete the mooring line definitions, go to the **Mooring line data tab** and set the line profile, line pretension, tail profile and tail length values for all the mooring lines (Line 1 – Line 16) according to the values shown in Table 4.14.

Figure 4-15   Vessel view after definition of all mooring lines

Table 4.14   The full mooring line definitions

Name	Vessel	Bollard	Fairlead	Winch	Line profile	Line pretension [t]	Tail profile	Tail length [m]
Line 1	Vessel 1	Bollard 1	Fairlead 1	Winch 9	HMPE	10	Polyester	11
Line 2	Vessel 1	Bollard 1	Fairlead 1	Winch 1	HMPE	10	Polyester	11
Line 3	Vessel 1	Bollard 1	Fairlead 2	Winch 2	HMPE	10	Polyester	11
Line 4	Vessel 1	Bollard 2	Fairlead 2	Winch 10	HMPE	10	Polyester	11
Line 5	Vessel 1	Bollard 2	Fairlead 3	Winch 11	HMPE	10	Polyester	11
Line 6	Vessel 1	Bollard 2	Fairlead 3	Winch 3	HMPE	10	Polyester	11
Line 7	Vessel 1	Bollard 3	Fairlead 4	Winch 12	HMPE	10	Polyester	11
Line 8	Vessel 1	Bollard 3	Fairlead 4	Winch 4	HMPE	10	Polyester	11
Line 9	Vessel 1	Bollard 4	Fairlead 5	Winch 5	HMPE	10	Polyester	11
Line 10	Vessel 1	Bollard 4	Fairlead 5	Winch 13	HMPE	10	Polyester	11
Line 11	Vessel 1	Bollard 5	Fairlead 6	Winch 6	HMPE	10	Polyester	11
Line 12	Vessel 1	Bollard 5	Fairlead 6	Winch 14	HMPE	10	Polyester	11
Line 13	Vessel 1	Bollard 5	Fairlead 7	Winch 15	HMPE	10	Polyester	11

Line 14	Vessel 1	Bollard 6	Fairlead 7	Winch 7	HMPE	10	Polyester	11
Line 15	Vessel 1	Bollard 6	Fairlead 8	Winch 8	HMPE	10	Polyester	11
Line 16	Vessel 1	Bollard 6	Fairlead 8	Winch 16	HMPE	10	Polyester	11

Dialog: Environmental Conditions
Expand the nodes and go to Physical Parameters.

1. Dialog: Physical Parameters
   In this dialog, you should use the default values:
2. Density of water = 1020 [kg/m^3]
3. Density of air   = 1.28 [kg/m^3]
4. Viscosity (water) = 10^{-6} [m^2/s]
5. Viscosity (air)   = 1.48 • 10^{-5} [m^2/s]

6. Dialog: Drift Forces
   Drift forces are not included.

7. Dialog: Current
   Do not include a current field.

8. Dialog: Wind
   Do not include a wind field.

Dialog: Convergence Parameters
In this dialog, specify the following values:
a. Max no. of iterations = 5000
Line convergence threshold = 1 %

Dialog: Vessel Displacements
Apply default parameters for both Vessel 1 and Vessel 2.

Dialog: Outputs
Apply the default output file names. All output files except 'External forces' and 'Chain forces' are included.

Ensure that all data in the (.m21ma) setup are saved.

4.6 Step 6 – Run the Simulation in Convergence Mode

Save the (.m21ma) editor and launch the M21 MA simulation.

4.7 Step 7 – Populate the MIKE 21 Mooring Analysis setup with final execution data

Dialog: Simulation Mode
Switch from Convergence Mode to Final Computation Mode. This will then open the Wave dialog under the Environmental Conditions dialog in the navigation tree.

Dialog: Wave
Use the following parameters:
a. Select Wave type = Wave field

  b. Check 'Include warm up/down'
  c. Select Format = Varying in time, constant in domain
  d. Set Number of waves to 1
  e. Set Power (N) to 12 (leading to FFT size = 4096)

Load in the wave data from the following file:
< example installation folder >\MIKE Zero
Projects\MIKE_21\Maritime\MooringAnalysis\Multibody\M21MA\Inputs\Waves\
waves.dfs0

Dialog: Vessel Displacements

Click on the 'Load' button. The initial vessel displacement fields for the six degree of motions (i.e. Surge, Sway, Heave, Roll, Pitch and Yaw) will be automatically populated with data from the final iteration of the output files:

  a. Convergence_Vessel_1_motions.dfs0 for Vessel 1
  b. Convergence_Vessel_2_motions.dfs0 for Vessel 2

This action ensures that the final time-domain computation starts from vessel states where there is sufficient tensional equilibrium in all mooring lines.

Dialog: Final Output

Leave the output file settings as default. All output files except 'External forces' and 'Chain forces' are included.

Set the vessel motion type to 'Relative to displaced position'.

Figure 4-16   Final output dialog

4.8   Step 8 – Run the Final Time-Domain Simulation

Save the (.m21ma) editor and launch the M21 MA simulation. From the log result file, you will see that the critical load for the vessel-berth lines and vessel-vessel lines were never exceeded.

```
============== Operability Summary: Vessel-Berth Lines ==============

Line    Status    Cause
1       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
2       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
3       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
4       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
5       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
6       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
7       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
8       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
9       Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
10      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
11      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
12      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
13      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
14      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
15      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
16      Pass      critical load never exceeded (line-part: 50% of TBS = 85 [ton], tail-part: 50% of TBS = 120 [ton])
====================================================================



============== Operability Summary: Vessel-Vessel Lines ==============

Line    Status    Cause
17      Pass      critical load never exceeded (50% of TBS = 120 [ton])
18      Pass      critical load never exceeded (50% of TBS = 120 [ton])
19      Pass      critical load never exceeded (50% of TBS = 120 [ton])
20      Pass      critical load never exceeded (50% of TBS = 120 [ton])
21      Pass      critical load never exceeded (50% of TBS = 120 [ton])
22      Pass      critical load never exceeded (50% of TBS = 120 [ton])
23      Pass      critical load never exceeded (50% of TBS = 120 [ton])
24      Pass      critical load never exceeded (50% of TBS = 120 [ton])
25      Pass      critical load never exceeded (50% of TBS = 120 [ton])
26      Pass      critical load never exceeded (50% of TBS = 120 [ton])
27      Pass      critical load never exceeded (50% of TBS = 120 [ton])
28      Pass      critical load never exceeded (50% of TBS = 120 [ton])
====================================================================
```

| 100 % | 1990-01-01 13:00:00 | Timestep: 7200 of 7200 | 121 |

Figure 4-17   Log result file after the simulation was completed

Notice that in the result folder, there are two sets of output files for the two vessels respectively. To inspect the model results, you can plot the vessel motions on MIKE Zero Plot Composer. As the two vessels are moored to each other, the surge responses are similar between them. However, Vessel 1 is predicted to have larger roll than Vessel 2 and Vessel 2 a larger sway than Vessel 1.
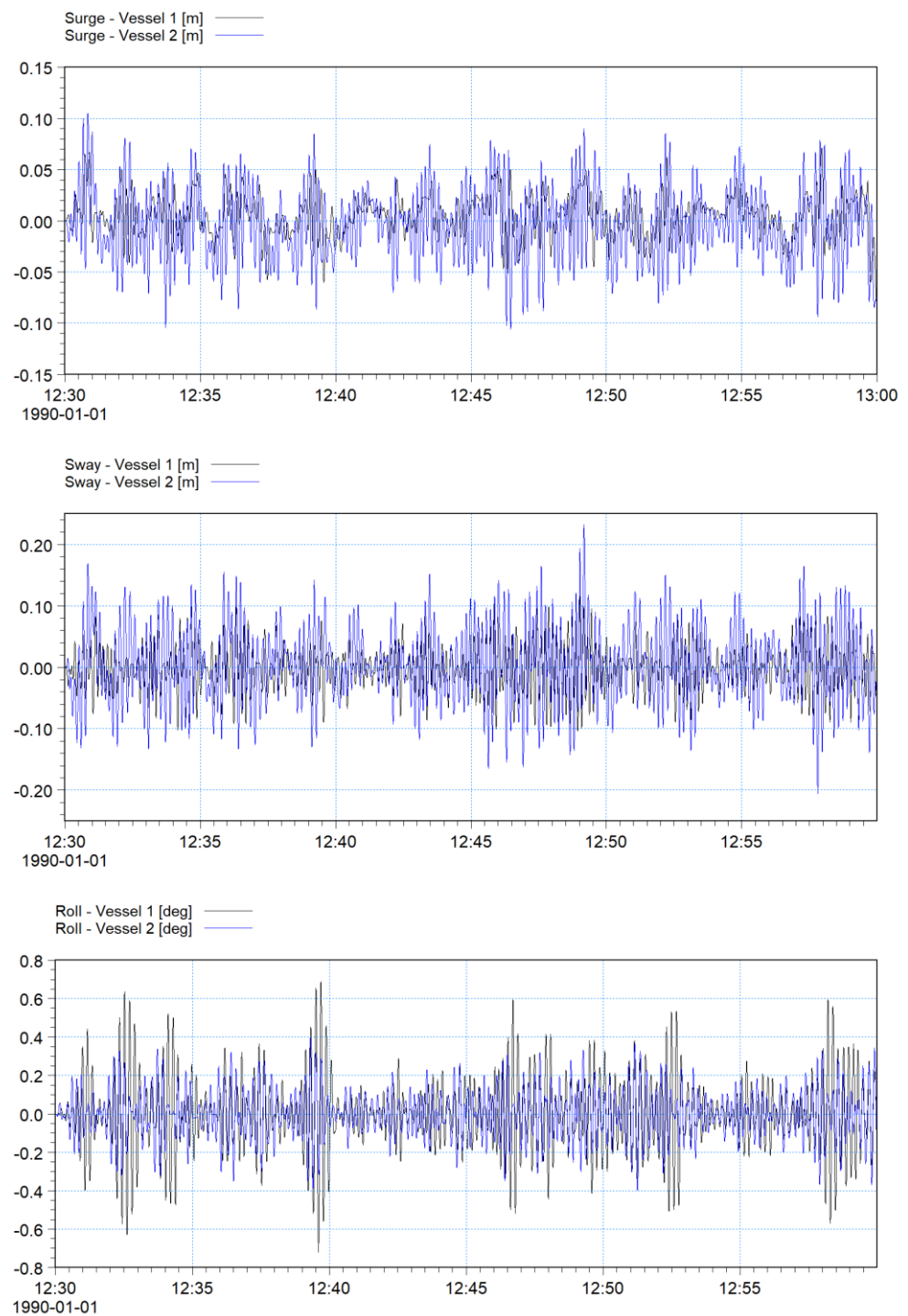
Figure 4-18   Time series of vessel motions for Vessel 1 and Vessel 2

5 Example: AutoMoor Mooring System

The step-by-step example in this Chapter demonstrates how to set up a Trelleborg AutoMoor automated mooring unit in a MIKE 21 Mooring Analysis simulation. The example calculates vessel motions in 6-DoF (Six Degrees of Freedom) of a moored tanker subjected to a time series of incident wave conditions (0D wave forcing).

This example is the same as the example provided in section 2 (Example: 0D Wave Forcing on a Vessel) but instead of mooring lines, two AutoMoor units are used. Therefore, the vessel frequency response and mooring analysis setup are taken from that example.

This example's files are available in the folder
<example installation folder>\MIKE_21\Maritime\MooringAnalysis\AutoMoor Mooring

5.1 Step 1 – Open the MIKE 21 Mooring Analysis Setup

This example is based on the example in section 2, so you can open that setup if you want to create the Automoor settings yourself.

Note: if you open the setup file for the example in section 2, delete the mooring line connections in the Mooring Setup dialog.

5.2 Step 2 - Create an AutoMoor Profile

An AutoMoor profile is defined by a file name and the percentage of the maximum force that an AutoMoor can bear (Figure 5-1).
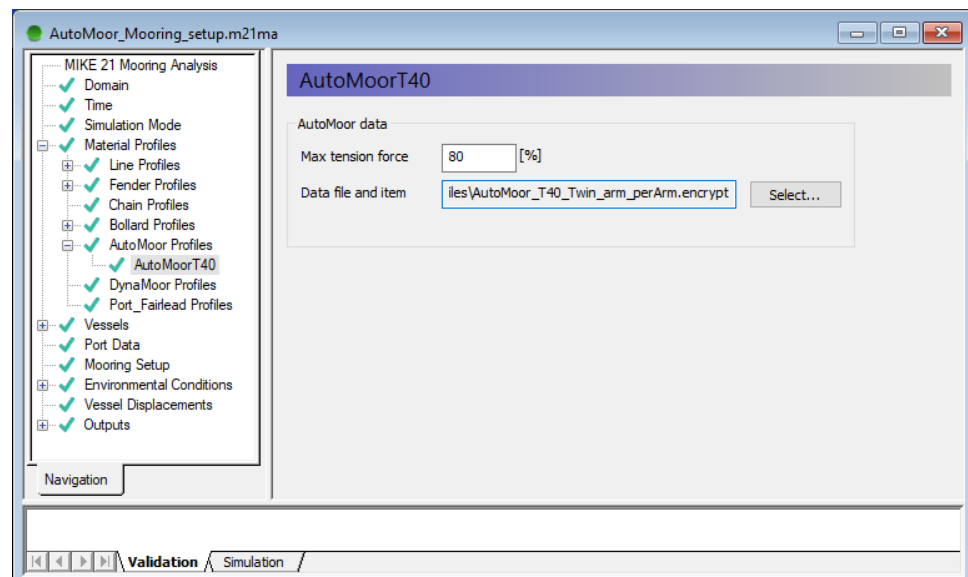

Figure 5-1 Definition of an AutoMoor Profile

Under 'Material Profiles' create a new AutoMoor profile. Select the following AutoMoor profile from the templates folder: '**C:\Program Files (x86)\DHI\MIKE Zero\2022\Templates\Maritime\MooringAnalysis\Material Profiles\AutoMoor**

Profiles\AutoMoor_T40_twin_arm_perArm.encrypt' or the appropriate address containing this file**.** The maximum tensile force of AutoMoor class AutoMoor_T40_twin, used in this example is 20 ton per arm. This value is available in the encrypted force-table file provided by Trelleborg. In this example the maximum tension force is reduced to 80%.

It should be noted that the current configuration is "per arm", meaning that a full T40 unit will require the addition of two profiles , as the T40 consists of two 20 ton arms. The user is referred to Trelleborg Product Datasheets for separation dimension.

5.3　Step 3 – Create AutoMoor Units

Now define the properties for the two AutoMoor profiles in the Port Data dialog tab (Figure 5-2.).

Enter the values as shown in Table 5.1.
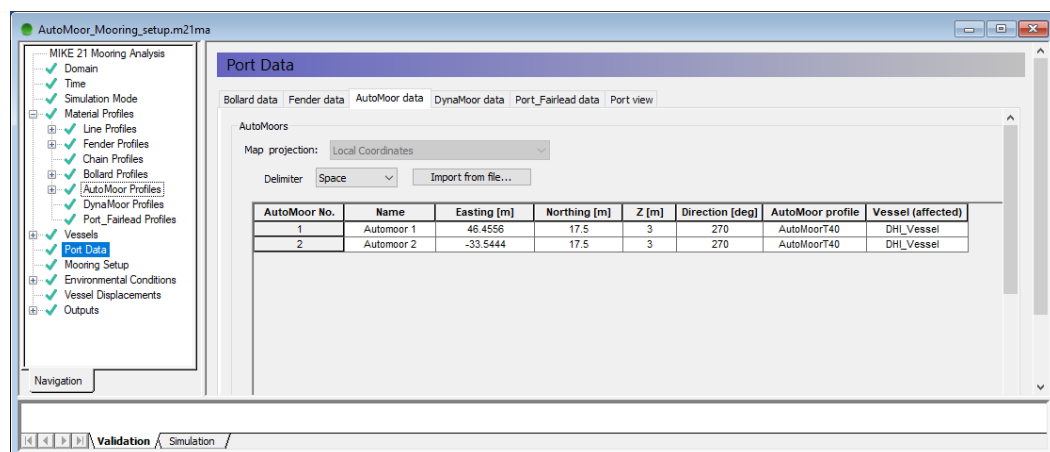
Figure 5-2 Definition of AutoMoor positions in port

Table 5.1　AutoMoor's characteristics in port

Parameter	AutoMoor No. 1	AutoMoor No. 2
Name	Automoor 1	Automoor 2
Easting	46.4556	-33.5444
Northing	17.5	17.5
Z	3	3
Direction	270	270
AutoMoor profile	AutoMoorT40	AutoMoorT40
Vessel (affected)	DHI_Vessel	DHI_Vessel

The same coordinate system used for fenders and bollards is also used for AutoMoor's. The AutoMoor direction is best defined as the same as the direction of the nearest fender,

as AutoMoor and the fender together control the vessel motions (with the fender working under compression while AutoMoor works under tension).
It is advised to arrange AutoMoors through a flat section of the vessel hull.

Based on the 'AutoMoorlength' and fender thickness, the connection point of AutoMoor on the berth is defined. The parked end of an AutoMoor is recommended to be behind the compressed fender-line.

5.4   Step 4 – Define the AutoMoor output files

Select to output the results for AutoMoor.

The output type 'AutoMoor forces' defines the result file name which includes the resulting forces on the AutoMoor's.

The output type 'AutoMoorXYZ' defines the result file name which includes the 'X', 'Y', 'Z', 'L', 'horizAngle', and 'vertAngle' of all AutoMoors. Here, Y is the AutoMoorlength in the initial 'AutoMoordirection', X is the deviation from initial direction on a horizontal plane and 'Z' is the deviation from the initial position in the vertical plane. horizAngle=ATAN(X/Y), vertAngle=ATAN(Z/Y), and L is the total length of AutoMoor, extended between the berth-based point of AutoMoor and its attachment point to the vessel.

5.5   Step 5 – Run MIKE21 MA with AutoMoor

To execute the (.m21ma) setup , click
Run → Start simulation … → Click OK in the Launch settings' pop-up window.

The simulation will be fast and produce the relevant output files for the time-domain analysis. However, the run time increases by increasing the number of AutoMoor units. If there is any error, it will be shown on the screen and will also be printed in the (.log) file.

5.6   Step 6 – Inspect Results

The Operability Summaries for AutoMoor at the end of the simulation can be found at the end of the (.log) file produced by the engine. For this simulation the critical AutoMoor extension and deflections were never exceeded, indicating the mooring system dynamics is within the specified operating limits.

```
================ Operability Summary: Port Automoors ================

Automoor Status    Cause
1    Pass    horizontal   angle never exceeded (100% of Max Horizontal (x/y plane) angle (0.28 [Rad])
1    Pass    vertical     angle never exceeded (100% of Max Vertical (z/y plane)  Angle (0.4 [Rad])
1    Pass    automoor     Y never exceeded (100% of Max allowed Y (4.27 [m])

2    Pass    horizontal   angle never exceeded (100% of Max Horizontal (x/y plane) angle (0.28 [Rad])
2    Pass    vertical     angle never exceeded (100% of Max Vertical (z/y plane)  Angle (0.4 [Rad])
2    Pass    automoor     Y never exceeded (100% of Max allowed Y (4.27 [m])

====================================================================
```

Figure 5-3   Operability Summaries for AutoMoor from the engines .log file.

In this example, the AutoMoor force time series are recorded in file "*aforces.dfs0*". The file can be easily opened in MikeZero (Figure 5-4).
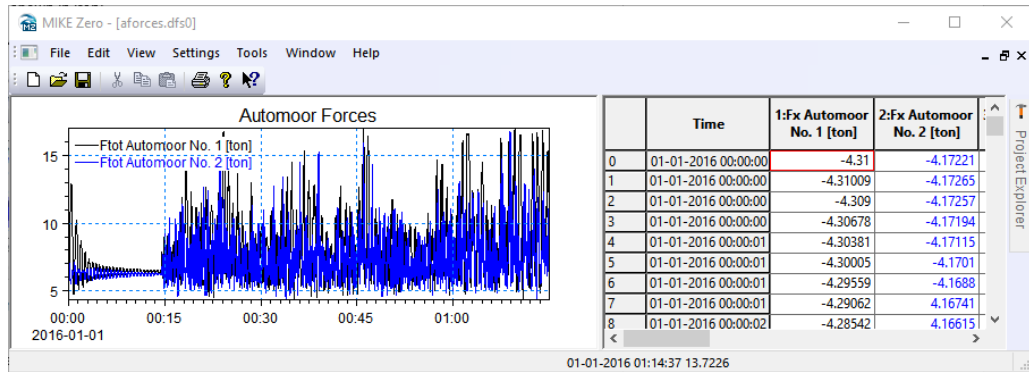

Figure 5-4   AutoMoor force results

The forces are shown in [ton].
The AutoMoor displacement time series are saved in file "automoorXYZ.dfs0". The file is readable in MIKE ZERO (Figure 5-5).
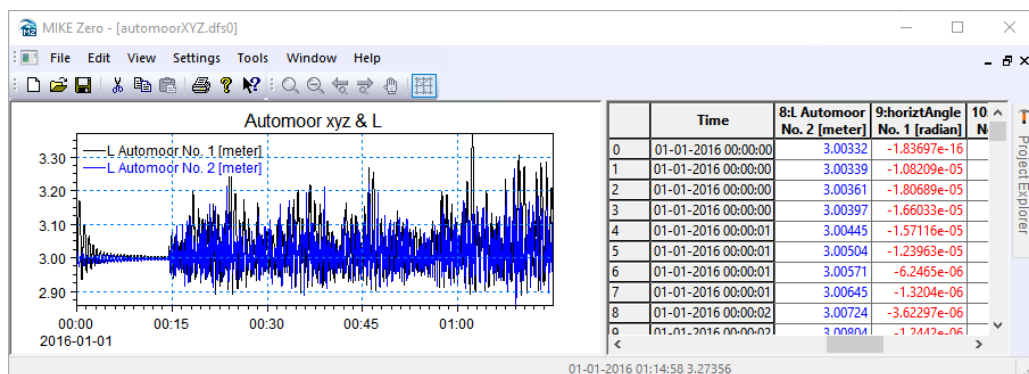

Figure 5-5   Time series of AutoMoor's displacements.

The lengths are shown in [meter] and angles in [Radian].

6 Example: DynaMoor Mooring System

The step-by-step example in this Chapter demonstrates how to set up a Trelleborg DynaMoor automated mooring unit in a MIKE 21 Mooring Analysis simulation. The example calculates vessel motions in 6-DoF (Six Degrees of Freedom) of a moored tanker subjected to a time series of incident wave conditions (0D wave forcing).

This example is the same as the example provided in section 2 (Example: 0D Wave Forcing on a Vessel) but instead of 12 mooring lines connected to bollards, two lines are connected to the port_fairlead and DynaMoor hooks. Therefore, the vessel frequency response and mooring analysis setup are taken from that example. In this section, it is shown how to add DynaMoor to the setup file.

This example's files are available in the folder
<example installation folder>\MIKE_21\Maritime\MooringAnalysis\ AutoMoor Mooring

6.1 Step 1 – Open the MIKE 21 Mooring Analysis Setup

This example is based on the example in section 2, so you can open that setup if you want to create the DynaMoor settings yourself.

6.2 Step 2 - Create a DynaMoor Profile

A DynaMoor profile is defined by a file name and the percentage of the maximum force that an AutoMoor can bear (Figure 6-1).
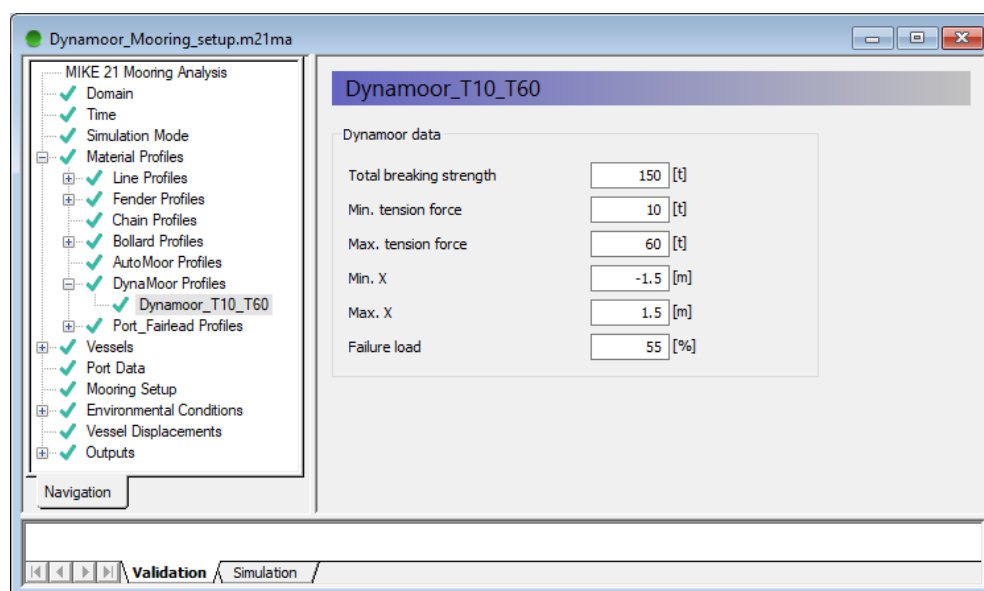

Figure 6-1 Definition of a DynaMoor Profile

The user is referred to Trelleborg Product Datasheets for product information. The maximum DynaMoor strength and the percentage of this ultimate strength used in the safety check of the device are defined by the "Total breaking strength", and "Failure load", respectively. The minimum and maximum stroke of the DynaMoor hook are defined in

"Xmin" and "Xmax". It is assumed that at the start of simulations, DynaMoor's hook is at the mid-stroke position (X=0). The minimum and maximum tensile force which must be kept in the line by DynaMoor must also be defined. Pls. specify the values as shown in Table 6.1.

Table 6.1   DynaMoor profile characteristics

Name	DynaMoor_T10_T60
Total breaking strengh	150 [t]
Minimum tension force	10 [t]
Maximum tension force	60 [t]
Minium X stroke	-1.5 [m]
Maximum X stroke	1.5 [m]
Failure load	55 [%]

Note: This setup is only valid for a constant tension DynaMoor unit. Other modes of DynaMoor including a passive tension (spring type) DynaMoor will be presented in the future developments.

Port_fairlead

Each DynaMoor unit is connected to a berth-based fairlead to control the line direction, which is defined as Port_fairlead. The Port_fairlead profiles must therefore be defined in the 'Materials' dialogs. The default values are as shown in Figure 6-2.
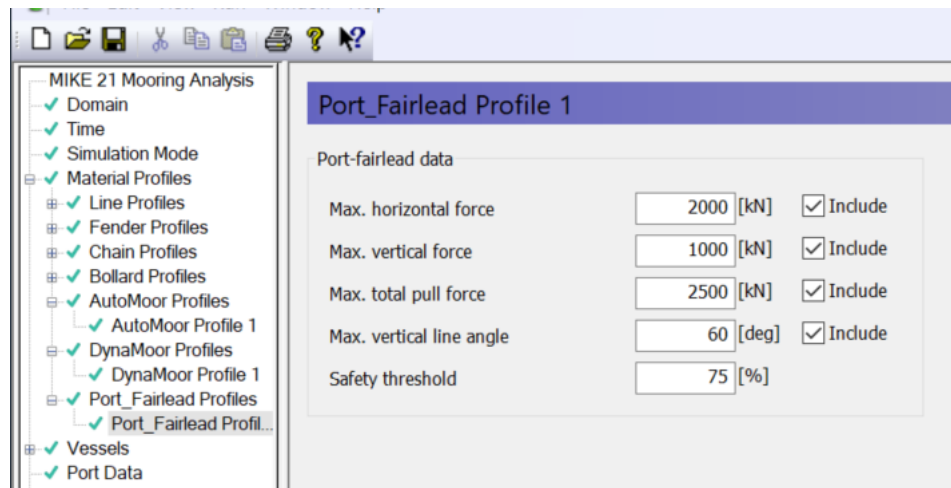

Figure 6-2 Default values for Port_fairlead profile

The Port_fairlead profile is only used for operability checks. The user has options to select if the various checks are to be included or not. Include all checks in this example.
The "Safety threshold" is the percentage of the maximum force used in the operability checks.

6.3   Step 3 – Create DynaMoor Units

Now define the properties for the DynaMoor's in the Port.

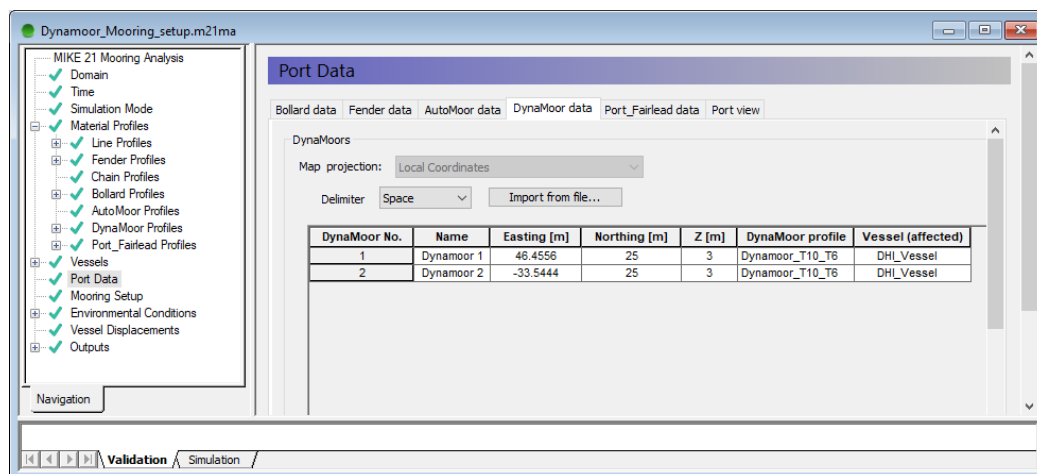Create two DynaMoor units as shown in Figure 6-3. Enter the values as shown in Table 5.1.

Figure 6-3 Definition of DynaMoor properties in port

Table 6.2   DynaMoor's characteristics in port

Parameter	DynaMoor No. 1	DynaMoor No. 2
Name	DynaMoor 1	DynaMoor 2
Easting	46.4556	-33.5444
Northing	25	25
Z	3	3
DynaMoor profile	DynaMoor_T10_T6	DynaMoor_T10_T6
Vessel (affected)	DHI_Vessel	DHI_Vessel

The coordinate of the DynaMoor's hook on the berth is defined by Easting, Northing and Z. The same coordinate system used for fenders and bollards is used for Dynamoor's.

The line that connects to the DynaMoor, will be defined in the Mooring setup dialog.
It is advised to connect DynaMoor's to the spring lines, however they can be used for any line connection.

Similar to the DynaMoor units, create two Port_Fairlead profiles units. Enter the values as shown in Table 6.3.

Table 6.3   Port_Fairleads characteristics in port

Parameter	Port_Fairlead No. 1	Port_Fairlead No. 2
Name	Port_Fairlead 1	Port_Fairlead 2
Easting	46.4556	-33.5444
Northing	20	20
Z	3	3
Port_Fairlead profile	Port_Fairlead profile	Port_Fairlead profile
Vessel (affected)	DHI_Vessel	DHI_Vessel

6.4   Step 4 – Specify Mooring Setup

In this example two of the 12 mooring lines (in Example: 0D Wave Forcing on a Vessel) is connected to a port_fairlead and a DynaMoor hook instead of a bollard.

Line 5 is connected to DynaMoor and Port_fairlead number 1, and Line 6 is connected to DynaMoor and Port_fairlead number 2, while the other lines remains connected to the berth bollards. Pls. update the setup as shown in Figure 6-4.
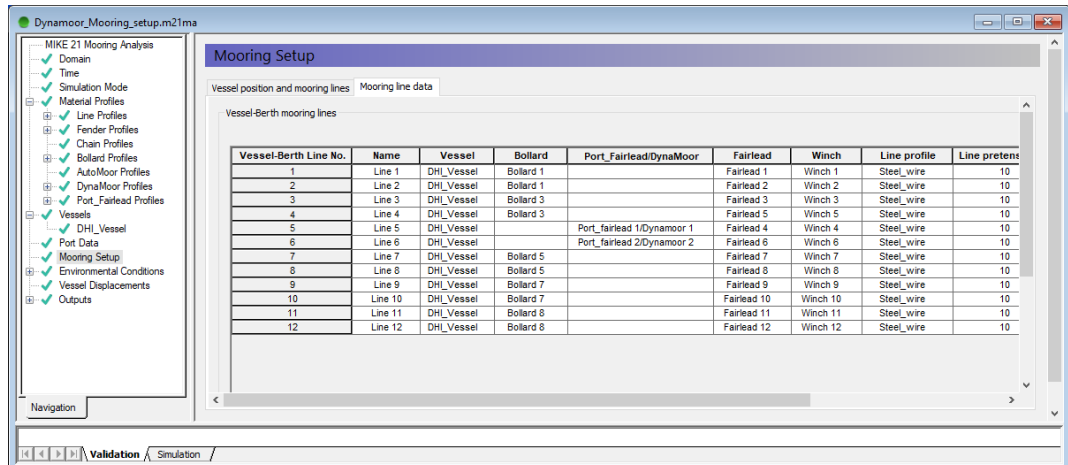

Figure 6-4 Vessel-Berth lines connected to Port_fairlead and DynaMoors as defined in dialog.

6.5   Step 5 – Define the DynaMoor output files

Select to output the results for DynaMoor.

The output type 'DynaMoor forces' defines the result file name which includes the resulting forces on the DynaMoor's.
The output type 'PortFairlead forces' defines the result file name which includes the resulting forces on the port_fairleads.

The output type 'DynaMoorXYZ' defines the result file name which includes the DynaMoor displacement. The displacement occurs along the DynaMoor hook and Port_fairlead orientation.

6.6   Step 6 – Run MIKE21 MA FOR DynaMoor

To execute the (.m21ma) setup , click
Run → Start simulation … → Click OK in the Launch settings' pop-up window.

The simulation will be fast and produce the relevant output files for the time-domain analysis. If there is any error, it will be shown on the screen and will also be printed in the (.log) file.

6.7   Step 7 – Inspect Results

The Operability Summaries for DynaMoor and Port_fairlead at the end of the simulation can be found at the end of the (.log) file produced by the engine. For this simulation the DynaMoor and Port_fairlead loads never exceeded, indicating the mooring system dynamics is within the specified operating limits.

```
=============== Operability Summary: Port Dynamoors ===============

Dynamoor    Status  Cause
1       Pass    dynamoor load never exceeded (55% of Max Dynamoor force (150 [ton]))
2       Pass    dynamoor load never exceeded (55% of Max Dynamoor force (150 [ton]))
==================================================================




=============== Operability Summary: Port_Fairleads ===============

Fairlead Status  Cause
1       Pass    horizontal force never exceeded (75% of Max Horizontal Force (2000 [kN])
1       Pass    vertical   force never exceeded (75% of Max Vertical   Force (1000 [kN])
1       Pass    Port_fairlead   force never exceeded (75% of Max Fairlead   Force (2500 [kN])
1       Pass    Port_fairlead   line angle never exceeded (60[deg])

2       Pass    horizontal force never exceeded (75% of Max Horizontal Force (2000 [kN])
2       Pass    vertical   force never exceeded (75% of Max Vertical   Force (1000 [kN])
2       Pass    Port_fairlead   force never exceeded (75% of Max Fairlead   Force (2500 [kN])
2       Pass    Port_fairlead   line angle never exceeded (60[deg])

==================================================================
```

Figure 6-5   Operability Summaries for DynaMoor and Port_fairlead from the engines .log file.

In this example, the DynaMoor force time series are recorded in file "*dynamoor_forces.dfs0*". The file can be easily opened in MikeZero (Figure 6-6).
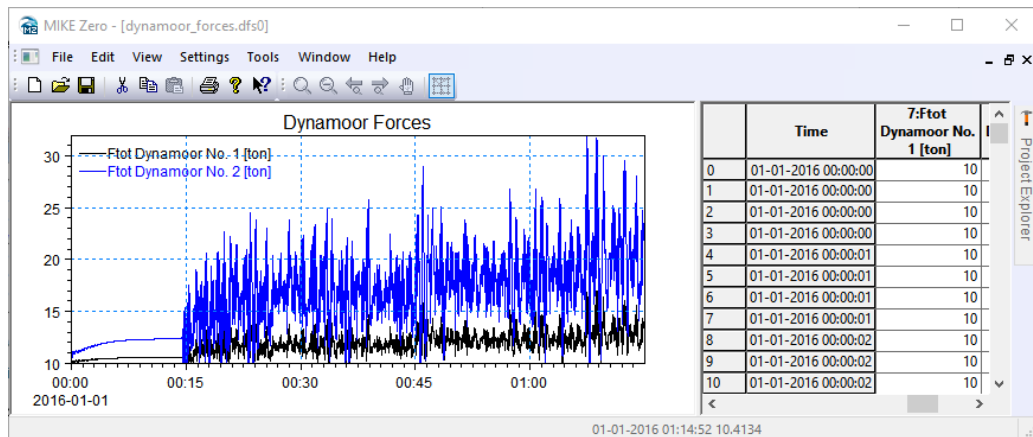
Figure 6-6   DynaMoor force results

The forces are shown in [ton].

The DynaMoor displacement time series are saved in file "*DynaMoorXYZ.dfs0*". The file is readable in MIKE ZERO (Figure 6-7).
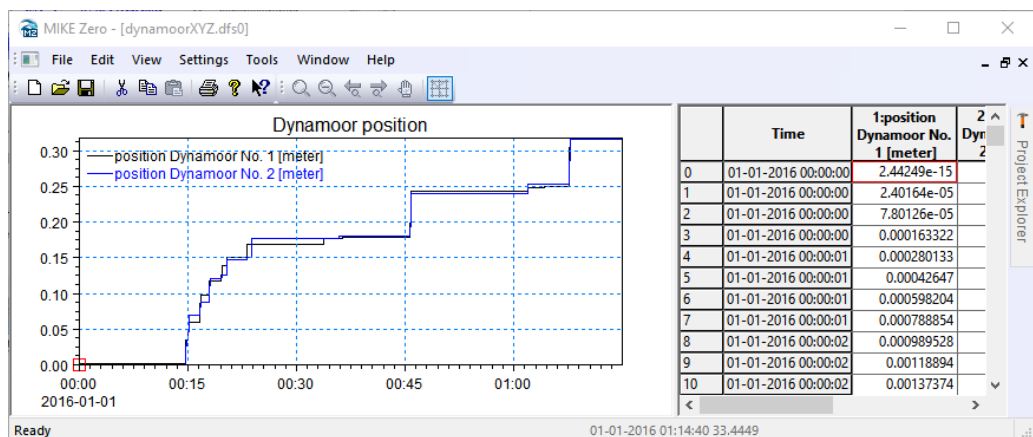

Figure 6-7   Time series of DynaMoors' displacements.

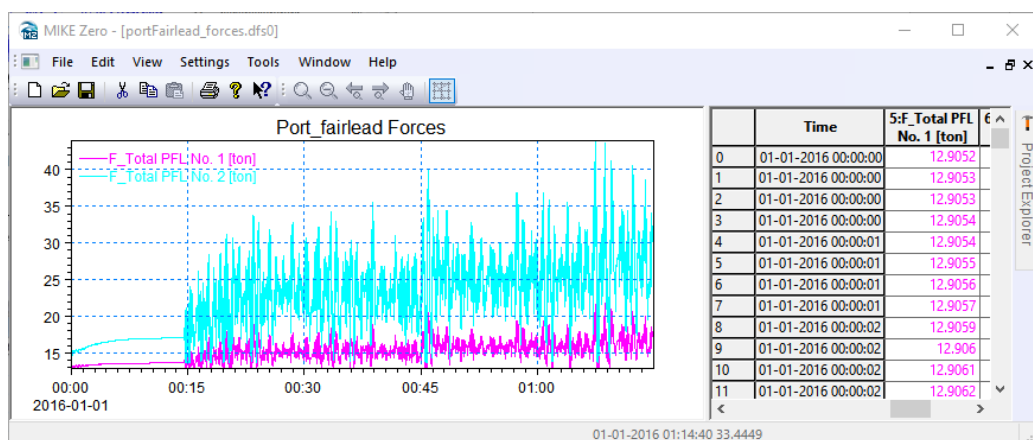The Port_fairlead forces are saved to "*portFairlead_forces.dfs0*" file .

Figure 6-8   Port_fairlead force results.