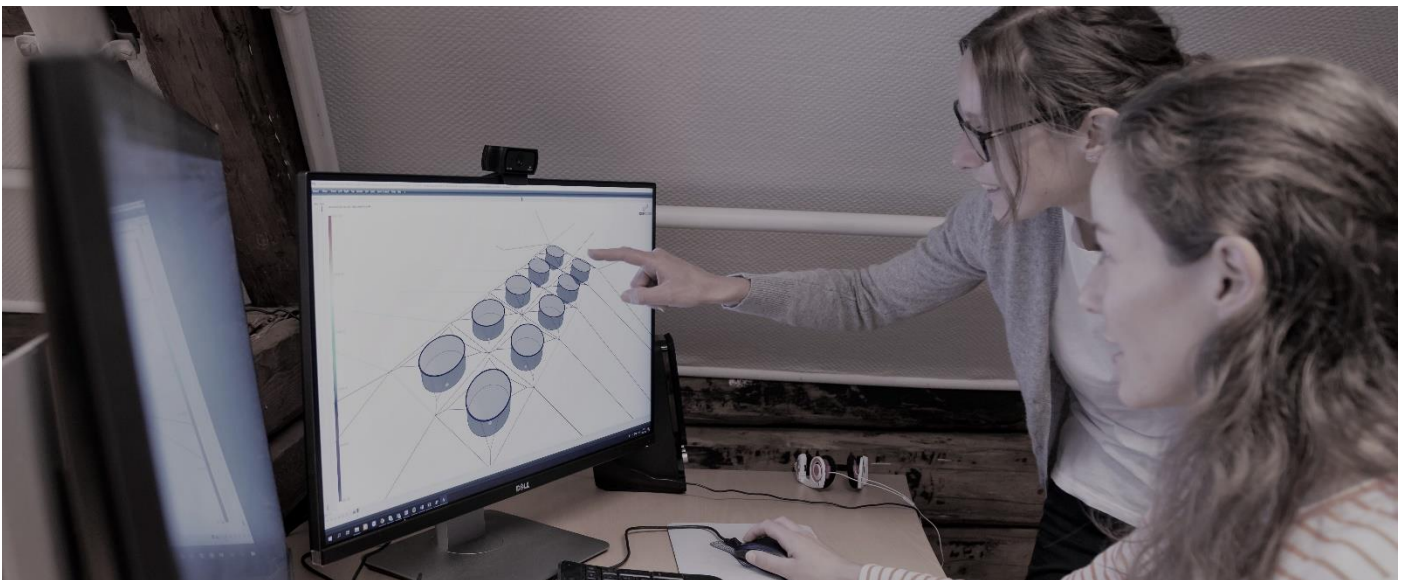


AquaSim training courses



AquaSim version: 2.17

Aquastructures AS
Kjøpmannsgata 21, 7013 Trondheim
Norway

Content

1	Introduction.....	4
1.1	About this course.....	4
1.2	Knowledge and prerequisites	4
1.3	Training files	4
1.4	Conventions and fonts in this course	4
1.5	What is AquaSim?.....	5
1.6	What is Finite Element Method?.....	5
1.7	The AquaSim package	5
1.8	Errors in FEM.....	7
1.9	Analysis process	7
2	Case study 01 – Cantilever beam with force	8
2.1	Learning objectives	8
2.2	Introduction	8
2.3	Preprocessing	9
2.4	Calculations.....	15
2.5	Postprocessing	18
2.6	Discretization	21
2.7	Summary	25
3	Case study 02 – Winch system	26
3.1	Learning objectives	26
3.2	Introduction	26
3.3	Preprocessing	27
3.4	Calculations static analysis.....	39
3.5	Postprocessing static analysis.....	41
3.6	Calculations dynamic analysis	44
3.7	Postprocessing dynamic analysis	47
3.8	Summary	50
4	Case study 03 – Aquaculture cage and mooring.....	51
4.1	Learning objectives	51
4.2	Introduction	51
4.3	Main components of aquaculture facility.....	52
4.4	Preprocessing	53
4.5	Calculations static analysis.....	75

4.6	Postprocessing static analysis.....	77
4.7	Calculation dynamic analysis.....	82
4.8	Postprocessing dynamic analysis	85
4.9	Summary	89
5	Case study 04 – Simplified aquaculture cage and mooring	90
5.1	Learning objectives	90
5.2	Introduction	90
5.3	Main components of aquaculture facility.....	91
5.4	Preprocessing	92
5.5	Calculations static analysis.....	117
5.6	Postprocessing static analysis.....	118
5.7	Calculation dynamic analysis.....	122
5.8	Postprocessing dynamic analysis	124
5.9	Summary	126
6	CaseStudy 05 – Simplified aquaculture cage and lice skirt.....	127
6.1	Learning objectives	127
6.2	Introduction	127
6.3	Preprocessing	128
6.4	Calculations dynamic analysis	130
6.5	Postprocessing dynamic analysis	132
6.6	Summary	133

1 Introduction

1.1 About this course

The objective of this course is to teach you how to apply the AquaSim software for static and dynamic analyses of structures. You will be learning fundamental aspects about preprocessing, running analysis and postprocessing of results. The course is especially aimed at the aquaculture industry, with focus on modeling and analysis of moored systems.

You will be presented a set of case studies. The complexity of the case studies increases, it is therefore recommended to complete them in sequence, starting with CaseStudy 01.

This course should be viewed as an introduction and supplement to the courses held by Aquastructures, and not as a replacement. Aquastructures offer attendance courses and online training for beginners and advanced users. Visit us on our website aquasim.no.

1.2 Knowledge and prerequisites

The course is designed so that you do not need prior knowledge of the AquaSim software. It is expected that you are confident within data processing (MS Excel and similar), and have basic skills in the following:

- Mechanics
- Knowledge of principles of hydrodynamic
- Knowledge of principles of Finite Element Method and analysis.

To start the course, it is assumed that the AquaSim package is installed locally on your computer.

1.3 Training files

The complete collection of files from this course is found in your AquaSim folder on your computer.

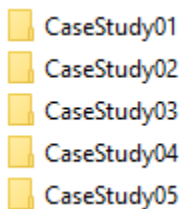


Figure 1

1.4 Conventions and fonts in this course

Conventions and fonts applied in this course are described in the table below.

Convention	Description
> (<i>greater than</i>)	Meaning the next step in your selection procedure. Example: Right click node > Nodes > Pointload.
ConsoLas	Is applied for commands and options. Example: Right click component > Type > Beam .
Oxygen	Is applied for headings that separates the main steps in each course: introduction, preprocessing, calculation and postprocessing
Trade Gothic Next Light	Indicates names, folders and files.

1.5 What is AquaSim?

AquaSim is an analysis tool developed by Aquastructures. The software is based on the calculation method Finite Element Method (FEM). It is applied for structural analysis of marine- and land-based structures.

The AquaSim package provides a complete solution from preprocessing with drawing and specification of material properties, calculations of response due to environmental loads, to postprocessing and presentation of results.

AquaSim is based on time domain analysis, meaning it accounts for time-varying loads.

1.6 What is Finite Element Method?

Finite Element Method (FEM) is a numerical technique for solving of partial differential equations. FEM is based on subdividing a structure into a discrete number of smaller elements. This is done through discretization. Simple equations are established for each element and assembled into larger systems of equations to solve the problem. Fundamentals of the Finite Element Method in AquaSim is found in the AquaSim Package Theory User Manual <https://aquasim.no/resources/documentation.html>.

1.7 The AquaSim package

AquaSim consist of three modules:

- Preprocessing with use of AquaEdit and AquaCross
- The calculation engine, also known as solver
- Postprocessing with use of AquaView and AquaTool

1.7.1 AquaEdit and AquaCross

AquaEdit is applied for preprocessing and preparation of analysis. The numerical model is established through a graphical interface. The components that constitute the model is given properties in terms of geometric cross-section, stiffness, weight, and material properties. AquaCross is an add-on in AquaEdit for drawing of complex cross-sections. Discretization of your model is done in the preprocessing phase, where the model is divided into a discrete number of elements.

The user interface is set up as most computer aided programs – with symbols and options. In AquaEdit, the analysis parameters are defined in terms of environmental loads, time series, result- and export selections.

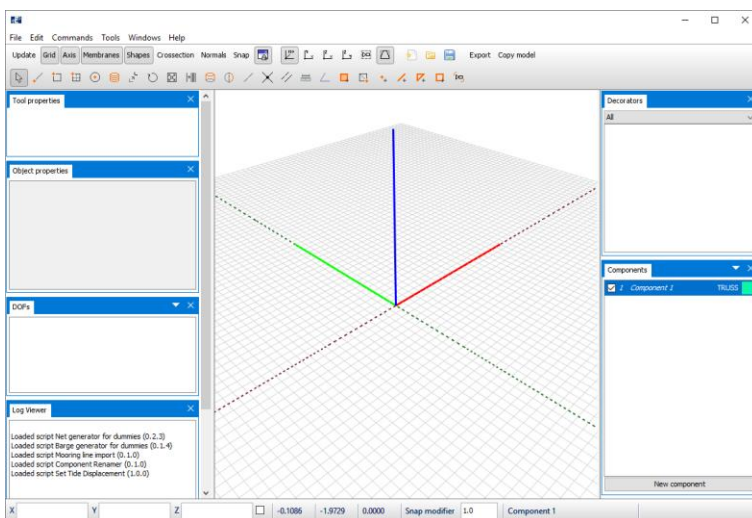


Figure 2 AquaEdit

AquaSim utilize four main component types to resemble structures, these are:

- TRUSS component: applied for wire, rope, chain and similar structures with circular cross sections. Capable of withstanding tensile and compressive loads.
- BEAM component: applied for structures with more complex cross sections, such as I-beams, floating collars, barges and so on. Capable of withstanding axial-, torsional-, and bending loads.
- MEMBRANE component: applied for rendering of net pens, tarpaulins, shell structures and so on.
- NODE2NODE component: applied for contact forces and similar. This is a component with a spring in each end of the discrete element.

1.7.2 Calculations

The numerical model is exported to the calculation engine which solves the partial differential equations based on the input from AquaEdit.

1.7.3 AquaView and AquaTool

The AquaSim package provide two regular postprocessing tools: AquaView and AquaTool. Good visualization of the results is important for the understanding and safety of the analyses. AquaView provides a 3D presentation of your numerical model along with resulting deformations, forces, and stresses.

AquaTool give access to the same results as in AquaView, only the presentation is different. AquaTool provides results in terms of tables and graphs, useful when handling large datasets.

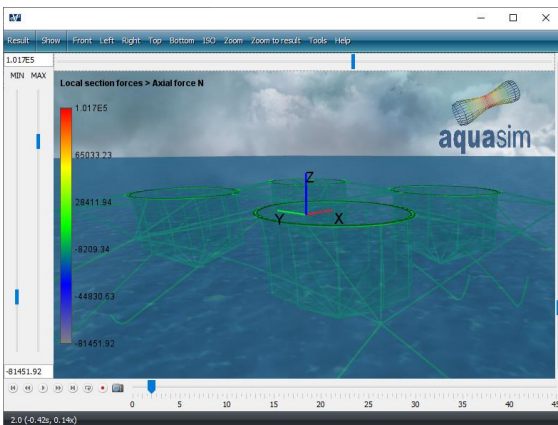


Figure 3 AquaView

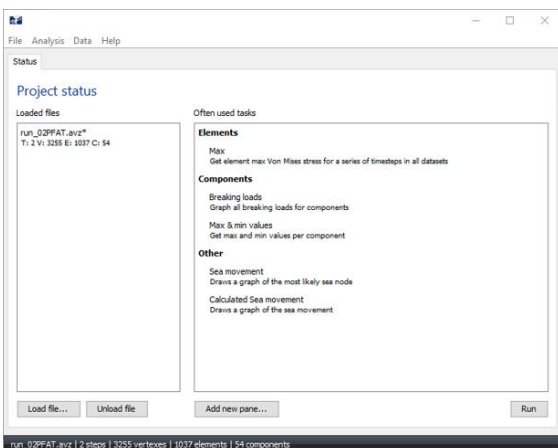


Figure 4 AquaTool

1.8 Errors in FEM

When building and discretizing models, errors may occur. Errors can originate from the model or in the calculations. Model errors are typically connected to discretization of the model, properties of components or inappropriate boundary conditions. This often leads AquaSim unable to calculate equilibrium between internal forces (cross section properties) and external forces (environmental loads), and the calculations tend to diverge. It is therefore important prior to building a model, to assess the nature of the problem: what is the purpose of your analysis, what are you going to investigate, can the problem be simplified?

Errors originated from the calculations are often connected to round-off problems from the solver. These errors are for most practical purposes very small.

1.9 Analysis process

Regardless of the problem investigated, you are going through the same process every time. Problem solving in AquaSim may be divided in the following main steps:

Preprocessing:

Creating a model:

The problem is defined, and a model is established in AquaEdit.

Cross section properties:

The model components are given cross sectional properties in terms of geometric cross-section, stiffness, weight, and material properties.

Discretization:

Discretization is conducted parallel to establishing the model.

Boundary conditions:

Boundary conditions are set mostly in terms of constraints at nodes. Allowing, or disallowing, translations, and rotations in 3D space.

Calculations:

Analysis parameters are defined prior to export of the model, these are set in terms of environmental conditions (wind, current and waves), time domain parameters, result selections and export alternatives. The model is exported, and calculations starts.

Displacements and stresses are calculated.

Postprocessing:

Results are assessed in either AquaView or AquaTool.

2 Case study 01 – Cantilever beam with force

2.1 Learning objectives

Upon completion of this case study, you will be able to:

- Navigate in AquaEdit
- Specify properties of beam component
- Verify model
- Know basic properties of analysis parameters
- Establish an analysis folder and run analysis
- Open, and navigate, in AquaView
- Assess the influence of model discretization

2.2 Introduction

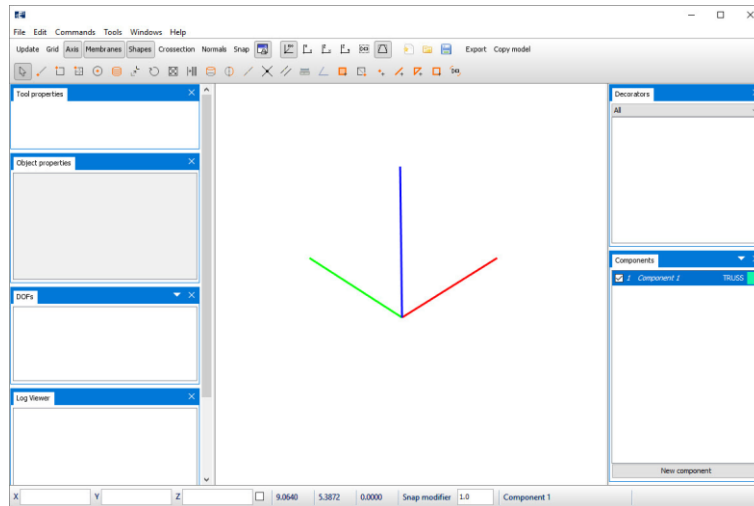
In this case study you will determine the shear force of a cantilever beam with rectangular cross section. The beam is exposed to a horizontal point load, giving rise to shear forces.



2.3 Preprocessing

1 Load preprocessing tool

From the Start menu on your computer search AquaEdit, or double click on AquaEdit shortcut from your desktop. The preprocessor AquaEdit will be loaded.



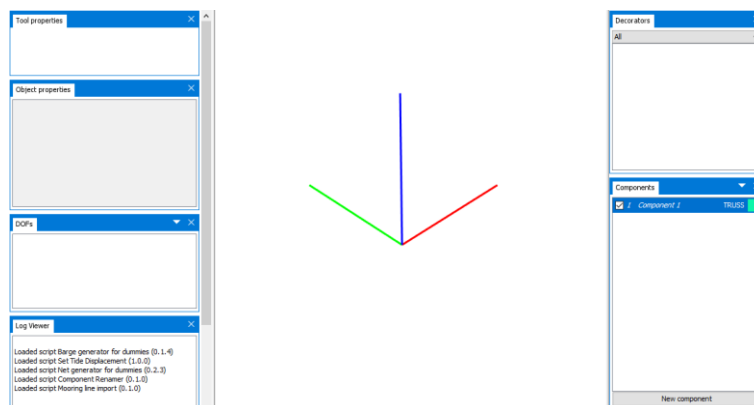
2 Review interface of AquaEdit

In this section, an introduction to the interface of AquaEdit is presented. For more detailed information, please see the AquaEdit User Manual found in your installation folder Aquastructures\AquaSim\doc or at aquasim.no.

The Toolbar provides quick access to the most common used features: settings for 3D view, camera views, drawing intents and export menu.



The main view is where the model is created, showing nodes and elements with respect to the global coordinate system.



The left and right columns in the main view are docking areas for object-, decorator- and component property windows. These windows can be dragged and dropped for customized views.

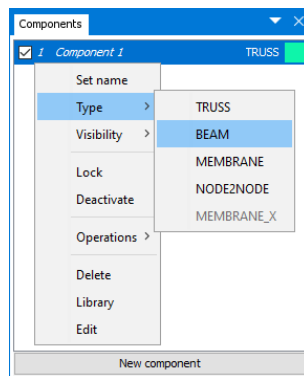
The Components window provides a list of all the components in your model. AquaSim provides four main component types (TRUSS, BEAM, MEMBRANE, NODE2NODE) as described in section 1.7.1. Each component represents a collection of element properties in your model.

The lower most toolbar provides access for typing coordinates of nodes and elements (useful when you need accurate positioning). Further, coordinates of your mouse pointer in the 3D view and tooltips are found here.





3 Generate beam component

You are to establish a BEAM component. In the Components window, AquaEdit by default present the first component as type TRUSS. To change this, right click 1 Component > **Type**, select **BEAM**.



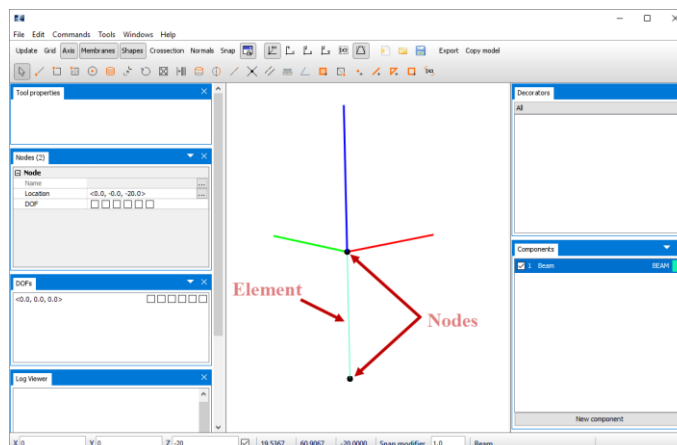
4 Draw beam

Select the intent **Draw line**  from the Toolbar menu. Move the mouse pointer down to the lower most toolbar to type the coordinates of the first node, **X=0 Y=0 Z=0** , press **ENTER** on your keyboard.

Now, select the checkbox to the right of coordinate Z. This will activate relative coordinates to the first defined node, type **X=0 Y=0 Z=-20**

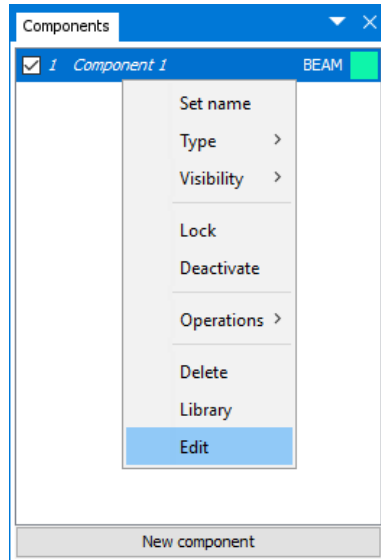


, press **ENTER** on your keyboard. Then, exit the Draw line intent by selecting **Pointer**, or press **ESC** three times. You should now have one element connected with two nodes, in your 3D viewer.

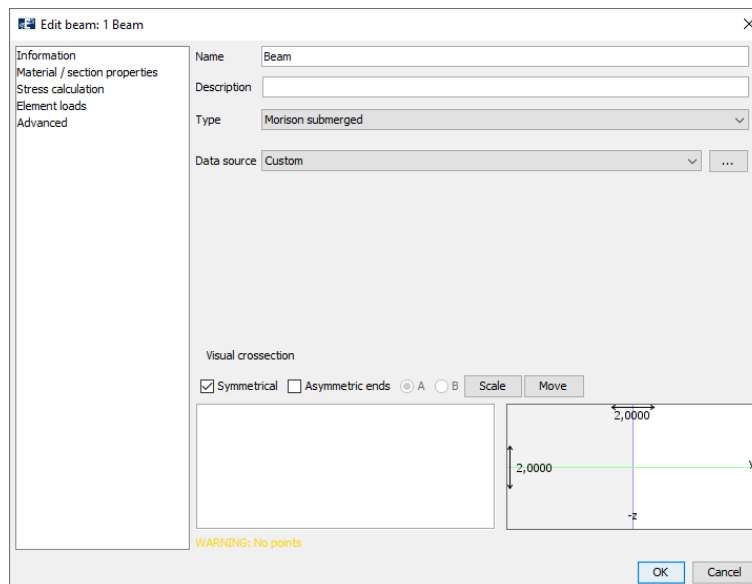


5 Properties of Beam component

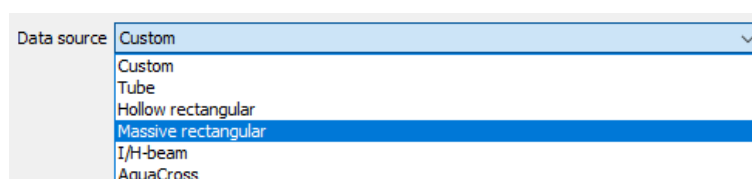
You should give the beam cross sectional properties by accessing the Edit beam window. Note that the component name is in *italic*, this indicates that it lacks essential properties for analyzes to be performed. Right click 1 Beam > **Edit**.



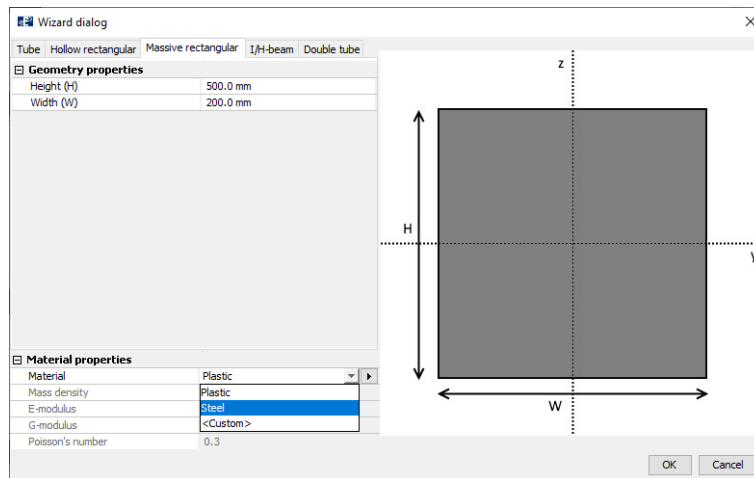
In the Edit beam window, the first view you meet is the Information-tab. In the Information-tab you may assign general information to your beam and either draw a cross section from predefined sections or access AquaCross for customized sections. In the row **Name**, type the component name Beam.



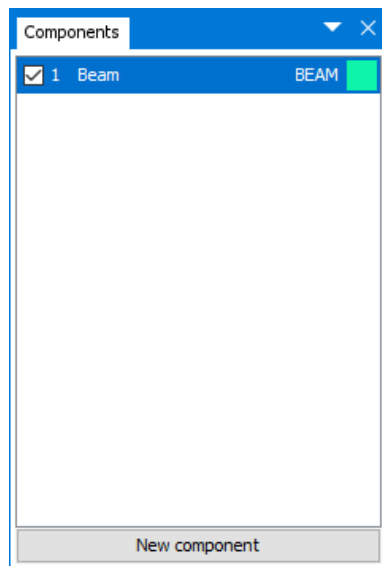
Press the drop-down menu from **Data source** > **Massive rectangular**.



The wizard dialogue for predefined sections is opened. Change **Material** from **Plastic** to **Steel**. Press **OK**, and then **OK** in the Edit beam window.

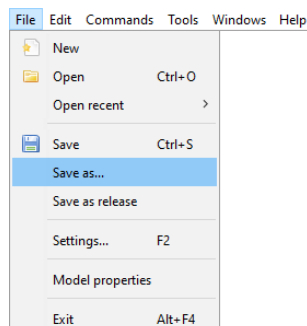


The name of the component should have changed from italic to straight font, meaning the beam has adequate properties.



6 Save the model

Good saving routines during modelling is important to ensure that work is not lost. In the top menu, select **File > Save as...**

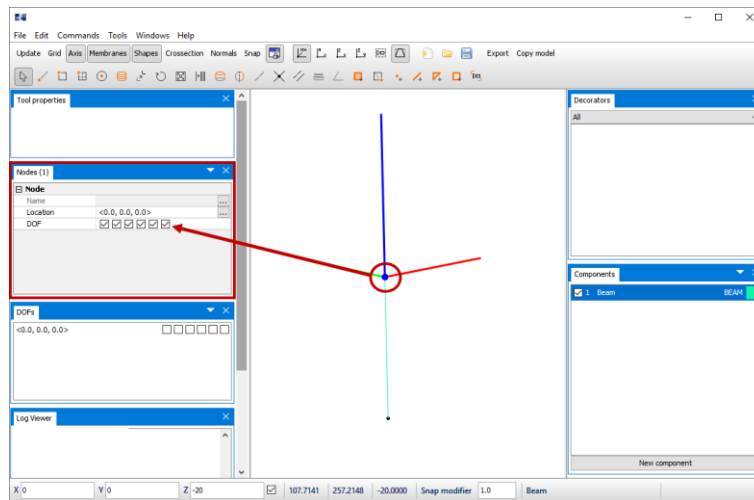


Navigate through the File Explorer to a suitable place to save your model. In this case study, we have created a folder in Documents\AquaSim called Analyses\CaseStudy01. Assign model name Beam and press **Save**.

7 Assign boundary conditions

The cantilevered beam should be restrained from translation and rotation at the top end and exposed to a point load in x-direction at the lower end.

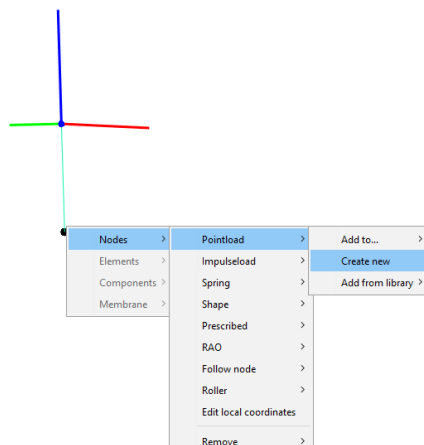
In the 3D view, use the mouse pointer and click on the node in the origin of the global coordinate system. The Nodes property window will be activated in your left docking area. In the row **DOF** (Degrees Of Freedom), select all the checkboxes.



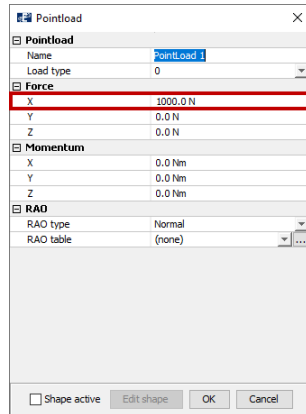
In the DOF row (counted from left to right): the first three checkboxes represent translation about x-, y- and z-axis respectively. The three last checkboxes represent rotation about x-, y- and z-axis respectively. One selection represents a constraint, so when all the checkboxes are selected the node is restrained in all Degrees Of Freedom. Having one or more restraints, the node in the 3D view will change from black to blue.

8 Assign point load

The point load is to be assigned to the free end of the beam. In the 3D view, use the mouse pointer to select the free end of the beam. **Right click > Nodes > Pointload > Create new**.

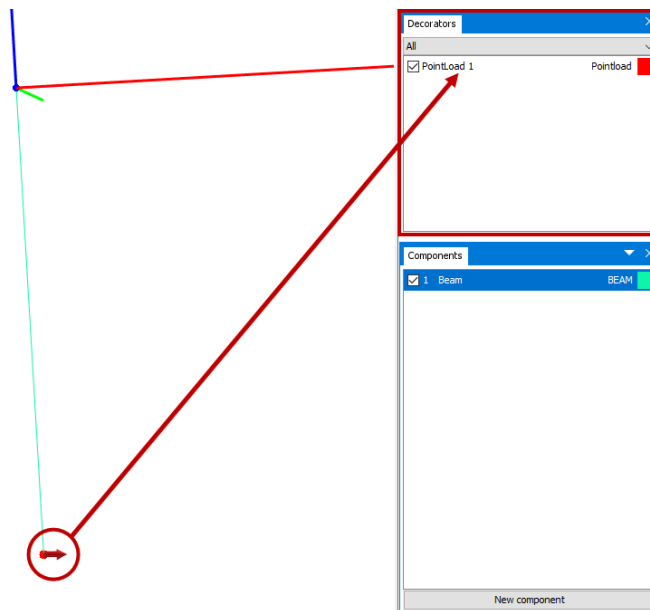


In the Pointload window, type **Force X=1000**. The directions for force and momentum are applied with respect to the global coordinate system. A positive value in x-direction correspond to a force acting in the positive x-direction. Click **OK**.



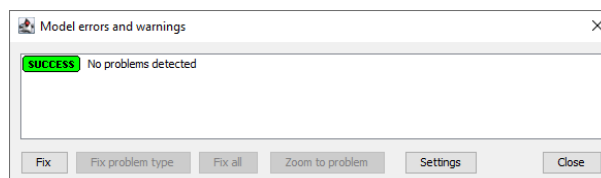
The Decorators window in the docking area to the right lists the node decorators in the model. In addition to Pointload, nodes may have decorators such as impulse load, spring, shape and more.

In the 3D view, the node with the Pointload change color from black to red. In addition, the arrow indicates the direction of the applied load.




9 Verify model

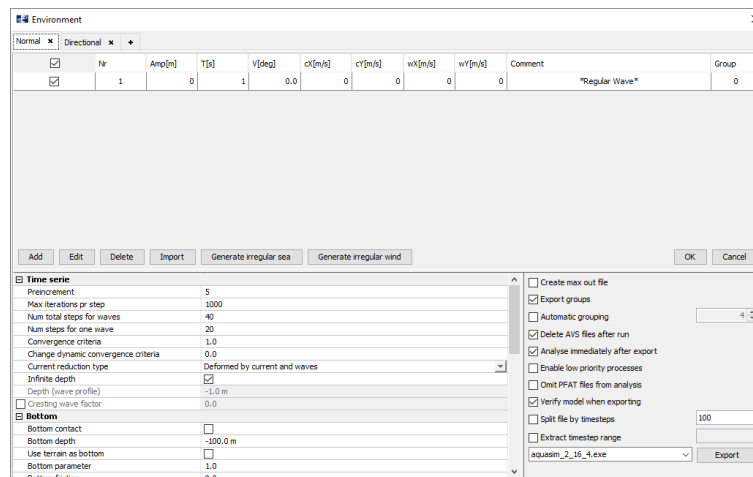
The model is complete. To verify the model is correct and without errors, select **Commands > Verify model**. If the model has errors, AquaSim will suggest solution to fix it. Read more about errors and warnings in the AquaEdit User Manual.



2.4 Calculations

10 Enter Environment window

Now that you have a complete model without errors, the next phase is to prepare the analysis. Before analysis can start, export parameters must be defined. In the Toolbar menu, select **Export** .



The Environment window is opened. Here, parameters for environmental loads (wave height, current and wind) are defined. In addition, analysis parameters such as discretization of water surface, configuration of seabed, type of results to be reported and so on is set.

11 Environmental loads

In the upper part of the Environment window parameters for environmental data is defined. AquaSim is based on analysis where the user defines sets with combination of wind, current and waves. These sets are referred to as load conditions or sea states. Details of this will be discussed later in this course.

12 Time serie parameters

AquaSim apply static and dynamic loads in incremental steps. This is to ensure a stable analysis and that a numerical solution is found. Static-, (quasi)static and dynamic loads are applied in order:

Static loads: static equilibrium is calculated between internal and external loads. Examples: weight, buoyancy, hydrostatic pressure, point loads.

(Quasi)static loads: are applied gradually in a predefined number of steps, until it is fully developed. These steps are defined in **Preincrement**.

Examples: current and regular wind.

Dynamic loads: are applied on top of the static and (quasi)static loads, following the same principle with gradually increasing loads. These steps are defined in **Num total steps for waves** and **Num steps for one wave**. Examples: regular- and irregular waves and wind.

You are to conduct a static analysis, so Preincrement is the parameter to define. In the row **Preincrement**, type 5. In the row **Num total steps for waves**, type 0.

Time serie	
Preincrement	5
Max iterations pr step	1000
Num total steps for waves	0
Num steps for one wave	20
Convergence criteria	1.0
Change dynamic convergence criteria	0.0
Current reduction type	Deformed by current and waves
Infinite depth	<input checked="" type="checkbox"/>
Depth (wave profile)	-1.0 m
<input type="checkbox"/> Cresting wave factor	0.0

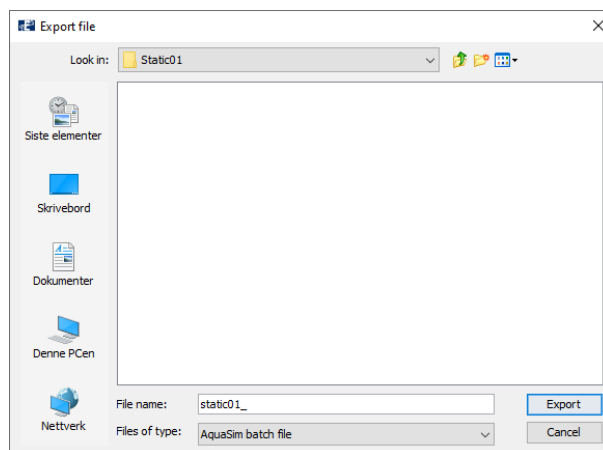
13 Export selections

In the lower right corner of the Environment window, export selections are available. Here you may choose what types of files to be generated/ deleted upon export, priority and grouping of analysis and selection of available solver versions. Make the same selections as in the figure below.

<input type="checkbox"/> Create max out file	
<input type="checkbox"/> Export groups	
<input type="checkbox"/> Automatic grouping	4
<input checked="" type="checkbox"/> Delete AVS files after run	
<input checked="" type="checkbox"/> Analyse immediately after export	
<input type="checkbox"/> Enable low priority processes	
<input type="checkbox"/> Omit PFAT files from analysis	
<input checked="" type="checkbox"/> Verify model when exporting	
<input type="checkbox"/> Split file by timesteps	100
<input type="checkbox"/> Extract timestep range	
aquasim_2_16_4.exe	<input type="button" value="Export"/>

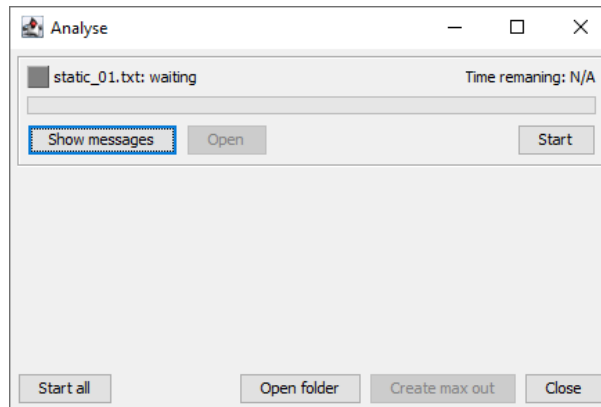
Select **Export**.

You must decide for where you want to save the analysis. Navigate to an appropriate place on your computer. In this case study, we have created an analysis folder in Documents\AquaSim\Analyses\CaseStudy01 called Static01. Type an analysis name e.g., static01_.

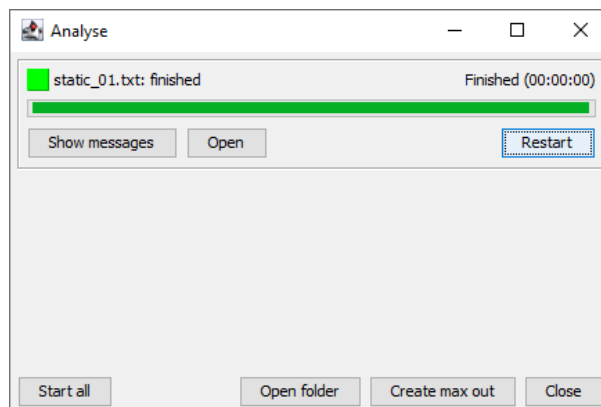


Select **Export**.

14 Run analysis Upon selecting Export, you will be presented with an analysis window. To start the analysis, press **Start** or **Start all**.



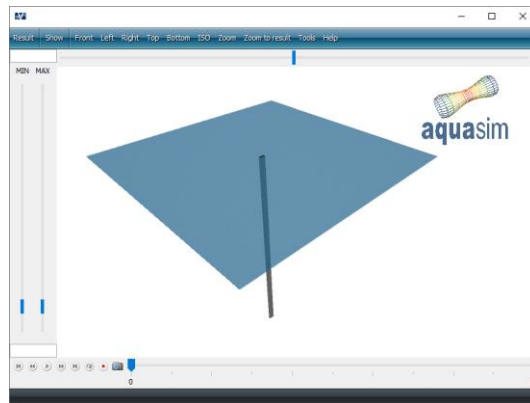
The progress of the analysis is shown with a green bar moving from left to right. When the analysis is finished, the analysis window will have status **finished**. You have now successfully completed the analysis.



2.5 Postprocessing

15 Open result file

In the Analyse window, select **Open**. This will load the postprocessing tool AquaView. The model is presented in a 3D view as it was built in AquaEdit.



The blue square seen in the 3D view, is the water surface. This is adjustable in size and transparency.

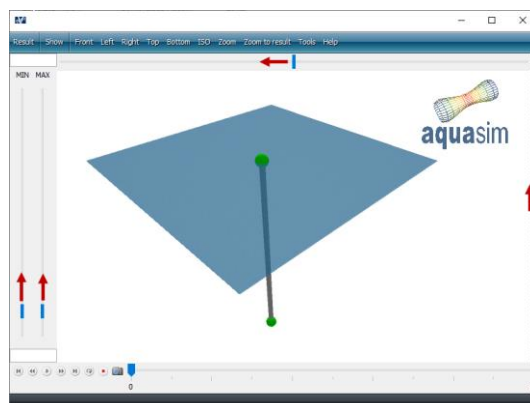
16 Review interface of AquaView

In this section, an introduction to the interface of AquaView is presented. For more detailed information, please see the AquaView User Manual found in your installation folder Aquastructures\AquaSim\doc or at aquasim.no. The top Toolbar menu provides access to view analysis results, camera views and tools.

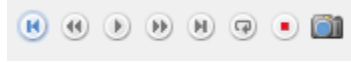


The main view is where the model is presented with results in time-domain. Several blue bars are seen to the left, right and above the main view:

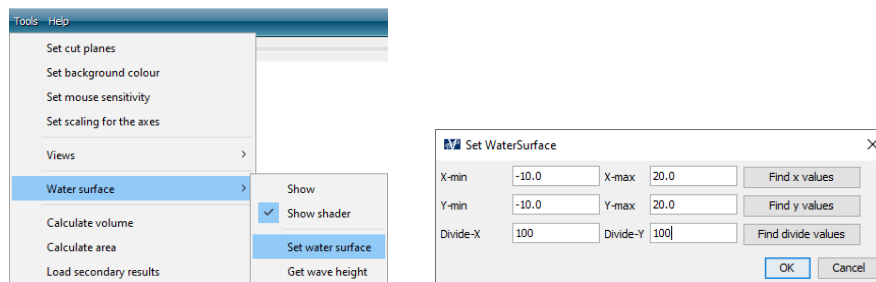
- Left side: controls the Graphic color palette. By click and drag, the color rendering of results in the main view is adjusted.
- Right side: controls the visual size of the nodes for a selected element in the model.
- Top: controls the transparency of the water surface.



At the lower most part of AquaView, you find the playback controls. It enables for continuous playback of the analysis, as well as choosing specific timesteps. Options for recording of videos and screen shots are also found here.

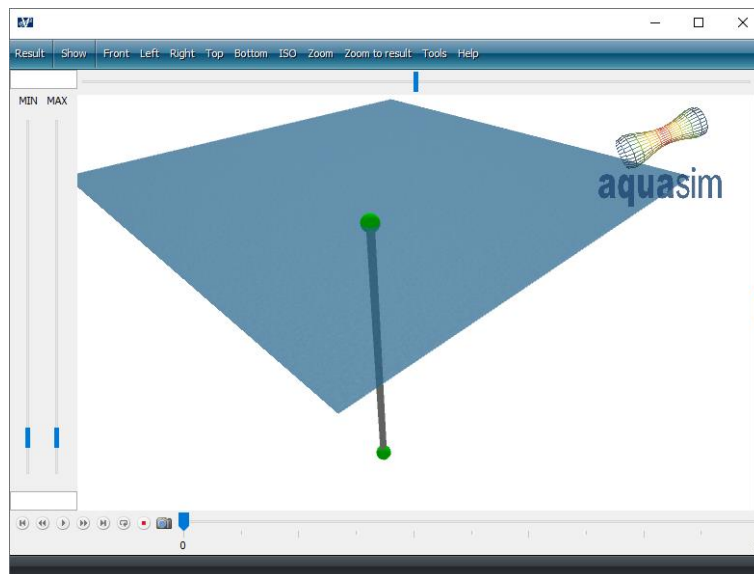


17 Water surface You should adjust the size of the water surface. From the top Toolbar menu, select **Tools > Water surface > Set water surface**.




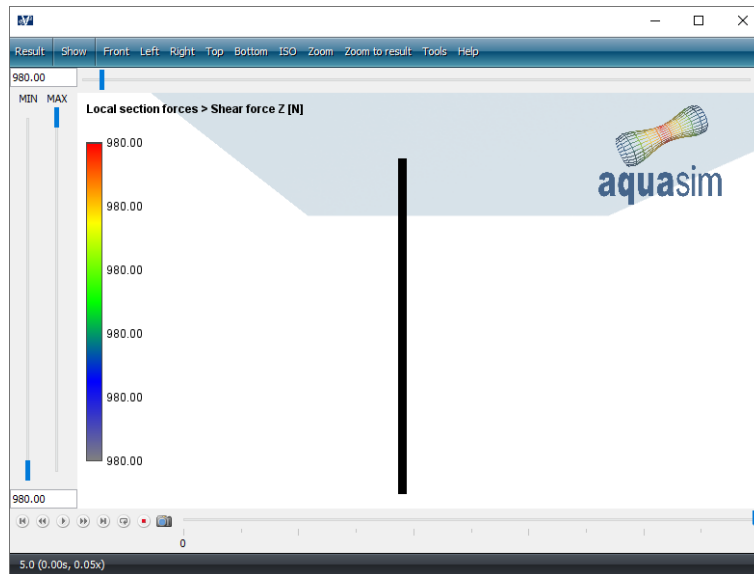
In the Set water surface window type **X-min=-10 Y-min=-10 X-max=20 Y-max=20**. Select **OK**.

The size of the water surface is updated.



18 Plot shear force

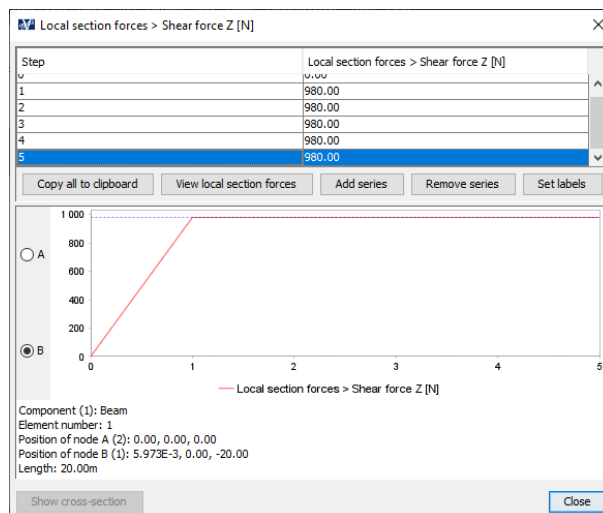
From the top Toolbar menu select **Result > Local section forces > Shear force Z [N]**. You should see the shear force in z-direction for the beam in the 3D view. By pressing  the results from time domain analysis are played. Observe how the beam deform and the colors change as a function of analysis steps.



Step 1 is results from calculation of static equilibrium between internal force in the beam (weight, buoyancy) and the external force from the point load. Step 2-5 are identical to the results in step 1, this is due to only static forces are defined prior to calculation. (Quasi)static loads (current, regular wind) are not included in the analysis.

19 Graphing results for element

The results may also be presented in a graph as a function of analysis steps. Use your mouse pointer and **right click** on the beam element in the 3D view. The Element result window will appear, presenting the results both as a table and in a graph.





As mentioned earlier, each element is connected by two nodes. In the Element result window, results are presented for node A (the first node of the element) and node B (the second node of the element). By selecting the checkbox **B**, shear force for the free end is presented. Take note of the maximum shear force at node B is 980.0 N at step number 5.

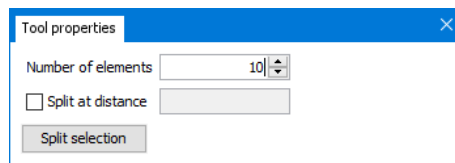
This section is now completed, **close** the Analyse window.

2.6 Discretization

20 Discretization One important aspect was not considered when the model was established in AquaEdit: discretization. In this section, you are going to see the effect of discretization of the model on analysis results.

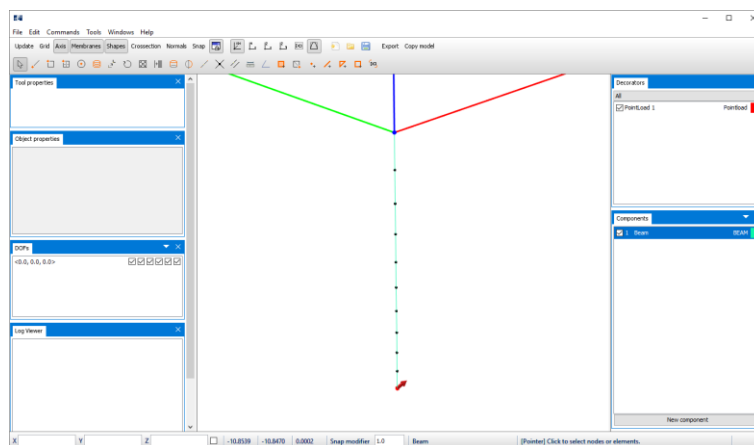
Go back to your model in AquaEdit or chose the premade model from Documents\AquaSim\Demos\CaseStudy_01\CaseStudy01.amodel. The premade model is loaded in AquaEdit by **double click** on the file.

You should divide the element into smaller entities. From the intents in the Toolbar menu, select **Split line** . The Tool properties window is activated in the left docking area, with settings for splitting of elements. In the row **Number of elements**, type 10 and press **ENTER** (alternatively select the arrow  until you get 10).



Move the mouse pointer to the 3D view until it is situated above the beam element, then **left click**.

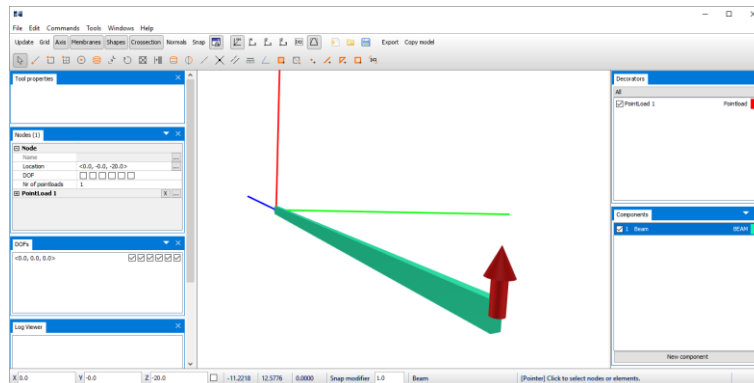
Your beam should now consist of 10 elements.



All the elements that constitute your beam share the same properties because they are grouped within the same component Beam.

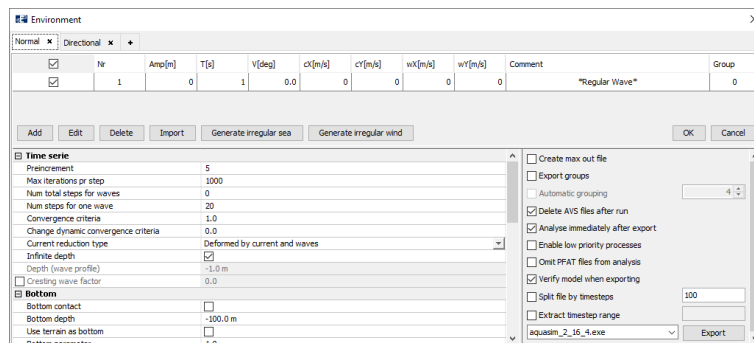
21 Visual cross section

Until now, your beam is just presented as a straight line in the 3D view. From step number 5 in this case study, you remember the beam was assigned a visual cross section. To turn on the display of the cross section, select **Crosssection** from the Toolbar. Then, zoom into the beam by **Scrolling** upwards.



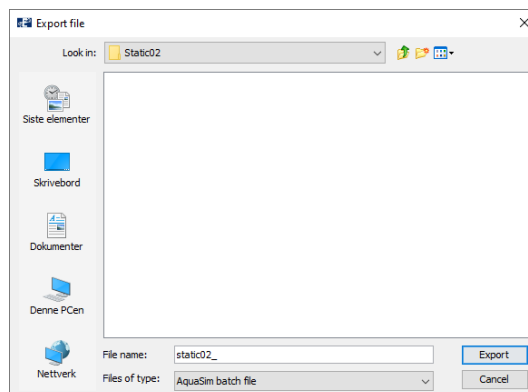
22 Enter Environment window

The model is now ready for analysis. Access the Environment window by selecting **Export** from the Toolbar. Leave the Time serie parameters and Export selections as described in section 2.4.



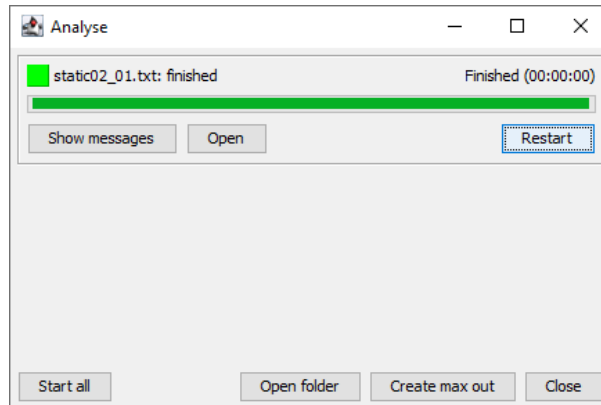
23 Export selections

In the Environment window, select **Export**. Create a new folder for analysis. In this case study, we have created an analysis folder in Documents\AquaSim\Analyses\CaseStudy01 called Static02. Give the analysis a name e.g., static02_.




Select **Export**.

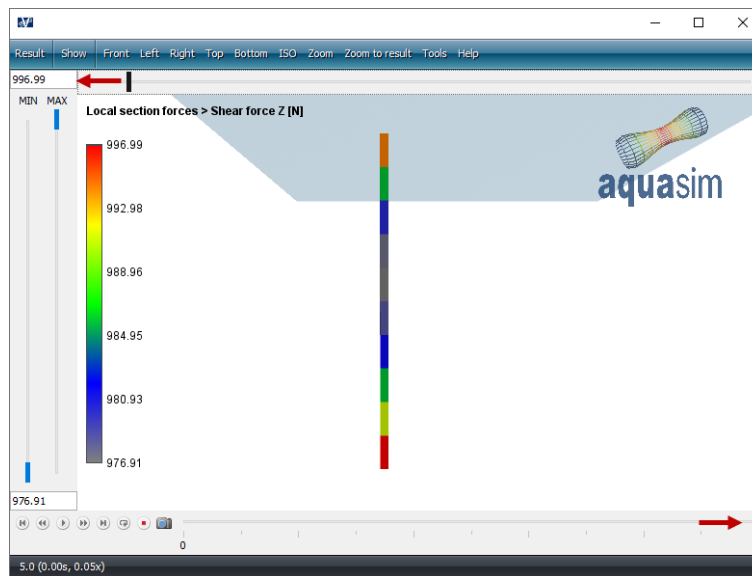
24 Run analysis The Analysis window appears. To start the analysis, press **Start** or **Start all**.



When the analysis changes the status from **waiting** to **finished**, select **Open** in the Analysis window. AquaView should be loaded.

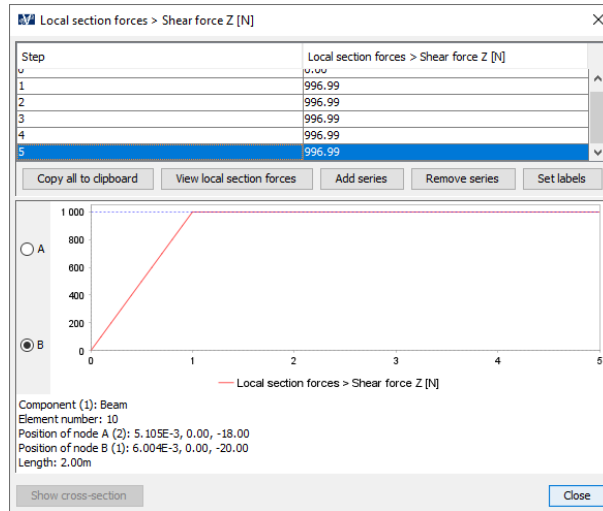
25 Plot shear force

From the top Toolbar menu in AquaView, select **Result > Local section forces > Shear force Z [N]**. Jump to the last analysis step by selecting  from the Playback control in the lower left corner of AquaView. Adjust the transparency of the water surface by dragging the blue bar in the top of the 3D view to the left.



You may observe that the color of the beam is different for the first analysis.

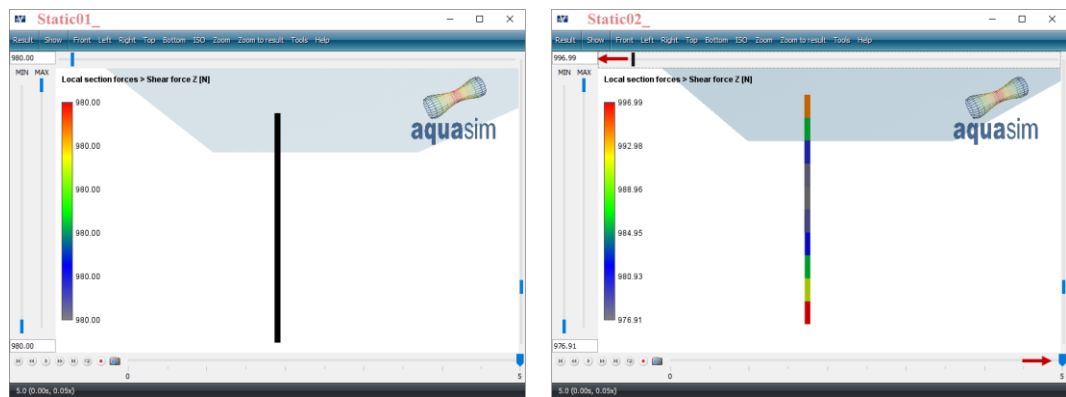
Use your mouse pointer and **right click** on the free end of the beam in the 3D view. The Element result window will appear, presenting the results both as a table and a graph.



By selecting node B, the shear force for the free end is presented. Take note of the maximum value at node B is 996.99 N at step number 5.

26 Comparing results and summary

By comparing the two analyses static01_ and static02_ it is seen that the discretization affects the results. Both color rendering and the magnitude of the shear force is affected.



Model	Shear force Z [N]
1 element (Static01_)	980.0
10 elements (Static02_)	996.99

You can experience smaller differences between your own results, and the one presented here. This may be due to different solver versions being applied. We see that the shear force increases with more elements in the model. This is because the beam becomes less stiff with increased number of elements.

This case study is now finished.

2.7 Summary

We modelled a simple beam to introduce the AquaSim interface and analysis functionality. A static analysis was conducted with two different discretization of the model: one with 1 element and the second with 10 elements.

We have seen that the number of elements in the beam affect results from the analysis, it becomes more accurate with more elements. If you had divided the beam into more than 10 elements, you might see the shear force increase slightly more.

At some point you will see that further increase of elements in the model will not affect the results; it converges to a final value. However, the more elements your model contain, the more comprehensive and time-consuming the calculations becomes.

3 Case study 02 – Winch system

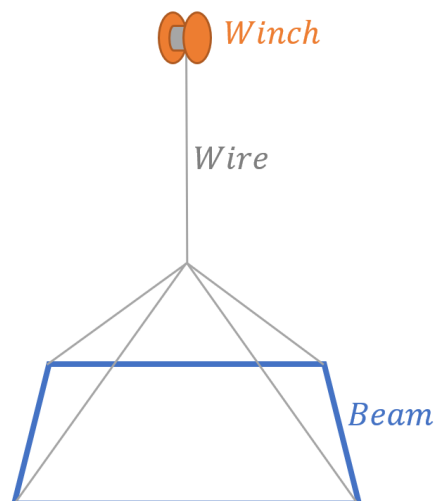
3.1 Learning objectives

Upon completion of this case study, you will be able to:

- Assemble a model with different component groups
- Specify properties of TRUSS and BEAM elements
- Establish winch as node decorator
- Add built-in shape to node
- Verify model input through static analysis
- Execute static analysis
- Execute multiple non-linear dynamic analyses
- Copy results from AquaView to Excel

3.2 Introduction

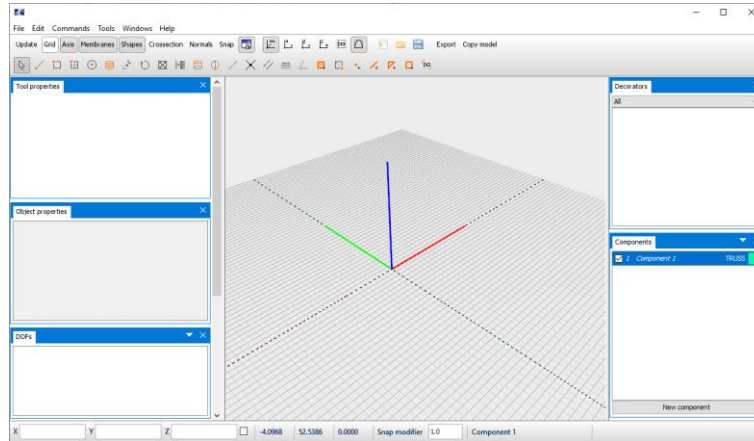
In this case study you will establish a system consisting of wire, beam, and winch. The wire is winching the beam from air through the water surface. You are going to assess the forces in the wire for two combinations of environmental loads (also referred to as load conditions). Before dynamic analysis is conducted, the weight of the system should be controlled through a static analysis.




3.3 Preprocessing

1 Load AquaEdit

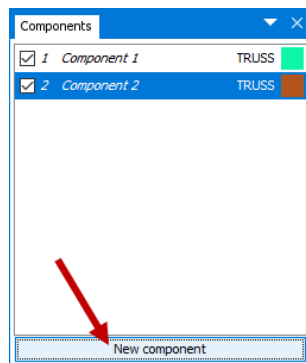
As you learned from the previous case study, AquaEdit is the preprocessing tool, and where the analysis parameters are defined prior to export. From the Start menu on your computer, search, and load AquaEdit.



The grid in the 3D view can be toggled on or off by selecting  from the Toolbar menu.

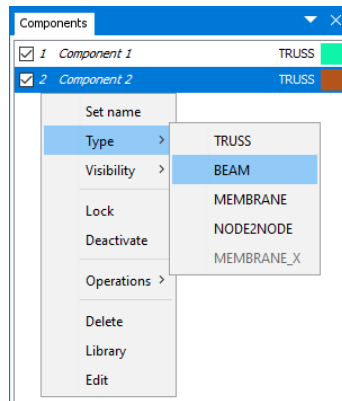
2 Create component groups

In this case study you will need two component groups: one for the wire and one for the beam structure. In the Components window, select **New component**.



The wire should be of component type TRUSS, and the beam should be type BEAM. Component type TRUSS is a structural element capable of only resisting tensile and compressive forces. Since a wire is flexible and bends easily, TRUSS is selected. The beam in the end of the winch system, is intended to be stiff, resisting both axial- and bending forces. Therefore, the BEAM is selected.

By default, AquaSim generate new components of type TRUSS. You need to change one of the component types. In the Components window, **right click** Component 2 > **Type** > **BEAM**.



Note that when changing component type to BEAM, Component 2 jumps to the top of the list. AquaSim order the component types with BEAM at the top, then MEMBRANE, TRUSS and last NODE2NODE.

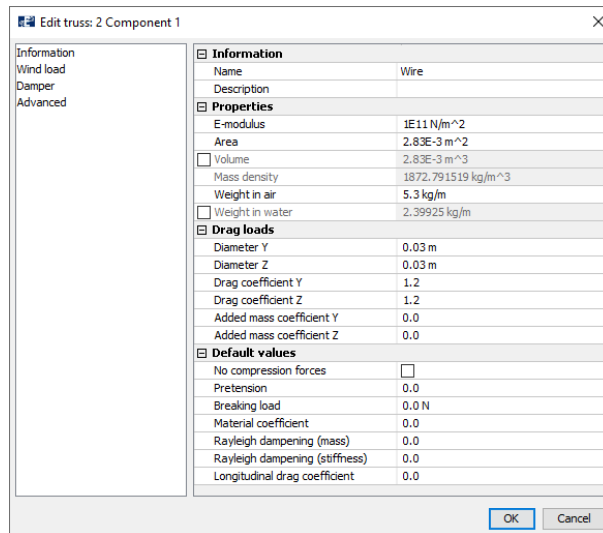
3 Component properties

The wire and beam have cross sectional properties as specified in the table below.

Properties of wire	Value
E-modulus	1.0E+11 N/m ²
Area	2.83E-03 m
Weight in air	5.3 kg/m
Diameter	0.03 m
Properties of beam	Value
Length x Width	1000 mm x 1000 mm
Density	1200 kg/m ³
E-modulus	2.1E+11 N/m ²
G-modulus	8.08E+10 N/m ²
Poisson's number	0.29 -
Weight in air	1200.0 kg/m

In the Components window, **double click** Component 1 to access the Edit truss window.

Give the component a suitable name, we have called it Wire. Type the information from the table in the Edit truss window.

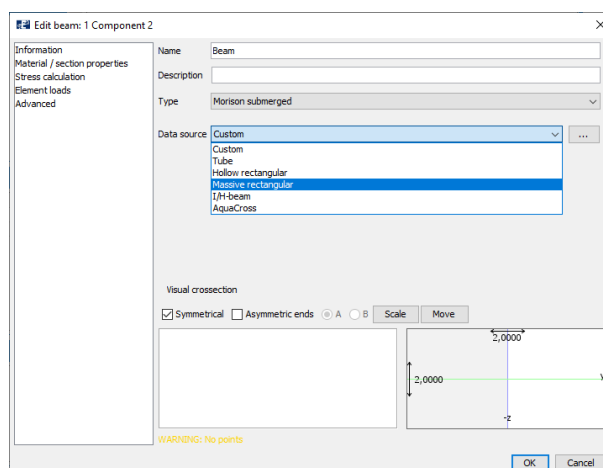


Information	
Name	Wire
Description	
Properties	
E-modulus	1E11 N/m ²
Area	2.83E-3 m ²
<input type="checkbox"/> Volume	2.83E-3 m ³
Mass density	1872.791519 kg/m ³
Weight in air	5.3 kg/m
<input type="checkbox"/> Weight in water	2.39925 kg/m
Drag loads	
Diameter Y	0.03 m
Diameter Z	0.03 m
Drag coefficient Y	1.2
Drag coefficient Z	1.2
Added mass coefficient Y	0.0
Added mass coefficient Z	0.0
Default values	
No compression forces	<input type="checkbox"/>
Pretension	0.0
Breaking load	0.0 N
Material coefficient	0.0
Rayleigh dampening (mass)	0.0
Rayleigh dampening (stiffness)	0.0
Longitudinal drag coefficient	0.0

Select **OK**.

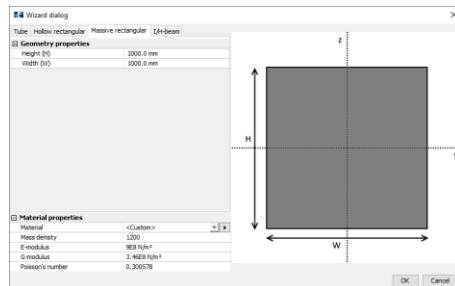
In the Components window, **double click** Component 2 to access the Edit beam window. In the row **Name**, give the component a suitable name. We have chosen Beam. The row **Type** provides a dropdown menu for selection of load formulation. A load formulation is a set of differential equations for calculation of external loads on the component. Morison submerged calculates the loads based on Morison equation. Hydrodynamic calculates the loads based on strip theory in combination with Morison equation. Choose **Morison submerged**.

From the drop-down menu in **Data source** select **Massive rectangular**.



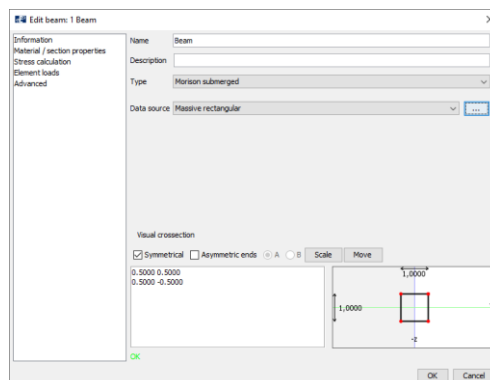
WARNING: No points

The Wizard dialogue for cross sections will be loaded. In the section for **Material properties**, select **<Custom>** from the **Material** list. Type the values provided for density, E-modulus, G-modulus and Poisson's number. In the **Geometry properties section**, type the dimensions of the cross section.

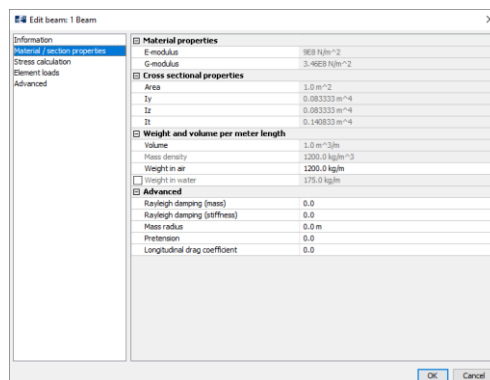


Select **OK**.

A preview of the visual cross section is found in the lower part of the Information tab. It is possible to edit this by manually typing coordinates of the cross section and move it up or down with respect to the local coordinate system. This will not affect the calculated cross section parameters as specified in the Wizard dialogue.



One parameter is remaining for the beam: the weight in air. Select the **Material / section parameters**-tab to the left in the Edit beam window. Here, cross sectional parameters for the beam are presented. Navigate to the row **Weight in air** and make sure it says 1200.0 kg/m.



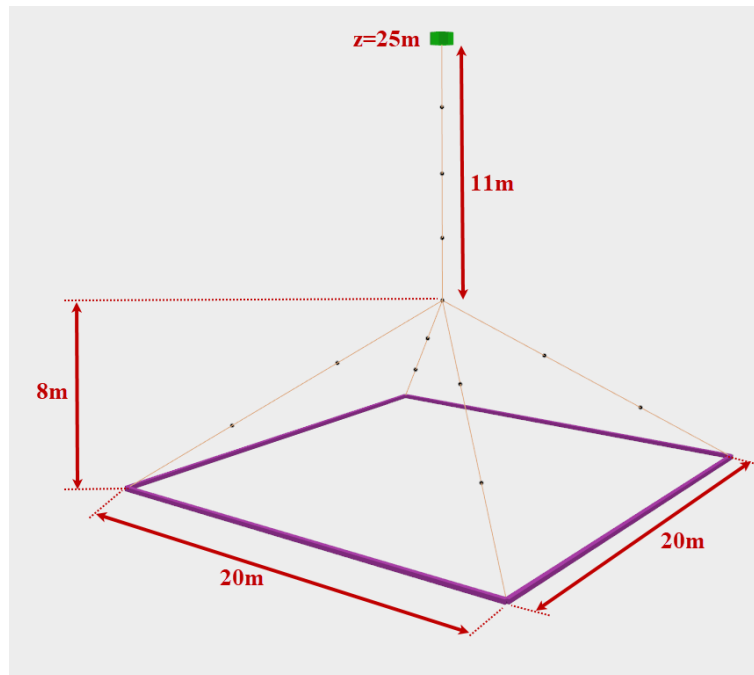
Select **OK** in the Edit beam window.

Notice the name of the component is changed from Component 2 to Beam, and that the font is changed from italic to straight. Indicating the component has necessary properties for analysis to be conducted.



4 Winch system, drawing

The winch system should have the dimensions as described in the figure below.




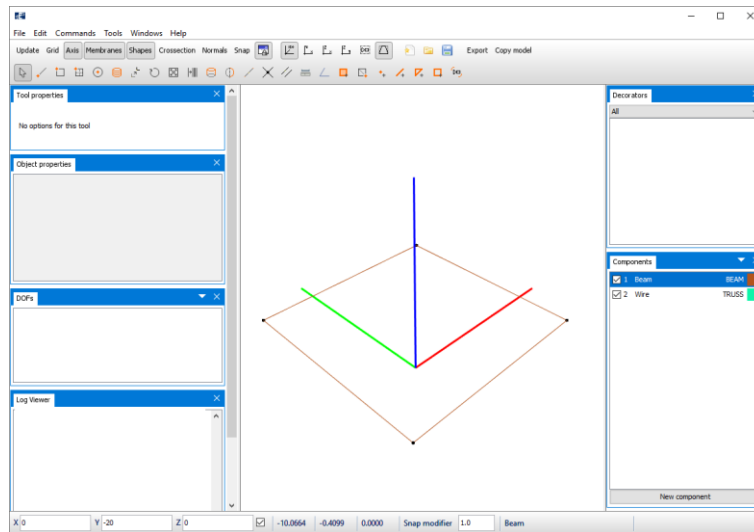
You are first to model the beam, then the wire.

5 Draw beam

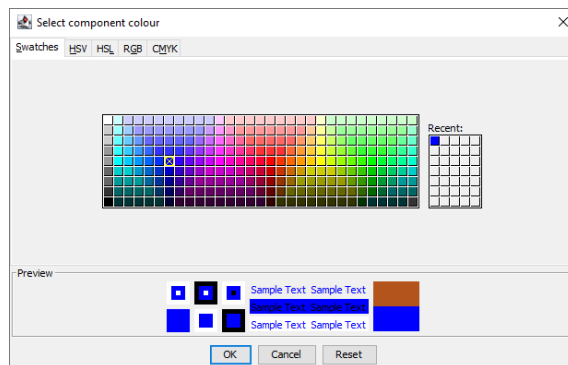
When drawing structures, the current component must first be activated. Activate the beam component by **left click** on 1 Beam in the Components window.



From the Toolbar menu, select the intent **Draw rectangle** . Move the mouse pointer down to the lower most toolbar to type the coordinates of the first node, **X=-10 Y=10 Z=6** , press **ENTER** on your keyboard. Select the checkbox to the right of coordinate Z. This will activate relative coordinates to the first defined node, type **X=20 Y=0 Z=0** , press **ENTER** on your keyboard. For coordinates of the last node, type **X=0 Y=-20 Z=0** , press **ENTER** on your keyboard.



The color of the 1 Beam can be changed by **left click** on the brown rectangle in the Components window. We have chosen blue.





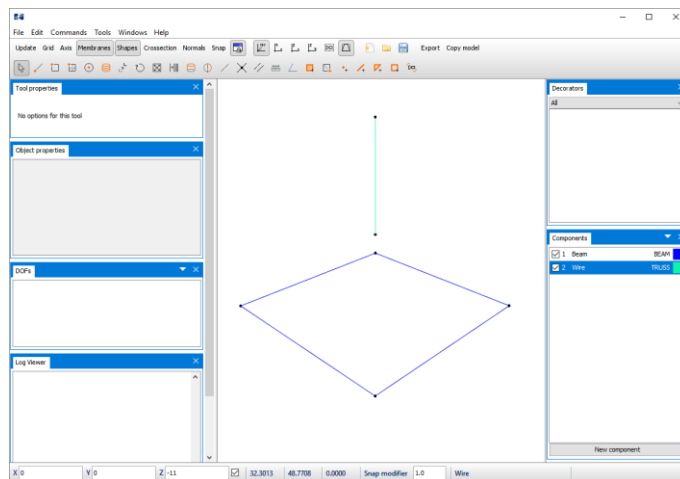
Select **OK**.



6 Draw wire

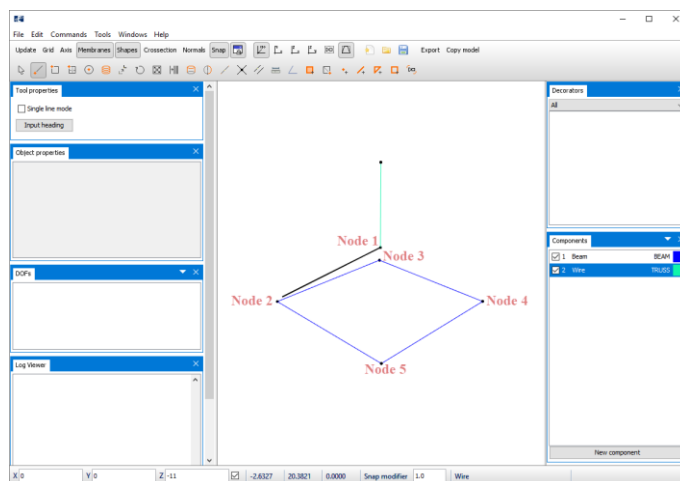
Activate the wire-component by **left click** on 2 Wire in the Components window.



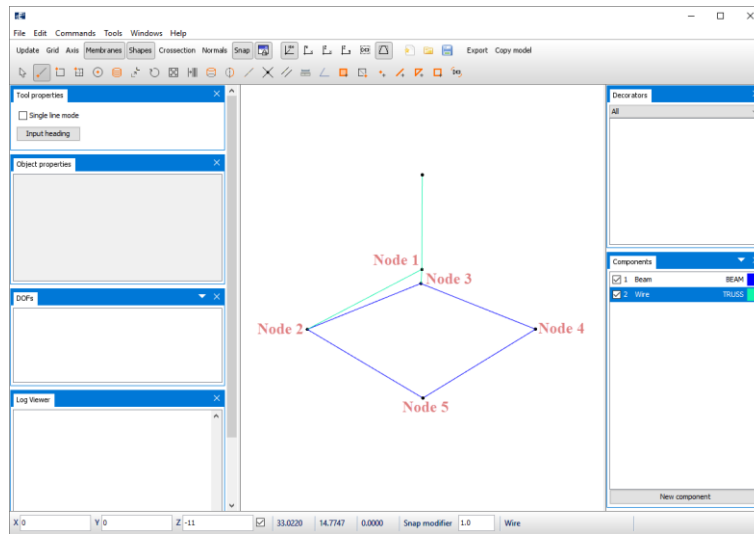
From the Toolbar menu, select the intent **Draw line** . Move the mouse pointer down to the lower most toolbar to type the coordinates of the first node, **X=0 Y=0 Z=25** , press **ENTER** on your keyboard. For the next node, make sure the checkbox for relative coordinates is still activated. Type **X=0 Y=0 Z=-11** , press **ENTER** on your keyboard. To exit the Draw line intent, select **Pointer**  or press **ESC** three times.



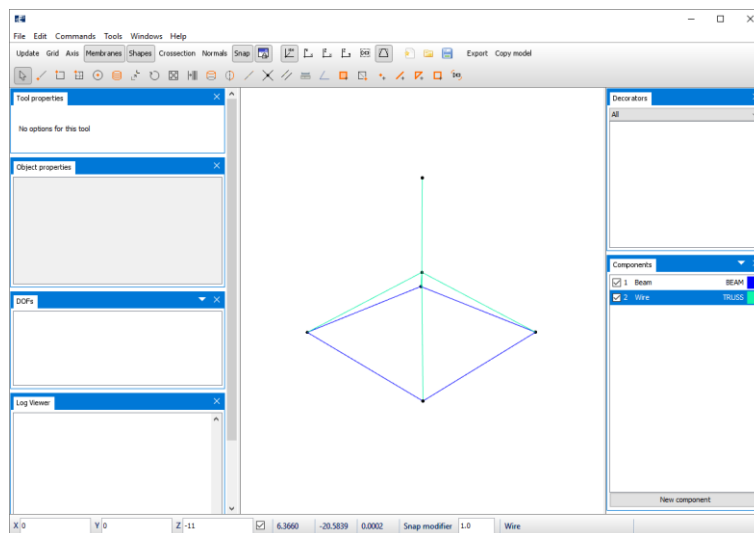
To draw the wire legs, select **Draw line**  and activate **Snap** . The snap function forces the selected intent to snap to the node closest to your mouse pointer. Move your mouse pointer to node number 1 (as indicated in the figure below) and **left click**.



The black line between Node 1 and Node 2 is a preview of the drawn line. Move the mouse pointer to Node 2 and **left click** on it. To escape the continuous line press **ESC once**. Then reselect Node 1 and move your mouse pointer to Node 3 and **left click** on it.

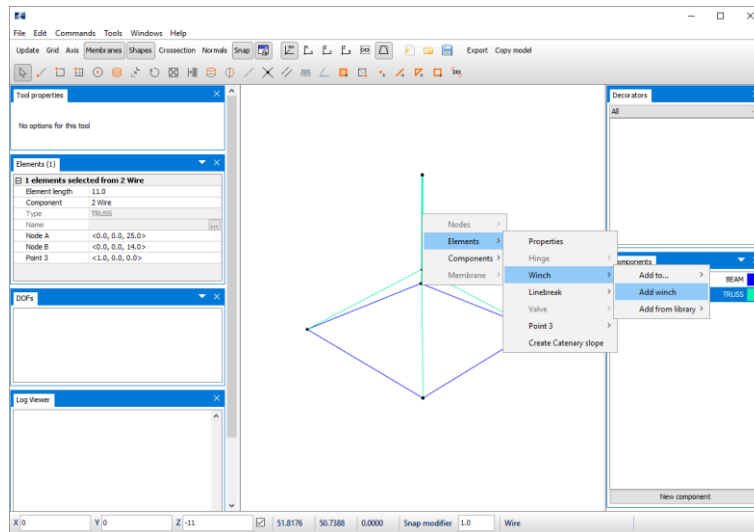


Repeat this for Node 4 and Node 5. When finished, select **Pointer** to escape the Draw line intent. The result should look like the figure below.



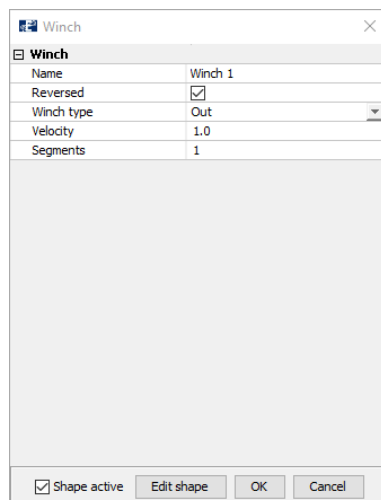
7 Model winch

A winch is applied to one end of an element. Select the vertical wire element in the 3D view by **left click > Elements > Winch > Add winch**.



The Edit winch window opens. The winch can either winch out or in, this is selected from the drop-down menu of Winch type. The velocity of winching in or out is in m/s. When winching out, AquaSim adds elements, and in Segments you can specify how many elements AquaSim should add. The selection Reversed allows to change which node the winch should be placed.

The winch type should be **Out**. Type winch velocity 1.0 m/s, and number of segments should be 1. By default, AquaSim places the winch at the second generated node of an element. You want to place the winch at the node with position $z=25\text{m}$. Since you started drawing the winch from the top, the second node of the element is situated at $z=6\text{m}$. To place the winch at the top node, activate **Reversed** by **left click** the checkbox. Then, activate a winch shape by selecting the **Shape active** checkbox.

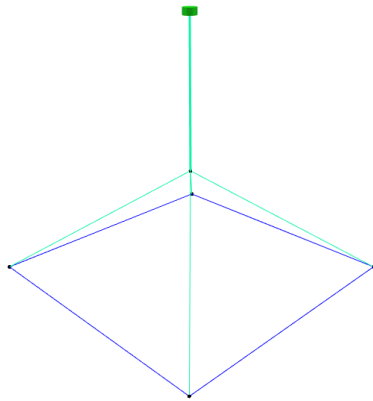



Select **OK**.

The modelled winch is available in the Decorators window in the right docking area.



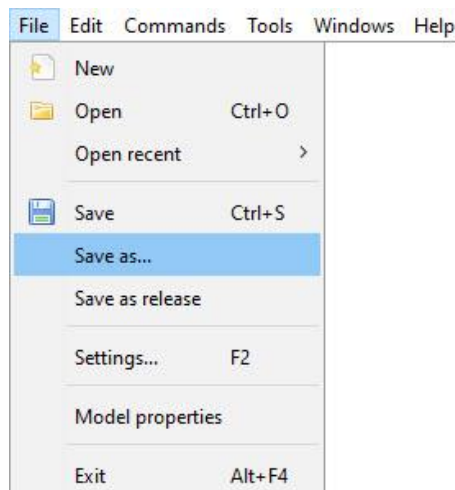
In the 3D view, the winch is visible as a green cylinder.



If you cannot see the green cylinder, turn on the display of shapes by selecting  in the Toolbar menu. If the winch has not ended up in the correct node, double click on Winch 1 in the Decorators window to reopen the Edit winch window. Make a selection in the row **Reversed**, then select **OK**.

8 Save the model

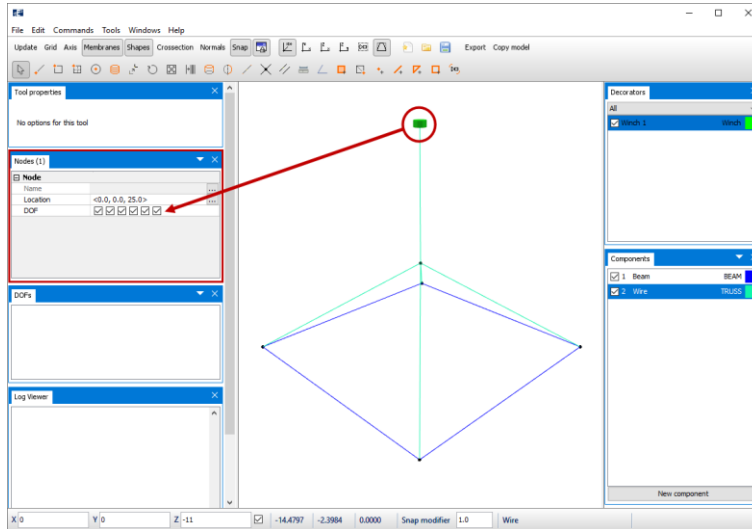
Make sure to save the model, to reduce risk of loosing data. In the top menu, select **File > Save as...**





Navigate through the File Explorer to a suitable place to save the model. We have chosen the folder Documents\AquaSim\Analyses\CaseStudy02. Assign the model name Winch and press **Save**.

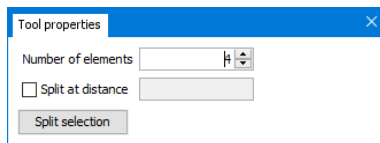
9 Assign boundary conditions

The wire should be restrained from translation and rotation at the winch. In the 3D view, select the node with the winch by **left click** on the node. Go to the Nodes window in the left docking area, **select all checkboxes** in the row **DOF**.

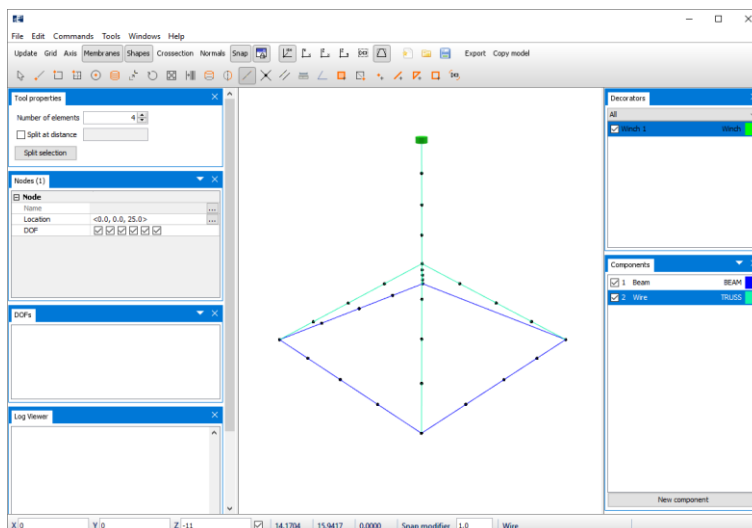


10 Discretization

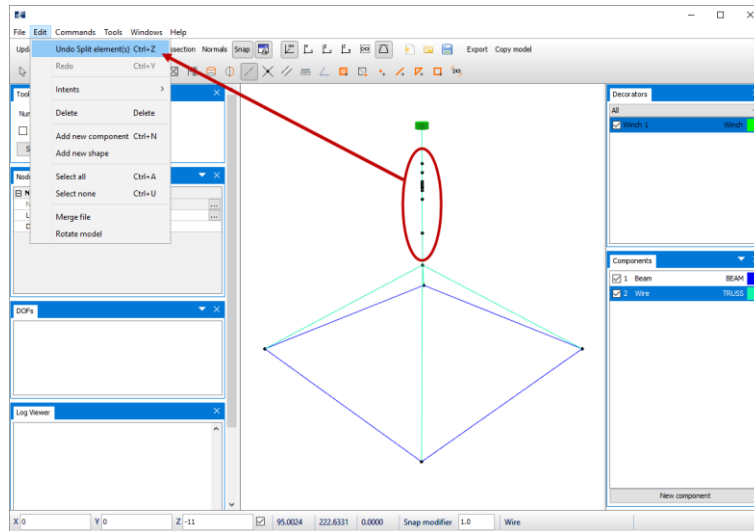
Your model is to be discretized. From the Toolbar menu, select the intent **Split line** . The Tool properties window is activated in the left docking area. In the row **Number of elements**, type 4 and press **ENTER** on your keyboard (alternatively select the upper arrow  until you get 4).



Move the mouse pointer to the 3D view to the vertical part of the wire, apply the **Split element** by **left click** on the element once. Repeat this for all elements in the model.

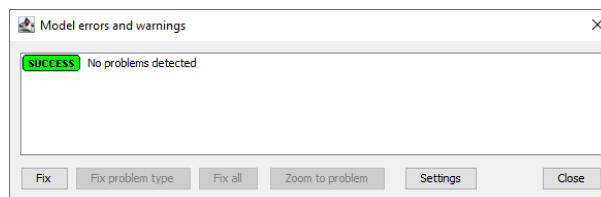


Be aware that if you left click an element more than once, AquaSim continue to divide the elements. If this happens, press **CRT + z** on your keyboard, or **Edit > Undo Split Element(s) Ctrl+z**.



11 Verify model


The model is now complete. To verify that the model is correct and without errors, select **Commands > Verify model**.



A complete model without errors is created, the next phase is to prepare the analysis.

3.4 Calculations static analysis

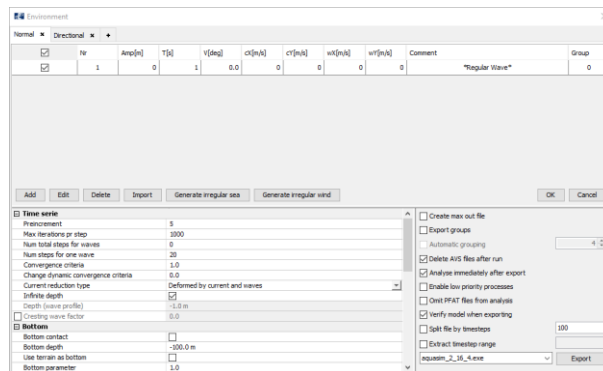
10 Enter Environmental window

Before analysis can start, export parameters must be defined. In the Toolbar menu, enter the Environmental window by selecting **Export** .

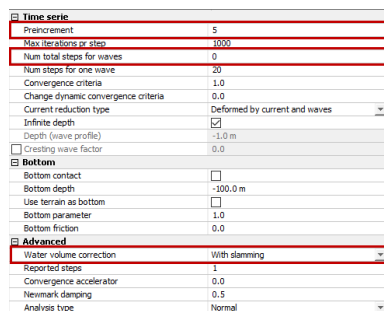
11 Time series parameters

Before a dynamic analysis is conducted, you are going to control the weight of the system through a static analysis. The weight of the system is controlled by consider the axial force in the wire.

In the section for time series, type **Num total steps for waves= 0**. This will leave only the static part of the analysis to be run. The static analysis will be conducted in 5 steps, as given in **Preincrement**. Make sure other settings are equivalent to the figure below.



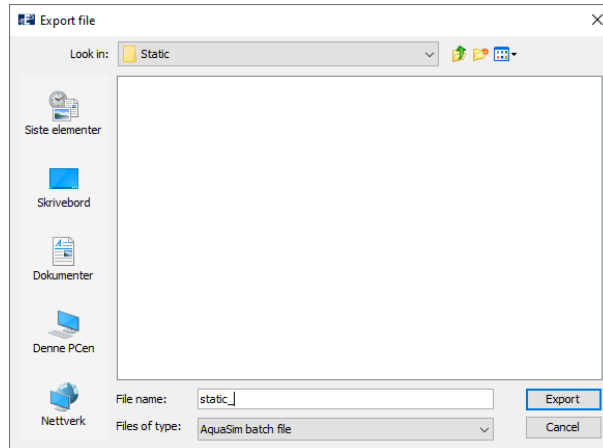
In the section **Advanced**, navigate to **Water volume correction**, select the drop-down menu and choose **With slamming**.



The Water volume correction is a feature for selecting how AquaSim should correct the calculation of hydrodynamic properties of elements when they move in and out of water. Choosing With slamming, AquaSim will identify if BEAM elements with load formulation Morison submerged, are above or below the water surface.

Hydrodynamic properties will be adjusted and updated for each analysis step. To get more insight in the different options for Water volume correction, please see the Theory Manual section Water volume correction.

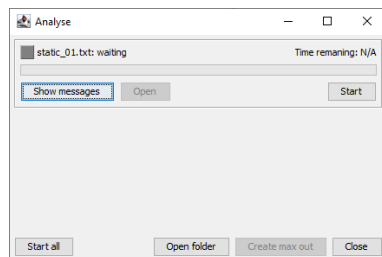
Select **Export** in the lower right corner of the Environmental window. Decide for where you to save the analysis. Navigate to an appropriate place on your computer. In this case study, we have created an analysis folder in Documents\AquaSim\Analyses\CaseStudy02 called Static. Type an analysis name e.g., static_.



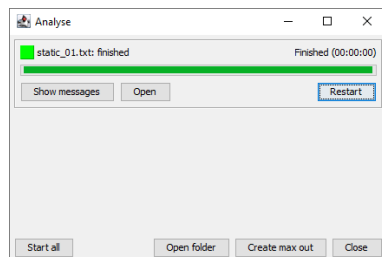
Select **Export**.

12 Run analysis

The Analysis window appears, select **Start** or **Start all**.



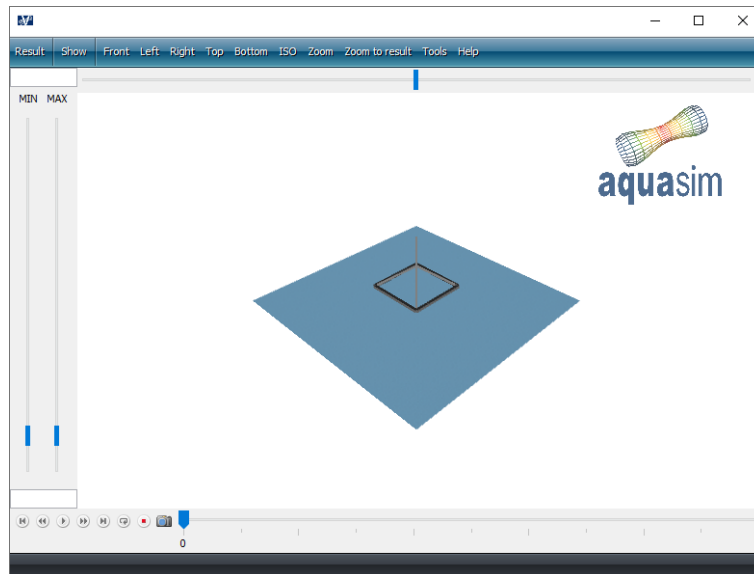
When the progress status in the Analysis window change to **finished**, and a green box appears the left of the status, the analysis is successfully completed without errors.



3.5 Postprocessing static analysis

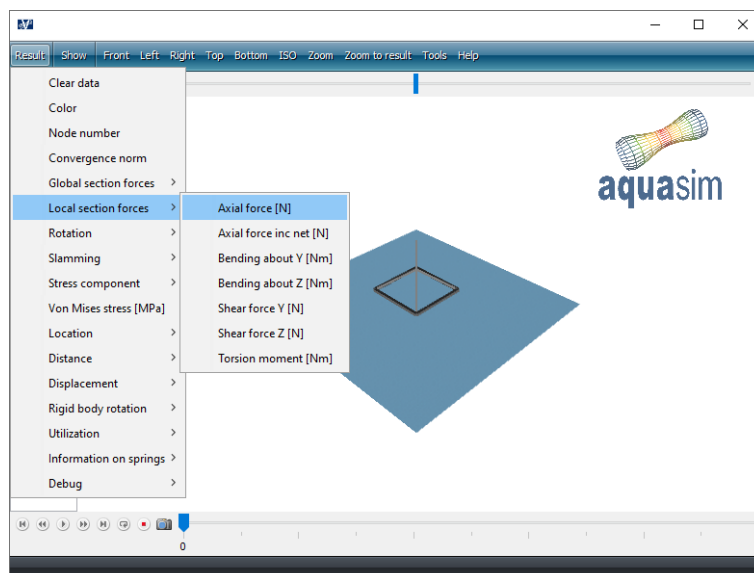
13 Open result file

In the Analysis window, select **Open**. This will load the postprocessing tool AquaView.

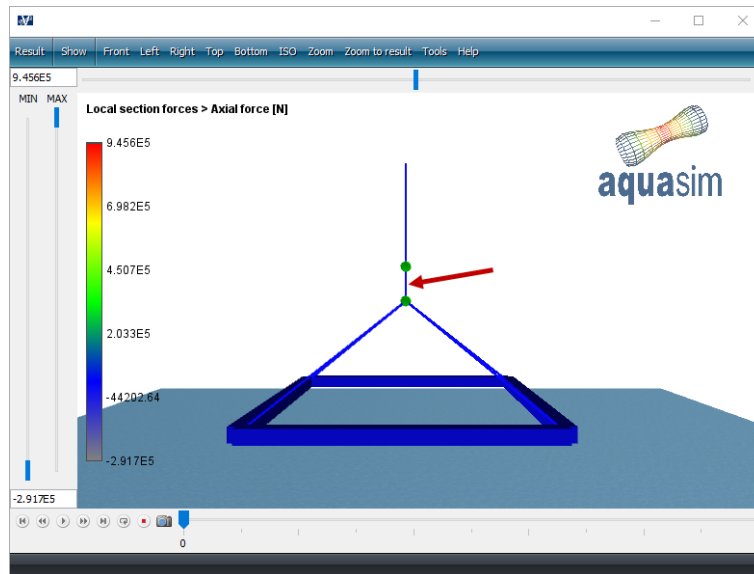


14 Plot axial force in wire

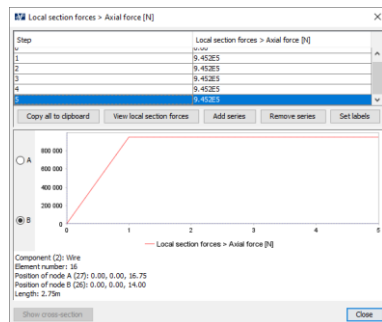
From the top Toolbar menu, select **Result > Local section forces > Axial force [N]**.



The axial force in the wire connection point is found by **left click** on the element between the two green spheres as indicated in the figure below.



Take note that the axial force in the element, at the lower most node (node B in our example), is $9.452E+05$ N.

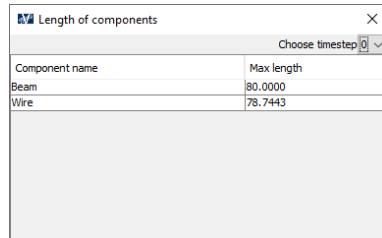


This should correspond to the total weight of beam and wire in air.

15 Compare results with hand calculations

From the input table presented in section 3.3 we know the weight per meter for both wire and beam. You need to find the total length of each element. There are several ways to do so, one method is presented below.

From the top Toolbar menu, select **Tools > Show component lengths**. The Length of components window opens.



The total mass of the components in air is calculated as:

$$m_{total} = m_{wire} + m_{beam} = 5.3 \frac{kg}{m} \cdot 78.744m + 1200 \frac{kg}{m} \cdot 80m = 96417.3kg$$

Converting this to Newton, we get the force due to gravity:

$$96417.3kg \cdot \frac{9.81m}{s^2} = 9.459E + 05 N$$

Hand calculations are compared in the table below:

AquaView [N]	Hand calculations [N]	Difference [N]
9.452E+05	9.459E+05	-0.07 %

The differences are considered neglectable small. We now know that AquaSim calculates the weight and axial force correctly.

Exit AquaView and **close** the Analyse window.

3.6 Calculations dynamic analysis

16 Enter Environmental window

You should investigate the response of the beam when it moves through the water surface for two different combinations of environmental loads. Each combination of environmental loads is referred to as a load condition (LC), or sea state.

Go back to the model in AquaEdit. If you have exited the program, the model can be reopened by **double click** on the Winch.amodel from the File explorer, AquaEdit will then be loaded.

From the Toolbar menu, enter the Environmental window by selecting **Export**

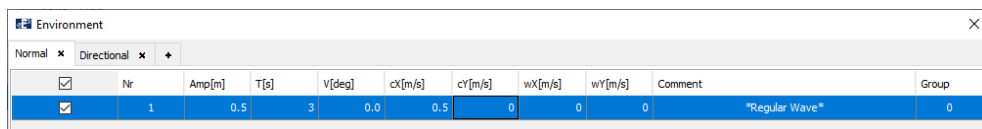


17 Load conditions

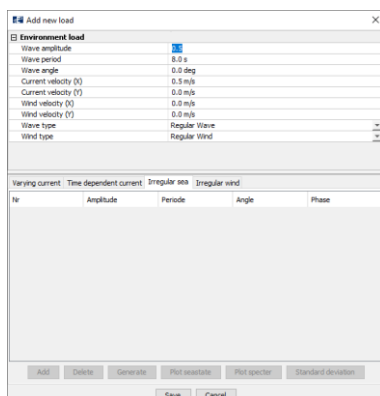
In the upper part of the Environmental window you find the parameters for the environmental loads. Each line represents one load condition. The input values for the load conditions are described in the table below.

Nr	Amp [m]	T [s]	V [deg]	cX [m/s]	cY [m/s]	wX [m/s]	wY [m/s]
Row number of the LC	Wave amplitude (ζ_A)	Wave period	Direction the wave propagates. A value of 0 correspond to along positive x-axis.	Current velocity along x-axis.	Current velocity along y-axis.	Wind velocity along x-axis.	Wind velocity along y-axis.
1	0.5	3	0	0.5	0	0	10
2	0.5	8	0	0.5	0	0	0

Type the values from row no. 1 in equivalent place in the Environmental window.



To add load condition no. 2, select **Add** and type the values in appropriate sections.



When this is done, select **Save**. The Add new load window will close, and a new row has been added to the list.

Environment										
Normal × Directional × +										
<input checked="" type="checkbox"/>	Nr	Amp[m]	T[s]	V[deg]	cX[m/s]	cY[m/s]	wX[m/s]	wY[m/s]	Comment	Group
<input checked="" type="checkbox"/>	1	0.5	3	0.0	0.5	0	0	0	*Regular Wave*	0
<input checked="" type="checkbox"/>	2	0.5	8	0.0	0.5	0	0	0	*Regular Wave*	0

18 Time series parameters

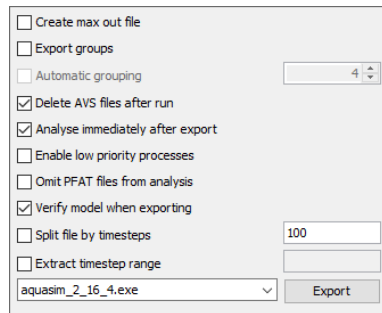
To conduct a dynamic analysis, type the parameters as presented in the **Time serie** section below.

Time serie	
Preincrement	5
Max iterations pr step	1000
Num total steps for waves	80
Num steps for one wave	20
Convergence criteria	1.0
Change dynamic convergence criteria	0.0
Current reduction type	Deformed by current and waves
Infinite depth	<input checked="" type="checkbox"/>
Depth (wave profile)	-1.0 m
<input type="checkbox"/> Cresting wave factor	0.0
Bottom	
Bottom contact	<input type="checkbox"/>
Bottom depth	-100.0 m
Use terrain as bottom	<input type="checkbox"/>
Bottom parameter	1.0
Bottom friction	0.0
Advanced	
Water volume correction	With slamming
Reported steps	1
Convergence accelerator	0.0
Newmark damping	0.5
Analysis type	Normal

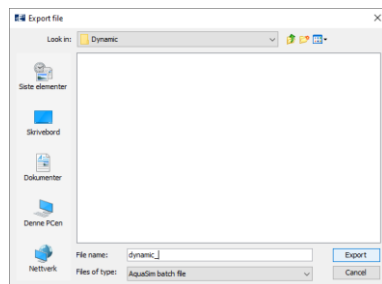
When **Num total for waves** = 80 and **Num steps for one wave** = 20, AquaSim will conduct the analysis over $80/20 = 4$ wave cycles in addition to the 5 preincremental steps. Make sure **With slamming** is activated as Water volume correction in the **Advanced** section.

19 Export selections

In the export selection section, apply the selections as provided in the figure below.



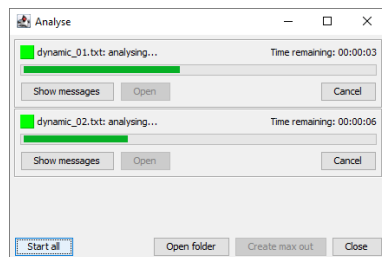
Select **Export**. Navigate to an appropriate place on your computer where you want to save the analysis. In this case study, we have chosen Documents\AquaSim\Analyses\CaseStudy02 and created a folder Dynamic. Type an analysis name e.g, dynamic_.



Select **Export**.

20 Run analysis

In the Analysis window you will see the load conditions ready to be analyzed. To run the analysis of both load conditions parallel, select **Start all**.



The analysis is finished when both load conditions change the status to **finished**. Depending on your computer, this analysis should be finished within approximate 10-15 seconds.


3.7 Postprocessing dynamic analysis

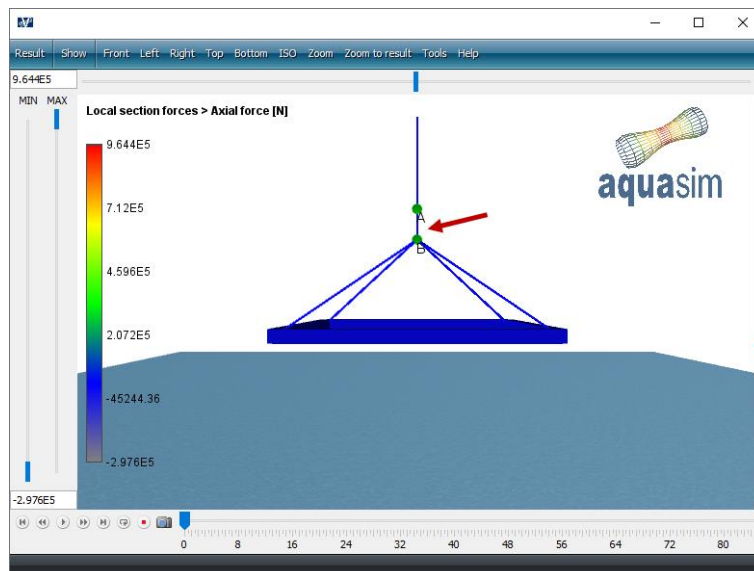
21 Open result file, load condition 01

In the Analysis window, select **Open** for the load condition called dynamic_01.

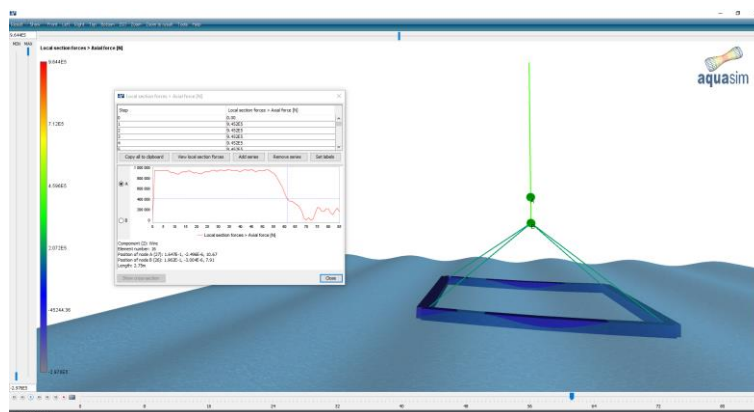
If your analysis is not completed, or an error occur, you may load the equivalent file found in AquaSim\Demos\CaseStudy02\Dynamic. In this folder, **double click** dynamic_01.avz, AquaView should be loaded.

22 Axial force in wire

You should observe how the axial force in the wire changes as the beam is lowered from air, through the water surface and below. From the top Toolbar menu in AquaView, select **Result > Local section force > Axial force [N]**. To get a better view of the lowering process, select **Front**  from the top Toolbar menu.



To view the axial force in the connection point of the wire **right click** on the element as indicated in the figure above. Select **Play** from the playback area in the lower left corner.



Observe how the axial force is more or less constant as long as the beam is in air. When the beam crosses the water surface a significant drop in the axial force is seen. This is due to buoyancy in the beam and damping effects from the water. Buoyancy is a force acting in the opposite direction of gravity forces, causing the load in the wire to be reduced.

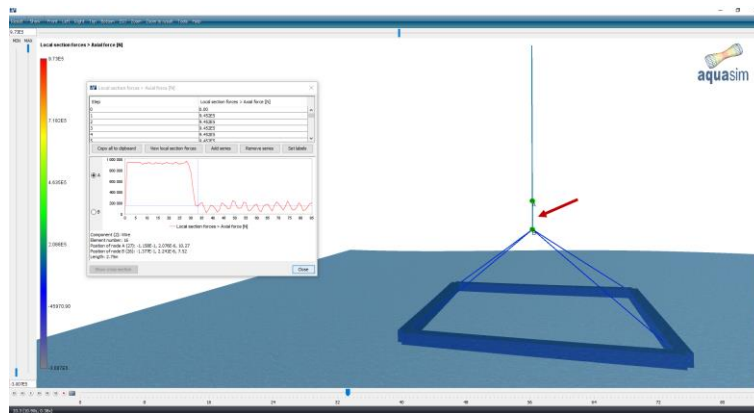
By selecting **j** from your keyboard, you will be provided with the maximum axial force in the components for the entire analysis. Which timestep the maximum axial force occur is also provided.

Component name	Component index	Value	Timestep
Beam	1	-4418.63	70.0
Wire	2	9.644E5	52.0

23 Open result file, load condition 02
24 Axial force in wire

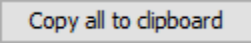
Navigate back to the Analysis window and open the result file for load condition 02 by selecting **Open** next to dynamic_02.

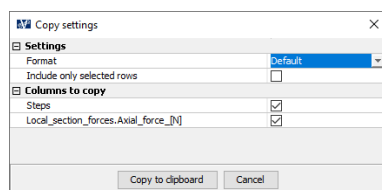
To view axial force in the wire, select **Result > Local section forces > Axial force [N]**. Select the same element as in load condition 01 by right click on the element as indicated in the figure below.



25 Comparison results

You should now extract the results from the selected element in AquaView to a calculation program after your preferences. In this case study, we have chosen MS Excel.

In the Element result window for load condition dynamic_01.avz, select **Copy all to clipboard** , the Copy settings window will open.

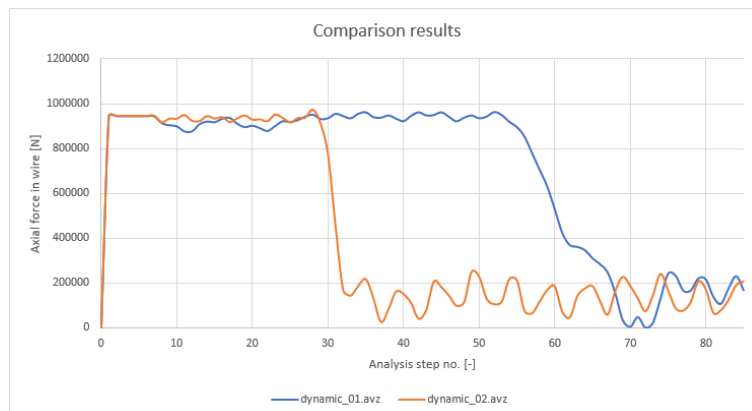


Select **Copy to clipboard**. Open the calculation program and paste by selecting **Ctrl + v**.

Repeat this for the same element in dynamic_02.avz, paste the results in the calculation program next to the results from dynamic_01.avz.

Step	Local section	Step	Local section	forces > Axial force [N]
0	0	0	0	
1	9.45E+05	1	9.45E+05	
2	9.45E+05	2	9.45E+05	
3	9.45E+05	3	9.45E+05	
4	9.45E+05	4	9.45E+05	
5	9.45E+05	5	9.45E+05	
6	9.45E+05	6	9.45E+05	
7	9.45E+05	7	9.45E+05	
8	9.14E+05	8	9.16E+05	
9	9.05E+05	9	9.33E+05	
10	9.00E+05	10	9.32E+05	
11	8.77E+05	11	9.49E+05	
12	8.77E+05	12	9.23E+05	
13	9.10E+05	13	9.22E+05	
14	9.21E+05	14	9.44E+05	
15	9.19E+05	15	9.33E+05	
16	9.34E+05	16	9.39E+05	
17	9.38E+05	17	9.17E+05	
18	9.11E+05	18	9.34E+05	
19	8.97E+05	19	9.47E+05	
20	9.03E+05	20	9.28E+05	

Create a scatter plot of both result sets. Comparing axial forces between the two load conditions, it is seen that the magnitude is more or less similar. Before the beam cross the water surface, axial force is equivalent to gravitational forces of the system. When the beam crosses the water surface, the axial force drops to about $2.0E+05$ N.



However, dynamic_01 has a more gradual decrease of axial force, and the drop occur at a later step than dynamic_02. This is due to the different wave periods in the load conditions. The wire is defined to winch out with a constant rate of speed. Since both load conditions are defined to be analyzed with the same number of analysis steps (**Preincrement + Number total for waves = 5+ 80**) the beam will pass the water surface at different analysis steps.

Dynamic considerations are now finished. Exit AquaView and AquaEdit.

3.8 Summary

We modelled a system with wire, beam, and winch to introduce the feature **Winch** in AquaSim. This required that two component groups were established.

A static analysis was conducted to control the weight of the system. Then two dynamic load conditions were established, and parallel analysis were conducted. The axial force in the wire was extracted from AquaView to MS Excel to compare the results.

4 Case study 03 – Aquaculture cage and mooring

4.1 Learning objectives

Upon completion of this case study, you will be able to:

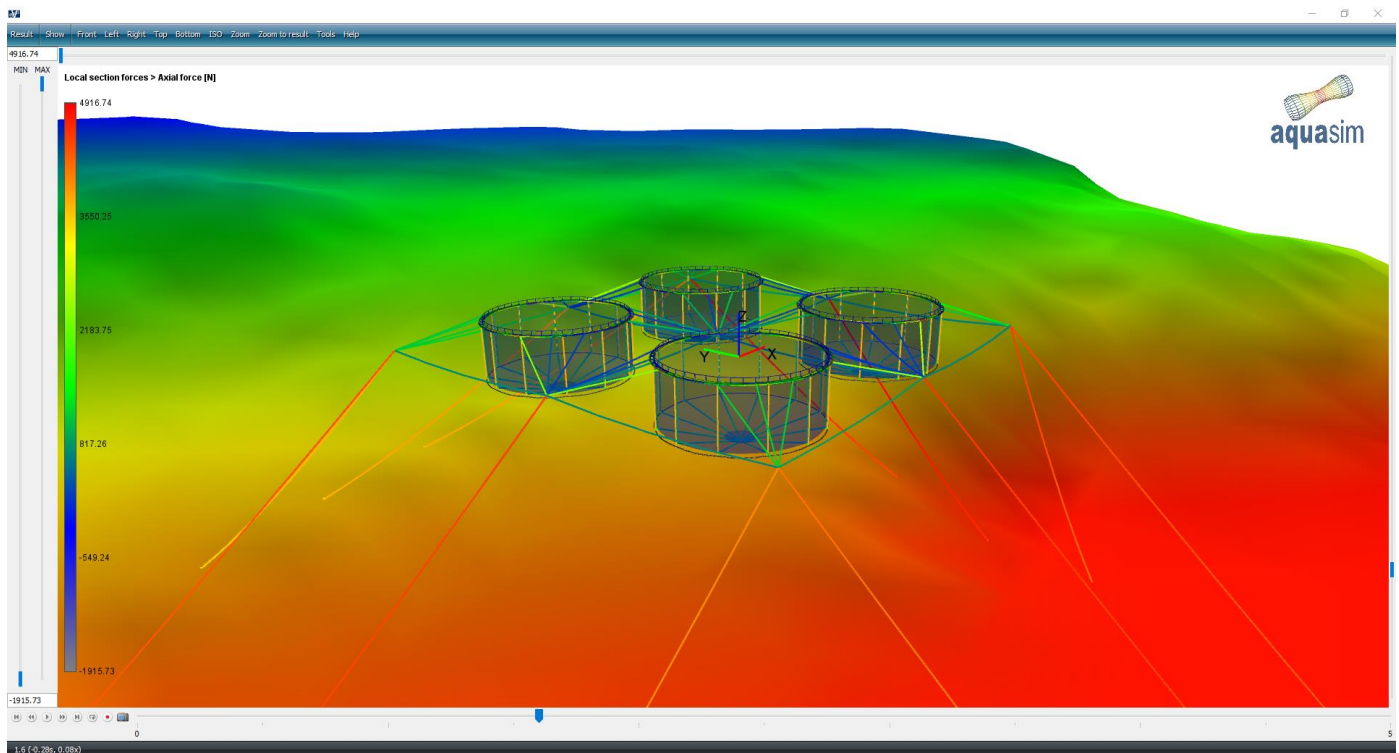
- Duplicate model components
- Model mooring components
- Mirror model components
- Model and specify properties of buoys
- Load bathymetric bottom (terrain)
- Snap nodes to terrain
- Apply the component library
- Execute static analysis
- Execute multiple non-linear dynamic analyses
- Have knowledge about system files
- Assess maximum forces in mooring components from analyses

4.2 Introduction

In this case study, you are to complete a model of an aquaculture facility and conduct a mooring analysis. You will first be introduced to the main components of the facility, then complete the model by adding mooring components and terrain. Through the snap function, the bottom chain is to be lowered to the bottom providing realistic representation of interaction between mooring components and the seabed.

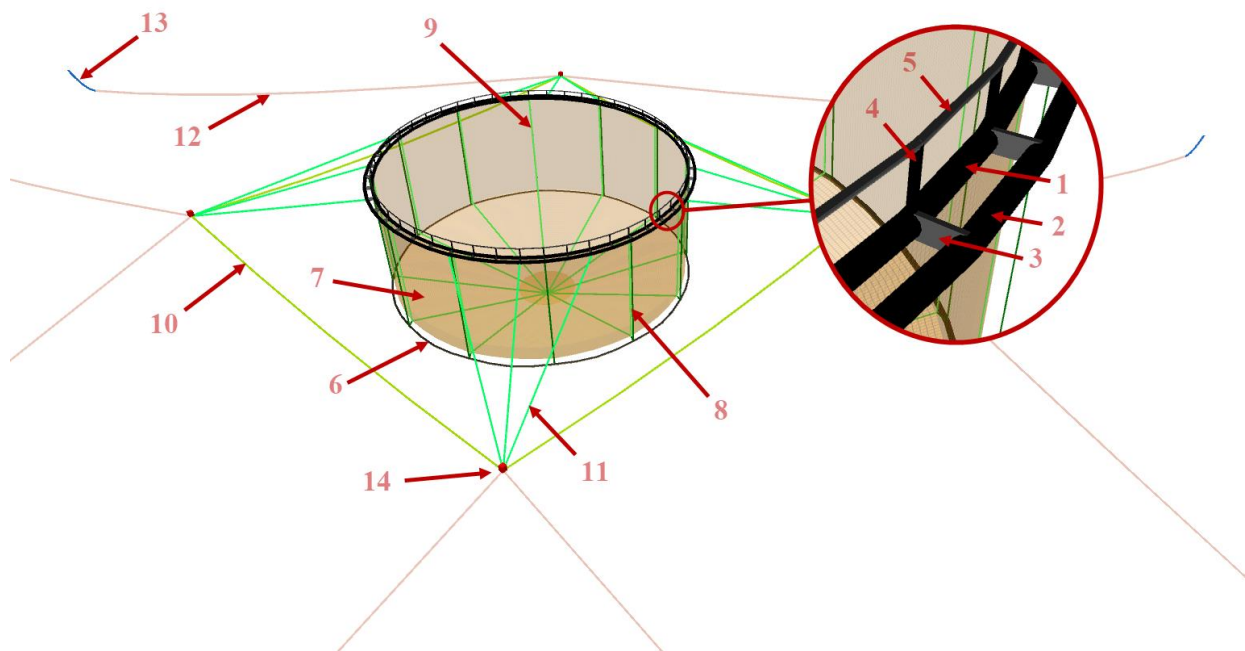
By a static analysis selected parameter, as mooring rope tension and buoyancy, should be checked.

Having conducted dynamic analysis, you are to find the maximum force in selected mooring components from these analyses.



4.3 Main components of aquaculture facility

Most aquaculture facilities consist of an enclosure, permeable or impermeable, connected to a floating unit in the waterline. The classical facility is equipped with a permeable net pen and a floating collar as the floating unit. Together these are referred to as cages. The figure below shows a single cage situated in a frame mooring system. In each corner of the frame, buoys are attached providing buoyancy to the frame and ease access to components during maintenance. Technical data of the components are described in the table below.

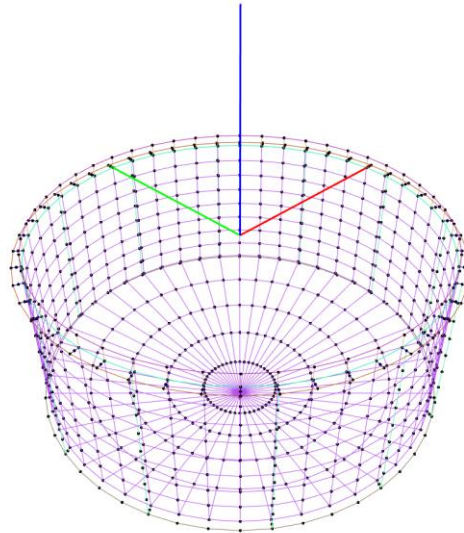


Component number	Description	Technical data*
1	Inner floating pipe of floating collar	PE100, Ø400, SDR16.7, CIR120m
2	Outer floating pipe of floating collar	PE100, Ø400, SDR16.7
3	Bracket, for positioning of floating pipes	PE100, L0.4m, W0.1m
4	Railing post	PE100, Ø160 SDR11
5	Railing	PE100, Ø140, SDR11
6	Suspension ring for net pen	PE100, Ø250, SDR11
7	Net pen	Sn15.4%
8	Net collector rope	20mm
9	Reinforcement rope	12mm
10	Frame rope	64mm
11	Bridle rope	48mm
12	Mooring rope	64mm
13	Bottom chain	32mm
14	Buoy	L1.0m, W1.0m

* Abbreviations: PE=polyethylene, Ø=pipe diameter, SDR=pipe diameter/wall thickness, CIR=circumference, Sn=solidity, D=depth, L=length, W=width.

4.4 Preprocessing

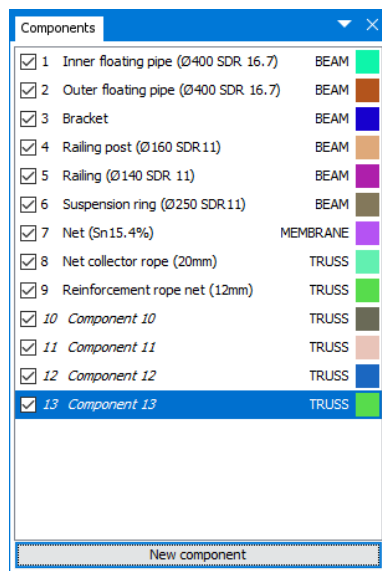
1 Open model Open CaseStudy03.amodel from Documents\AquaSim\Demos\CaseStudy03.



The model contains an aquaculture cage, and you are to model and add properties to the mooring components. This includes Frame rope, Brindles rope, Mooring rope, Bottom chain, and Buoys for positioning of the frame rope.

2 Create component groups

Create four new component groups by selecting **New component** from the Components window, one for each mooring component.



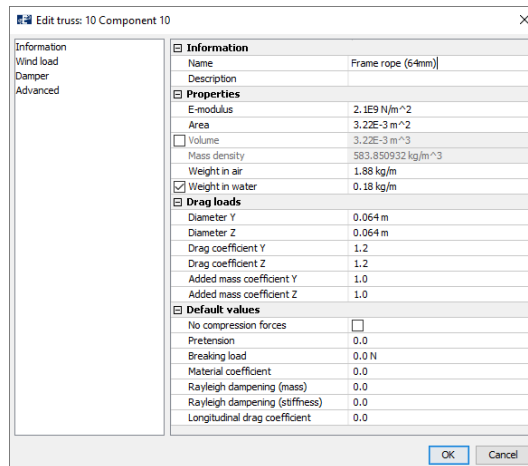
The new component groups are by default added as TRUSS with names Component 10 to Component 13. The ropes and chain are considered as flexible components, only capable of taking axial forces. Component type should then be TRUSS.

3 Component properties

Properties of the mooring components are presented below.

Properties of Frame rope	Value
E-modulus	2.1E+09 N/mm ²
Area	3.22E-03 m ²
Weight in air	1.85 kg/m
Weight in water	0.188 kg/m
Diameter	64 mm
Drag coefficient	1.2
Added mass coefficient	1.0
Properties of Bridle rope	Value
E-modulus	2.1E+09 N/m ²
Area	1.81E-03 m ²
Weight in air	1.95 kg/m
Diameter	48 mm
Drag coefficient	1.2
Added mass coefficient	1.0
Properties of Mooring rope	Value
E-modulus	2.1E+09 N/m ²
Area	3.22E-03 m ²
Weight in air	1.85 kg/m
Diameter	64 mm
Drag coefficient	1.2
Added mass coefficient	1.0
Properties of Bottom chain	Value
E-modulus	1.1E+11 N/m ²
Area	8.04E-04 m ²
Weight in air	20.4 kg/m
Diameter	32 mm
Drag coefficient	1.2
Added mass coefficient	1.0

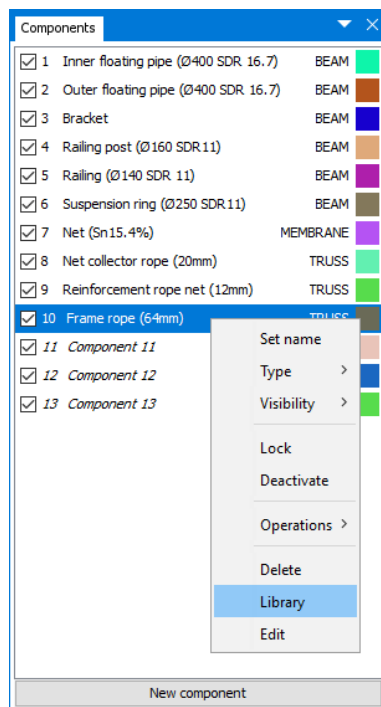
Start with adding properties for Frame rope by **right click** Component 10 > **Edit**. Type the information from table in the Edit component window. When defining the weight of rope in water, select the check box next to **Weight in water**. A warning dialogue appears, stating that it is not recommended to change the calculated value. We add a slight low weight to the truss, to eliminate the buoyancy of the truss (i.e. negative values).



Select **OK**.

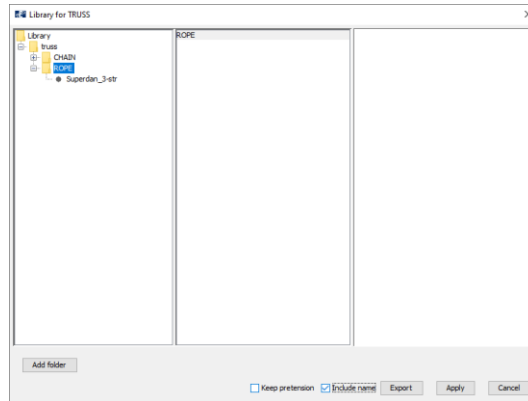
4 Save component properties to library

AquaEdit has the possibility to save and add properties to a library. Making modelling of components with identical properties more efficient. In the components window, **right click** Frame rope (64mm) > **Library**.

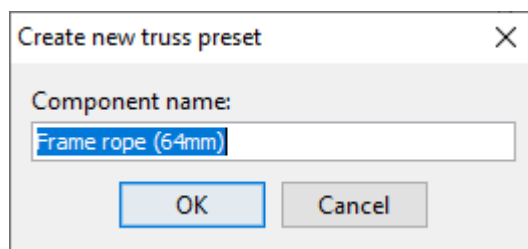


The Library for TRUSS is opened. In the library you may add, edit or apply properties to existing or new TRUSS elements.

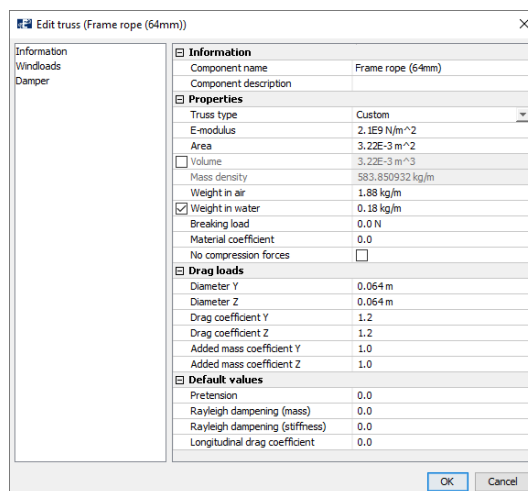
You should save the properties of Frame rope (64mm) to the library. First, you need to navigate to where you want to save the properties; select the folder **truss** and then click **ROPE**.



Select the check box for **Include name** and select **Export**. AquaEdit suggest a name according to the component name. Select **OK**.



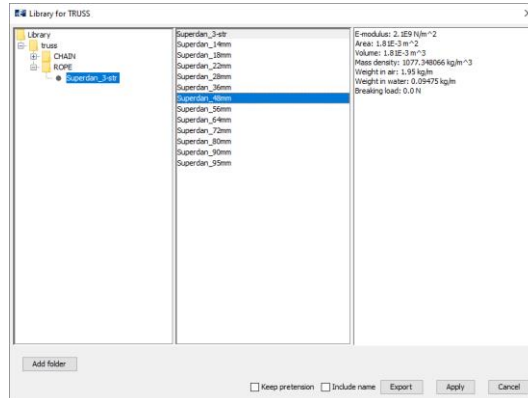
The Edit truss window opens. If you must conduct any modification before saving to library, this can be done here. Select **OK**.



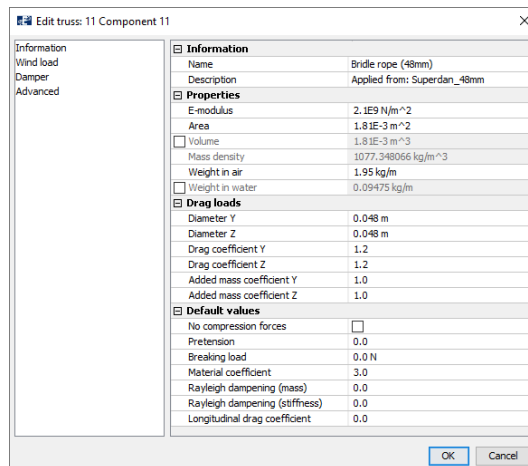
In the **Save change into library** window, select **OK**. Then, exit the Library for truss window.

5 Add properties from Library

For the properties of Bridle rope you should add them from the properties library of TRUSS elements. In the components window, **right click** Component 11 > **Library**. In the folder tree to the left, navigate to **truss > ROPE**, select **Superdan_3-str**. From the table in step number 3, we see that the diameter of the rope should be 48 mm. In the middle column, select **Superdan_48mm** and then **Apply**.

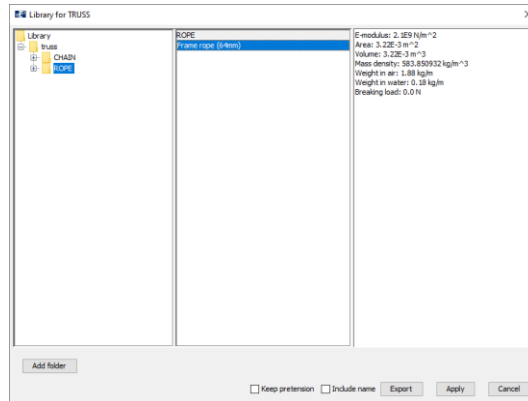


Open the Edit truss window by **double click** on Component 11. Change the name of the component to Bridle rope (48mm). Compare the values in the Edit truss window with the values provided in the table from step number 3. If no differences are observed, select **OK**.



6 Properties of Mooring rope

The Mooring rope should have identical properties as the Fram rope, i.e. 64 mm. In the Components window, **right click** Component 12 > Library. Navigate to where you saved the properties from Frame rope (64mm). Select saved properties by **left click** Frame rope (64mm) in the middle column, select **Apply**.

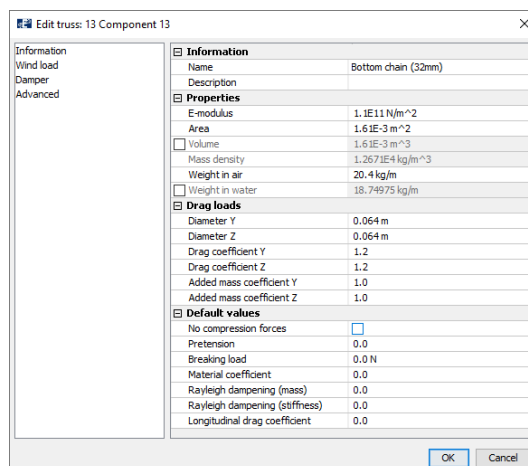


You may change the name of the component by **double click** Component 12 in the components window and assign the name Mooring rope (64mm).

7 Properties of Bottom chain

Enter the Edit TRUSS window for the last component by **double click** Component 13.

Trusses in AquaEdit assumes a single circular cross section, but chains are assembled by loops of steel rings. To account for this, both cross sectional area and diameter can be defined with twice the provided diameter from the table. In the Edit TRUSS window type values according to the figure below.




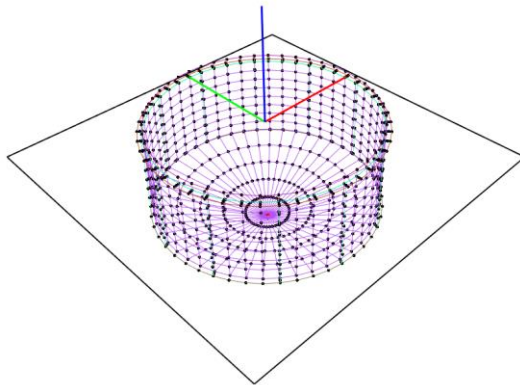
The font of all components should now have changed from *italic* to straight, indicating that required properties are added. If not, you should open the Edit TRUSS window for the components and check your parameters.

8 Draw Frame rope

The frame should be modelled as 60m x 60m at a depth of 7m. When drawing structures, the current component must first be activated.

Activate the component Frame rope (64mm) by **left click** in the Components window. There are several methods for modelling the frame, one is described below.



From the Toolbar menu, select the intent **Draw rectangle** . In the lower most toolbar, to the left, type the coordinates of the first node **x=-30 Y=-30 Z=-7** , press **ENTER** on your keyboard. Select the checkbox to the right for activation of relative coordinates. Type **x=60 Y=0 Z=0** , press **ENTER**. For the last node, type **x=0 Y=60 Z=0** . The preview of the drawn rectangle should resemble the figure below.

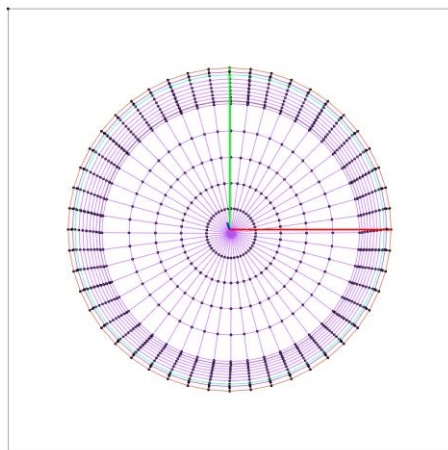



Press **ENTER** on your keyboard.

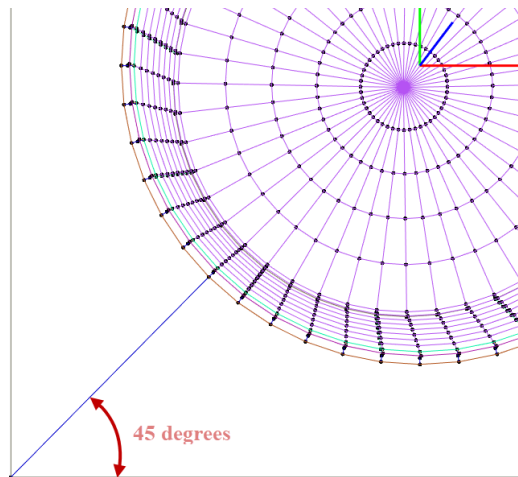
9 Draw Bridle rope

The bridle should be of type 3-part. Meaning that the floating collar is restrained by pairs of 3 bridle ropes in each frame corner. Activate the bridle component by **left click** Bridle rope (48mm) in the Components window. To ease modelling, select

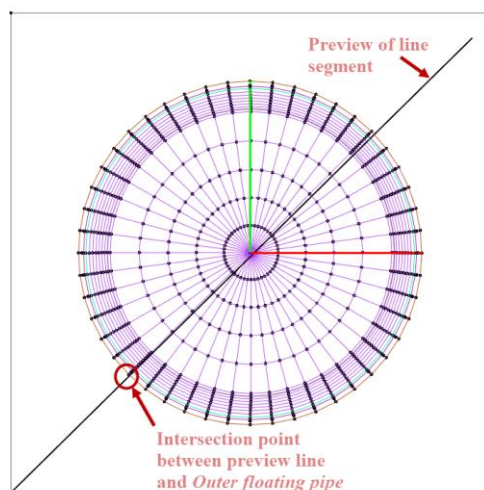
Top view  and then activate the **Snap** function .



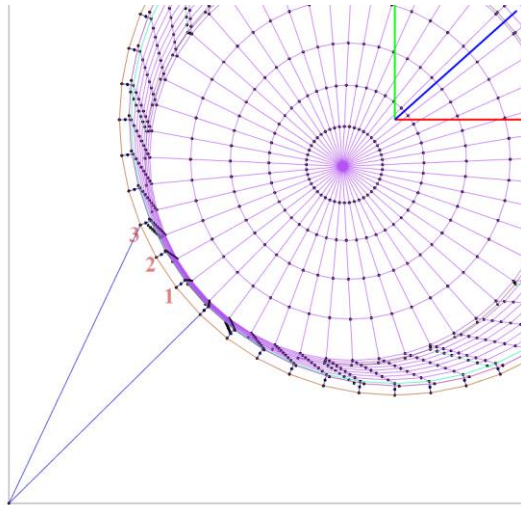
Select **Draw line**  and click on the lower left corner of the frame, then select the node on the Outer floating pipe that result in a line segment 45 degrees (in XY-plane) with respect to the horizontal frame. Press **Esc** once to escape the continuous line modelling. Do not mind if the color of the elements does not resemble our figures, it has no significance for the properties of your model.



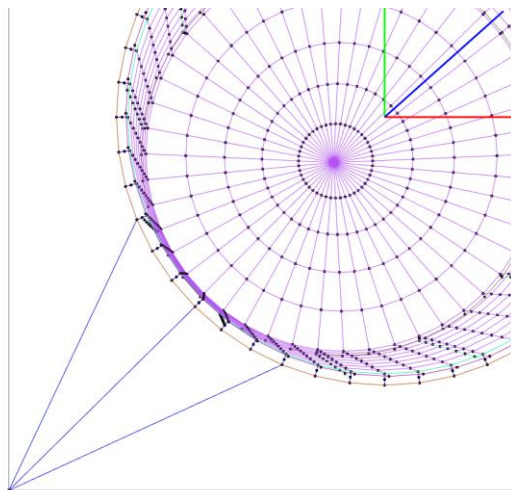
Tip! Select the correct node on the Outer floating pipe by first move your mouse pointer to the node diagonal in the frame. The correct node is the intersection point between the preview line and the Outer floating pipe.



The two other bridle ropes should be attached to the Outer floating pipe on each side of the first. Select the node on Outer floating pipe counting three brackets clockwise, and then select the corner node of the frame.



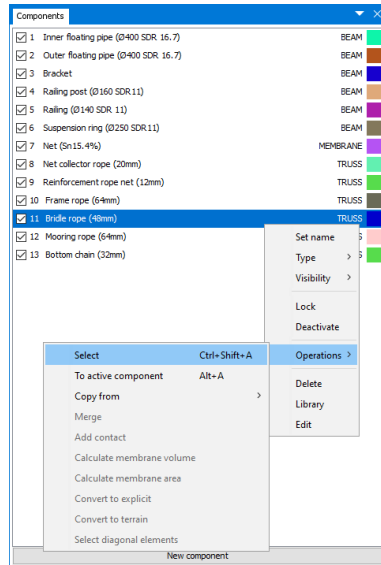
Repeat this for the last bridle rope.




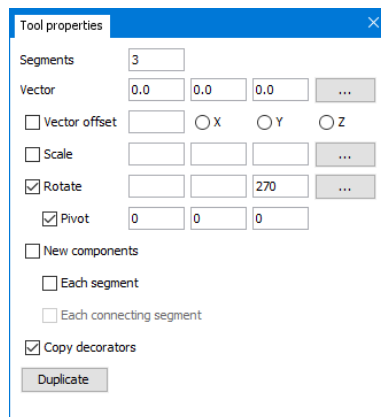
10 Duplicate segments

For repeating and time-consuming modelling steps, the Duplicate function is useful. You should now duplicate the bridle elements to the remaining frame corners.

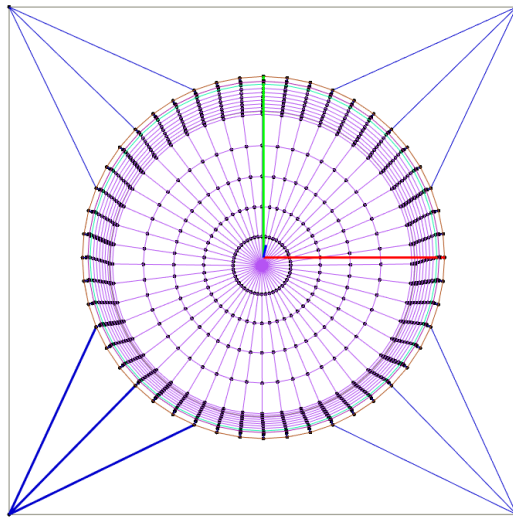
Select the bridle elements either by **left click + CTRL** and select the three elements. Or in the Components window **left click** Bridle rope (48mm) > **Operations** > **Select**.



In the Toolbar select **Duplicate** . In the Tool properties window, 3 pairs of segments should be duplicated, and rotated 270 degrees about the global z-axis about origin (Pivot).

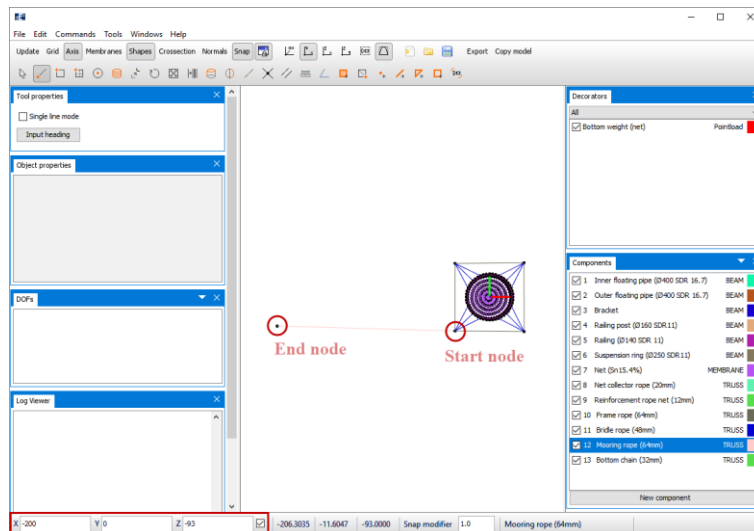



Select **Duplicate**. With these selections, you model should resemble the figure below.

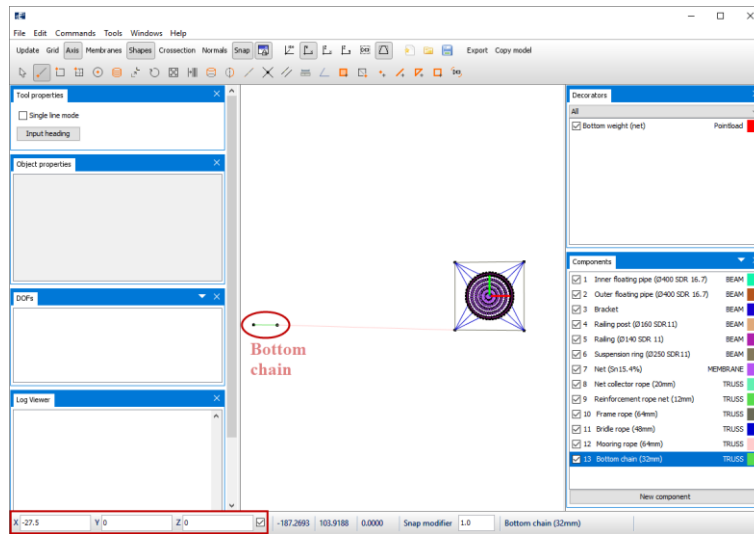


11 Draw Mooring rope and Bottom chain

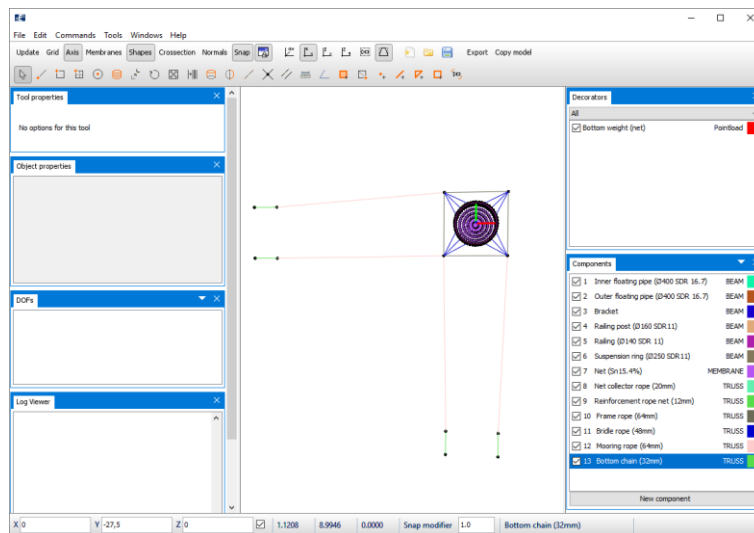
The mooring ropes should have a length of 220m at -100m depth. Activate Mooring rope (64mm) from the Components window and select **Draw line**. Start the line from the lower left node in the frame, activate relative coordinates in the checkbox and type **X=-200 Y=0 Z=-93**, this should result in a mooring rope approximate to length of 220m.



To model the Bottom chain go to the Components window and select Bottom chain (32mm). If you still have the **Draw line** intent activated, select End node as indicated in the figure above. If the intent is deactivated, reselect **Draw line** by pressing  from the Toolbar menu. The chain should be 27.5m long, this is done by typing **X=-27.5 Y=0 Z=0** in the lower toolbar. Make sure relative coordinates is activated.



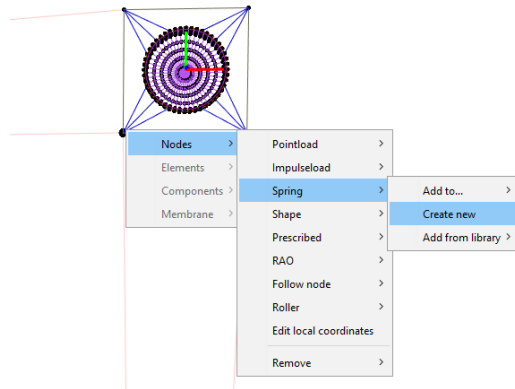
Repeat the procedure for modelling of mooring rope and bottom chain for additional three times, with the structure as shown in the figure below.



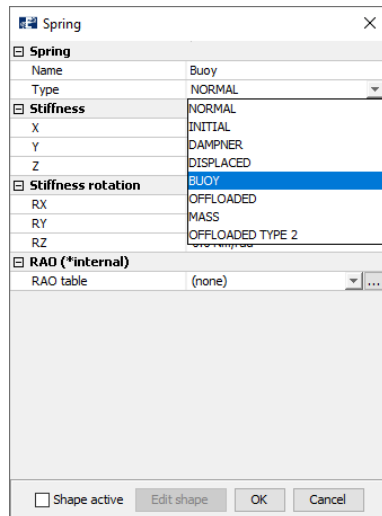
12 Buoys

You should now model buoys in each corner of the frame rope. For simplicity reasons, we model them in the corner nodes connection the frame and not in the water line.

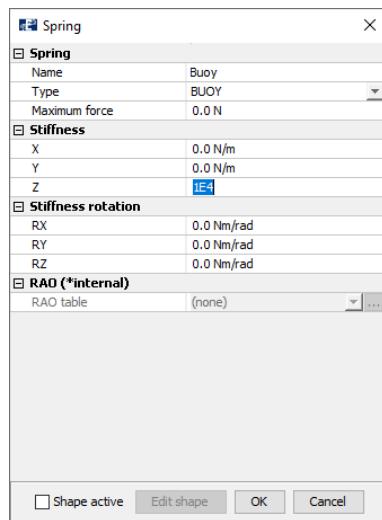
Select the lower left node of the frame, **right click** > **Nodes** > **Spring** > **Create new**.



This will open the Edit spring window. Dedicate a suitable name, e.g. Buoy (frame). There are several types of springs available in AquaEdit, you should choose **BUOY**.

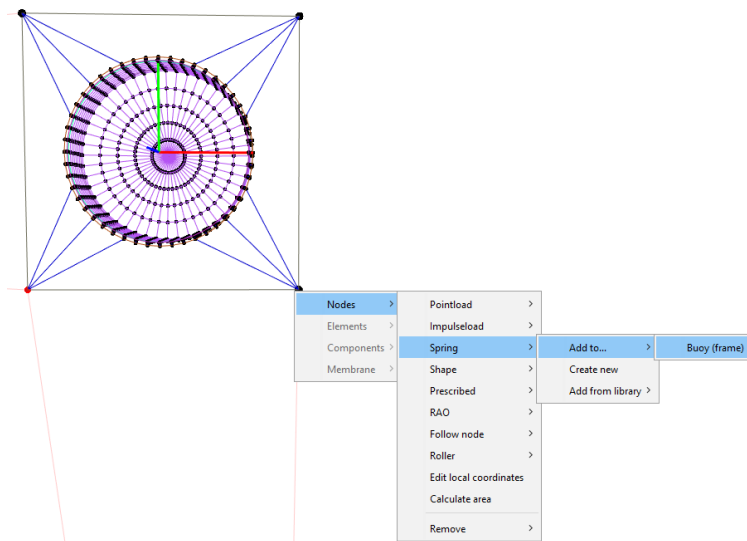


In real life, the buoy will intersect the water line. The intersection area will provide the buoy with a water plane stiffness. You are going to add a water plane stiffness in z-direction. This is the parameter significant for position of the frame. In the Stiffness section type **Z=1E+04 N/m**, this will correspond to a buoy with a waterplane area of approximate 1m².



Select **OK**. The node with the buoy should now have changed color to red.

To add buoys to the remaining corner nodes, you may apply multi-selection. Multi-selection is activated by hold **CTRL + left click** on nodes. Select the remaining three corner nodes **right click > Nodes > Spring > Add to... > Buoy (frame)**.

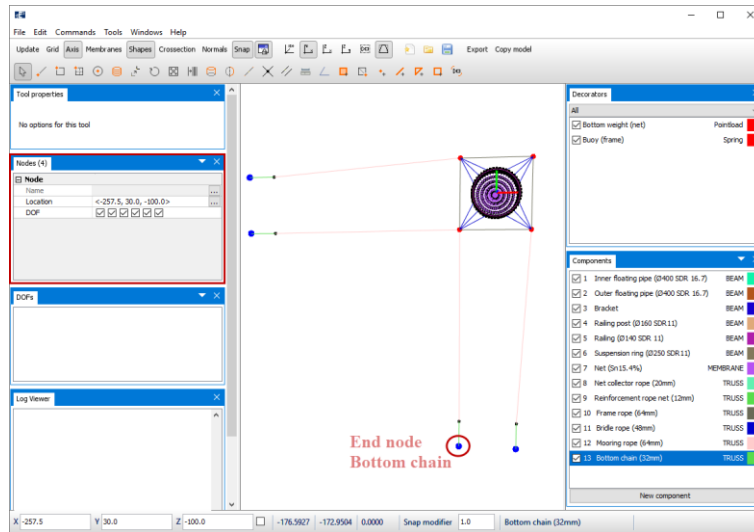


Buoys should now have been added to the three remaining nodes.

13 Boundary conditions


At the end of the chain, a mooring bolt or anchor is attached. To resemble this, the end nodes are restrained in all 6 Degrees of freedom. Multi-selection is activated by hold **CTRL + left click** on nodes.

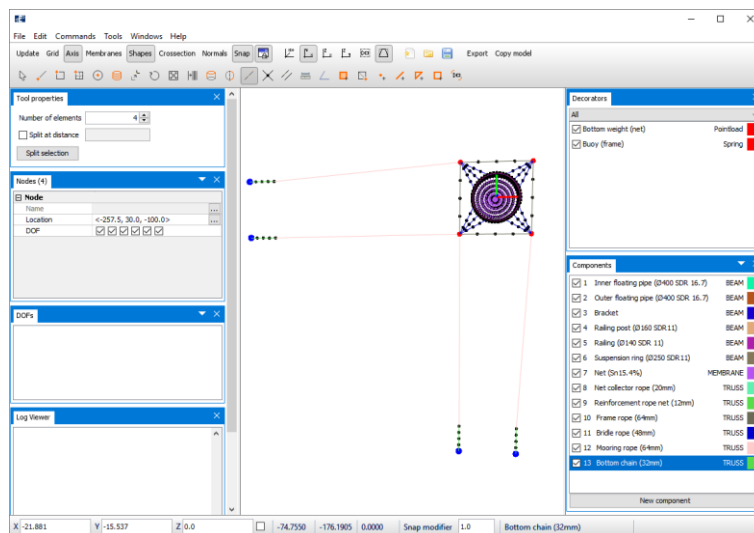
In the Nodes properties window select all the checkboxes in the row **DOF**.



14 Discretization The mooring components should be discretized with the number of elements according to the table below.

Component name	Number of elements
Frame rope (64mm)	4
Bridle rope (48mm)	4
Bottom chain (32mm)	4

Select the intent **Split line**  and discretize the component by left click on the components in the 3D view. After discretization, the model should resemble the figure below.

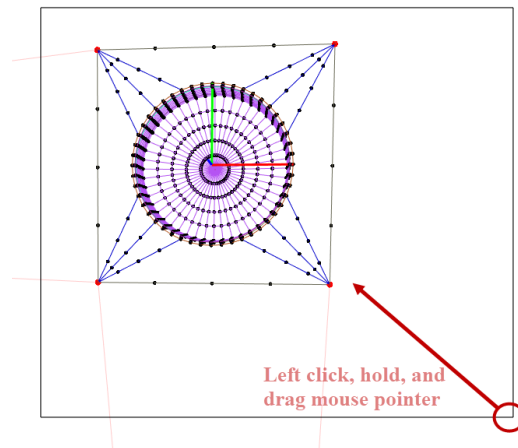


You should wait with discretization of Mooring rope (64mm) until terrain has been loaded and the chain nodes are snapped to terrain.

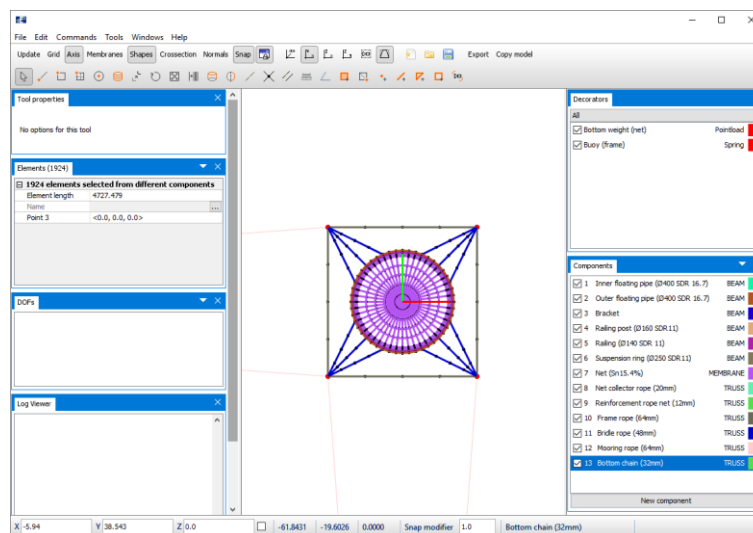
15 Duplicate cage

The model is to be extended to a 2 x 2 cage facility with identical net, floating collar, suspension, and bridle. Instead of model the same floating collar, net suspension, and bridles form scratch, you can apply the duplicate function. First, you need to select the elements for duplication.

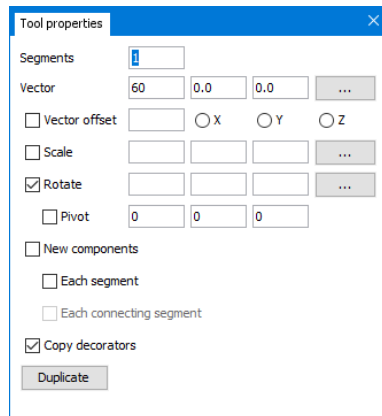
You can select the whole cage by left click in lower right corner of the 3D view, hold and drag the mouse pointer to the upper left corner.



At the moment you do NOT want to duplicate the mooring ropes. De-select these components by hold **CTRL** and **left click** on the mooring elements in the 3D viewer.

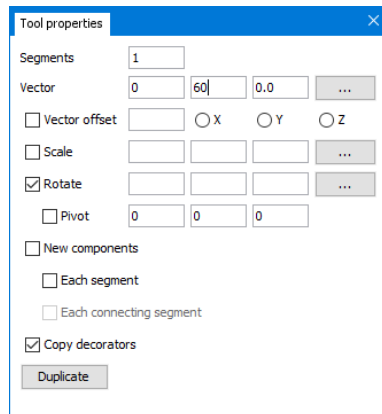


In the Toolbar menu, select Duplicate. You should duplicate the cage by 1 in the X-direction next to the existing. In the Properties window for Duplicate choose **Segments=1** and **Vector=60** (this corresponds to the size of the frame).

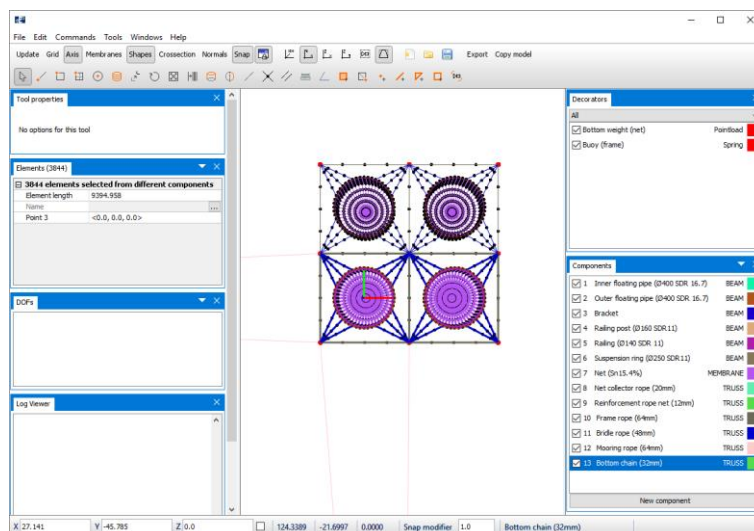


Select **Duplicate**.

Then select the two cages, except from the mooring ropes, and duplicate them with 1 segment in Y-direction.

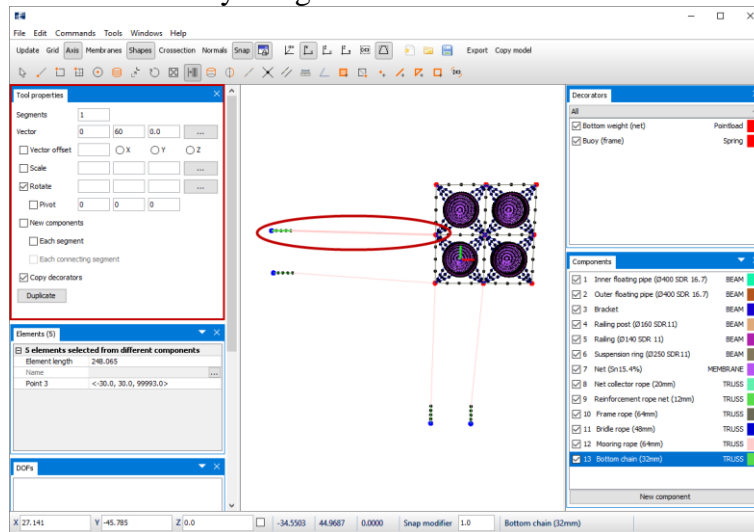


Select **Duplicate**. You should now have 2 x 2 cage system.



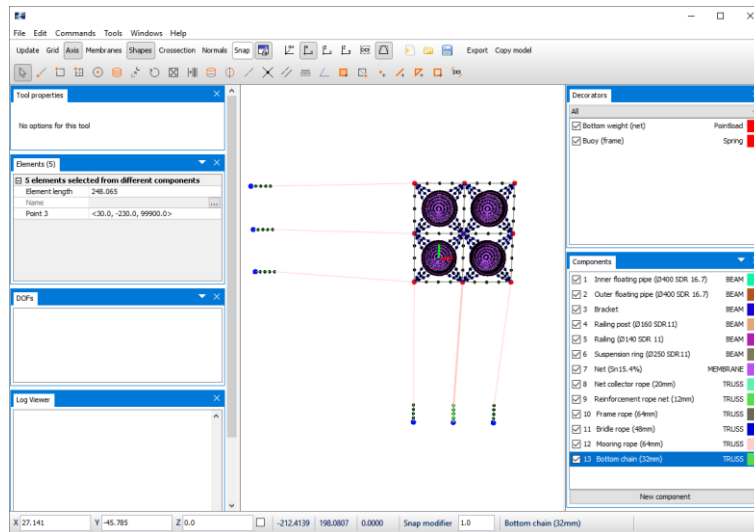
16 Duplicate Mooring rope and Bottom chain

Select the mooring rope and associated chain on the left side of the cages. Duplicate these elements by 1 segment and 60m in Y-direction.




In the Tool properties window, select **Duplicate**.

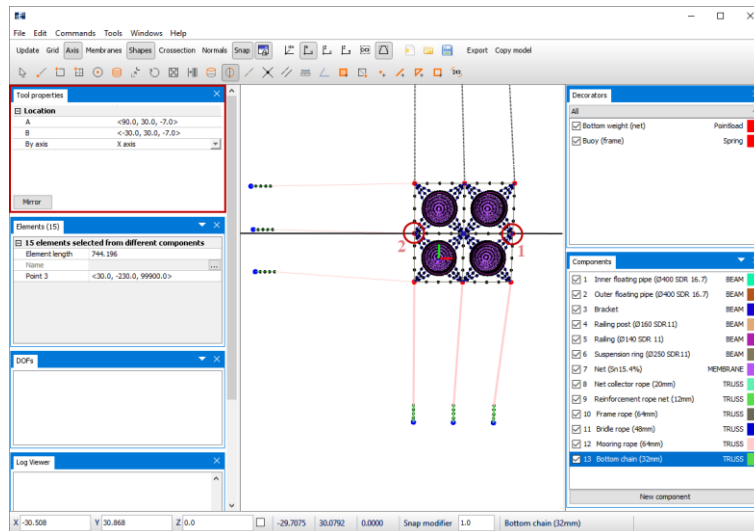
Repeat this for the mooring rope and chain in the lower right corner. Duplicate these elements by 1 segment and 60 m in X-direction.



In the Tool properties window, select **Duplicate**.

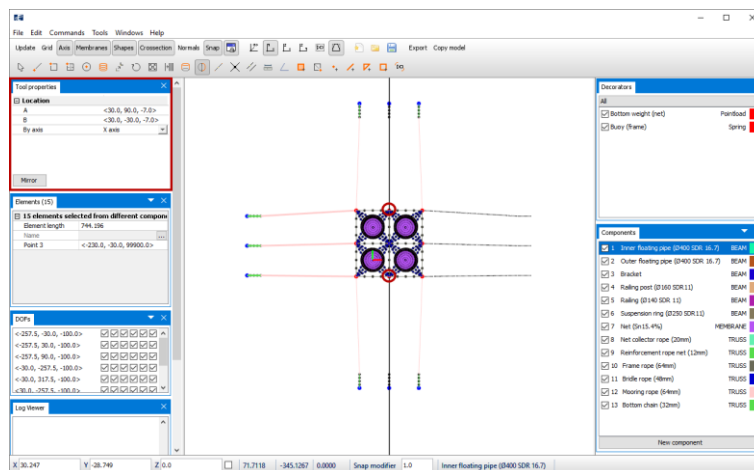
17 Mirror Mooring rope and Bottom chain

Elements can also be mirrored. Select the three lower most mooring ropes and associated chain in the 3D view. From Toolbar menu, select the intent **Mirror** . You must now draw an axis upon the elements is to be mirrored about. Select the nodes as indicated in figure below.

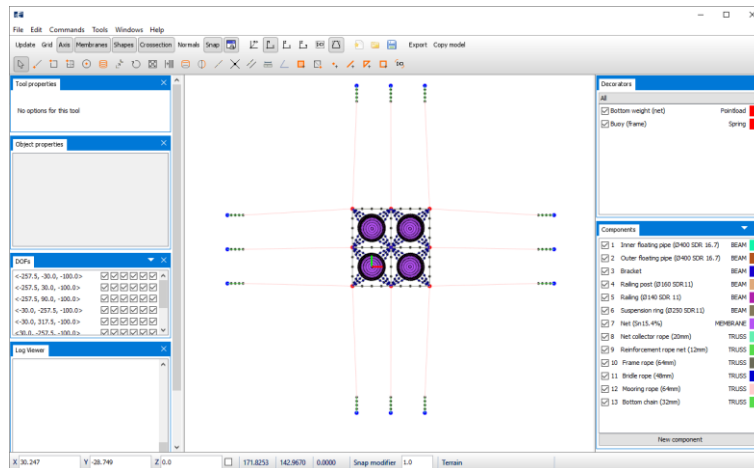


Select **Mirror** in the Properties window.

The three sets of mooring ropes and chain should also be mirrored. In the 3D view, select the three left most mooring ropes and associated chain. From the Toolbar menu, select **Mirror**. Select nodes as indicated below so that a mirror axis in Y-direction is generated.



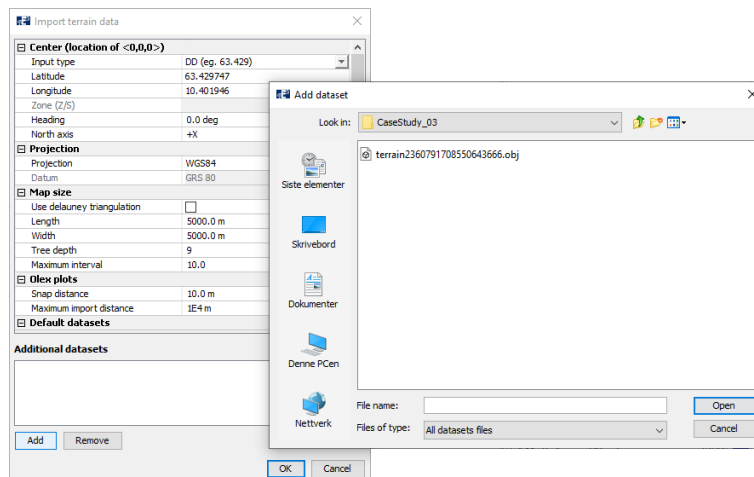
Select **Mirror** in the Properties window.



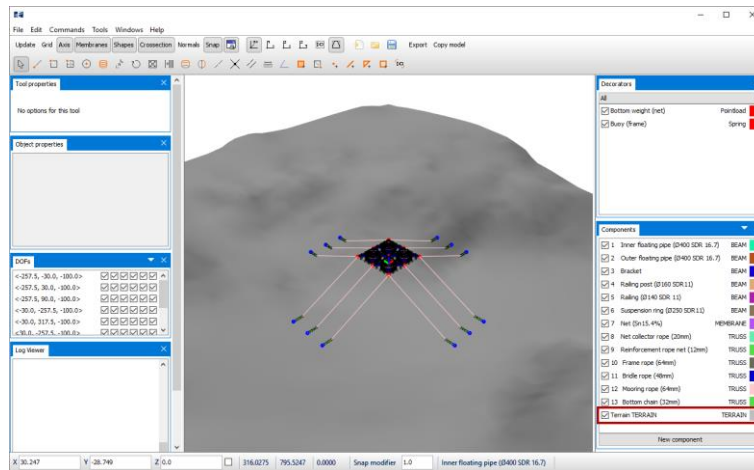
18 Import terrain The Import terrain options are found in **Commands > Import terrain**. A terrain model is prepared in terms of a **.obj**-file. This is a geometry definition file presenting vertexes and faces of polygons.

For AquaEdit to be able to position the terrain correctly, datum and position data is needed. In this case study, the **.obj**-file is prepared so you do not need to consider these parameters.

In **Additional datasets**, select **Add** and navigate to Documents\AquaSim\Demos\CaseStudy03 and select the **.obj**-file. Select **Open**.



In the Import terrain data window, select **OK**. The terrain is added to the Components window with the component type **TERRAIN**.

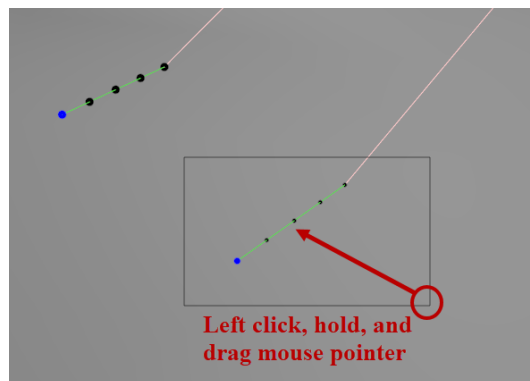


19 Save the model
20 Snap node to terrain

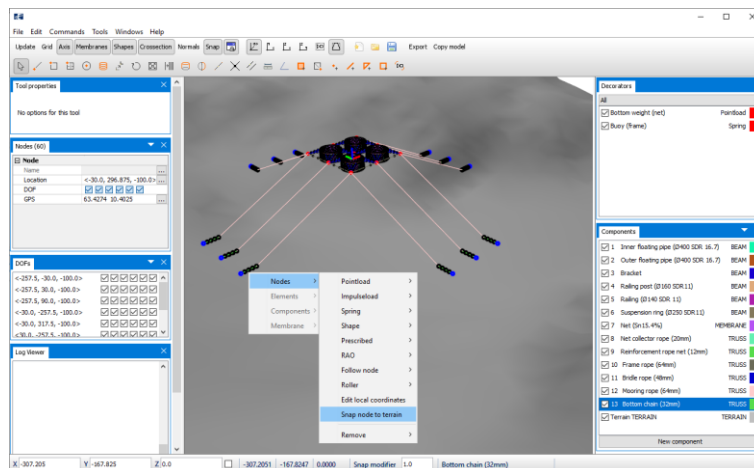
Remember to save your model regularly, so that data is not lost.

The bottom chain has been modelled at a depth of -100m, you should now snap the chain nodes to the terrain. Select all the nodes associated with the Bottom chain (32mm) in the 3D view.

For multi-selection of nodes, **left click** on one node and hold **CTRL**, then drag your mouse pointer from right to left so the selection square cover the chain.




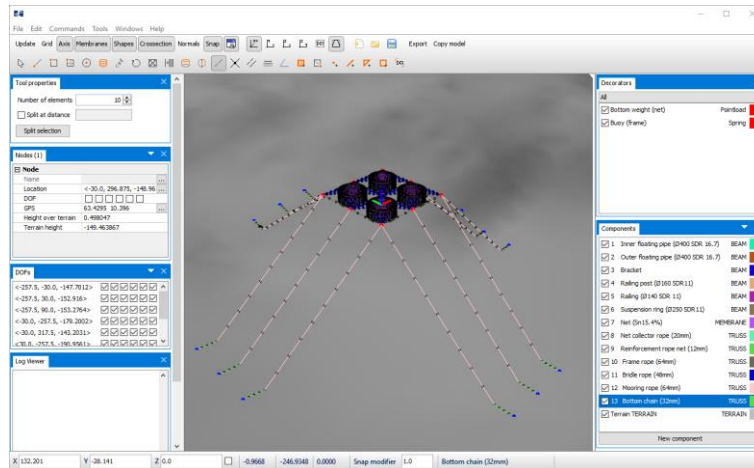
Right click in the 3D view > **Nodes** > **Snap node to terrain**.



The nodes connected with Bottom chain (32mm) is now moved from modelled position down to the terrain.

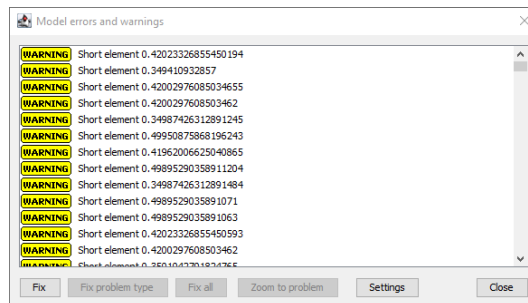
21 Discretization of Mooring rope

Split the elements associated with Mooring rope (64mm) in 10 elements. Select the intent **Split line**  with **Number of segments= 10**.



22 Verify model


Verify the input parameters in the model **Commands > Verify model**. AquaEdit gives a warning for short elements.



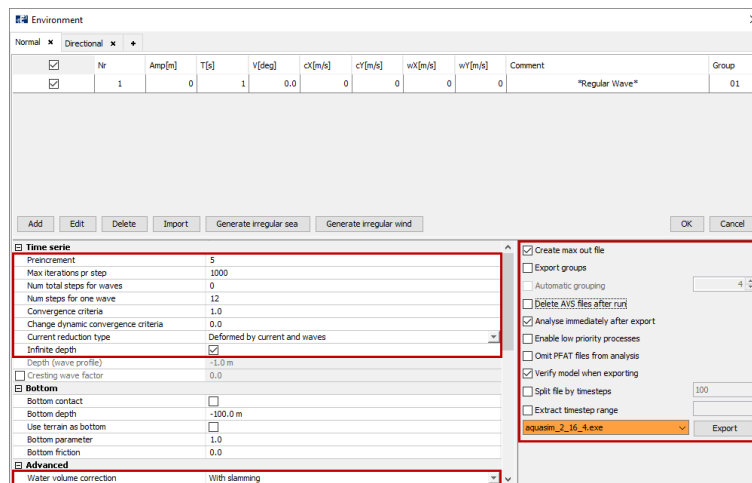
Tolerances for errors and warnings can be edited in **File > Settings > Errors and warnings**. Accept the warning, select **Close**.

4.5 Calculations static analysis

23 Enter environmental window

Enter the Environmental window by selecting **Export** . You should run a static analysis and control key parameters for equilibrium in the model.

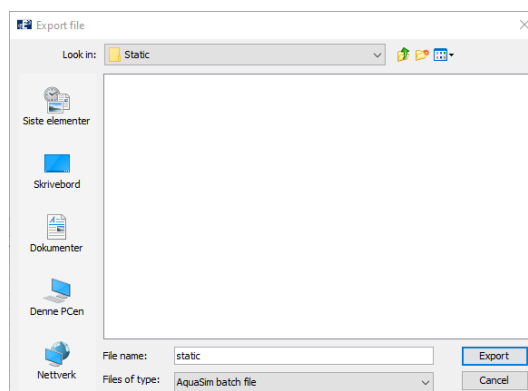
Select the tab **Normal**. Prepare the static analysis by making selections as described in the figure below. We have highlighted the selection for Water volume correction to be **With slamming**. This is due the components constituting the floating collar is dedicated the load formulation Morison submerged. AquaSim will identify if these elements are above or below the water surface and adjust the hydrodynamic properties for each analysis step. More detailed information about export parameters is found in CaseStudy01 or by reading the AquaEdit User Manual.



The area outlined in orange provides selection of solver versions. Your solver version may deviate from the figure above. You should choose the latest version from the dropdown menu.

Select **Export**. Navigate to a folder where you want to save the analysis. In this case study, we have chosen Documents\AquaSim\Analyses\CaseStudy03 and created a folder Static.

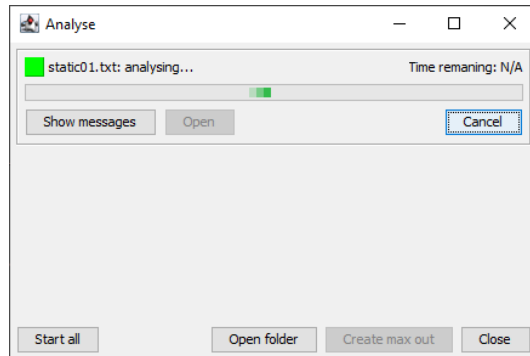
Type an appropriate name of the analysis file, we have chosen static.



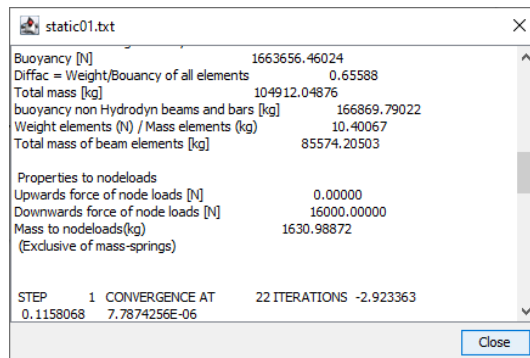
Select **Export**.

24 Run static analysis

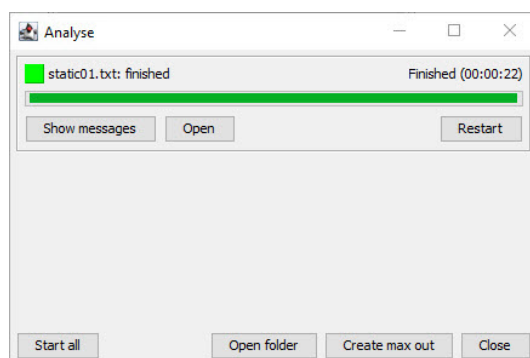
The **Analyse** window opens. To start the analysis select **Start** or **Start all**.



If you select **Show message**, you will be provided with key information about the analysis, which solver is run, statistics of input-parameters and status for each number of analysis step. It reports whether the analyzed step has converged or not, useful when assessing how the analysis evolves.



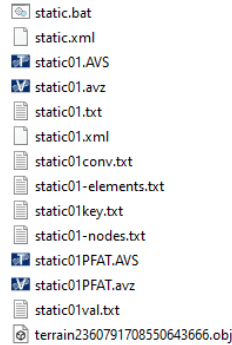
The analysis is successfully completed when the square in the upper left corner of the **Analyse** window gets green with the status **finished**.



4.6 Postprocessing static analysis

25 System files

In your file explorer, navigate to where you saved the analysis. In our case, Documents\AquaSim\Analyses\CaseStudy03\Static. Upon start and finish of analysis, several system files are generated. An introduction to these is provided here.



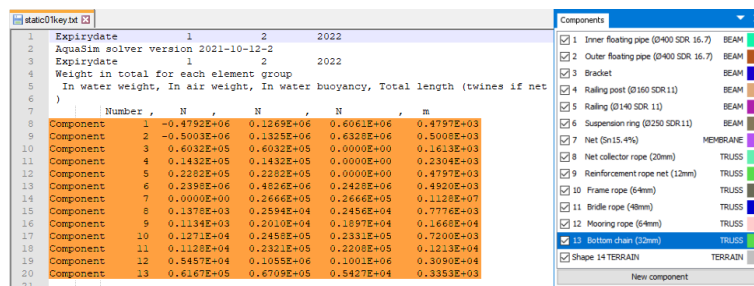
File name	File	Description
static.bat	Batch file	Script file for run of commands. Double click to start the analysis. This is consistent with starting analysis in AquaEdit from Export.
static.xml	XML file	This file holds the graphic information and other key data.
static01.AVS	Result file	Result file generated by the solver. Updated for each completed analysis step. Can be opened in AquaView and AquaTool during and after completion of analysis.
static01.avz	Result file	Result file generated by the solver after completion of analysis. This is a compressed version of the .AVS-file.
static01.txt	Input file	Holds information about the model and analysis parameters. Generated from AquaEdit upon start of analysis.
static01.xml	XML file	Holds graphic information and other key data for activated components from AquaEdit.
static01conv.txt	Result convergence file	Provides information about the analysis status. Consistent with the information provided in the Show message form the Analyse window.
static01-elements.txt	Result ID file	If individual elements are given custom names in AquaEdit, these are listed here.
static01key.txt	Result file key data	Contains a set of key data of the analysis model. This file is useful for self-checking of input parameters. Also holds information about license and the user.
static01-nodes.txt	Result ID file	If individual nodes are given custom names in AquaEdit, these are listed here.
static01PFAT.AVS	Result file	Result file generated by the solver upon completion of analysis. Holds information about key model data and max result values from the time series analysis.
static01PFAT.avz	Result file	Result file generated by the solver upon completion of analysis.

static01val.txt	Result file validation data	This file holds key data for the analysis. Useful for self-validation of input parameters.
terrain23[.].l66.obj	Object file	Contain information about terrain geometry.

26 Buoyancy of floating collar

The key.txt file is useful for self-check of key parameters from your model. Open the static01key.txt-file in a simple text editor reader e.g., Notepad or Notepad ++.

Navigate to the section saying Weight in total for each element group. These parameters are based on the input values from AquaEdit (i.e., before static equilibrium is calculated).



Components are listed according to the numbering from the Components window in AquaEdit. Component 1 correspond to Inner floating pipe (Ø400 SDR 16.7), component 2 correspond to Outer floating pipe (Ø400 SDR 16.7), and so on. Copy the parameters as indicated in orange and paste them in a calculation program after your preferences. We have chosen MS Excel.

Summarize the buoyancy for the components constituting the floating collar. And summarize the weight below water line for the mooring components.

Key data after static equilibrium, static analysis							
			Weight below	Weight above	Buoyancy	Total	
			water line	water line		length	Component name
CaseStudy03, detailed model	Component	1	-4.79E+05	1.27E+05	6.06E+05	4.80E+02	Inner floating pipe (Ø400 SDR 16.7)
	Component	2	-5.00E+05	1.33E+05	6.33E+05	5.01E+02	Outer floating pipe (Ø400 SDR 16.7)
	Component	3	6.03E+04	6.03E+04	0.00E+00	1.61E+02	Bracket
	Component	4	1.43E+04	1.43E+04	0.00E+00	2.30E+02	Railing post (Ø160 SDR11)
	Component	5	2.28E+04	2.28E+04	0.00E+00	4.80E+02	Railing (Ø140 SDR11)
	Component	6	2.40E+05	4.83E+05	2.43E+05	4.92E+02	Suspension ring (Ø250 SDR11)
	Component	7	0.00E+00	2.67E+04	2.67E+04	1.13E+06	Net (Sn15.4%)
	Component	8	1.38E+02	2.59E+03	2.46E+03	7.78E+02	Net collector rope (20mm)
	Component	9	1.13E+02	2.01E+03	1.90E+03	1.67E+03	Reinforcement rope net (12mm)
	Component	10	1.27E+03	2.46E+04	2.33E+04	7.20E+02	Frame rope (64mm)
	Component	11	1.13E+03	2.32E+04	2.21E+04	1.21E+03	Bridle rope (48mm)
	Component	12	5.46E+03	1.06E+05	1.00E+05	3.09E+03	Mooring rope (64mm)
	Component	13	6.17E+04	6.71E+04	5.43E+03	3.35E+02	Bottom chain (32mm)
Buoyancy floating collar					1238900 N		
Weight mooring components					69526 N		

For elements of type Morison submerged, the buoyancy is calculated based on the volume of the whole cross section (this is adjusted when static equilibrium is calculated). Note the total buoyancy of the floating collar and the total weight of the mooring components. Keep these figures for future reference when conducting CaseStudy04.

27 Check of key parameters

Other key parameters found in the static01key.txt file is information about weight and mass.

```

static01key.txt
63 Total, all elements
64 Weight in air [N] 1091155.76894
65 Weight in water[N] -572500.69130
66 The authomatic in and out of water option is used
67 Weight in air is used for part over water line
68 Weight in water is used for part under water line
69 Buoyancy [N] 1663656.46024
70 Diffrac = Weight/Bouancy of all elements 0.65588
71 Total mass [kg] 104912.04876
72 buoyancy non Hydrodyn beams and bars [kg] 166869.79022
73 Weight elements (N) / Mass elements (kg) 10.40067
74 Total mass of beam elements [kg] 85574.20503
75
76 Properties to nodeloads
77 Upwards force of node loads [N] 0.00000
78 Downwards force of node loads [N] 16000.00000
79 Mass to nodeloads (kg) 1630.98872
80 (Exclusive of mass-springs)
81

```

Navigate to the section saying Total, all elements. Two parameters indicating the state of your model is highlighted in the figure above:

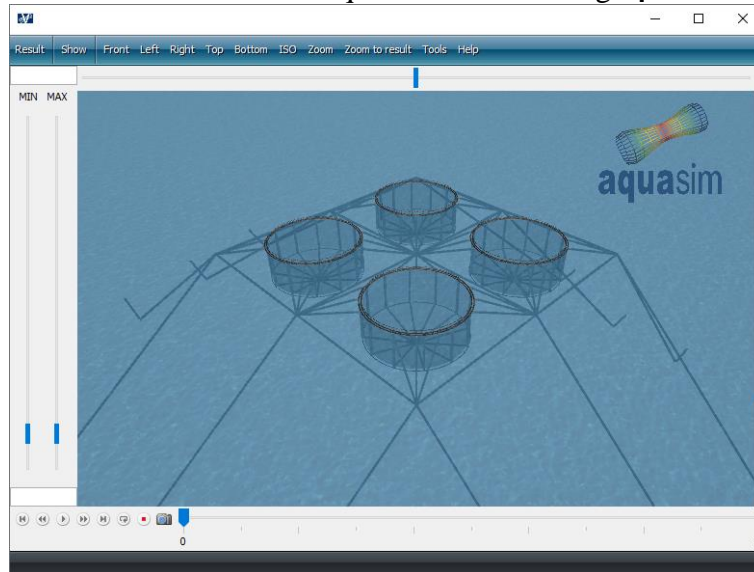
- Diffrac = Weight / Buoyancy of all elements, provides information about the relation between weight and buoyancy of components in your model. If this balance is perfect, Diffrac should equal 1.0. If this number is far from 1.0 this can indicate e.g., inadequate definition of volume of components compared to the weight of it.
- Weight elements (N) / Mass elements (kg), provides information about the relation between weight and mass of components in your model. If this balance is perfect, this parameter should equal 9.81. You should not accept significant deviances from 9.81, which indicates e.g., inadequate definition of Weight in water. If Weight in water is overridden, as we did for component Frame rope (64mm) in step number 3, deviance from 9.81 must be expected.

Further self-checking can involve controlling Downwards force of node loads [N], which is reported to be a total of 16000.0N. This can be validated through controlling the input defined in AquaEdit. Go back to your AquaEdit model, and to the Decorators window. Here you find a Pointload which is located at the bottom of each net pen. **Double click** Bottom weight (net) in the Decorators window. A force of -4000N is defined, this will result in a total of 4000 times 4 net pens = 16000N in total downwards force in your model. This matches the value in the static01key.txt-file.

Exit the static01key.txt-file.

28 Open results in AquaView

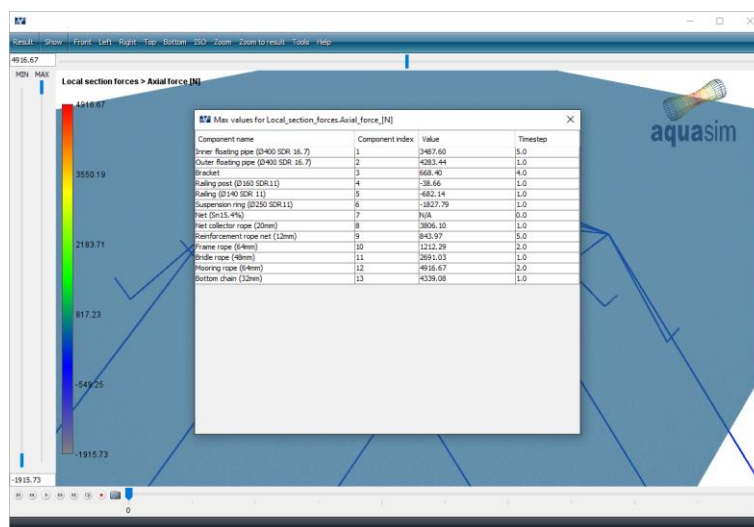
From the analysis folder Static, open the analysis in AquaView by **double click** on static01.avz. This is equivalent to selecting **Open** from the Analyse window.



Press **Play** from the playback menu and observe the response of your model. Zoom in to structural details and see how it displaces. If your analysis, against conjecture, should have failed to complete successfully, or an unexpectedness have occurred, you can open the equivalent result file from Documents\AquaSim\Demos\CaseStudy03\Static.

29 Pretension of mooring components

As a result of the mooring component's weight in water, they will achieve a pretension. The pretension is found by plotting the axial forces. In the Toolbar menu, select **Result > Local section forces > Axial force [N]**. You should find the maximum pretension for the mooring components; these can be found by pressing **j** on your keyboard.



The first column in the Max values for- window list the component names according to the definition you made in AquaEdit. The second column is the component numbering, or index. The third column is the maximum value for the respective

component group, and the last column indicate which analysis step this max value occurred.

Copy the rows for component number 10 to 12 and paste it in the same sheet you established for the key-data.


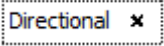
	J	K	L	M	N	O
1	Pretension in mooring components, static analysis					
5	CaseStudy03, detailed model					
6						
7						
8		Component name		Component no.	Pretension [N]	Step no.
9						
10		Frame rope (64mm)		10	1212.29	2
11		Bridle rope (48mm)		11	2691.03	1
12	Mooring rope (64mm)		12	4916.67	2	
13						
14						
15						
16						
19						

Keep these figures for future reference when conducting CaseStudy04.

Static considerations are now finished. Exit AquaView and the Analyse window from AquaEdit.

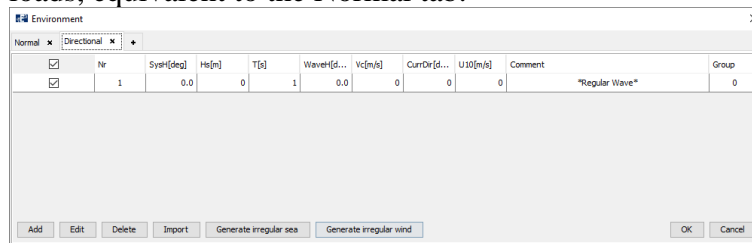
4.7 Calculation dynamic analysis

30 Enter Environmental window

Go back to your model in AquaEdit. You should investigate the response and forces in mooring components for two combinations of waves and current. Enter the Environmental window by selecting **Export**  from the Toolbar menu. Select the tab **Directional** . In Directional, the environmental loads are based on defining directions from where AquaSim decomposes the loads into vectors.

31 Load conditions and Export parameters

Each row in the Directional window represents a combination of environmental loads, equivalent to the Normal tab.

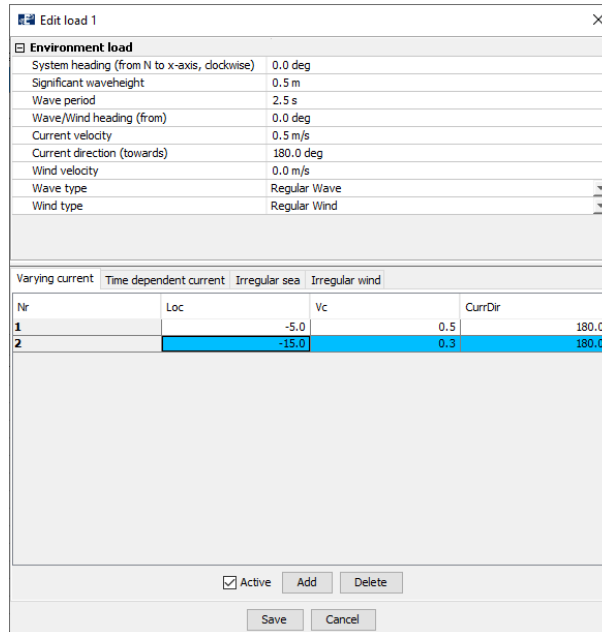


The input values for the load conditions (LC) are described in the table below.

Nr	SysH[deg]	Hs[m]	T[s]	WaveH[d eg]	Vc[m/s]	CurrDir [deg]	U10[m/s]
Row number of the LC	System heading, orientation of the model from compass north to the global x-axis. Clockwise, in degrees.	Significant wave height. The average wave height (from trough to crest) if the highest one-third of the waves. In meters.	Period of the associated wave height. In seconds.	Heading of the wave. Waves are defined with the direction it originates from. In degrees.	Current speed. In meters per second.	Direction of current. Current is defined with the direction it propagates to. In degrees.	Wind speed at 10 meters altitude. In meters per second.
1	0	0.5	2.5	0	0.5 / 0.3	180 / 180	0
2	0	1	3.0	45	0.3 / 0.2	225 / 225	0

The slashes for Vc and CurrDir indicates a non-uniform current profile in the water column. The system heading provides a global orientation of the environmental loads. This facilitates modelling because you do not have to consider the orientation of the aquaculture facility when you are building the system. Note that the definition of wave direction is *from*, meaning that the input direction is from where waves originate. Current is opposite, where the definition of current direction is *to*, meaning that the input direction is to where the current propagates.

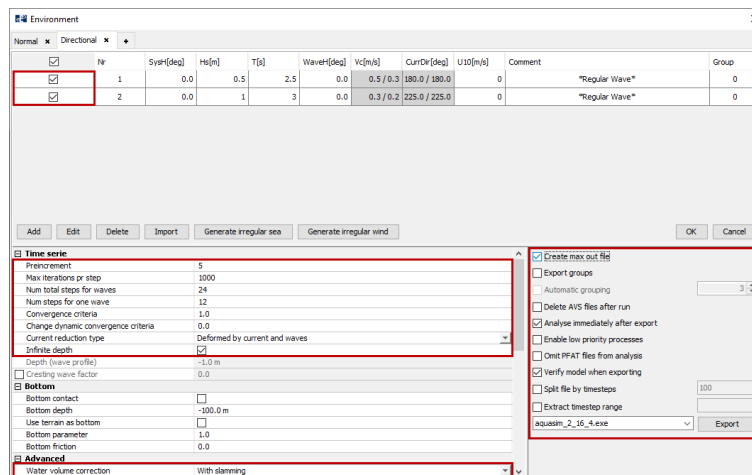
In the Environmental window, click on row number 1 and select **Edit**. In the Edit load window, type the values for LC 1. To add varying current, go to the **Varying current** tab, select the checkbox for **Active** and **Add** two rows.



Nr	Loc	Vc	CurrDir
1	-5.0	0.5	180.0
2	-15.0	0.3	180.0

Select **Save**.

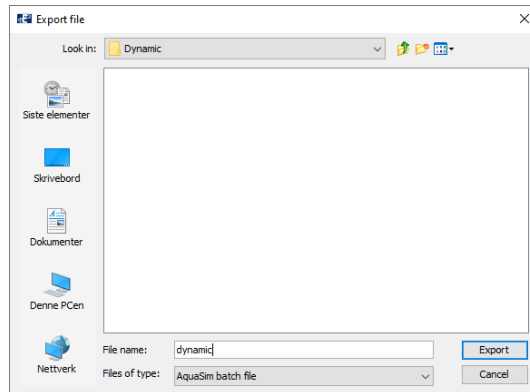
Add a new row for load condition 2 and fill in the environmental values from the table, following the same procedure as described above. When finished, select **Save**.



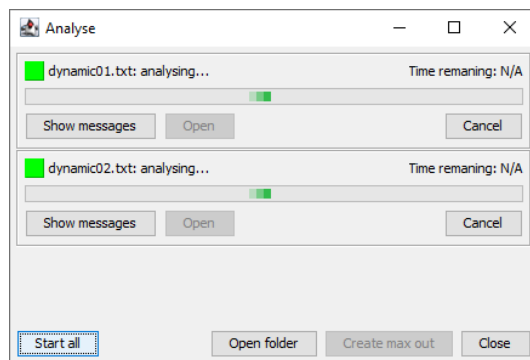
Nr	Syst(deg)	Hs[m]	T[s]	WavePer(deg)	Vc(m/s)	CurrDir(deg)	U10(m/s)	Comment	Group
1	0.0	0.5	2.5	0.0	0.5 / 0.3	180.0 / 180.0	0	"Regular Wave"	0
2	0.0	1	3	0.0	0.3 / 0.2	225.0 / 225.0	0	"Regular Wave"	0

Make sure load conditions are activated by selecting the checkboxes in upper left corner of the window. Dynamic analysis with two regular waves should be conducted; **Num total steps for waves = 24** and **Num steps for one wave = 12**. This corresponds to a wave profile defined with 12 points and $24/12=2$ waves. The remaining settings in your Environmental window should be similar to the figure above.

Select **Export**. Navigate to an appropriate place to save the analysis, we have chosen Documents\AquaSim\Analyses\CaseStudy03\Dynamic. Type an analysis name e.g., dynamic.



Select **Export**. In the Analyse window, select **Start all**.



Dynamic analysis of the aquaculture facility has now started. Depending on your computer, this analysis should be finished within approximate 10-15 minutes. Follow status and development of the analysis by selecting **Show message**.

32 Create max out file

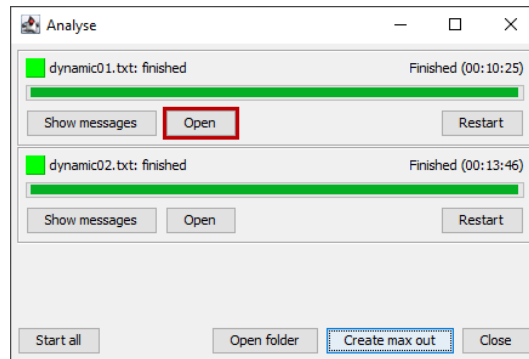
The maximum resulting forces from a set of analyzed load conditions are for the most of it applied when designing and dimensioning mooring systems. AquaSim finds the maximum values and store them in a max-out-file. This can be generated by selecting **Create max out** in the Analyse window upon completion of analysis.

When both analyses are finished, select **Create max out** from the Analyse window. We leave this for later application.

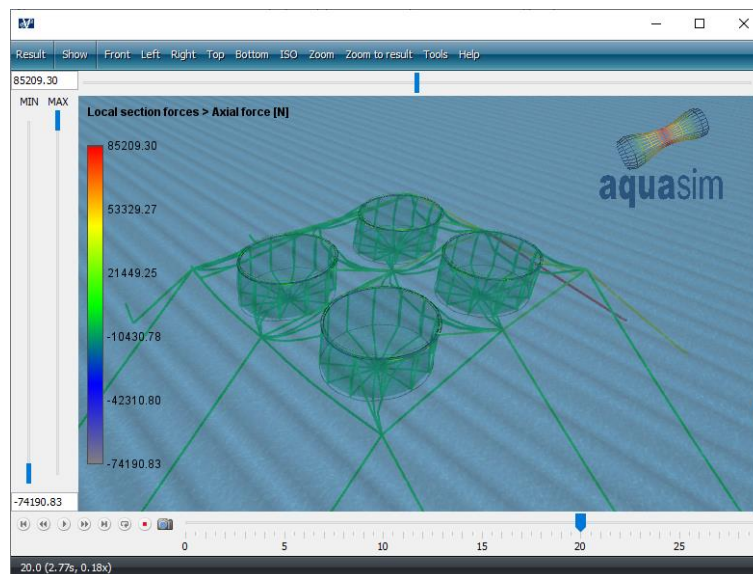
4.8 Postprocessing dynamic analysis

33 Visual interpretation of results

Load the result file from the first load condition by selecting **Open** from dynamic01.txt in the Analyse window.



Load condition 01 will be loaded in AquaView. Plot **Axial force [N]** and play the analysis by selecting **Play** from the playback menu. Observe how the axial forces in the model evolve as waves and current passes the facility.



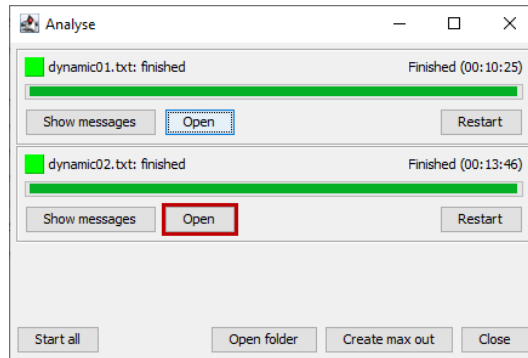
The playback speed can be adjusted by selecting **Set duration** from the playback menu. Unit is seconds, if you type 30 it will take 30 seconds to play from the first step to the last step.

When observing the analysis playback, you should observe the model behaving as expected. That is; forces in components rise when wave passes, the net deforms due to current and so on. This is a type of visual validation of the analysis. If you observe “jumping” elements or other abnormal response, this can indicate that the analysis has potential for improvements. An analysis can produce erroneous results despite achieved convergence for all steps. “Jumping” elements or other abnormal fluctuations may indicate a lack of damping in the system or eigen period issues.

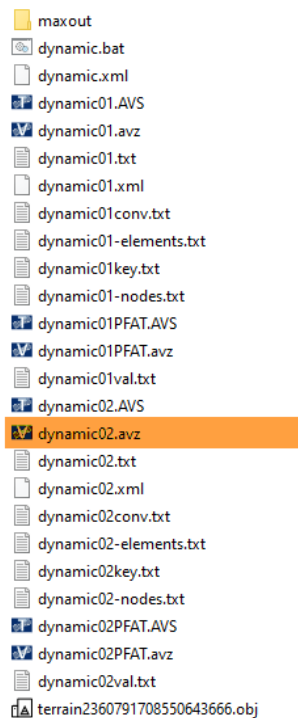
These considerations are not the scope of this case study but is highlighted as something to be aware of when working with dynamic problem solving.

Exit AquaView.

Load the result file from the second load condition by selecting **Open** from dynamic02.txt in the Analyse window.



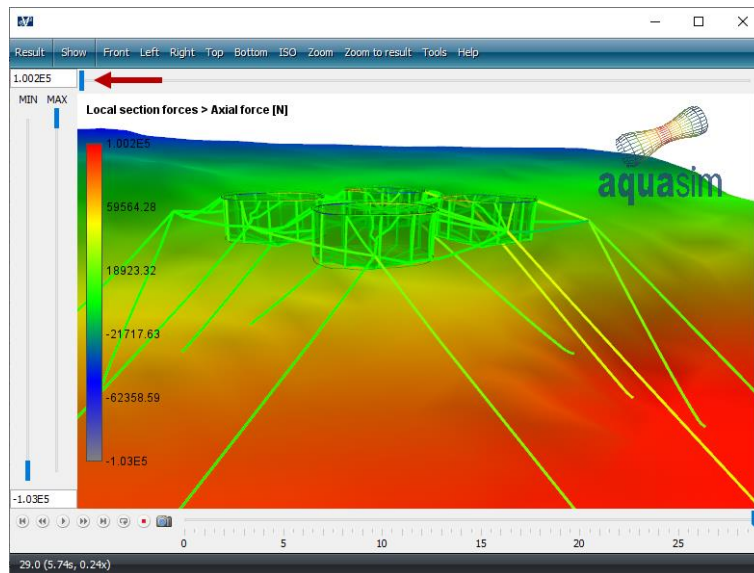
If you have closed the Analyse window, you may load the result file from your analysis folder in the File Explorer. **Double click** on dynamic02.avz.



In AquaView, you should plot the **Axial force [N]**. Follow the same procedure as for load condition 01, play the analysis and observe the response of your model. This is for visual validation of your model.

34 View terrain in AquaView






The terrain may be loaded in AquaView as well. In the Toolbar menu, select **Show > Shapes > Terrain**. Depth color can also be activated by selecting **Toggle depth color** in the Shapes menu. Adjust the transparency of the water surface by dragging the upper blue bar to the left.



Exit AquaView.

35 Maximum resulting forces

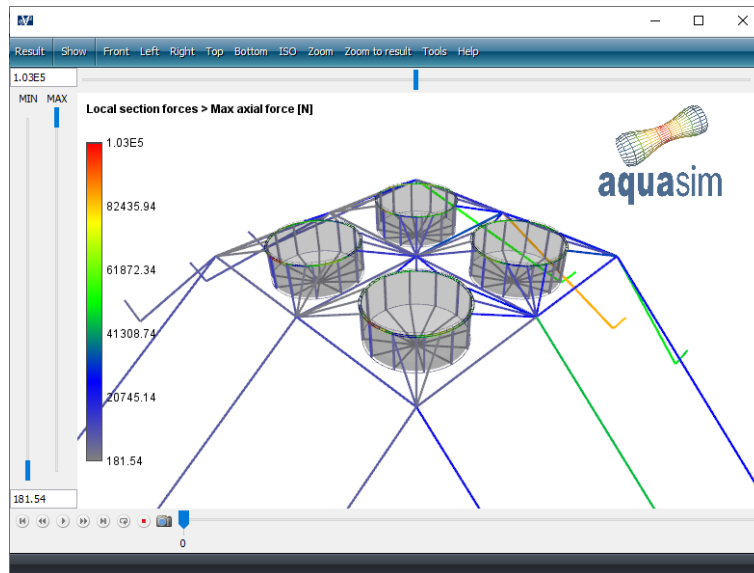
In this section you are to find maximum forces in the mooring components from the two analyzed load conditions. In your analysis folder, you will find a folder called maxout. Enter this folder. Here you will find files that contain maximum loads from both analyzes. These were generated when you selected **Generate max out** from the Analyse window in step number 32.

-  dynamic.xml
-  dynamic_max_out.avs
-  dynamic_max_out.avz
-  dynamic01PFAT.avz
-  dynamic02PFAT.avz

Load dynamic_max_out.avz by **double click** the file. AquaView will be loaded. If you failed to generate the maxout-files, you can find equivalent file in Documents\AquaSim\Demos\CaseStudy03\Dynamic\maxout.

The maxout-file only contains maximum values from load condition 01 and 02, and consequently playback functionality is not available.

You should now find max axial force in the mooring components Frame rope (64mm), Bridle rope (48mm) and Mooring rope (64mm). From the Toolbar menu in AquaView, select **Result > Local section forces > Max axial force [N]**.



Press **j** on your keyboard to enter the window for maximum values of each component group.

Component name	Component index	Value	File index
Inner floating pipe (Ø400 SDR 16.7)	1	1.03E5	dynamic02PFAT....
Outer floating pipe (Ø400 SDR 16.7)	2	1.00E5	dynamic02PFAT....
Bracket	3	18190.17	dynamic02PFAT....
Railing post (Ø160 SDR11)	4	745.62	dynamic02PFAT....
Railing (Ø140 SDR 11)	5	8404.33	dynamic02PFAT....
Suspension ring (Ø250 SDR11)	6	14557.25	dynamic02PFAT....
Net (Sn15.4%)	7	N/A	N/A
Net collector rope (20mm)	8	14949.11	dynamic02PFAT....
Reinforcement rope net (12mm)	9	4396.10	dynamic02PFAT....
Frame rope (64mm)	10	33650.51	dynamic01PFAT....
Bridle rope (48mm)	11	32381.21	dynamic02PFAT....
Mooring rope (64mm)	12	85209.30	dynamic01PFAT....
Bottom chain (32mm)	13	84799.53	dynamic01PFAT....

What the different columns in this window represent is provided in step number 29 in this case study. In addition, we have the column File index. This column indicates from which load condition the max values originate.

Copy the abovementioned components to clipboard by selecting component number 10 to 13 and **CTRL + c**. Paste the data in the same sheet you established from the static analysis.

	Q	R	S	T	U	V	W
1	Axial force in mooring components, dynamic analysis						
5							
6							
7							
8	CaseStudy03, detailed model	Component name		Component no.	Axial force [N]	File index	
9							
10		Frame rope (64mm)		10	33650.51	dynamic01PFAT.avz	
11		Bridle rope (48mm)		11	32381.21	dynamic02PFAT.avz	
12		Mooring rope (64mm)		12	85209.30	dynamic01PFAT.avz	
13	Bottom chain (32mm)		13	84799.53	dynamic01PFAT.avz		
14							
15							
16							
19							

Keep these figures for future reference when conducting CaseStudy04.

Dynamic considerations are now finished. Exit AquaView and AquaEdit.

4.9 Summary

In this case study we have modelled an aquaculture facility in a 2 x 2 configuration of cages. Mooring components and terrain have been added to a prepared model. To simplify and increase effectiveness in modelling you have learned how to apply the properties library, duplication, and mirror of elements.

By adding a bathymetric bottom (terrain) to the model you have seen that the bottom chain achieves a more realistic configuration. You have also learned about the different system files that is generated when performing analyses. And that these are useful when evaluating the input and condition of your model.

Several results have been copied and transferred to a calculation sheet for future reference. In the next case study, you are to model the same aquaculture facility in a more simplified manner and compare results from this case study.

5 Case study 04 – Simplified aquaculture cage and mooring

5.1 Learning objectives

Upon completion of this case study, you will be able to:

- Apply additional modeling intents, as draw circle, extrude elements
- Establish routines for transferring experience and information from one analysis case to the next
- Understand, and evaluate strategies, for choosing degree of model details

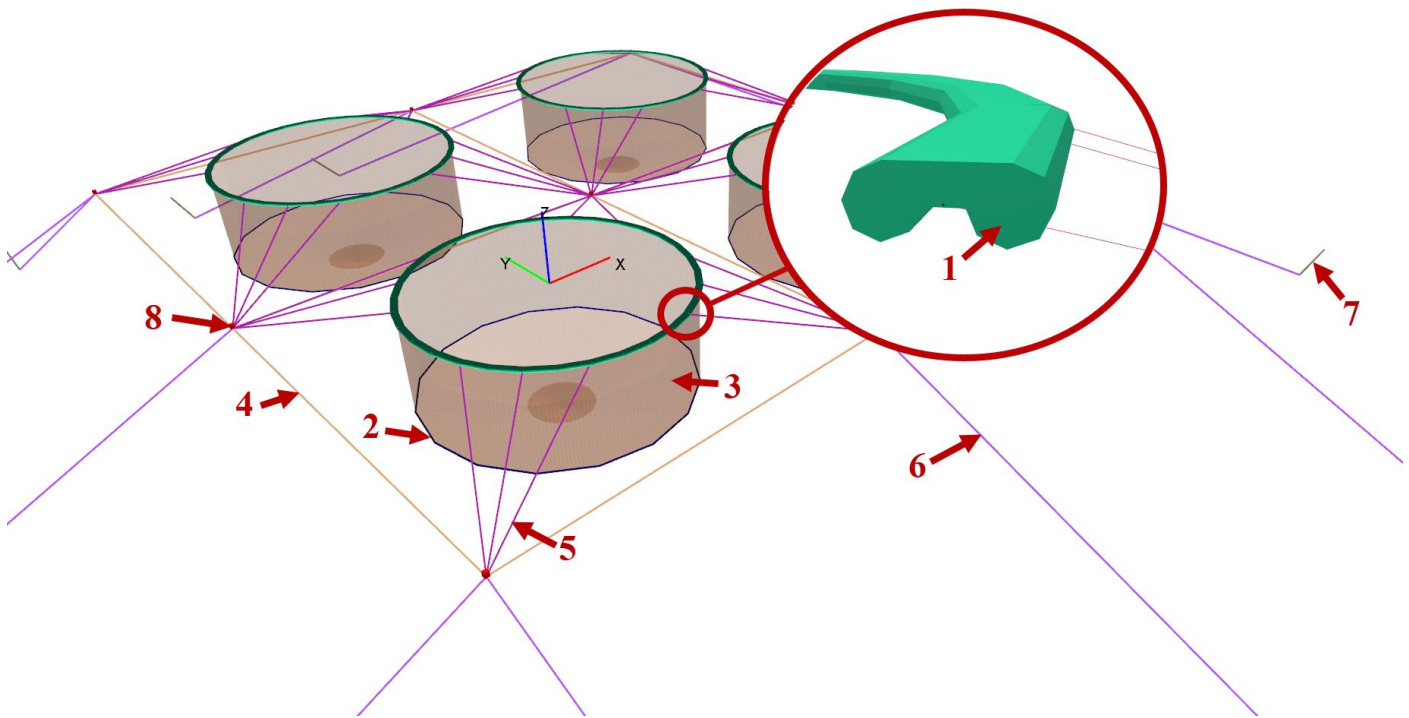
5.2 Introduction

You are to model the same aquaculture facility as in CaseStudy03 in a simplified manner. Through static analysis selected key parameters, as mooring rope tension and buoyancy of the floating collar, should be checked. These parameters are to be compared to corresponding figures from the previous case study, in order to ensure these correspond.

By dynamic analysis, you are to find maximum axial force in selected mooring components and evaluate the results.

5.3 Main components of aquaculture facility

The main simplifications of the aquaculture facility in this case study lies in how the floating collar is modelled and the discretization. Instead of modelling each floating pipe, with brackets, railing post and railing, the floating collar is represented with a double tube. The model and main technical specifications are presented in the figure and table below.

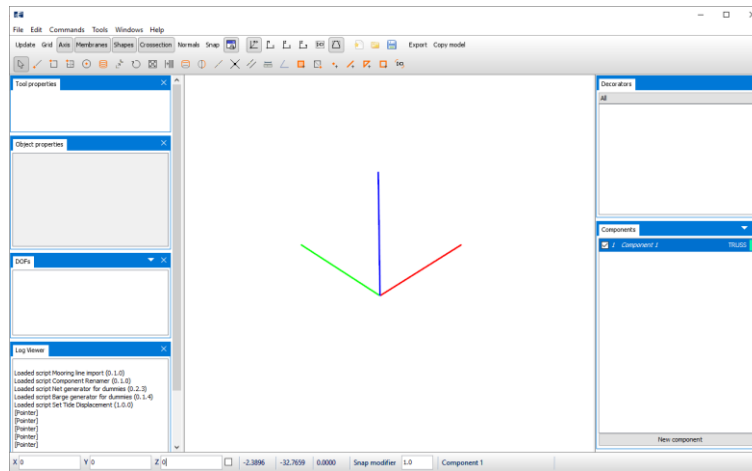


Component number	Description	Technical data*
1	Floating collar, double tube	PE100, Ø400, SDR16.7, CIR120m
2	Suspension ring for net pen	PE100, Ø250, SDR11
3	Net pen	Sn15.4%, CIR120m, D15m
4	Frame rope	64mm, L60m, W60m
5	Bridle rope	48mm
6	Mooring rope	64mm, L220m
7	Bottom chain	32mm, L27.5m
8	Buoy	L1.0m, W1.0m

* Abbreviations: PE=polyethylene, Ø=pipe diameter, SDR=pipe diameter/wall thickness, CIR=circumference, Sn=solidity, D=depth, L=length, W=width.

5.4 Preprocessing

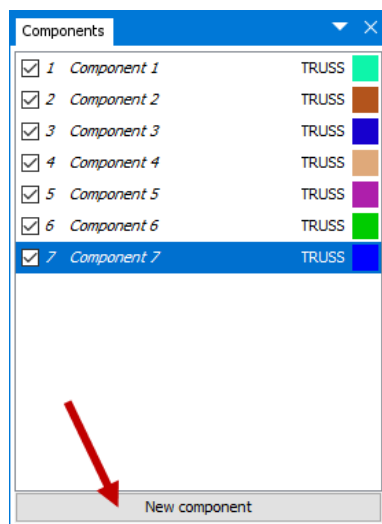
1 Load AquaEdit The model should be built from scratch. Load AquaEdit.



2 Generate component groups

From the table in section 5.3 you see the model consist of 7 components, in addition to buoys. Buoys are considered as a node decorator in AquaEdit, and not as a component.

From the Components window, generate six additional new component groups by selecting **New component**.

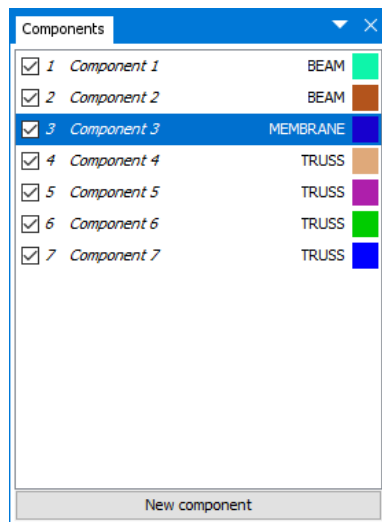


We need to decide for the component types. The type depends on the cross-sectional characteristics of the various components. You should evaluate if the component is able to withstand only axial/ compressive forces, is it able to withstand bending and torsion moments, is it a fabric type?

Below you find evaluation of the different components and the component types.

Component number	Evaluation	Component type
1 Floating collar (Ø400 SDR16.7)	A floating collar consist of polyethylene pipes and brackets with massive thickness. These cross sections can withstand axial/compressive-, bending-, and torsional forces. The component type should then be BEAM.	BEAM
2 Suspension ring (Ø250 SDR11)	The suspension ring is made of polyethylene pipe with a massive thickness. It is able to withstand axial/compressive-, bending-, and torsional force. The component type should then be BEAM.	BEAM
3 Net pen (Sn15.4%)	The net pen is an assembly of threads, either sewn or tied together. The assembly will form a network of meshes in which water can flow through. The net is flexible, and as for ropes, only capable of withstanding axial- and compressive forces. The component type should be MEMBRANE.	MEMBRANE
4 Frame rope (64mm)	As presented in CaseStudy03 the mooring components are considered flexible, only capable of withstanding axial- and, to some extent, compressive forces. The component type should be TRUSS.	TRUSS
5 Bridle rope (48mm)		TRUSS
6 Mooring rope (64mm)		TRUSS
7 Bottom chain (32mm)		TRUSS

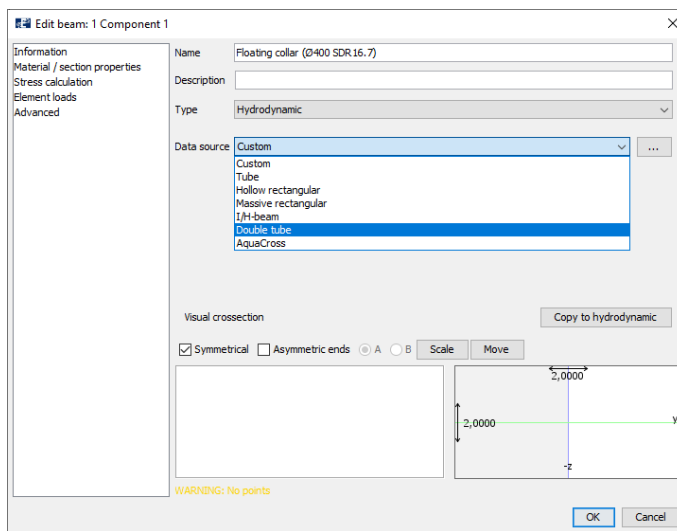
In the Components window change the component types according to the table above. This is done by **right click > Type >** select appropriate component type.



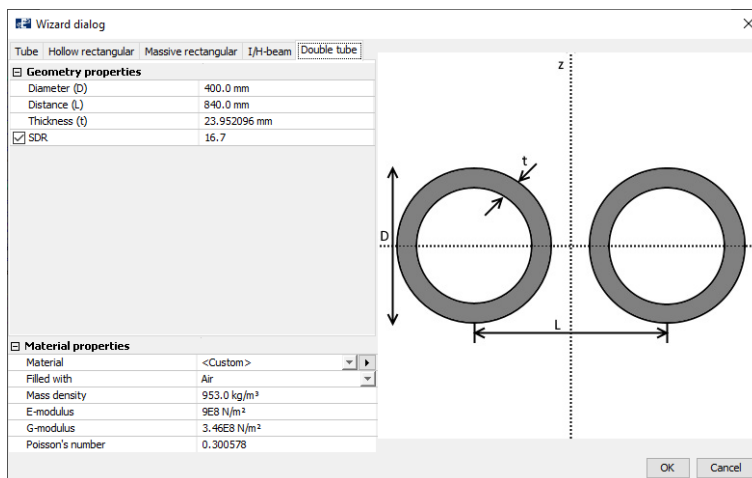
3 Properties of Floating collar

You are to assign cross sectional properties to the floating collar. The floating collar is modelled in a simplified manner; you are to model them as one cross section with similar properties as two individual pipes and brackets. In the Components window, **double click** Component 1 to enter the Edit beam window.

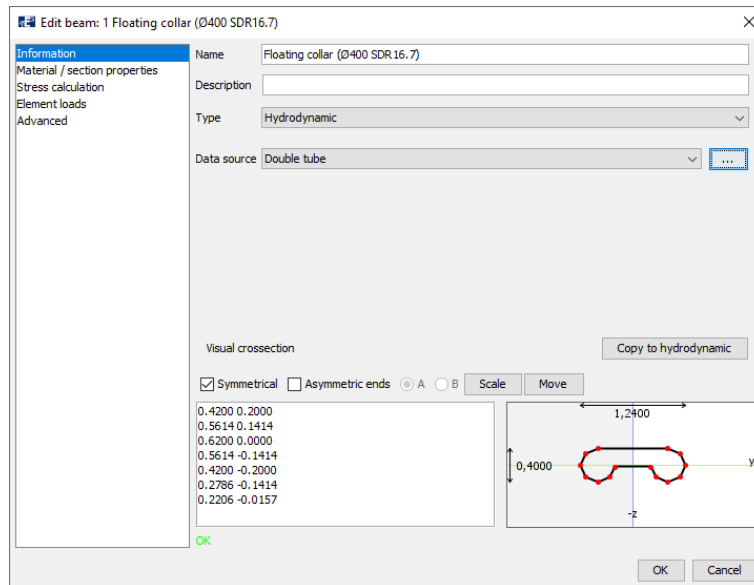
Assign the component an appropriate name e.g., Floating collar (Ø400 SDR16.7). The double tube is only available for the load formulation Hydrodynamic. In the row **Type**, select **Hydrodynamic** from the drop-down menu. From the drop-down menu in **Data source** select **Double tube**.



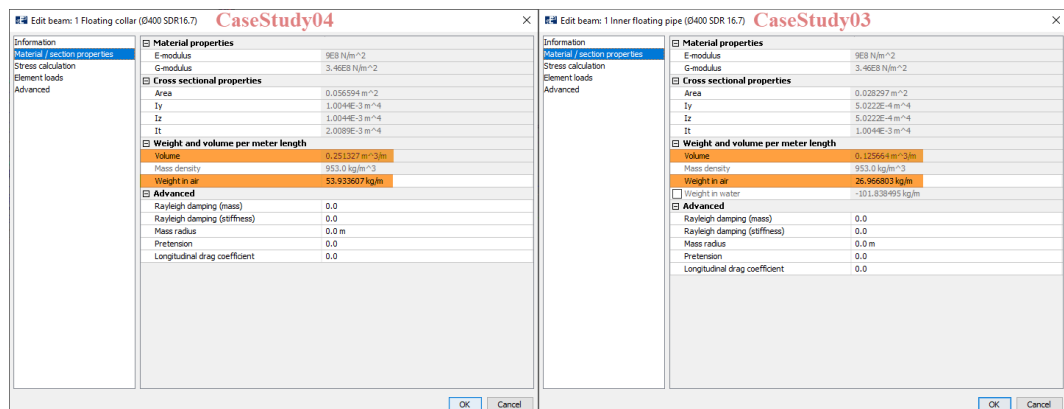
The wizard dialogue for cross sections will be loaded. The pipe diameter is 400mm, and the SDR is 16.7. Type **Diameter (D)**= 400mm, and **SDR**= 16.7. Distance (L) is the center-to-center distance between the floating pipes. By measuring the length of a bracket from CaseStudy03, we find this distance to be 840mm. Type **Distance (L)**= 840mm.



In the **Material properties** section select **Plastic** as the material, and that the pipe should be filled with **Air**. Select **OK**.



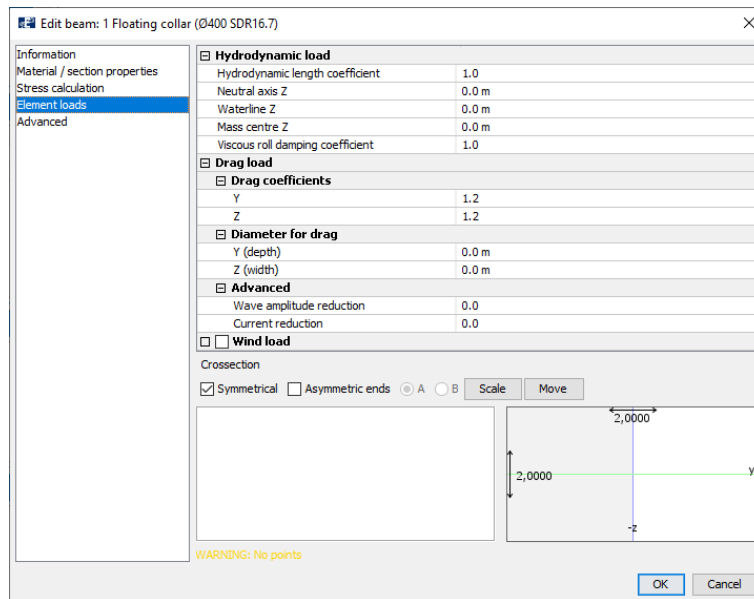
The **Information** tab will be updated with a visual cross section of the double tube, found in the lower right corner of the window. Note that the cross section has its own local coordinate system with y- and z-axis. You should now control that the volume and weight of the double tube is consistent with the detailed model from CaseStudy03. Cross section properties are automatically calculated by the cross section wizard. Go to the **Material/ section** properties tab to review the parameters. **Volume** and **Weight in air** should be twice the corresponding parameters as for a one floating pipe from CaseStudy03. In the figure below we have compared the component Floating collar (Ø400SDR16.7) from CaseStudy04 and component Inner floating pipe (Ø400 SDR 16.7) from CaseStudy03.



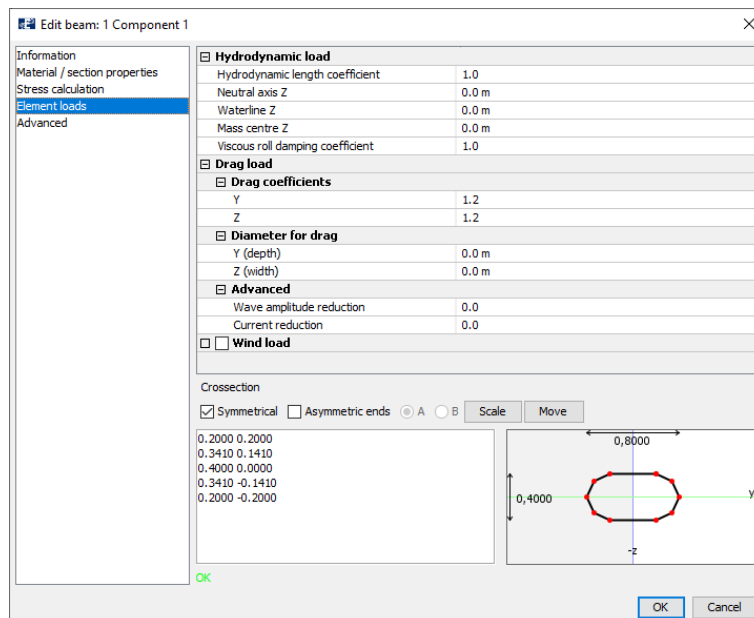
The **Stress calculation** tab provides parameters needed to calculate stresses in the component. Input values here are distances from neutral axis of your cross section to the edge of your cross section.

Go to the **Element loads** tab. Here, hydrodynamic coefficients needed for the analysis is defined. Applying the load formulation Hydrodynamic, AquaSim calculates the forces based on both strip theory and Morison equation. For AquaSim to calculate the part for strip theory, a calculation cross section is needed. This is found in the **Crosssection**-part of the Element loads tab. As you see, this part has

no cross section. You can either define it by typing coordinates, or you can copy the visual cross section you generated in the wizard.



Go back to the **Information** tab and select **Copy to hydrodynamic**. Re-enter the **Element loads** tab to see an approximation of the visual cross section has appeared in the **Crossection**-part.

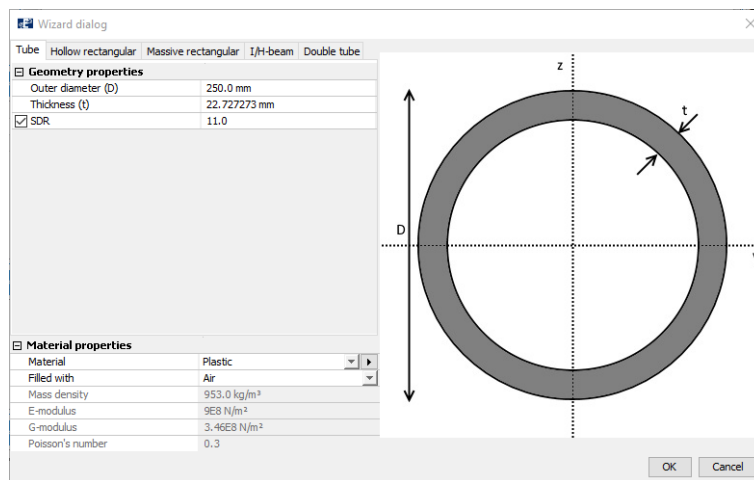


Select **OK**.

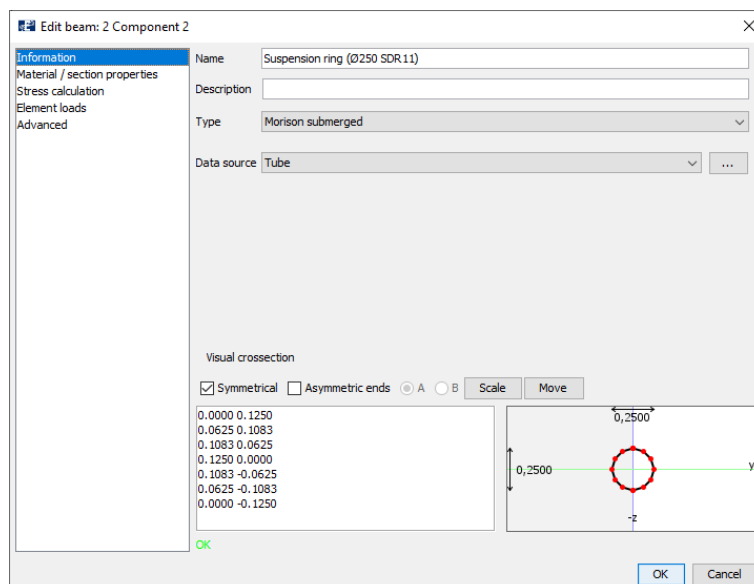
4 Properties of Suspension ring

The suspension ring should consist of a single PE-pipe with a weight of 100 kg/m in air. The purpose of the suspension ring is to suspend the net pen in vertical as well as in horizontal direction, ensuring the fish to have sufficient volume to move in. To achieve a reasonable weight for the suspension, the pipes are often equipped with chain or similar.

Double click Component 2 in the Components window. Assign the name Suspension ring (Ø250 SDR11), and select **Morison submerged** as **Type**. The ring should be modelled as a tube. From **Data source**, select **Tube**. In the wizard dialogue for cross section, type **Outer diameter(D)**= 250mm and **SDR**= 11. Leave the **Material properties** with default values.



Select **OK**.



The weight in air is defined in the **Material/ section properties** tab. Navigate to this tab and in the row **Weight in air** type 100 kg/m. Select **OK**.

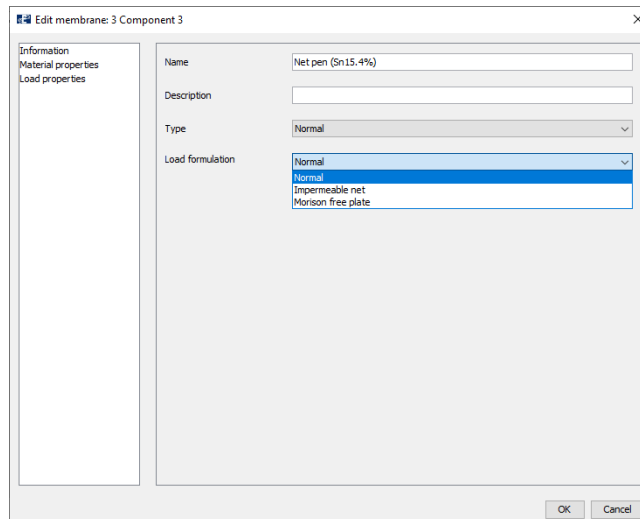
5 Properties of Net pen

The net pen is represented by the component type MEMBRANE. With a solidity of 15.4%. The solidity indicates the size of the mesh and the permeability. Two-dimensional solidity is commonly calculated as:

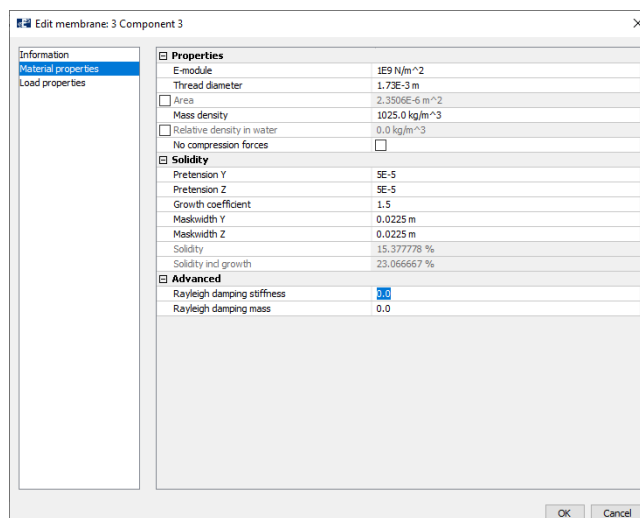
$$Sn = \frac{d}{L_y} + \frac{d}{L_z}$$

where Sn is the solidity, d is the diameter of the thread, L_y is the mesh width and L_z is the mesh height. In this case study, a thread diameter of 1.73E-03m and mesh size of 2.25E-02m in y-and z-direction will correspond to a solidity of approximate 15.4%.

In the Components window, **double click** the component with component type MEMBRANE. In the Edit membrane window, you will be met with an **Information** tab. In the row **Name**, type a name e.g., Net pen (Sn15.4%). **Type** provides the possibility to choose between several types of membranes. For classical net pens, as in this case study, select **Normal** from the drop-down menu in the row **Type**. The row **Load formulation** allow for selecting different methods of how hydrodynamic forces should be calculated, form the drop-down menu select **Normal**.



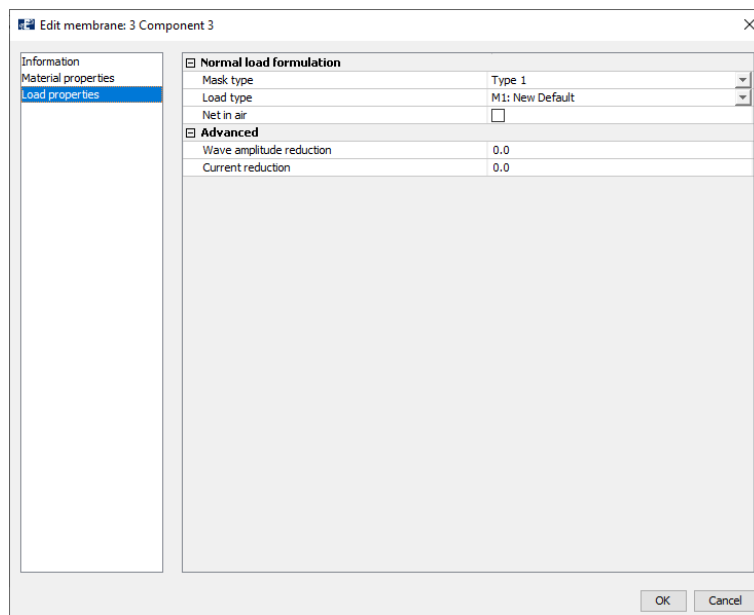
To define the cross sectional properties of the net pen, you should enter the **Material properties** tab.



E-module is the elasticity modulus of the thread, type $1.0E+09N/m^2$. This is approximately to nylon material. The **Thread diameter** is $1.73E-03m$. The size of the mesh is **Maskwidth Y** = $2.25E-02m$ and **Maskwidth Z** = $2.25E-02m$. The **Solidity** should now be close to 15.4%.

Notice the option **Growth coefficient**. As for all types of structures at sea, the net pen will be exposed to growth in terms of shells and algae. To account for growth, AquaSim provides a multiplication factor to increase the solidity of the net. **Growth coefficient** should be equal to 1.5, and the solidity including the growth coefficient is found in the row **Solidity incl growth**.

Navigate to **Load properties** tab. Options for mask types in the mesh is found in **Mask type**. You can choose between meshes ordered as horizontal squares, or like diamonds. From the drop-down menu, select **Type 1**, this corresponds to horizontal squares. From the drop-down menu in **Load type**, select **M1: New default**. This selection defines how drag forces is calculated with respect to how the membrane deforms under influence of current.

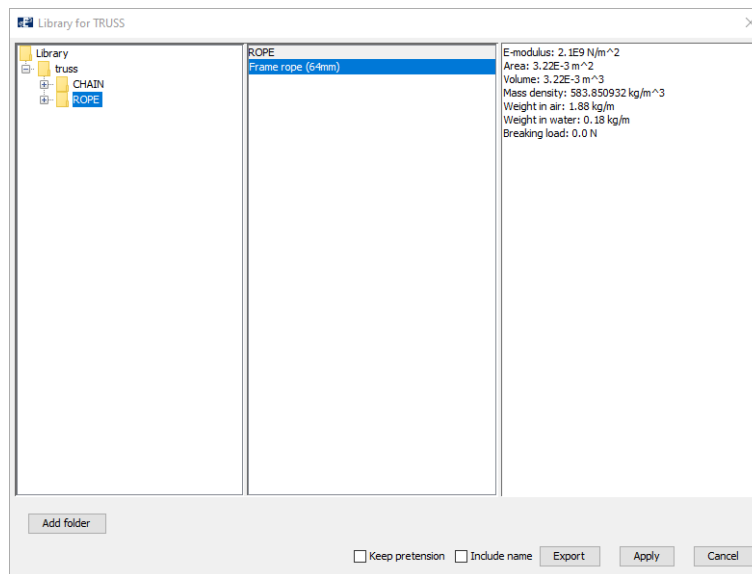


The necessary properties for the net pen are now defined, select **OK**.

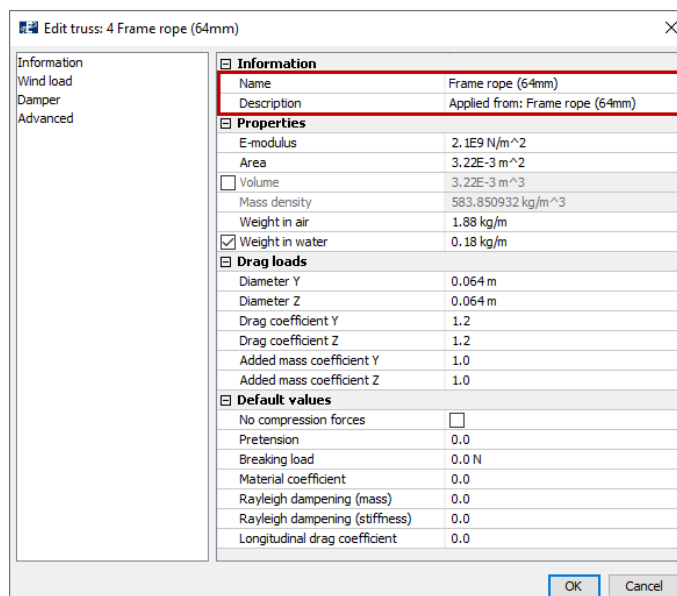
5 Properties of Mooring components

The mooring components should have similar properties as in CaseStudy03. Apply the library to assign the mooring components properties as described in the table in section 5.3.

Start with the frame rope, which should have the dimension 64mm. In the components window, **right click** Component 4 > **Library**. In the folder tree, navigate through **truss** > **ROPE**, in the middle column select Frame rope (64mm) as you generated in CaseStudy03. Then select **Apply** in the lower right corner of the window.



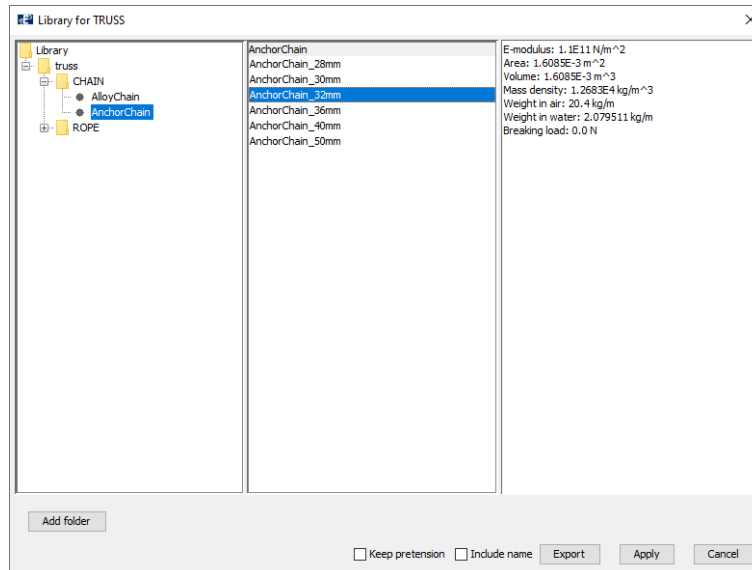
Change the name of the component by **double click** Component 4 and type the name Frame rope (64mm).



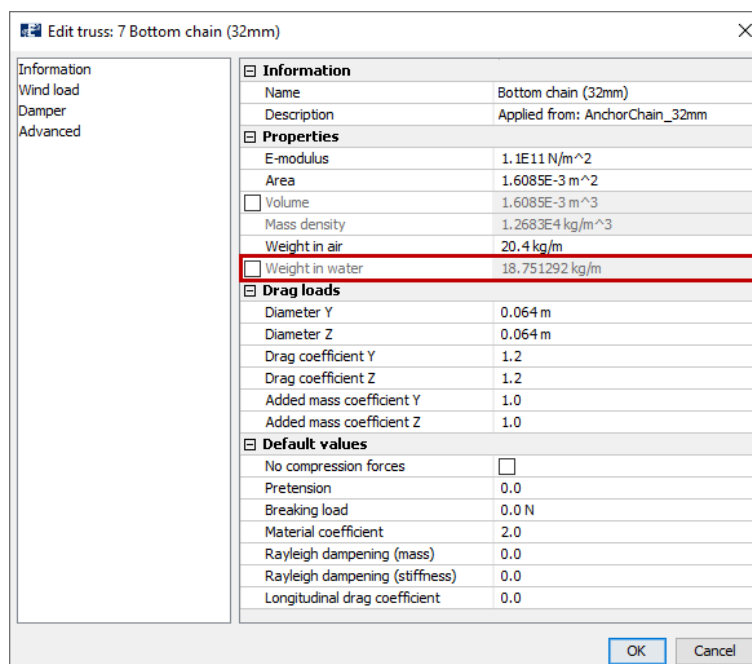
Select **OK**.

Repeat the use of library for the bridle rope (48mm) and mooring rope (64mm). Change the names; accordingly, Bridle rope (48mm) and Mooring rope (64mm).

The last component should be the bottom chain (32mm). **Right click** Component 7 > **Library**. In the Library for truss folder tree, select **truss** > **CHAIN** > **Anchor chain**, and in the middle column, select AnchorChain_32mm.

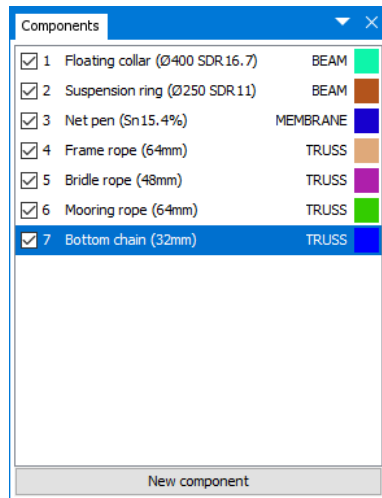


Then select **Apply** in the lower right corner of the window. **Double click** Component 7 in the Components window. Apply a suitable name to the component e.g., Bottom chain (32mm) and **de-select the checkbox for Weight in water**.





Select **OK**.

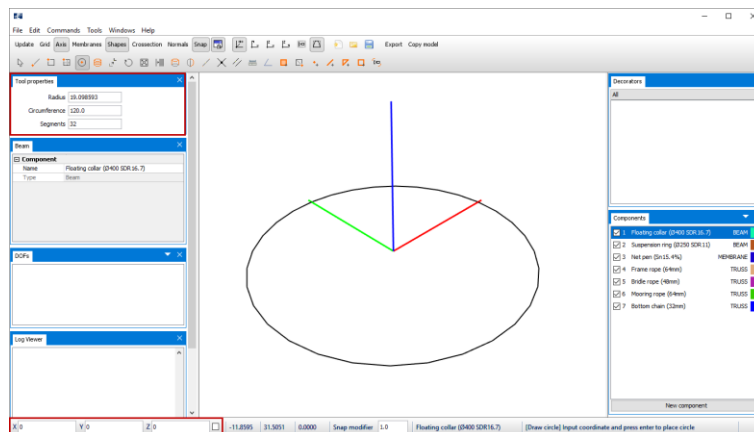
The Components window should resemble the figure below.



6 Draw Floating collar

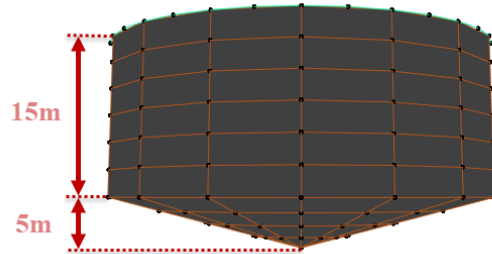
Since all the cages in the aquaculture facility is identical, you can model the first cage and then duplicate the remaining cages. When you start modeling, it is convenient to draw based on the origin of the global coordinate system. From the Toolbar menu, activate the **Snap** function .

The floating collar is circular and situated in the mean water line. In the Components window, **left click** Floating collar (Ø400 SDR16.7). Select the intent **Draw circle** . In the Tool properties window type **Circumference= 120** and **Segments= 32**. This will generate a circle with a circumference of 120 meters and 32 elements. Place the circle in **X= 0 Y= 0 Z= 0**, by **SHIFT + left click** in the 3D view. Alternatively, type the coordinates in the fields in the lower most toolbar.

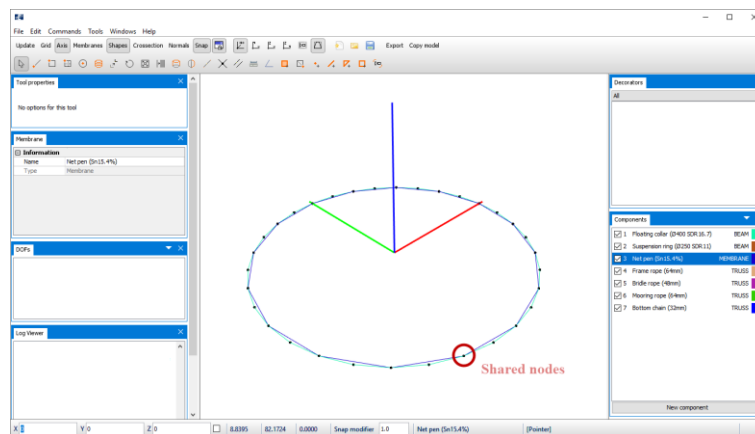


Press **ENTER**.


7 Draw Net pen The net pen is attached to the floating collar by shared nodes. Shared nodes are nodes that coincide between two components. Dimensions of the net pen is indicated in the figure below.

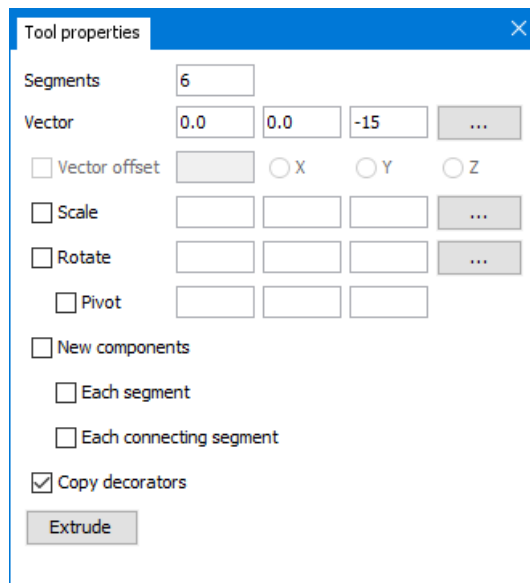


In the Components window, activate Net pen (Sn15.4%). Select the intent **Draw circle**, and in the Tool properties window type **Circumference= 120** and **Segments= 16**. Place the circle in **X= 0 Y= 0 Z= 0**. Press **ENTER**.



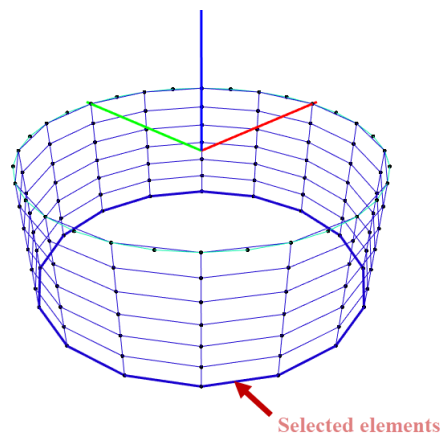
The sides and bottom of the net pen should be extruded from the net pen circle. From the Components window, select the net pen circle by **right click** Net pen (Sn15.4%) > **Operations** > **Select**. The net pen elements in the 3D window should be highlighted.

From the Toolbar menu, select the intent **Extrude** . In the Tool properties window, type **Segments= 6** and **Vector= -15** in Z-direction. From this, the net pen sides will be generated with 6 elements in the vertical direction 15 meters downwards.

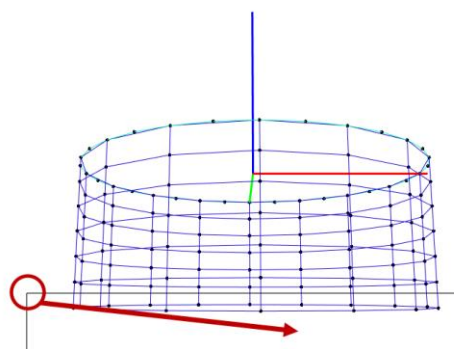


Select **Extrude**.


To generate the net pen bottom, select the lower most horizontal elements of the net pen. Apply multi-selection by **CTRL + left click** on elements in the 3D view.

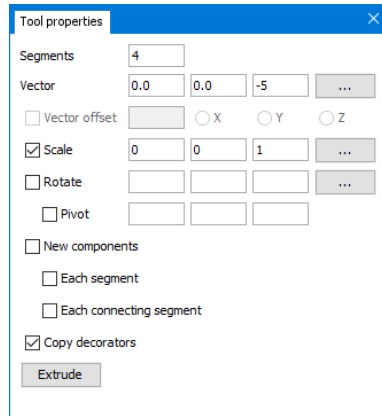


Tip! Orient your model in XZ-plane. Select the elements by click, hold, and drag the mouse pointer from left to right to select the elements.

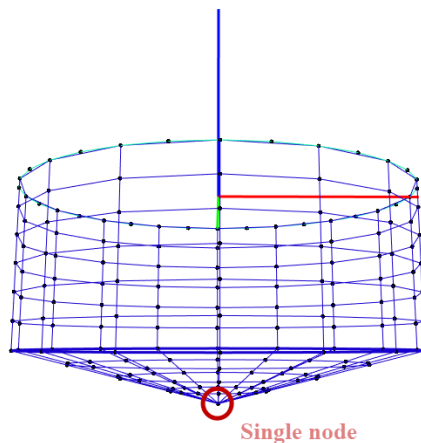


Left click, hold, and drag mouse pointer

Select the intent **Extrude** . The bottom has a depth of 5 meters and should be scaled in X- and Y-direction so that the tip consists of a single node.

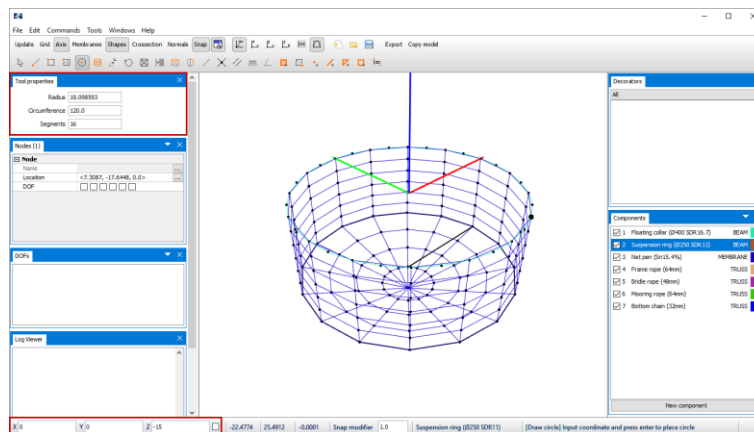


Select **Extrude**.



8 Draw Suspension ring

From the Components window, select Suspension ring (Ø250 SDR11). Select the intent Draw circle and type **Circumference= 120** and **Segments= 16**. Apply the field for coordinates in the lower left corner of AquaEdit and type **X= 0 Y= 0 Z= -15**.



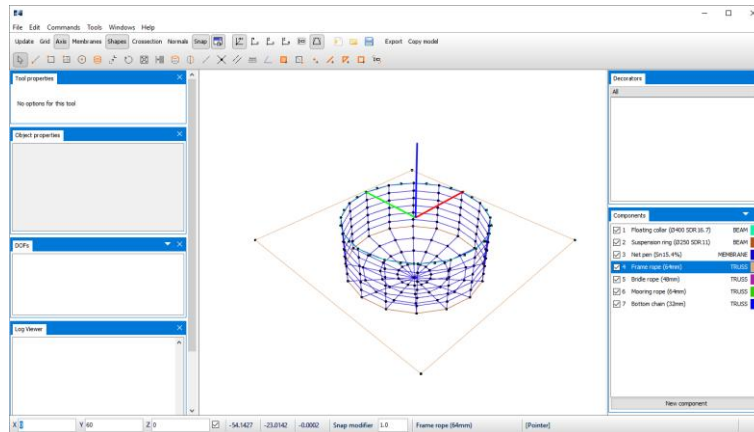
Press **ENTER**.

9 Save the model

Remember to save your model regularly, so that data is not lost. We have chosen to save the model at Documents\AquaSim\Analyses\ and generated the folder CaseStudy04.

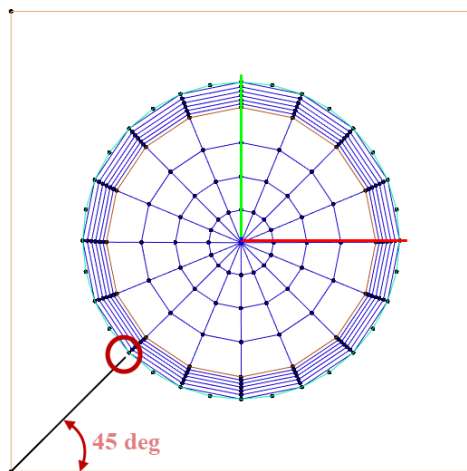
10 Draw Frame rope

The frame should be modelled as 60m x 60m at depth of 7m. Activate the component Frame rope (64mm) from the Components window. Apply the intent **Draw line**, or **Draw rectangle** and follow the procedure as described in step number 8 in CaseStudy03.

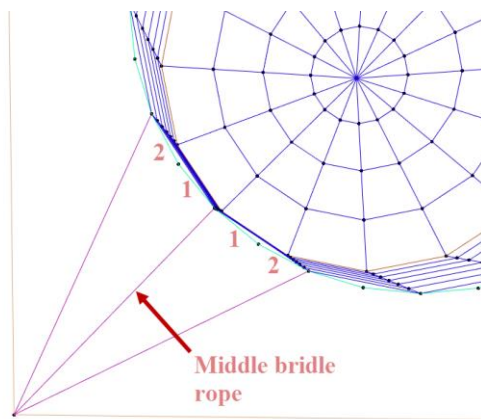


11 Draw Bridle rope


The bridle should be of type 3-part. Active the component Bridle rope (48mm) from the Components window. Select the intent **Draw line**. Draw a line from the lower left corner of the frame to the node on the floating collar that result in a line 45 degrees (in XY-plane) with respect to the horizontal frame.

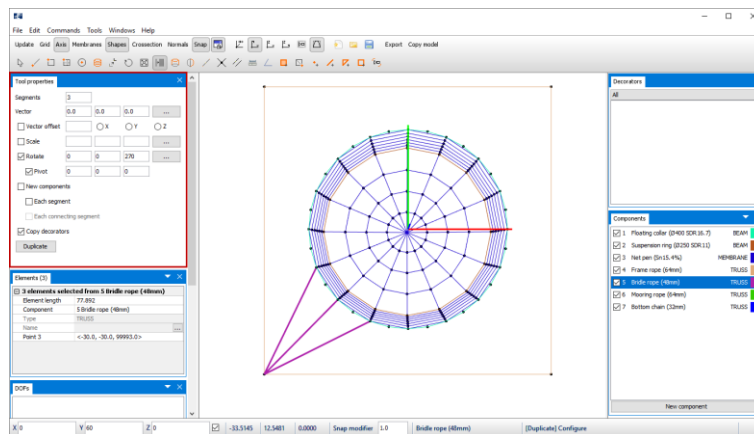


Press **ESC** once to escape the continuous line modelling. The two other bridle ropes should be attached to the floating collar counting two elements on each side of the middle bridle rope.

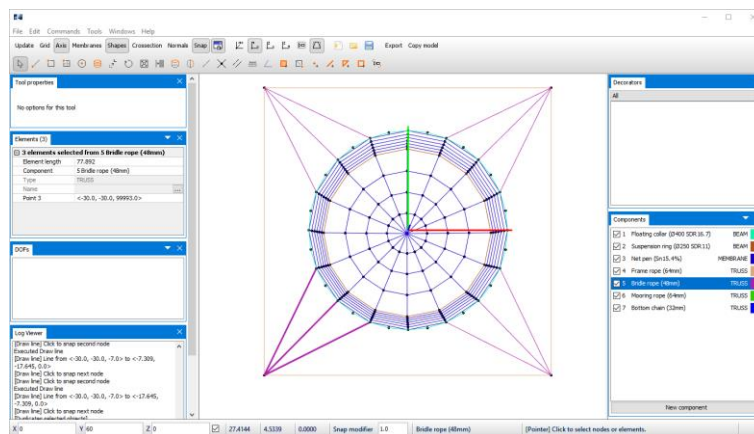


Duplicate the 3-part bridle rope to the three remaining corners of the frame rope.

Select the three bridle ropes in the 3D view. Select the intent **Duplicate**  and make the selections as shown in the figure below.



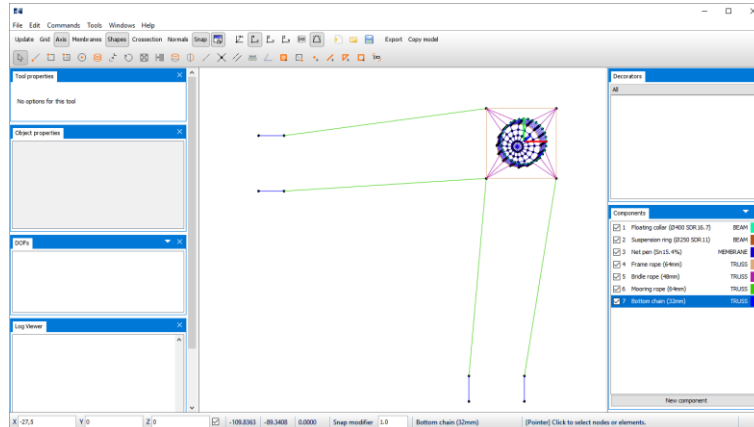
Select **Duplicate**.



12 Draw Mooring rope and Bottom chain

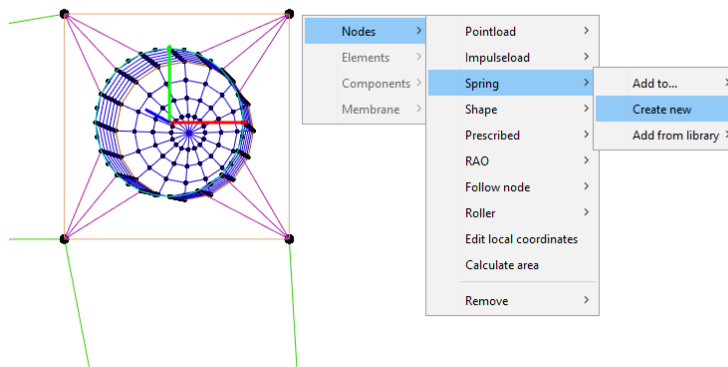
The mooring ropes should have a length of 220m at -100m depth. The bottom chain should be 27.5m of length.

Follow the procedure as described in step number 11 from CaseStudy03. The model should resemble the figure below when the mooring rope and bottom chain is modelled.

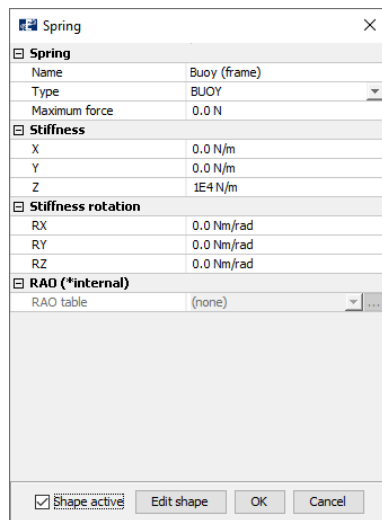


13 Buoys

Each corner of the frame should be assigned buoys. Select the corner nodes of the frame by **CTRL + left click** on the nodes. In the 3D view, **right click > Nodes > Spring > Create new**.



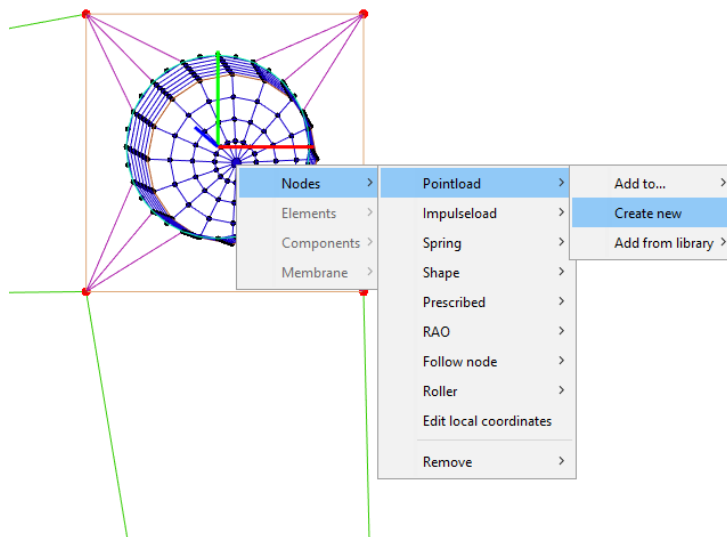
In the Edit spring window, dedicate a suitable name e.g., Buoy (frame). In the row **Type**, select **BUOY** from the drop-down menu. Provide the buoy a vertical stiffness of $1.0E+04$ N/m.



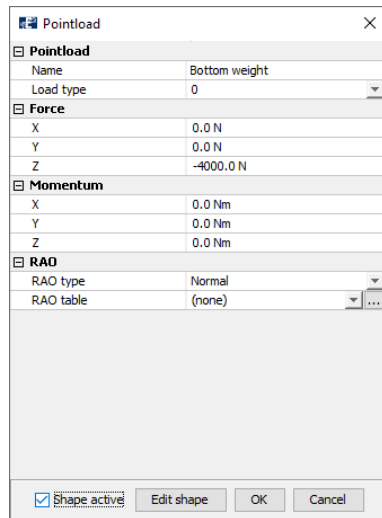
Activate the buoy shape by selecting the checkbox and then select **OK**. The corner nodes of the frame will change color from black to red, indication a node decorator is added.

14 Bottom weight of net pen

In addition to the suspension ring, you should add a bottom weight to the tip of the net pen. Select the node in the bottom tip of the net pen, **right click** > **Nodes** > **Pointload** > **Create new**.



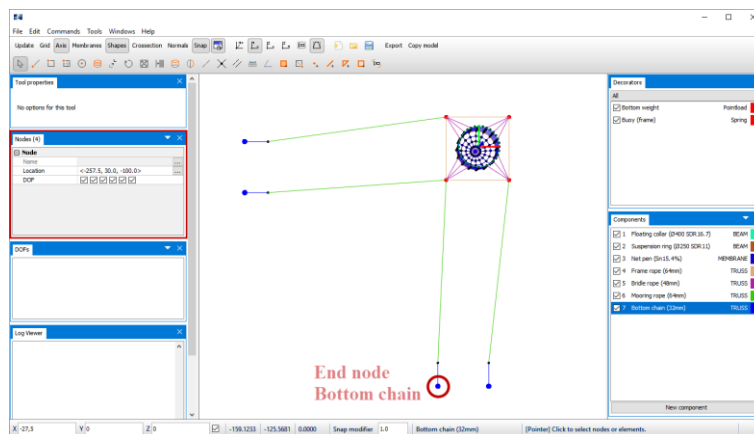
Dedicate the node a vertical load in negative Z-direction of -4000N, similar as to the figure below.



Activate the shape by selecting the checkbox and then select **OK**.

15 Boundary conditions


At the end node of the chain, a mooring bolt or anchor is attached. To resemble this, the end nodes are restrained in all Degrees of freedom. Select all four end nodes of the bottom chain and select all the checkboxes in the row **DOF** of the Nodes window.

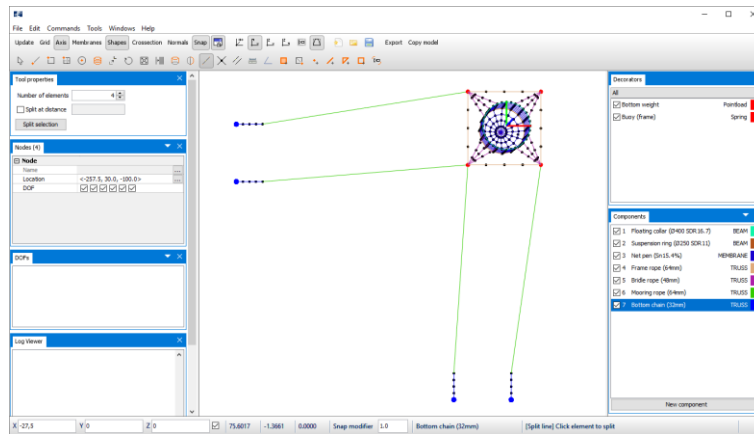


16 Discretization

The mooring components should be discretized with the number of elements according to the table below.

Component name	Number of elements
Frame rope (64mm)	4
Bridle rope (48mm)	4
Bottom chain (32mm)	4

Select the intent **Split line**  and discretize the component by left click on the components in the 3D view. After discretization, the model should resemble the figure below.

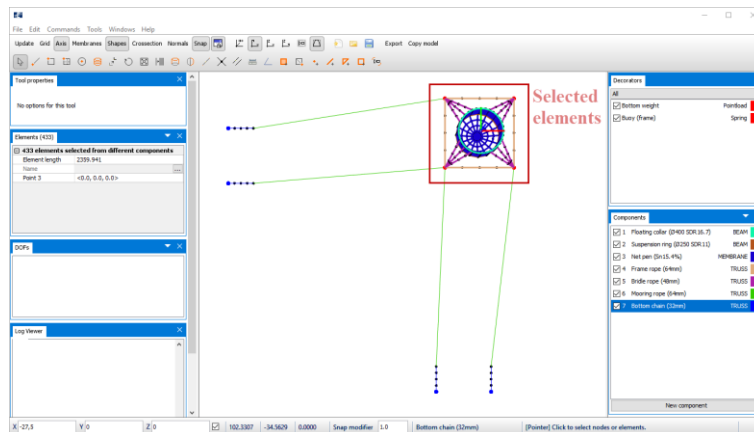


You should wait with discretization of Mooring rope (64mm) until terrain has been loaded and the chain nodes are snapped to terrain.

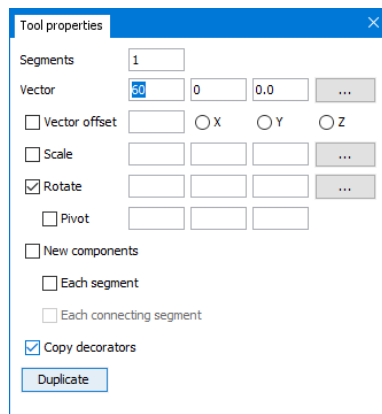
17 Duplicate cages

The model is to be extended to a 2 x 2 cage facility with identical components. You are to duplicate the cage in both X- and Y-direction.

First, you need to select the elements for duplication in the 3D view. Deselect the mooring ropes if necessary.

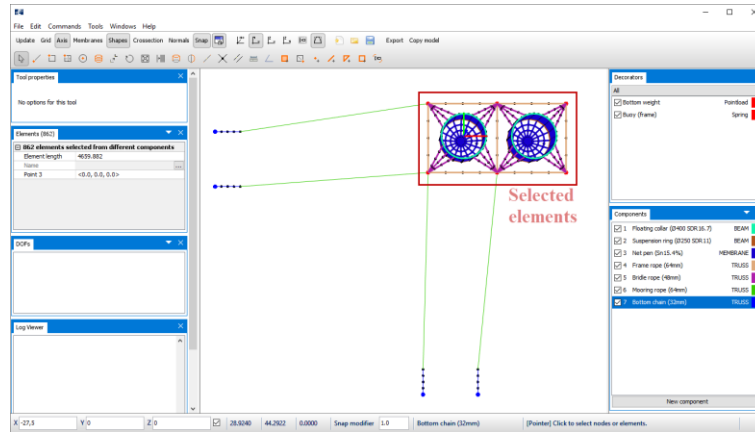


From the Toolbar menu, select **Duplicate** and make selections as described in the figure below.

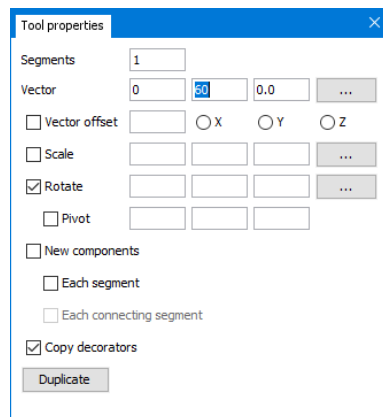


Select **Duplicate**.

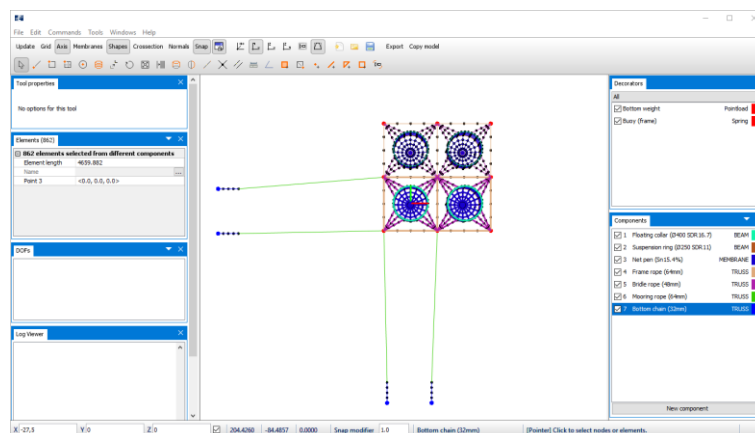
Then select the two cages, except from the mooring ropes.



From the Toolbar menu, select **Duplicate** and make selections as described in the figure below.

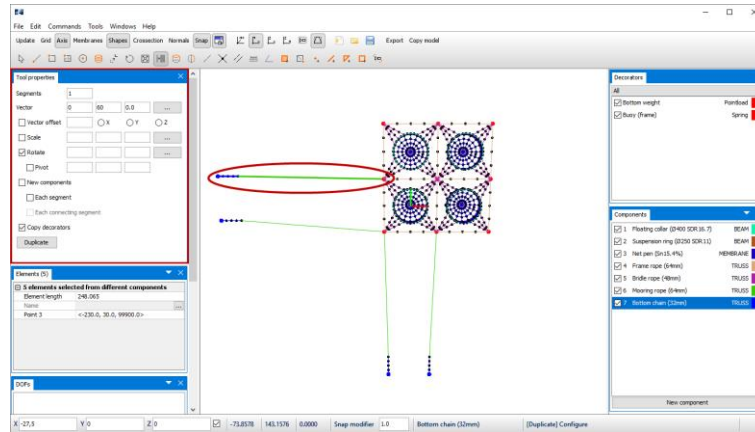


Select **Duplicate**.



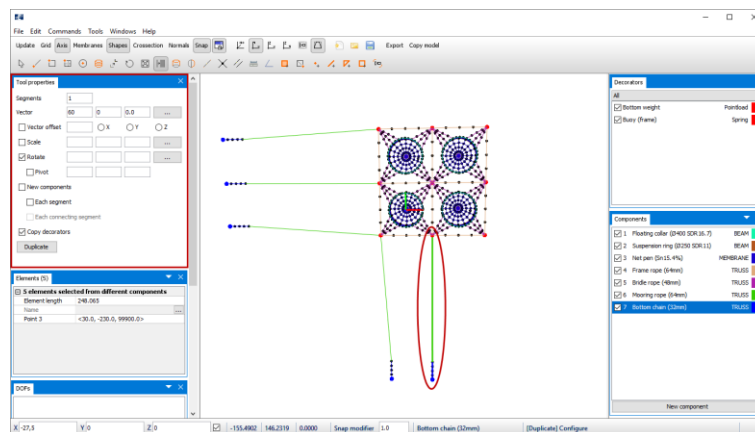
18 Duplicate Mooring rope and Bottom chain

The remaining mooring rope and bottom chain should be added to the facility. From the 3D view, select the mooring rope and associated chain on the left side of the cages. Duplicate these elements by 1 segment and 60m in Y-direction.



In the Tool properties window, select **Duplicate**.


Repeat this for the mooring rope and chain in the lower right corner. Duplicate these elements by 1 segment and 60m in X-direction.

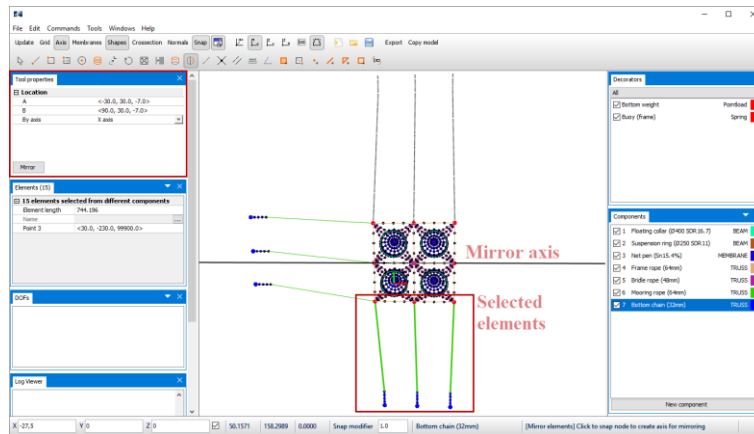


In the Tool properties window, select **Duplicate**.

19 Mirror Mooring rope and Bottom chain

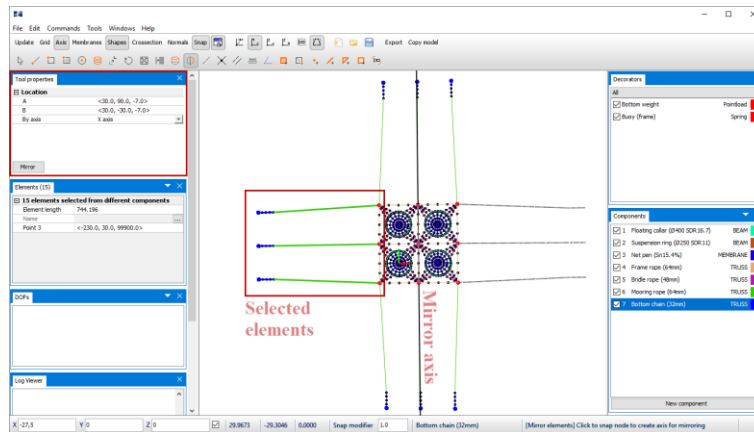
You should model the remaining mooring ropes and bottom chain by applying the mirror intent.

From the 3D view, select the three lower most pairs of mooring rope and chain and select **Mirror** . Choose the mirror axis by node selection in the 3D view.



In the Tool properties window select **Mirror**.

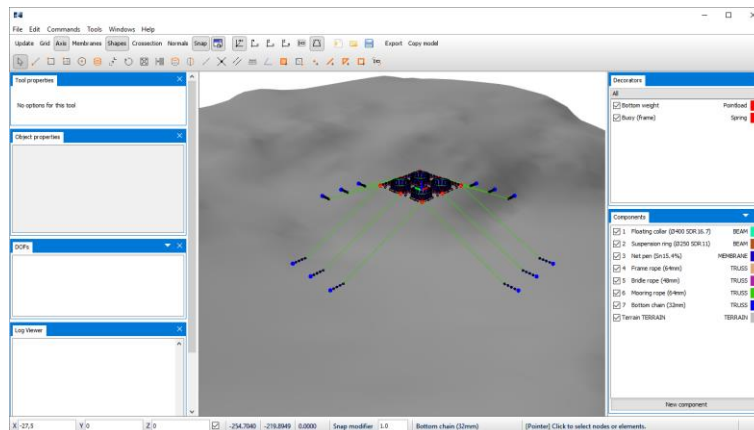
Select the three left most pairs of mooring rope and chain and select **Mirror**. Choose the mirror axis by node selection in the 3D view.



In the Tool properties window select **Mirror**.

20 Import terrain

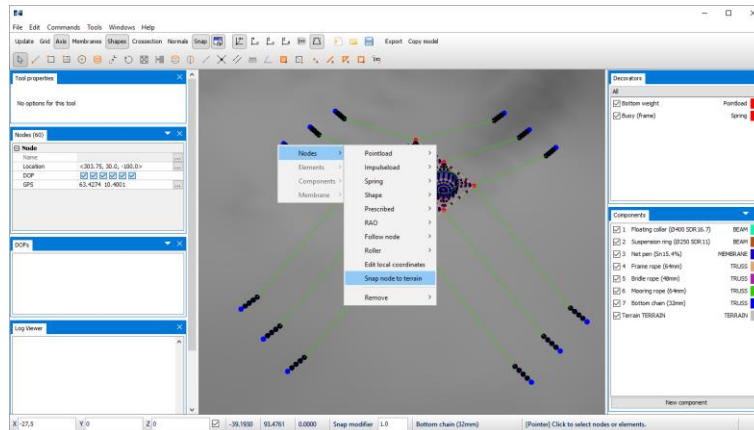
You should import the same terrain as applied in CaseStudy03. Follow the same procedure to import terrain as described in step number 18 in CaseStudy03. The terrain .obj-file is found in Documents\AquaSim\Demos\Casestudy03.



21 Snap nodes to terrain

The bottom chain has been modelled at a depth of -100m, you should snap the chain nodes to the terrain. From the 3D view, select all the nodes associated with the component Bottom chain (32mm).

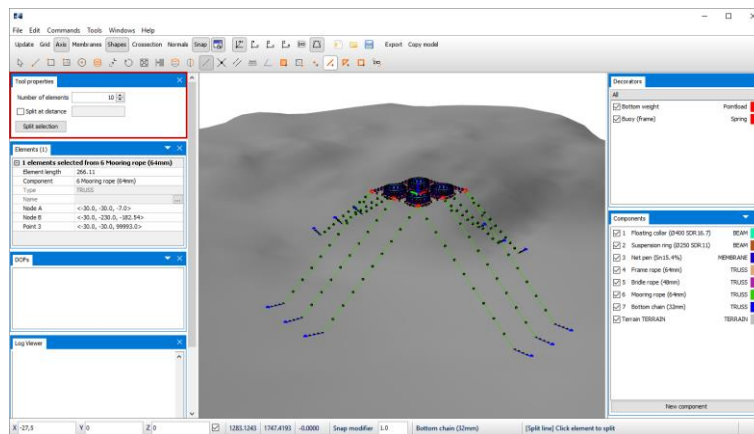
For multi-selection of nodes, **left click** on one node and hold **CTRL**, then select the remaining nodes. **Right click** > **Nodes** > **Snap node to terrain**.



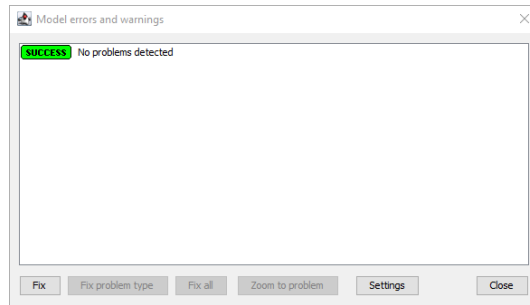
The nodes connected with Bottom chain (32mm) is now moved from modelled position down to the terrain.

22 Discretization of Mooring rope

Split the elements associated with Mooring rope (64mm) in 10 elements. Select the intent **Split line**, and in the Tool properties window type **Number of segments= 10**.




23 Verify model Verify the model from **Commands > Verify model**.



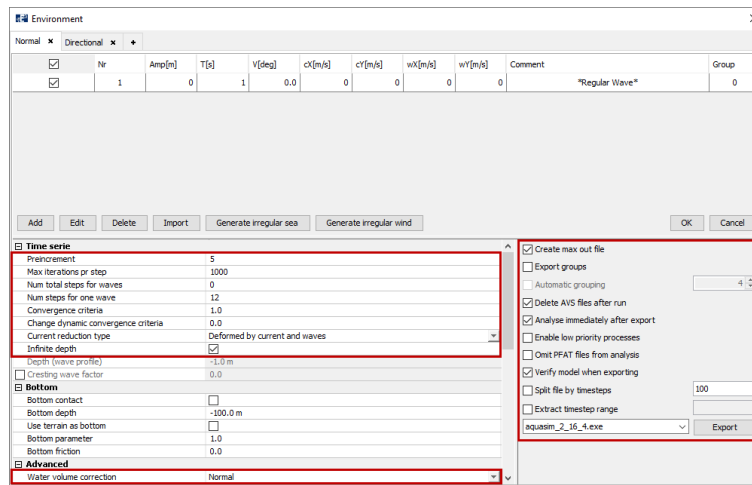
Your model is now successfully completed.

5.5 Calculations static analysis

24 Enter Environmental window

From the Toolbar menu in AquaEdit, select **Export** . You should run a static analysis and control key parameters for equilibrium in the model.

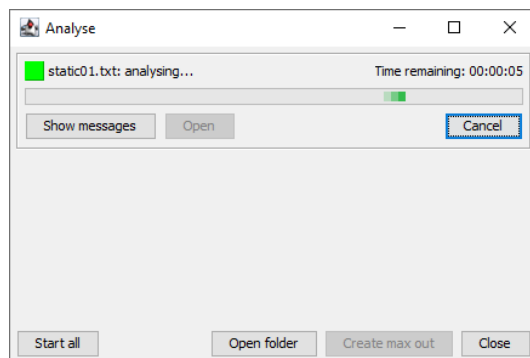
Select the tab **Normal**. Prepare the static analysis by making selections as described in the figure below. We have highlighted the selection for Water volume correction to be **Normal**. This is due to the type Hydrodynamic, for the floating collar, is not supported with the adjustments provided for the With slamming-option.



Select **Export**. Navigate to a folder where you want to save the analysis. We have chosen Documents\AquaSim\Analyses\CaseStudy03\Static. Type an analysis name e.g., static. Select **Export** from the folder navigation window.

25 Run static analysis

The **Analyse** window opens. Start the analysis by selecting **Start** or **Start all**.



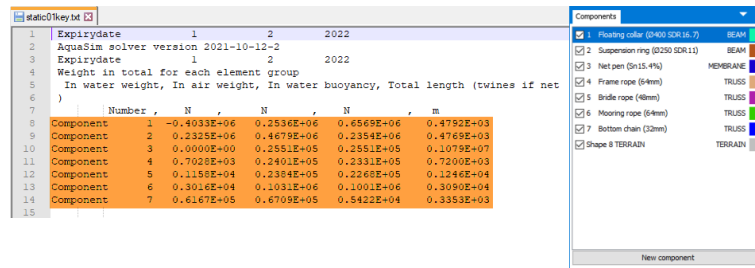
The analysis is successfully completed when the square in the upper left corner of the Analyse window gets green with the status **finished**.

5.6 Postprocessing static analysis

26 Check of buoyancy

You should now check and compare key parameters from the static analysis. Go to your analysis folder Documents\AquaSim\Analyses\CaseStudy04\Static and load the file static01key.txt.

Navigate to the section Weight in total for each element group and copy the weight statistics to the calculation sheet as established in CaseStudy03.



The In water buoyancy of the floating collar and In water weight of mooring components should be compared to the figures achieved in CaseStudy03.

The resulting buoyancy of the floating collar in CaseStudy04 is approximate half of what achieved in CaseStudy03. This is due to how the volume is calculated. For elements of type Morison submerged (CaseStudy03) volume of the whole floating pipe is included, upon static equilibrium this is adjusted. For elements of type Hydrodynamic (CaseStudy04) the submerged volume as defined in the input is included. Remember, for type Hydrodynamic AquaSim calculates hydrostatic properties based on the values defined in AquaEdit.

Key data after static equilibrium, static analysis							
		Weight below water line	Weight above water line	Buoyancy	Total length	Component name	
CaseStudy03, detailed model	Component 1	-4.79E+05	1.27E+05	6.06E+05	4.80E+02	Inner floating pipe (Ø400 SDR 16.7)	
	Component 2	-5.00E+05	1.33E+05	6.33E+05	5.01E+02	Outer floating pipe (Ø400 SDR 16.7)	
	Component 3	6.03E+04	6.03E+04	0.00E+00	1.61E+02	Bracket	
	Component 4	1.43E+04	1.43E+04	0.00E+00	2.30E+02	Railing post (Ø160 SDR11)	
	Component 5	2.28E+04	2.28E+04	0.00E+00	4.80E+02	Railing (Ø140 SDR11)	
	Component 6	2.40E+05	4.83E+05	2.43E+05	4.92E+02	Suspension ring (Ø250 SDR11)	
	Component 7	0.00E+00	2.67E+04	2.67E+04	1.13E+02	Net (Sn15.4%)	
	Component 8	1.38E+02	2.59E+03	2.46E+03	7.78E+02	Net collector rope (20mm)	
	Component 9	1.13E+02	2.01E+03	1.90E+03	1.67E+02	Reinforcement rope net (12mm)	
	Component 10	1.27E+03	2.46E+04	2.33E+04	7.20E+02	Frame rope (64mm)	
	Component 11	1.13E+03	2.32E+04	2.21E+04	1.21E+03	Bridle rope (48mm)	
	Component 12	5.46E+03	1.06E+05	1.00E+05	3.09E+03	Mooring rope (64mm)	
	Component 13	6.17E+04	6.71E+04	5.43E+03	3.35E+02	Bottom chain (32mm)	
Buoyancy floating collar				1238900 N			
Weight mooring components				69526 N			
CaseStudy04, simplified model	Component 1	-4.03E+05	2.54E+05	6.57E+05	4.79E+02	Floating collar (Ø400 SDR16.7)	
	Component 2	2.33E+05	4.68E+05	2.35E+05	4.77E+02	Suspension ring (Ø250 SDR11)	
	Component 3	0.00E+00	2.59E+04	2.59E+04	1.09E+02	Net pen (Sn15.4%)	
	Component 4	1.27E+03	2.46E+04	2.33E+04	7.20E+02	Frame rope (64mm)	
	Component 5	1.16E+03	2.38E+04	2.27E+04	1.25E+03	Bridle rope (48mm)	
	Component 6	5.46E+03	1.06E+05	1.00E+05	3.09E+03	Mooring rope (64mm)	
	Component 7	6.17E+04	6.71E+04	5.42E+03	3.35E+02	Bottom chain (32mm)	
Buoyancy floating collar				656900 N			
Weight mooring components				69556 N			

The in water weight of mooring components compare well between CaseStudy04 and CaseStudy03. This will affect the pretension of the mooring components.

27 Check of key parameters

In your static01key.txt-file, navigate down to the section saying Total, all elements. You are to compare the differences in how Diffrac and Weight elements (N) / Mass elements (kg) have changed between CaseStudy04 and CaseStudy03.

static01key.txt		CaseStudy04
57	Total, all elements	
58	Weight in air [N]	968311.46915
59	Weight in water[N]	-101288.03285
60	Buoyancy [N]	1069599.50200
61	Diffrac = Weight/Bouancy of all elements	0.90530
62	Total mass [kg]	92608.22204
63	buoyancy non Hydrodyn beams and bars [kg]	39436.87801
64	Weight elements (N) / Mass elements (kg)	10.45600
65	Total mass of beam elements [kg]	73539.57870
66		
67	Properties to nodeloads	
68	Upwards force of node loads [N]	0.00000
69	Downwards force of node loads [N]	16000.00000
70	Mass to nodeloads(kg)	1630.98872
71	(Exclusive of mass-springs)	
72		
static01key.txt		CaseStudy03
63	Total, all elements	
64	Weight in air [N]	1091155.76371
65	Weight in water[N]	-572500.66587
66	The authomatic in and out of water option is used	
67	Weight in air is used for part over water line	
68	Weight in water is used for part under water line	
69	Buoyancy [N]	1663656.42959
70	Diffrac = Weight/Bouancy of all elements	0.65588
71	Total mass [kg]	104912.04986
72	buoyancy non Hydrodyn beams and bars [kg]	166869.78710
73	Weight elements (N) / Mass elements (kg)	10.40067
74	Total mass of beam elements [kg]	85574.20503
75		
76	Properties to nodeloads	
77	Upwards force of node loads [N]	0.00000
78	Downwards force of node loads [N]	16000.00000
79	Mass to nodeloads(kg)	1630.98872
80	(Exclusive of mass-springs)	

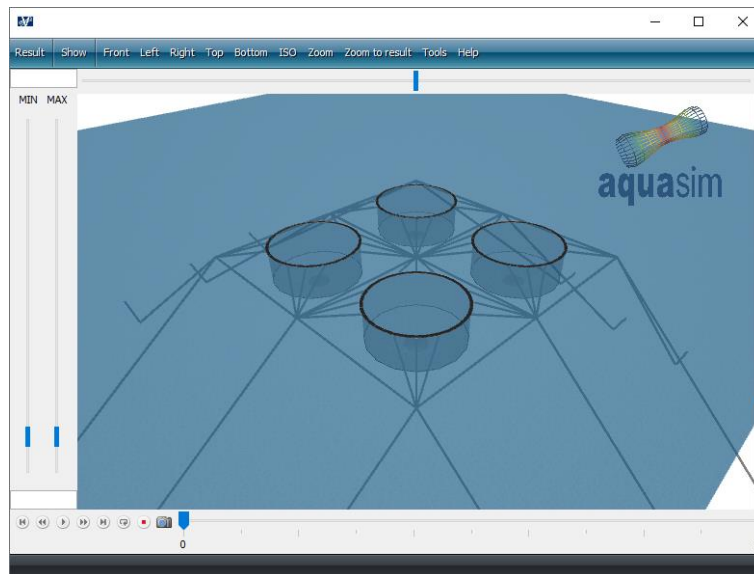
The Diffrac has increased in CaseStudy04 compared to CaseStudy03. This is due the buoyancy in the simplified model has reduced, as described in step number 26.

The Weight elements (N) / Mass elements (kg) is more or less the same. This is because the weight and mass of the two models compare well.

Exit the static01key.txt-file.

28 Open results in AquaView

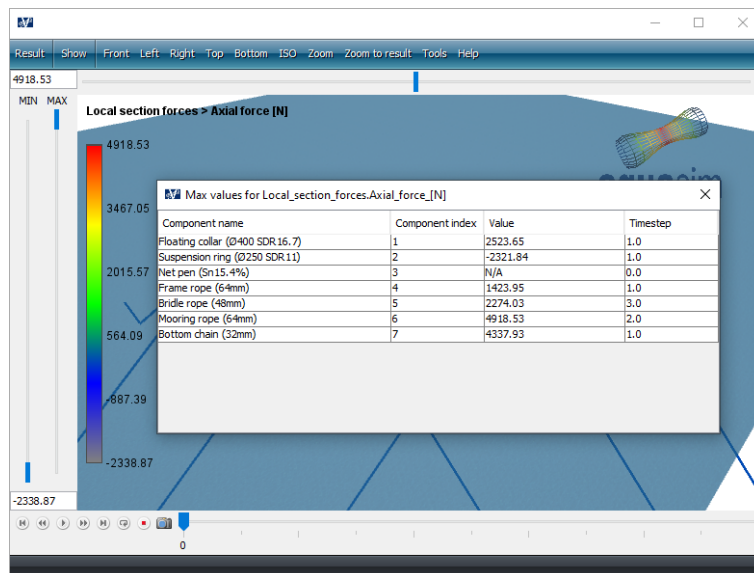
Load static01.avz in AquaView. Press **Play**  from the playback menu and observe the response of your model.



Zoom in on structural details and see how the model displaces.

29 Pretension of mooring components

As a result of the mooring component's weight in water, they will achieve a pretension. The pretension is found by plotting axial forces. From the Toolbar menu, select **Result > Local section forces > Axial force [N]**.



Copy the rows for component number 4 to 6 and paste it in the calculation sheet.
Compare the figures between CaseStudy04 and CaseStudy03.

	J	K	L	M	N	O
1	Pretension in mooring components, static analysis					
5	CaseStudy03, detailed model					
6						
7						
8						
9		Component name	Component no.	Pretension [N]	Step no.	
10		Frame rope (64mm)	10	1212.29	2	
11		Bridle rope (48mm)	11	2691.03	1	
12		Mooring rope (64mm)	12	4916.67	2	
13						
14						
15						
16						
19						
20						
21	CaseStudy04, simplified model					
23						
24						
25						
26		Component name	Component no.	Pretension [N]	Step no.	
27		Frame rope (64mm)	4	1423.95	1	
28		Bridle rope (48mm)	5	2274.03	3	
29		Mooring rope (64mm)	6	4918.53	2	
30						
31						
32						
33						
34						
36						


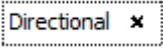
As seen, the figures compare well between the two case studies.

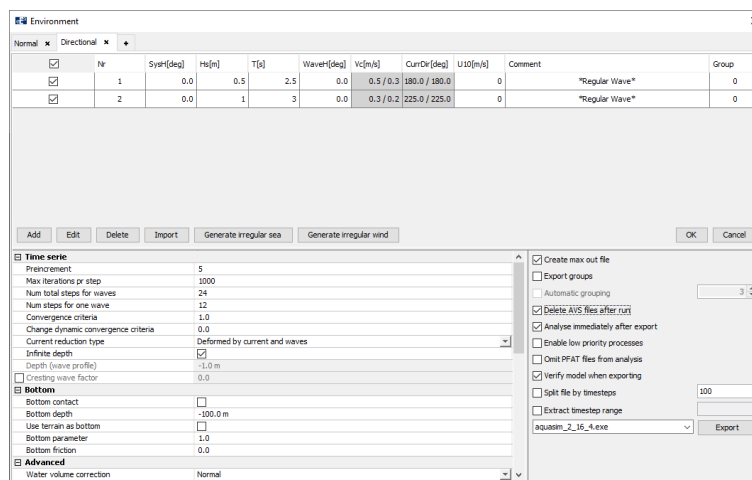
Static considerations are now finished. Exit AquaView and the Analyse window from AquaEdit.

5.7 Calculation dynamic analysis

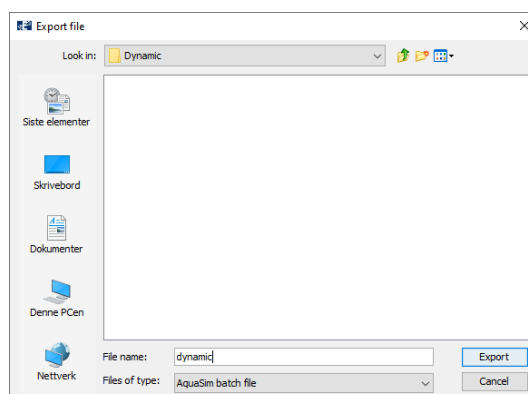
30 Load conditions and Export parameters

Go back to your model in AquaEdit. You should investigate the response and forces in mooring components for the same two load conditions as in CaseStudy03.

Enter the Environmental window be selecting **Export** . Select the tab **Directional** . Input the same load conditions by follow the steps as described in step number 31 from chapter 4.7. Apply time series parameters and Export options as described in the figure below. The Water volume correction should be **Normal**.

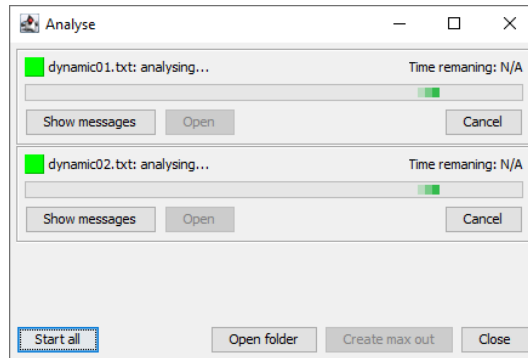


Select **Export**. Navigate to an appropriate place to save the analysis, we have chosen Documents\AquaSim\Analyses\CaseStudy04 and created the folder Dynamic. The analysis is called dynamic.



Select **Export**.

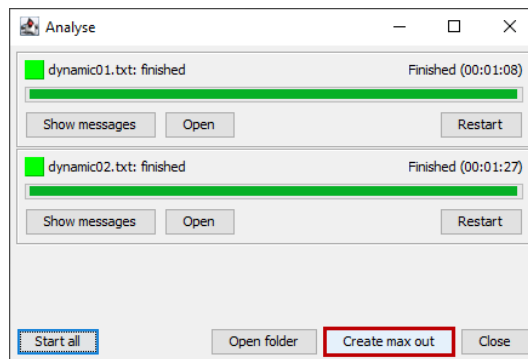
In the Analyse window, select **Start all**.



The dynamic analysis has started. Depending on your computer, this analysis should be finished within approximate 1-2 minutes. Follow the status and development of the analysis by selecting

31 Create max out file

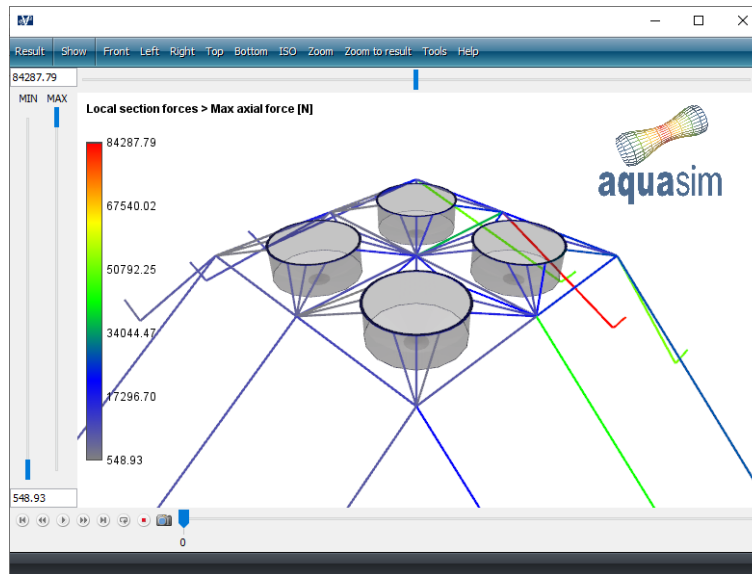
Upon completion of the analysis, select **Create max out** from the Analyse window.



5.8 Postprocessing dynamic analysis

32 Open max out results In the Analyse window, select **Open folder**. Enter the folder maxout and open dynamic_max_out.avz by **double click**.

From the Toolbar menu in AquaView, select **Result > Local section forces > Max axial force [N]**.



If problems have occurred in your model or analysis, you can retrieve similar results from folder Documents\AquaSim\Demos\CaseStudy04\Dynamic.

Press **j** on your keyboard to enter the window for maximum values of each component group.

Component name	Component index	Value	File index
Floating collar (Ø400 SDR16.7)	1	30242.64	dynamic02PFAT....
Suspension ring (Ø250 SDR11)	2	7525.23	dynamic02PFAT....
Net pen (Sn15.4%)	3	N/A	N/A
Frame rope (64mm)	4	36174.14	dynamic01PFAT....
Bridle rope (48mm)	5	27380.02	dynamic02PFAT....
Mooring rope (64mm)	6	84287.79	dynamic01PFAT....
Bottom chain (32mm)	7	83664.83	dynamic01PFAT....

Copy the rows for component group number 4 to 7 and paste it in the calculation sheet you have established.

33 Evaluation of results

You should compare the maximum achieved axial forces from the two load conditions with CaseStudy03. In the figure below, the axial forces in the mooring components are reported for CaseStudy03 and CaseStudy04.

	Q	R	S	T	U	V	W
1	Axial force in mooring components, dynamic analysis						
5	CaseStudy03, detailed model						
6							
7							
8							
9		Component name	Component no.	Axial force [N]	File index		
10		Frame rope (64mm)	10	33650.51	dynamic01PFAT.avz		
11	Bridle rope (48mm)	11	32381.21	dynamic02PFAT.avz			
12	Mooring rope (64mm)	12	85209.30	dynamic01PFAT.avz			
13	Bottom chain (32mm)	13	84799.53	dynamic01PFAT.avz			
14							
15							
16							
17							
18							
19							
20							
21	CaseStudy04, simplified model						
22							
23							
24							
25							
26		Component name	Component no.	Axial force [N]	File index		
27	Frame rope (64mm)	4.00	36174.14	dynamic01PFAT.avz			
28	Bridle rope (48mm)	5.00	27380.02	dynamic02PFAT.avz			
29	Mooring rope (64mm)	6.00	84287.79	dynamic01PFAT.avz			
30	Bottom chain (32mm)	7.00	83664.83	dynamic01PFAT.avz			
31							
32							
33							
34							
35							
36							

By comparing, we see that the forces generally compare well, but some differences are observed. The reason for this is complex, and come from, among other things, differences in discretization of the model that will lead to a different response when exposed to dynamic loads.

An aquaculture facility is mainly exposed to drag forces, this is due to the net pen having a large area exposed to current and waves. These forces will propagate further into the mooring components. In this case study, the net pen was modelled with fewer elements compared to CaseStudy03. Leading to a slightly different deformation of the net pen.

Dynamic considerations are finished. Exit AquaView, the Analyse window and close AquaEdit.

5.9 Summary

This case study was intended to provide understanding of the consequences of the level of details in your model. You have modelled an aquaculture facility identical to the one established in CaseStudy03, but in a more simplified manner.

To ensure that hydrostatic properties are consistent, key parameters of both models are compared after completed static analysis. By comparing axial forces achieved in CaseStudy04 with CaseStudy03, we see that they compare well. At the same time, you have reduced the complexity in the model and reduced computational time significantly. From 10-15 minutes in CaseStudy03 to 1-2 minutes in CaseStudy04 for dynamic analysis.

From this we can conclude that the level of detail of your model must be assessed based on the purpose of your analysis.

6 CaseStudy 05 – Simplified aquaculture cage and lice skirt

6.1 Learning objectives

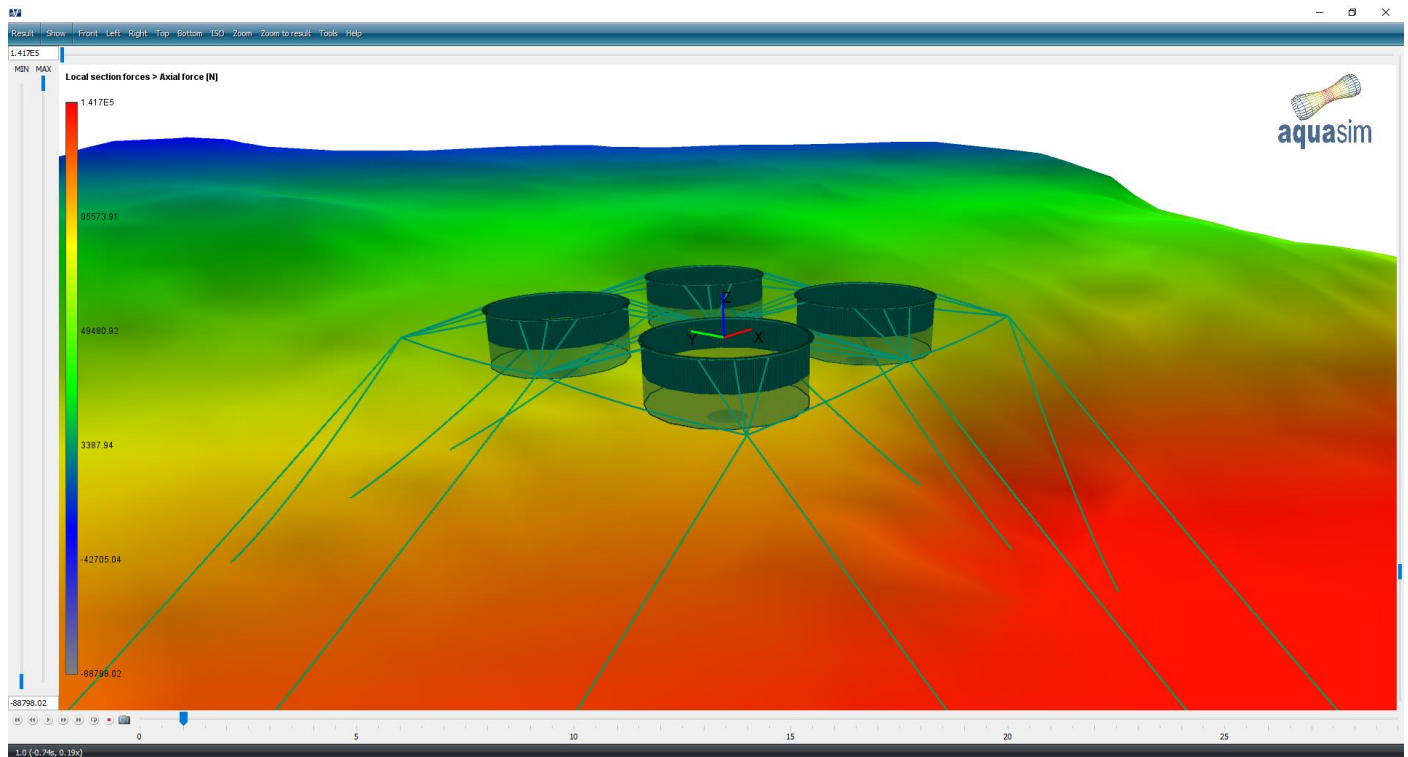
Upon completion of this case study, you will be able to:

- Apply properties of an impermeable tarpaulin (lice skirt)
- Execute non-linear dynamic analysis
- Assess the impact lice skirt have on loads in mooring components

6.2 Introduction

In this case study you are to complete a model of an aquaculture facility equipped with impermeable tarpaulins. Impermeable tarpaulins are commonly applied to prevent and reduce lice infestation and are referred to as lice skirts. The name originates from how it is installed at the facility – it is attached to the floating collar and extends down along the outside of the net pen, as it was a skirt.

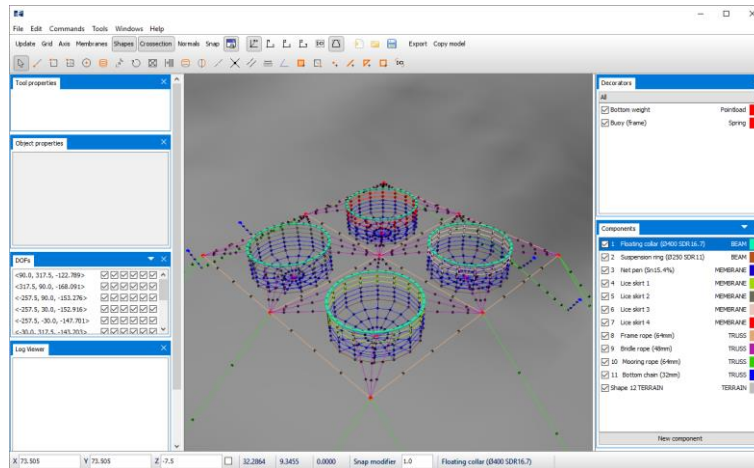
You are to conduct a dynamic analysis, applying the same environmental conditions as in CaseStudy04 and assess how forces in mooring component forces are affected by the presence of lice skirts by comparison.



6.3 Preprocessing

1 Open model

Open the prepared model CaseStudy05.amodel from Documents\AquaSim\Demos\CaseStudy05.

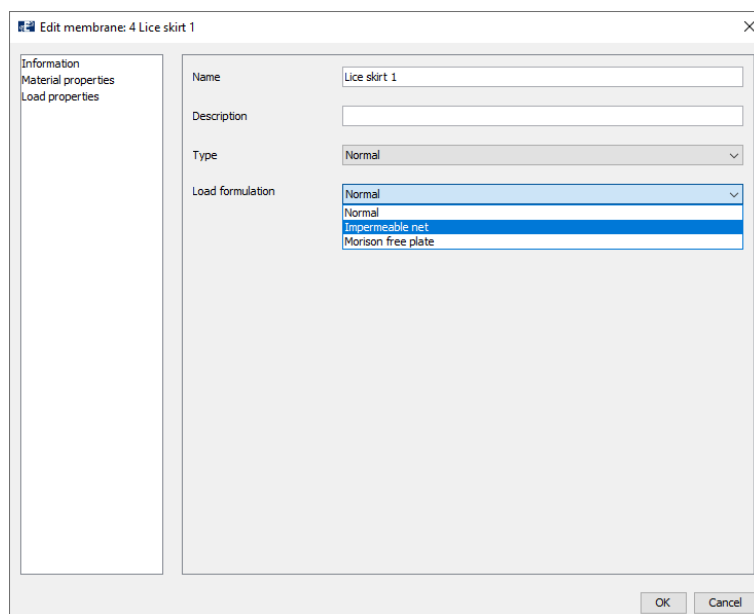


This model is identical to the one established in CaseStudy04, with floating collar, net pen, mooring components and terrain. The difference is that 4 lice skirts are included. The skirts extend 7.5m below the water surface.

2 Properties of Lice skirt

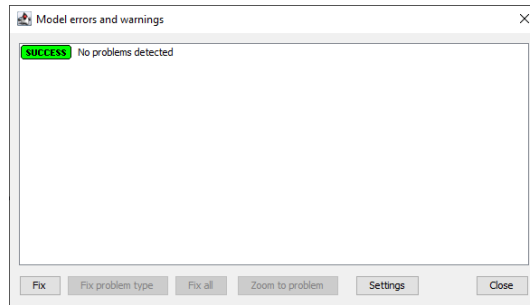
Lice skirts are impermeable and flexible tarpaulins, in oppose to net pens where water is allowed to flow through. You should change the load formulation of the associated lice skirts, so it resembles an impermeable tarpaulin.

In the Components window, **double click** Lice skirt 1. Select the drop-down menu for **Load formulation** and select Impermeable net.

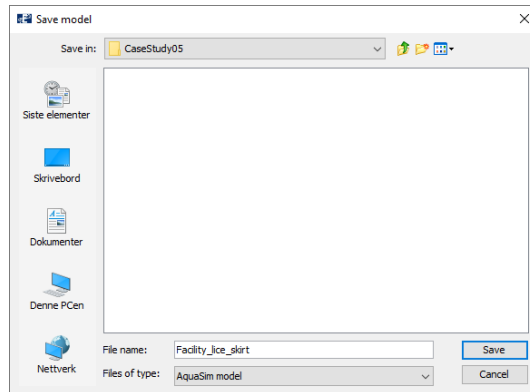


Repeat this procedure for Lice skirt 2, Lice skirt 3 and Lice skirt 4.

3 Verify model Verify the input of your model by selecting **Commands > Verify model1**.



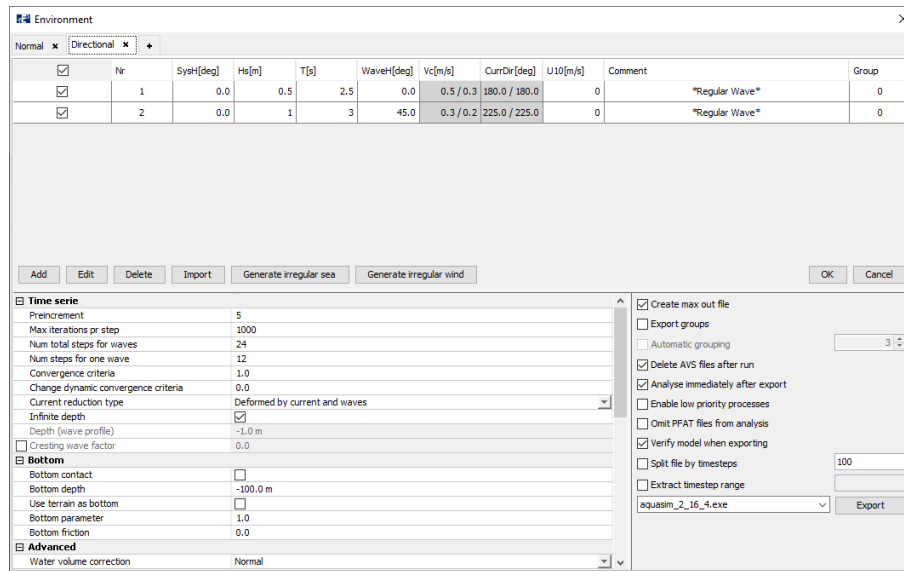
4 Save model Remember to save the model. Select **File > Save as....** Navigate to a folder where you want to save the model. We have chosen Documents\AquaSim\Analyses\CaseStudy05. Give the model a name e.g., Facility_lice_skirt.amodel.



6.4 Calculations dynamic analysis

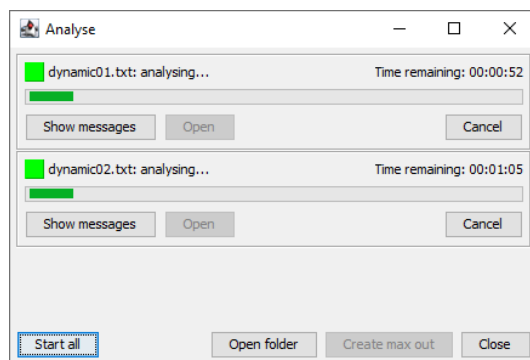
5 Load conditions and export parameters

From the Toolbar menu in AquaEdit, select **Export**. Necessary analysis parameters are prepared, check that these are consistent with the figure below.



Select **Export** and choose an appropriate place to save the analysis, we have chosen Documents\AquaSim\Analyses\CaseStudy05 and created the folder Dynamic. Give the analysis a name and select **Export**.

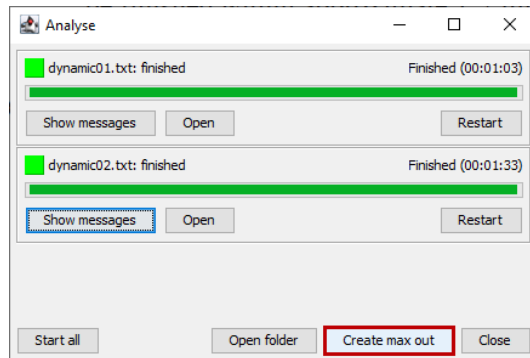
In the Analyse window, select **Start all**.



Depending on your computer, this analysis should be finished within approximate 3 minutes.

6 Create max out file

Upon completion of the analysis, select **Create max out** from the Analyse window.



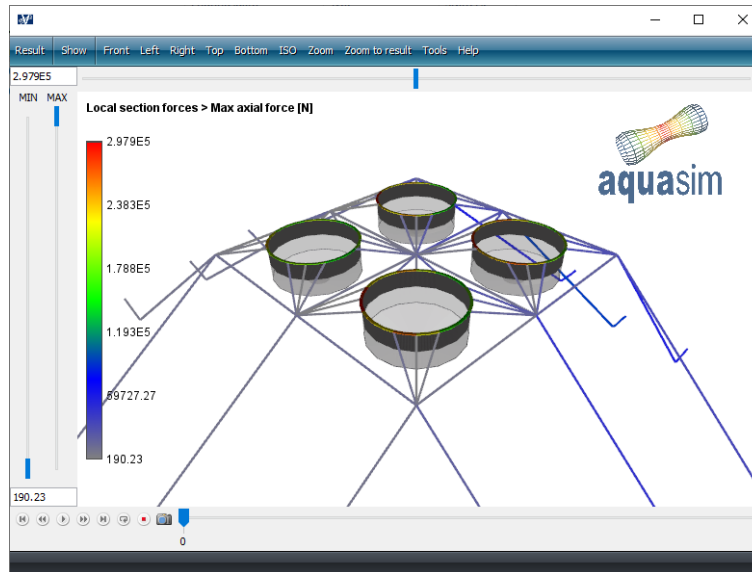
The analysis is now finished.

6.5 Postprocessing dynamic analysis

7 Open max out results

From the Analyse window, select **Open folder**. Navigate to the folder maxout and load the file dynamic_max_out.avz by double click.

From the Toolbar menu in AquaView, select **Result > Local section forces > Max axial force [N]**.



Press **j** on your keyboard to enter the window for maximum values of each component group.

Component name	Component index	Value	File index
Floating collar (Ø400 SDR.16.7)	1	2.979E5	dynamic02PFAT....
Suspension ring (Ø250 SDR.11)	2	5108.13	dynamic02PFAT....
Net pen (Sn15.4%)	3	N/A	N/A
Lice skirt 1	4	N/A	N/A
Lice skirt 2	5	N/A	N/A
Lice skirt 3	6	N/A	N/A
Lice skirt 4	7	N/A	N/A
Frame rope (64mm)	8	25955.75	dynamic01PFAT....
Bridle rope (48mm)	9	28923.50	dynamic02PFAT....
Mooring rope (64mm)	10	93170.69	dynamic01PFAT....
Bottom chain (32mm)	11	92703.38	dynamic01PFAT....

Copy the rows for component group number 8-11 and paste it in the calculation sheet you have established.

8 Evaluation of results

You should compare the maximum axial forces from the two load conditions with CaseStudy04. In the figure below, the axial forces in the mooring components are reported for CaseStudy04 and CaseStudy05.

	Q	R	S	T	U	V	W
21	CaseStudy04, simplified model						
23							
24							
25							
26							
27		Component name	Component no.	Axial force [N]	File index		
28		Frame rope (64mm)	4.00	36174.14	dynamic01PFAT.avz		
29		Bridle rope (48mm)	5.00	27380.02	dynamic02PFAT.avz		
30		Mooring rope (64mm)	6.00	84287.79	dynamic01PFAT.avz		
31		Bottom chain (32mm)	7.00	83664.83	dynamic01PFAT.avz		
32							
33							
34							
36							
37							
38	CaseStudy05, lice skirt						
40							
41							
42							
43		Component name	Component no.	Axial force [N]	File index		
44		Frame rope (64mm)	8.00	25955.75	dynamic01PFAT.avz		
45		Bridle rope (48mm)	9.00	28923.50	dynamic02PFAT.avz		
46		Mooring rope (64mm)	10.00	93170.69	dynamic01PFAT.avz		
47		Bottom chain (32mm)	11.00	92703.38	dynamic01PFAT.avz		
48							
49							
50							
51							
53							

By comparing, we see that the forces in Mooring rope (64mm) and Bottom chain (32mm) have increased in CaseStudy05. This is due to the lice skirts, where the fluid particles are forced to flow around the impermeable net, causing an increase of drag forces. The forces are transferred to the mooring ropes and bottom chain.

The axial force in the frame rope, is on the other hand, reduced. This can be explained by the added mass that is associated with the lice skirt. On the inside of the fish cage water mass is captured. It will take time to move all this water mass, and the consequence is a slower response of the cage in CaseStudy05 compared to CaseStudy04.

Dynamic considerations are finished. Exit AquaView, the Analyse window and close AquaEdit.

6.6 Summary

In this case study you have investigated the effect lice skirts have on forces in mooring components. By adding impermeable tarpaulin, the axial forces will increase.

--- End of document ---