» AquaEdit

AquaCross

AquaView

AquaTool

Other

User manual

Document ID: TR-20000-583-1
Date: 09/12/2024

AquaSim version: 2.20

Aquastructures AS
Kjøpmannsgata 21
7013 Trondheim
Norway

Contents

1. Introduction

AquaSim is an analysis tool developed by Aquastructures AS. It uses the Finite Element Method (FEM) for calculation and simulation of structural response. The software is well suited for slender, lightweight- and large volume structures, flexible configurations and coupled systems exposed to environmental loads such as:

- waves
- currents
- wind
- impulse loads
- operational conditions
- resonance

This manual describes the functionality of the post processing tool AquaEdit.

1.1 The AquaSim software package

The AquaSim package consists of several programs, as well as the AquaSim solver. Also included is several postprocessing tools.

1.1.1 AquaEdit

Building geometrical models through a graphical interface. Structural and hydrodynamic properties are defined and added to the graphical model.

1.1.2 AquaSim solver

Having prepared the input by an analysis model, one starts the AquaSim solver to derive results. This is the part that calculates the results from the given input of geometry, properties, loads and environment.

AquaSim handles global analysis and interactions of forces transmitted between stiff and flexible components. AquaSim establishes simultaneously a visual simulation of displacements, accelerations, and deformations in the structure AquaSim calculates for each step the local section forces, stresses, and stress ranges in each system component, applicable to local analysis and fatigue assessments.

AquaSim is based on real-time simulations. This implies that AquaSim considers the nonlinear effects, such as geometrical changes in the component's cross-section, to continuously maintain the correct relation between e.g. the applied forces and the resulting displacements.

AquaSim considers hydroelasticity, handling the interactions and coupled dynamics between the external loads and the construction. Deformations and changes in the global structural geometry will imply changes in the load scenario applied to the construction.

For more technical information about the AquaSim solver read the report in (Aquastructures, 2024a).

1.1.3 AquaView

The 3D results viewer, AquaView presents results from the solver graphically. For more information read the AquaView manual (Aquastructures AS, 2022f).

1.1.4 AquaTool

AquaTool presents the results from the AquaSim analysis in tables and diagrams. For more information read the AquaTool manual (Aquastructures AS, 2022e).

1.1.5 Post-processing tools

There are several post-processing tools included. These are covered in an own user manual which is available on request from the user (Aquastructures AS, 2022g).

1.2 Terminology

These terms appear throughout this document:

	Definition
Node	Point in 3D space that describes end points of element.
Element	Object between two nodes (straight element) or four nodes (membrane element). The simulation process will treat each element as a discrete object, so for higher resolution results, add more elements.
Beam	A beam is a structural element that is capable of withstanding load primarily by resisting torsion and axial loads.
Membrane	Structural element used for nets
Truss	Structural element where forces in the members are either tensile or compressive forces. I.e. Ropes.
Node2node	Spring element going from one node to another.
Component	A group of elements which is defined by the same material data. Components are defined either as beam, truss, membrane or node2node.
Shape	Shapes are predefined 3D models that can be imported and viewed as part of the main model but does not affect the simulation in any way.
Terrain	Terrain data can be imported from several sources. Point data type terrain data should be used. The terrain will be imported and optimized for viewing and usage.
Decorator	An attribute added to a node or an element, like pointloads, hinges, etc.
Intent	A function used in the main view; draw line, extrude, split line etc.

2. Interface

2.1 Main view

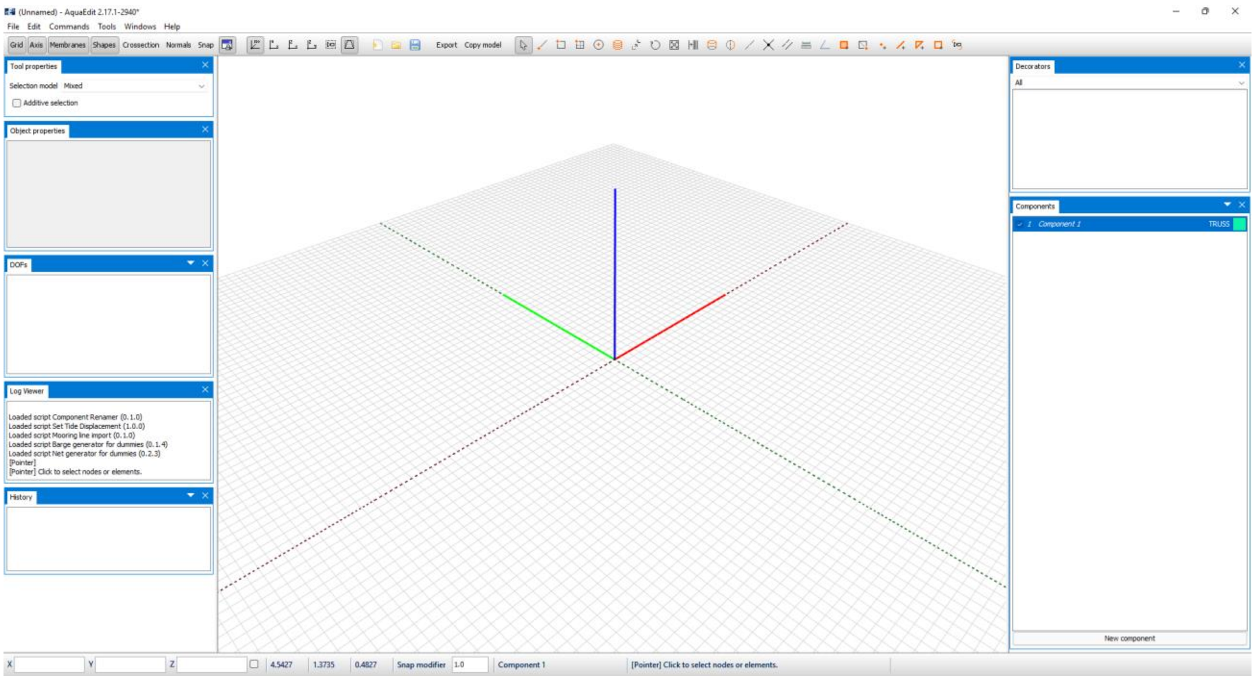
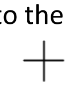
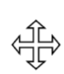


Figure 1 - The main application view

A picture of the main view is shown in Figure 1. More detailed description of the view is described in the following chapters.

2.2 Interacting with the 3D model

Name	Function
Select	Press left mouse button or hold down left mouse button for box select. Press ctrl and left mouse button to select more, or to de-select selected node/element. Hold ctrl and left mouse button add several nodes/elements to the selection. Hold shift to select only nodes when box selecting.
Rotate	Press and hold down middle mouse button.
Pan	Press and hold down right mouse button.
Zoom	Scroll with mouse wheel. Holding down **CTRL** at the same time as zooming will use the alternate zoom acceleration. See Settings: Drawing The keyboard can also be used for zooming. These shortcuts can be found under **`Commands > Zoom.`**
Lock 	Locks the location of Tool Windows to the Main View.
Cursor in the 3D view	The cursor changes related to the action selected by the user: Drawing intent is selected  Translation and rotation of model 

2.3 Toolbar

The toolbar allows for toggling of several functions:

Name	Function
Update	Forces a redraw of the screen in case a command does not update correctly
Grid	Toggles visibility of grid
Axis	Toggles visibility of axis
Membranes	Toggles visibility of membrane elements
Shapes	Toggles the visibility of shapes
Crosssection	Toggles the visibility of cross-sections on beams
Normals	Toggles the visibility of normals (the projections of point3)
Snap	Toggles whether commands are forced to snap to closest node. Snap modifier determines how close something must be to enable snapping. This can be set by entering a new value into the box behind the snap modifier box. Pressing enter will set a new snapping threshold.
Export	Opens the environment data and export to AquaSim window.
Copy model	See chapter Copy model.

Tip! Right click on the toolbar to unlock/ lock the functions. By unlocking, the user may drag-and-drop the Tool-windows to the docking area or place it in the 3D view.

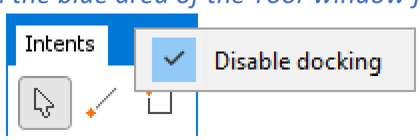
2.4 Intents

Tool window for selecting the active intent. A more detailed description of the intents can be found in chapter 3.

Figure 2 - Intent tool window

Pointer	Draw line	Draw rectangle
Draw membrane rectangle	Draw circle	Draw tube
Move	Rotate	Scale
Duplicate	Extrude	Mirror
Split line	Intersect	Merge lines
Measure distance	Measure angle	Select quad(s)
Draw membrane	Select node(s)	Select element(s)
Select component	Box select	Rotate view

Tip! Right click in the blue area of the Tool-window for more options.



2.5 Views

The view window can be used to view the model from different perspectives. The shortcuts for these can be found under **`Commands`** > **`Views.`**

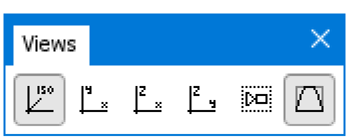


Figure 3 - View window

The default views are isometric, top-down, front and side. The second last button zooms to selected nodeselements.

The last button toggles between perspective (default) and orthogonal view.

2.6 File

The file window with shortcuts to some basic options. Shortcuts for these can be found under **`File`**.

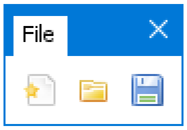


Figure 4 - File window

The options here are; New, Open and Save.

2.7 Object properties

This window will change accordingly to what is currently selected in the main window, examples below.

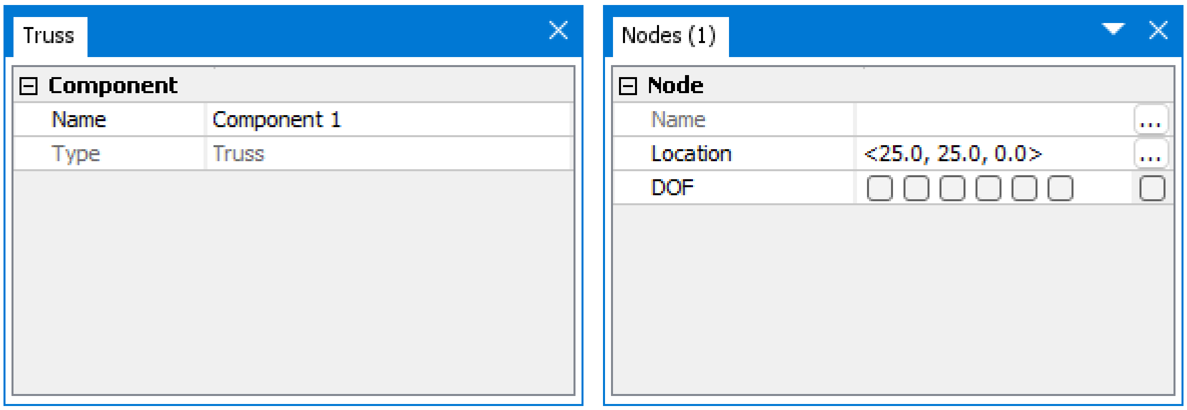


Figure 5 - Component selected on the left, node selected on the right

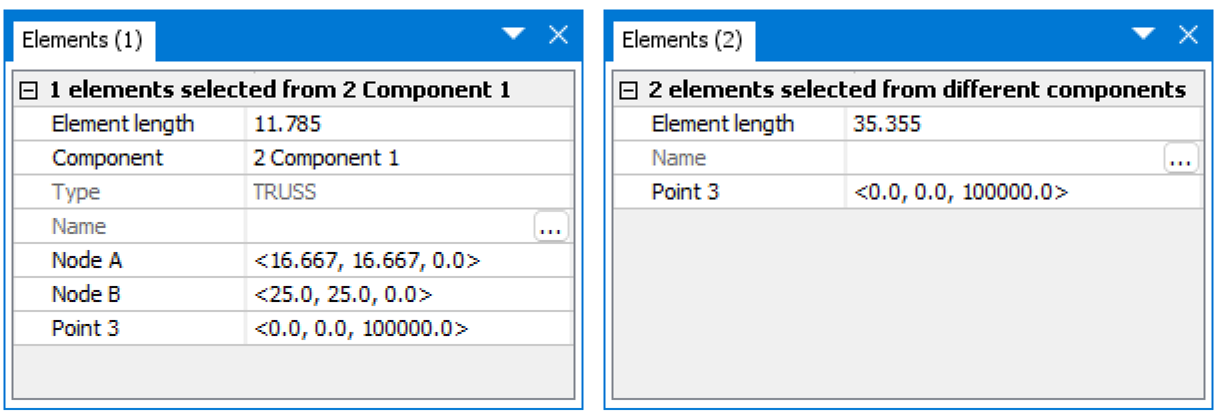


Figure 6 - One element selected on the left, two elements selected on the right

Figure 5 and Figure 6 shows what this window may look like depending on what is selected. A more detailed description is explained in chapter 4.

2.8 Tool properties

This window will contain properties regarding the current selected intent, the intents is shown in Figure 2 and is discussed further in chapter 3. A couple examples is shown below.

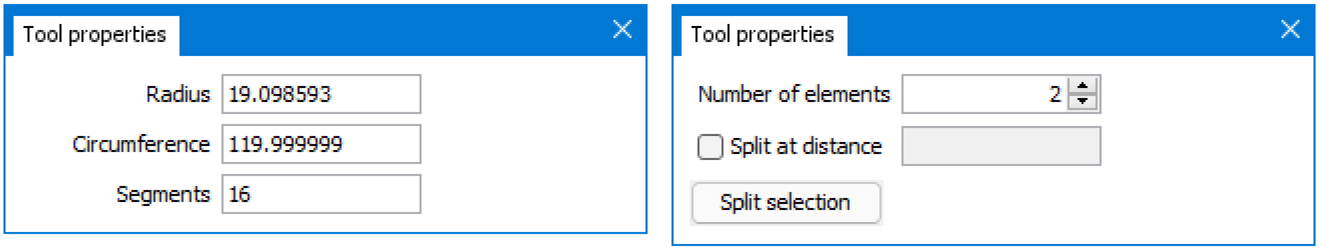


Figure 7 - Properties for drawing circle on the left, properties for splitting on the right

2.9 Decorators

All the decorators added to the model will be listed in this window. A dropdown menu can be used to filter the list to only show the decorators the user is interested in. Each decorator has a popup window when the user right clicks on an item in the list.

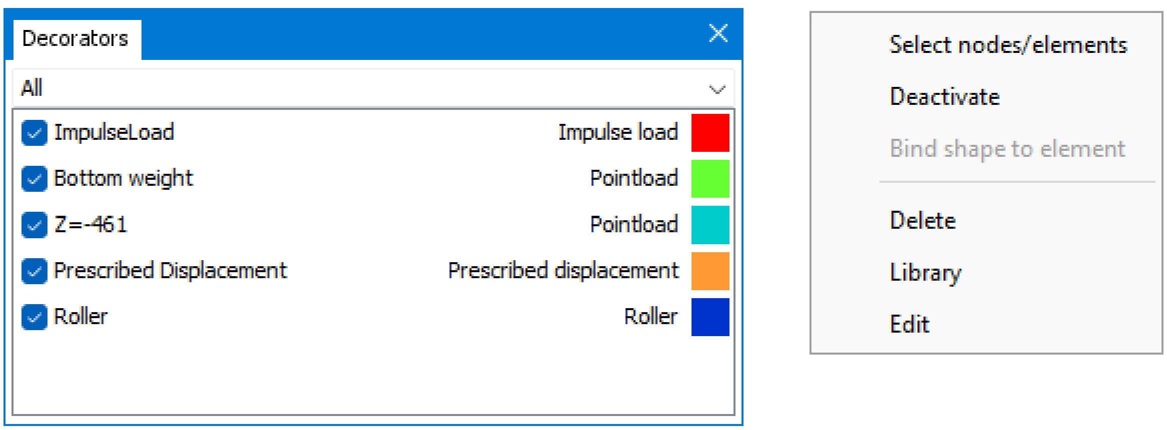


Figure 8 - Node decorators window with some decorators that are added to the model on the left, popup window on the right

The checkbox in front of each decorator toggles visibility in the 3D view.

By left clicking on the color behind a decorator will bring up a color palette where the color of the decorator can be altered. This color will be what the node(s) belonging to the decorator is drawn as. The default color is red. Note: This only works for decorators on nodes.

The property window of the decorator can be activated by double clicking an item in the list or selecting “Edit” in the popup window.

By selecting “Select nodes/elements” in the popup window the user can select all the nodes or elements that share the selected decorator.

By selecting “Deactivate” in the popup window the user can exclude the decorator from an analysis.

Selecting “Delete” removes the selected decorator from the model.

2.10 History

This window contains information of what the user has done while working on a model. An example can be seen in the picture below.

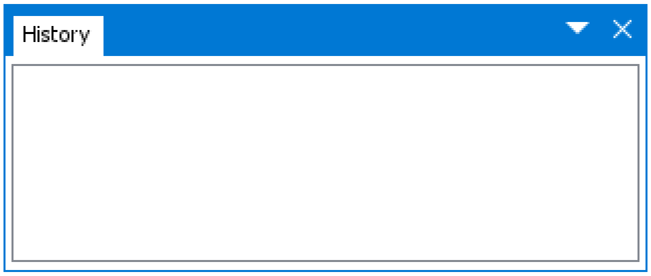


Figure 9 - History window

2.11 Log viewer

This window contains information on what the user has done while working. In contrary to the History window the Log viewer window is not reset when opening another model or creating a new model. It also contains some more information and when input is needed from the user the Log viewer will contain information about this. An example is shown below, where a rectangle has been drawn.

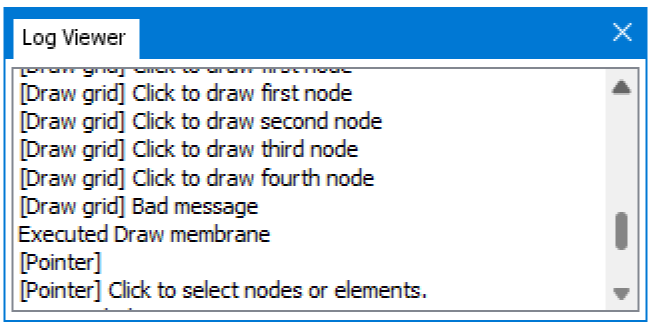


Figure 10 - Log viewer after a rectangle has been drawn

2.12 POIs

Shows the Points Of Interest. E.g. the node names.

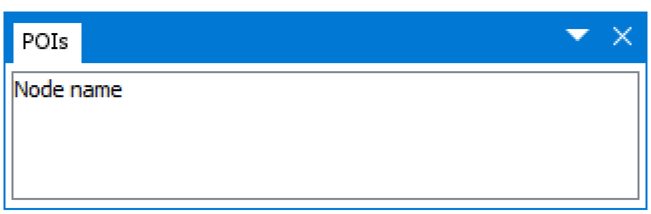


Figure 11 - POIs window

2.13 DOFs

Shows all DOFs in the current model. If a DOF is selected in the list, the corresponding node is selected and zoomed in. All DOFs can be edited directly in the list, and if a DOF is added/edited in the window for node properties then the DOF list will be automatically updated.

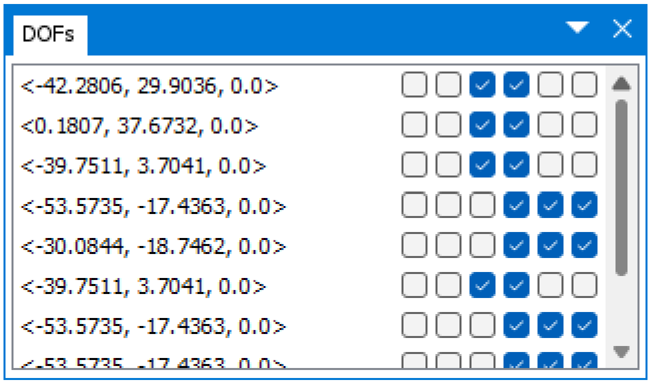


Figure 12 - DOFs window

2.14 Components

The components window shows a list of all the components in the current model. By default, the window sorts the components based on their type (beam, membrane, membrane_x, truss, node2node, shape, in that order), see Figure 13.

The checkbox in front of each component toggles the components visibility in 3D view. Components that are not visible cannot be interacted with.

If a component is lacking properties, the name will be shown in cursive. It is possible to rearrange the component list by dragging and dropping one or several selected items.

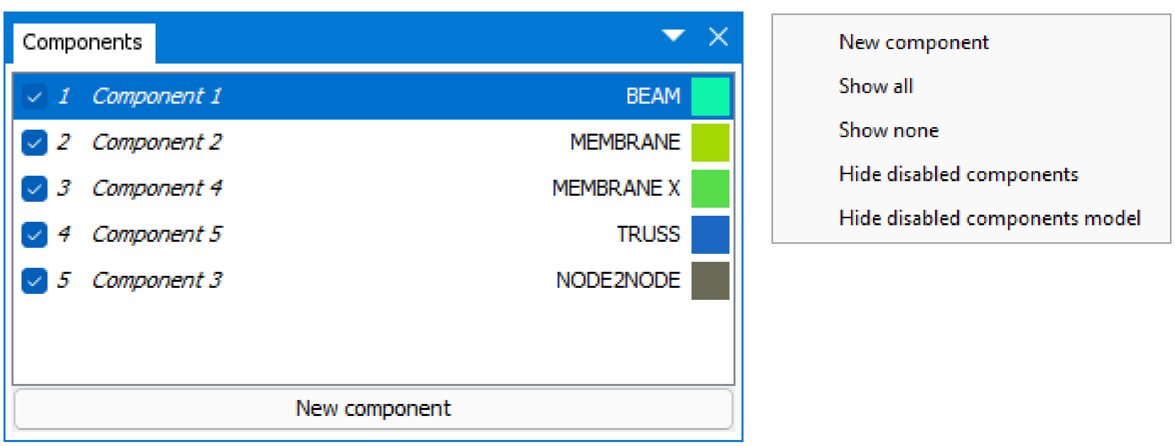


Figure 13 - The Components window on the left, popup window on the left

Figure 13 shows an example of a components window. The popup window on the right is shown when a user clicks on the ∨-button on the top-right corner of the window.

New components can be added by clicking the button “New component”, using the shortcut that can be found in **`Edit`**, or using the Library under **`Tools > Library`**. The user can drag and drop files from the Library window. If the user drags in one file over a component the user will get a question asking if the component should load this file into it, or if the user wants to create a new component based on the contents of the file. If the user drags in several files, the program will automatically create new components based on the content of these files.

2.14.1 Color

Left clicking on the color behind each component brings up a color palette where the color can be altered.

2.14.2 Component options

Right clicking on a component brings up a list of options for the selected component, see Figure 14.

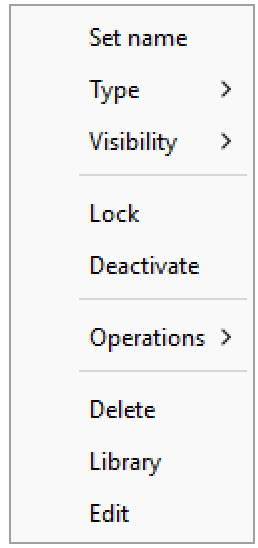


Figure 14 - Right click on a component

Set name

Sets the name of the component, keyboard shortcut F2. Name of component can also be changed in the property window for the component by selecting Edit.

Type

Sets the type of component; beam, membrane, membrane x, truss and node2node. If the component contains properties a warning will be shown before changing the type (which removes all pre-existing properties).

Visibility

Toggles the visibility, equal to using the check marks in front of the components. Using the “Show only this” option will turn the visibility of the component on, while turning visibility of the other components off.

“Show stress” will turn on/off stress coloring in AquaView when results are opened.

Lock

Disables any editing of the component.

Deactivate

Deactivates the component from analysis.

Operations

See chapter 2.14.3 (next chapter).

2.14.3 Operations

Select

Select all elements of the component.

To active component

Will move all selected elements to this component.

Copy from

Will copy properties from a valid selected component.

Merge

If multiple components of the same type are selected, they will be merged into one. Only the properties of the first selected component will be used.

Add contact

Adds a contact between two selected components. The Component contact table will be loaded, as illustrated in Figure 15.

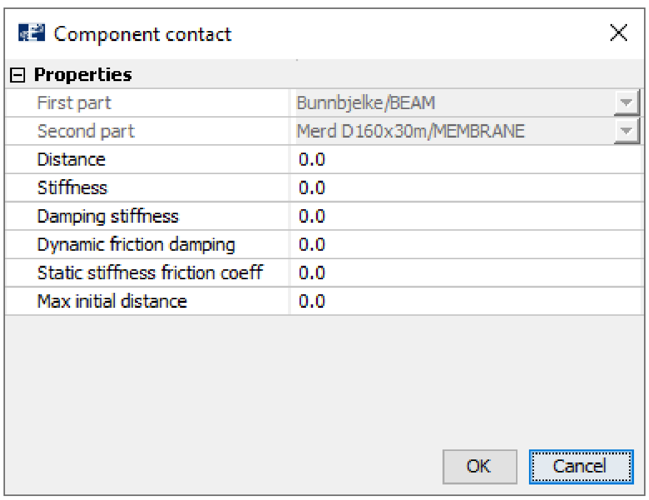


Figure 15 - Component contact

Properties	Description
Distance	Distance before center of node and center of element the contact starts.
Stiffness	Stiffness of contact.
Damping stiffness	Damping proportional to stiffness at contact, normal direction.
Dynamic friction damping	Dynamic friction damping proportional to stiffness parallel to element.
Static stiffness friction coeff	Static stiffness friction coefficient proportional to stiffness.
Max initial distance	When contact is established between two component groups, AquaSim will keep track of the distance between all elements within these two groups. For large models, this can slow down analysis time. For elements that are initially further away from “Max initial distance” contact will not be established.

Calculate membrane volume
Calculates the volume of the membrane (in drawn state). Available on Membrane and Membrane X components.

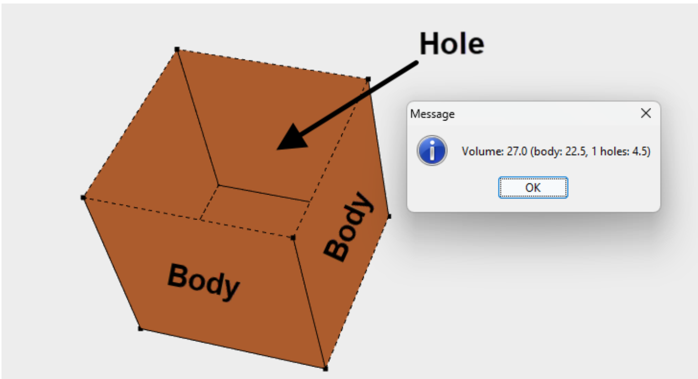


Figure 16 – Calculate membrane volume

Figure 16 how an example of a container with 5 walls and the top open. The container sides are the 'body' and the open top is 'hole'. The total volume of the container is reported as 'Volume'. Unit is [m3]. It should be noted that the volume-calculator algorithm is best suited for MembraneX in combination with *Membrane normal are verified*.

Calculate membrane area
Show the calculated surface area of the membrane (in drawn state). Only available on Membrane and Membrane X components. Unit is [m2].

Convert to explicit
Will turn a component with type membrane over to the type MembraneX (explicit membrane).

Convert to terrain
Will convert the component to terrain data, using the elements of the component as basis for the terrain (only available on membrane components).

Select diagonal elements
Select elements that are diagonal.

Set component colour
Enable to set the component colour for multiple component groups. The colour is changed for the component groups that are multiselected.

Delete
Deletes this component.

Library
Opens the library for exporting or applying a template to the component. This dialog is explained in more detail in chapter Library.

Edit

Brings up the property window for the component, this is explained further in the coming chapters. Double-clicking on the component or pressing ENTER will also open the property window.

2.14.4 Beam

Information

The first panel of the property window, see Figure 18, specifies the name and the description of the beam, as well as whether the beam has Morison or hydrodynamic load model. For more information about Morison and hydrodynamic load models, see (Aquastructures, 2024a).

Choosing one of the items in the dropdown list under “Data source” will auto-generate cross-section values for the beam. This function calculates cross-section values for: Tube, hollow rectangular, massive rectangular, I/H-beam and double tube. All auto-generated values will be locked, to edit fields manually choose “Custom” in the dropdown list after generating the cross-section. There is also a possibility to draw your own cross-section in a separate 2D drawing program that opens using the data source AquaCross. This is explained in detail in its own user manual (Aquastructures AS, 2022d).

The visual cross-section can be transferred to the hydrodynamic cross-section by clicking the button “Copy to hydrodynamic”.

The text area contains the points in the 2D cross-section. Each line represents one point, with the coordinate on the y-axis first and the coordinate on the z-axis second. For a cross-section that is symmetric around the z-axis, only half of the points are needed, enabling “Symmetrical” will copy the points on the right-hand side of the z-axis over to the left-hand side and create a symmetrical cross-section.

The cross-section can be visualized in the 3D-window with the cross-section applied along the beam elements local x-axis. AquaEdit follows the right-hand rule, see Figure 17. Direction **a** in the figure is the direction from node A to node B (local x-axis), direction **a** x **b** is the local z-axis (see chapter Point 3) and direction **b** is the local y-axis. This is important to remember when drawing a non-symmetrical cross-section and when editing point3 of the element.

The beam can have an asymmetric cross-section as well. This means that the cross-section at node A and node B is different and AquaEdit will interpolate between them to create an asymmetric cross-section. Figure 19 shows three beam elements with a square cross-section at node A and a straight-line cross-section at node B. The button “Scale” will take cross-section at node A and scale it and then apply it to node B.

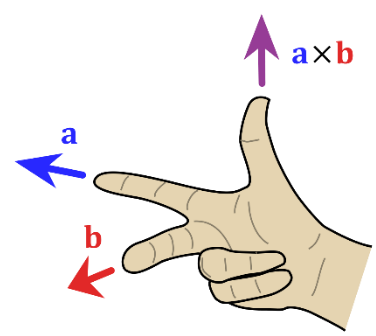


Figure 17 – Right-hand rule

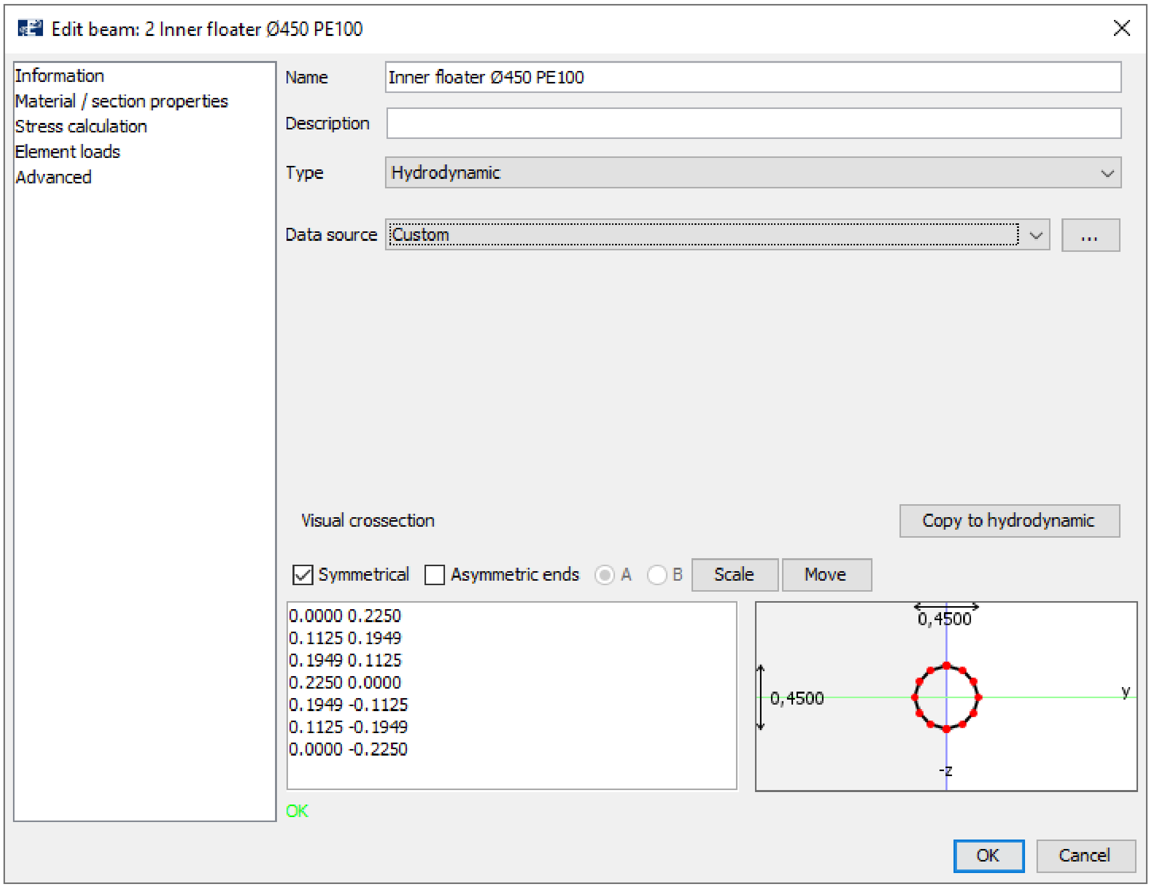


Figure 18 – Information panel

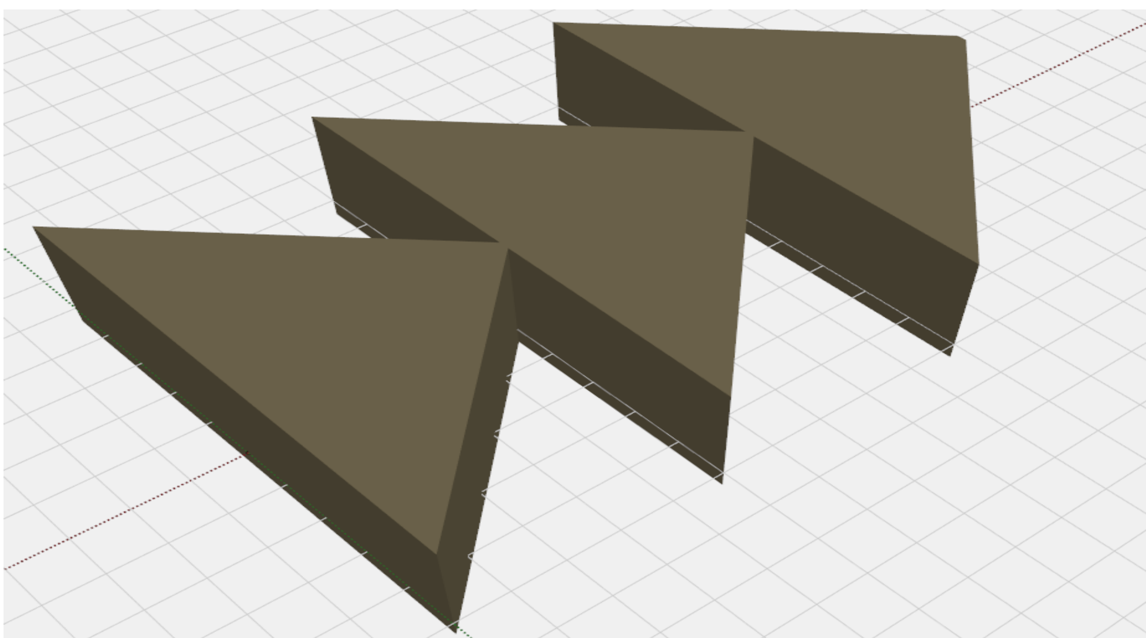


Figure 19 - Asymmetrical beam elements

Material/section properties

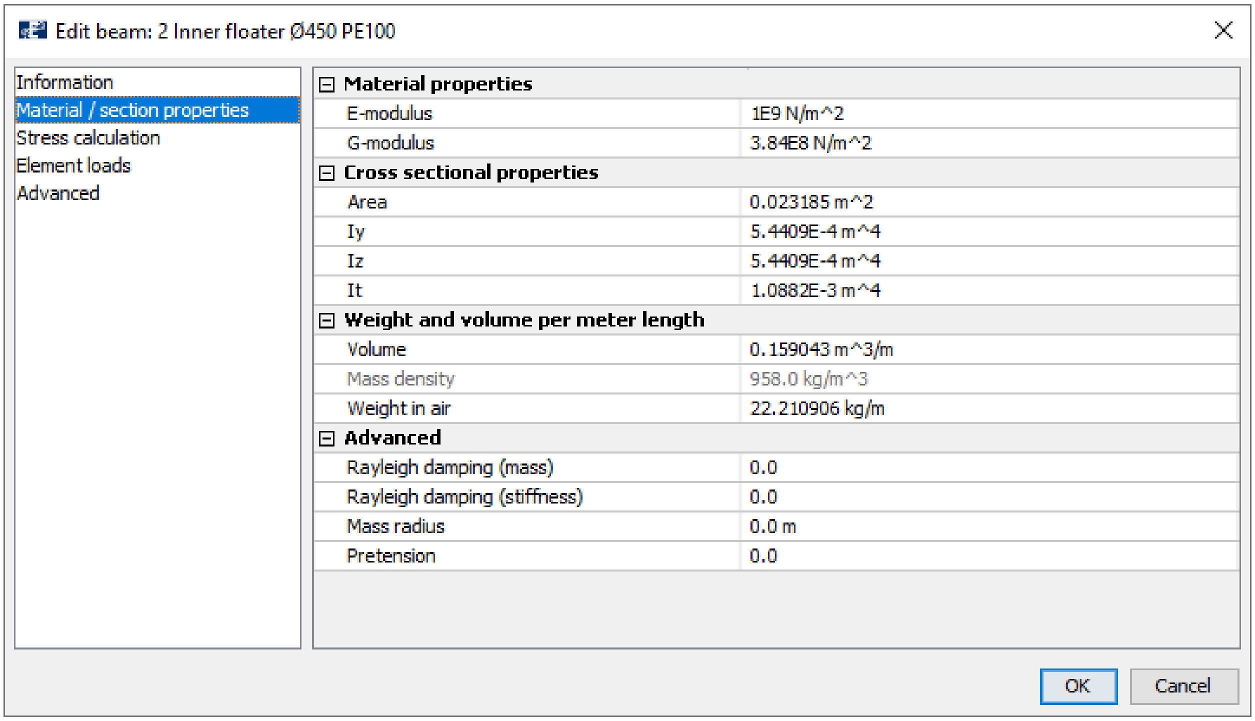


Figure 20 - Material/section properties panel

Properties	**Description**
E-modulus	Young's modulus.
G-modulus	Shear modulus.
Area	Cross sectional area.
Iy	Moment of area around local y-axis.
Iz	Moment of area around local z-axis.
It	Torsional stiffness.
Volume	Hydrodynamic: Maximum buoyancy per meter Morison: Amount of added mass. Also relates to the added mass of the component.
Mass density	Mass density of material.
Weight in air	Weight in air of element, per length.
Weight in water	$Weight\ in\ water = weight\ in\ air - buoyancy\ of\ element$ (if element is in air, then input is weight in air). If "With slamming" is chosen under Time series (see chapter 9.4) then Weight in water must have an input.
Rayleigh damping (mass)	Extra damping on acceleration on elements
Rayleigh damping (stiffness)	Extra damping on stiffness on elements.
Mass radius	Average radius the mass has with respect to local x-axis
Pretension	Inner tension in element (pre-strain).

A more detailed description can be found in (Aquastructures, 2024a).

Stress calculation

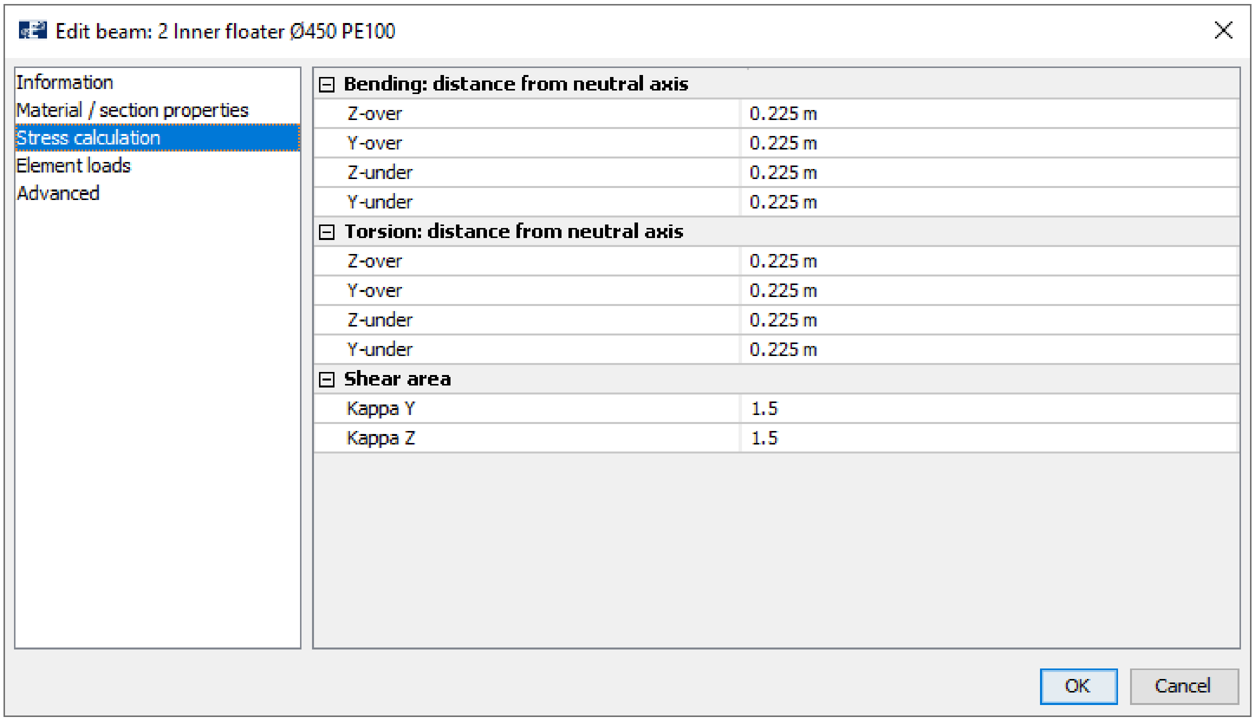


Figure 21 - Stress calculation panel

The arms for bending and torsion, and shear constants/shear area. This is used to calculate stresses in the beam dependent on forces and moments acting on it.

A more detailed description of bending, torsion and shear is found in (Aquastructures, 2024a).

Element loads

For Morison load model:

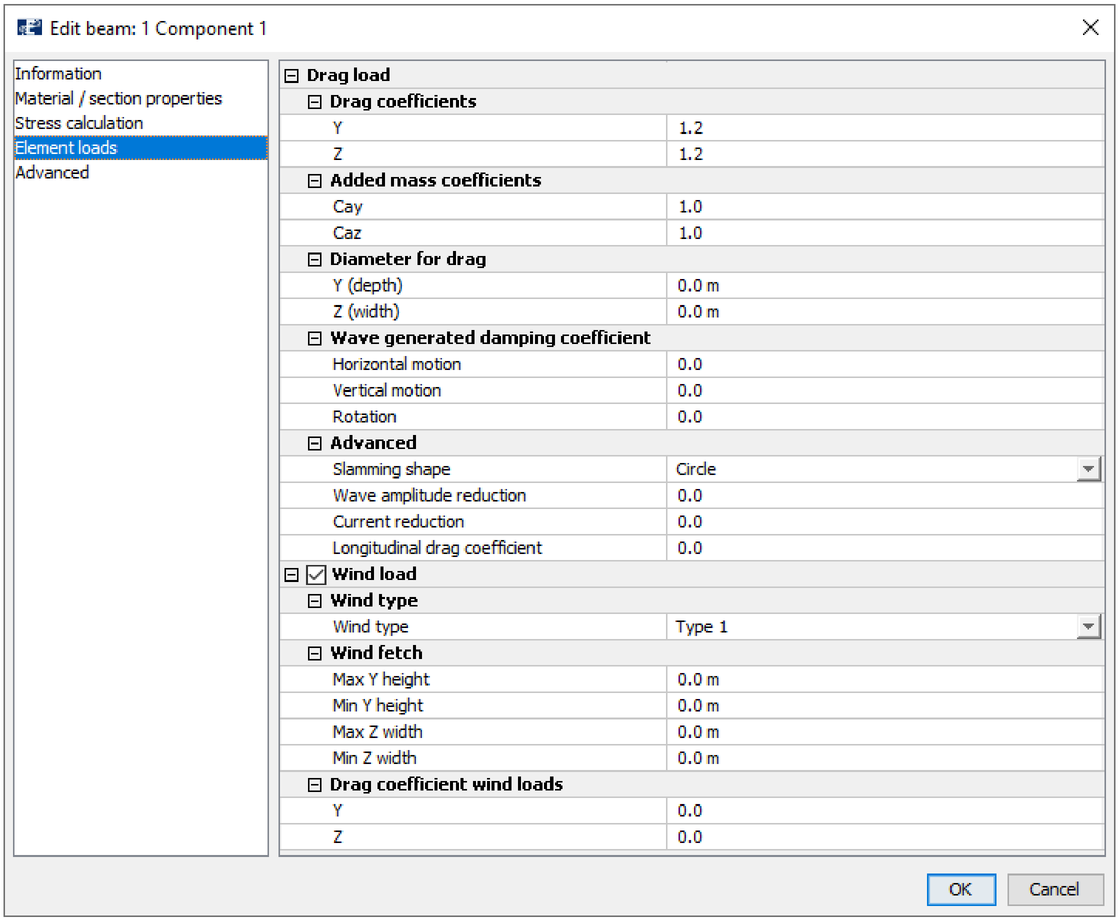


Figure 22 - Element loads panel, for Morison load model

Properties	**Description**
Drag coefficient Y/Z	Drag coefficient for local y- and z-axis.
Added mass coefficient C_{ay}/C_{az}	Added mass coefficient for local y- and z-axis.
Diameter for drag Y (depth)	This will together with element length define drag area for element in local y-axis.
Diameter for drag Z (width)	This will together with element length define drag area in local z-axis.
Wave generated damping coefficient	In case element is in wave zone, additional damping can be included. Horizontal motion, Vertical motion, and Rotation.
Slamming shape	Shape of object going thought the water line. Default circle, but rectangular volume with height based on the drag height.
Wave amplitude reduction	0 means full wave, 1 means no wave, 0.5 means half amplitude.
Current amplitude reduction	0 means full current, 1 means no current, 0.5 means half current.
Longitudinal drag coefficient	Drag coefficient in local x-direction.
Wind load	See description below.

Wind load

Wind load is activated/de-activated by clicking the checkbox. There are two alternatives to choose from in the drop-down list, Type 1 and Type 2.

Type 1:

Wind is added in local Y and Z directions on beam elements. The wind drag area is defined by Max/ Min Y height and Max/ Min Z width. AquaSim assumes the wind area to be rectangular. The wind drag force is then calculated from this area and the associated drag coefficients.

Wind is only possible to add lateral and transverse to the beam. To add wind in the direction of the beam (i.e. along local x-direction); use a “dummy beam” perpendicular for the “main beam”.

AquaSim uses the wind speed at an altitude of 10 meters as input. As the wind is reduced towards the surface, it is necessary to use the position of the top and bottom of the wind-affected area to get the correct wind force acting on the element. AquaSim takes into consideration the z position of the elements when calculating the correct wind force.

Max/Min Y height is the top and bottom of that area. For horizontal elements with a default horizontal local y-axis, this is the height of the element.

As some elements during the analysis tends to roll, or for some other reason the default y- and z-axis is changed, there would be a wind-affected area perpendicular to the local x-axis. For wind-affected elements, rolling, the Min/Max Z width describes the width of this element measured from the neutral axis of the element. See Figure 23 for illustration.

The wind fetch area is then calculated from the element length, the differential in Max Z width and Min Z width along the element’s **local** y-axis and the differential in Max Y height and Min Y height along the element’s **local** z-axis.

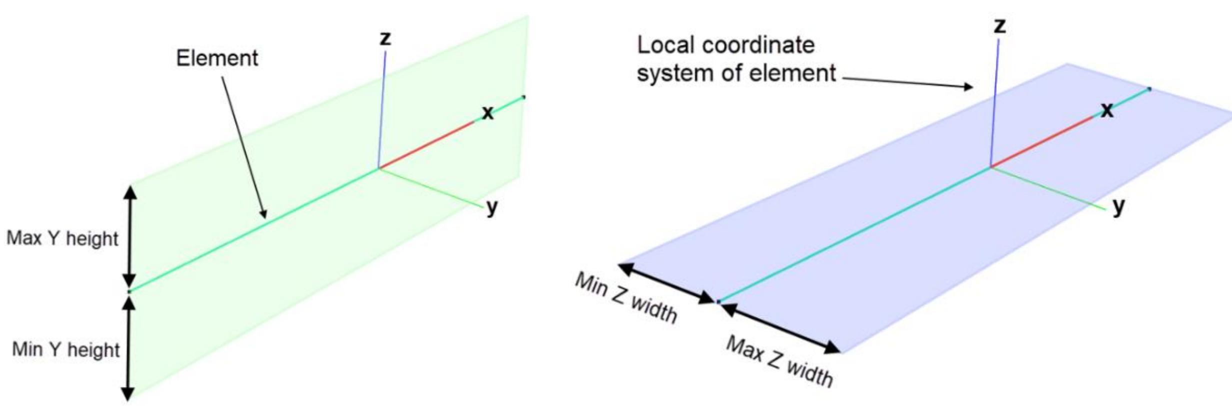


Figure 23 - Wind fetch area for an element Type 1

Type 2:

For Type 2, wind forces are treated in the same way as drag loads from current. Wind forces are calculated based on “Diameter for drag” and “Drag coefficient” under the tab “Drag Loads”. Instead of using current-speed and density of water, wind-speed and density of air is used.

AquaSim assumes the area to be rectangle. Although “Diameter for drag” indicates a circle, the area is calculated based on the assumption that Y (depth) and Z (width) represents one side of a rectangle.

Type 2 takes into consideration the Z position of the element. The wind profile is calculated according NORSOK N003, more information about this is found in (Aquastructures, 2024a).

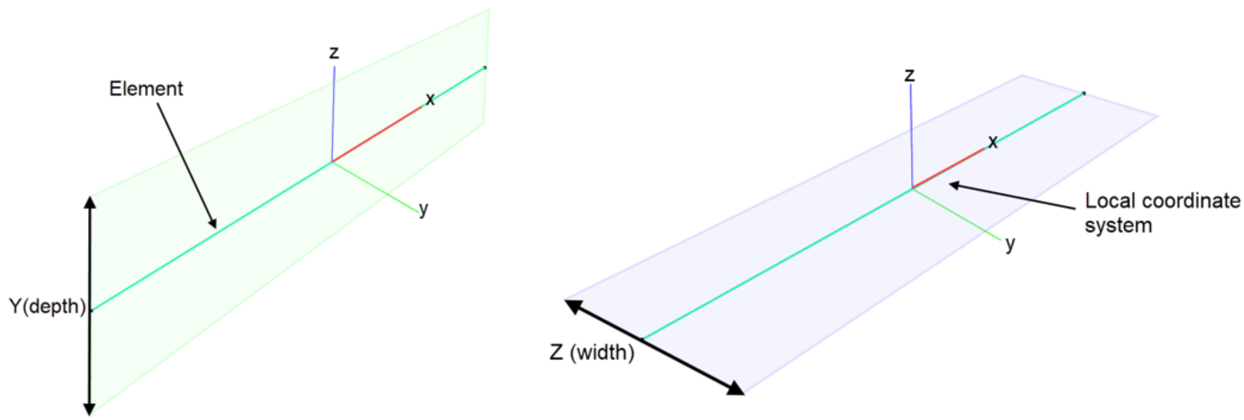


Figure 24 - Wind fetch area for an element Type 2

For Hydrodynamic load model:

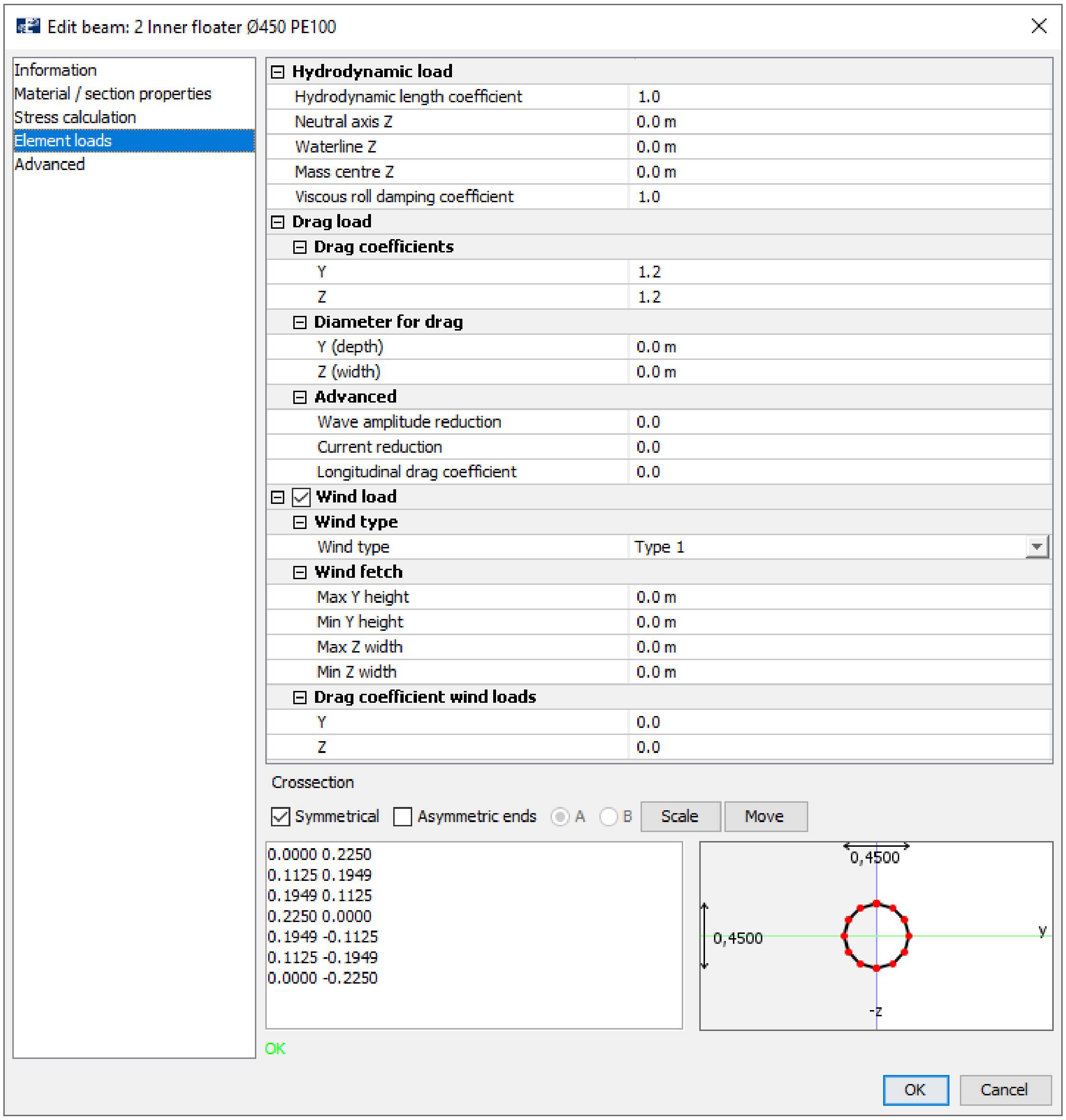


Figure 25 - Element loads panel, for Hydrodynamic load model

Properties	**Description**
Hydrodynamic length coeff.	Length of flotation on element (e.g. pontoons) divided by total length of element. Between 0 and 1.
Neutral axis Z	Vertical space between element and the elements cross-sectional centre.
Waterline Z	For adjusting the waterline level on the hydrodynamic profile. Positive level means more buoyancy. In addition, a blue line on the cross-section view indicates where the adjusted waterline is.
Mass centre Z	Vertical space between element and mass centre of the cross-section.
Viscous roll damping coeff.	Damping coefficient for roll motion on beam.

Wave amplitude reduction	Reduction of wave amplitude on element, due to e.g. shadow effects. A number between 0.0 and 1.0. 0 correspond to 0% reduction, 1.0 correspond to 100% reduction.
Current reduction	Reduction of current due to e.g. shadow effects. A number between 0.0 and 1.0. 0 correspond to 0% reduction, 1.0 correspond to 100% reduction.
Longitudinal drag coefficient	Drag coefficient in local x-axis.

A more detailed description of hydrodynamic loads can be found in (Aquastructures, 2024a).

The Scale button with create an asymmetric cross-section where the cross-section on the B side will be the A cross-section scaled by this factor.

The Move button translates the cross-section in the z and y dimension. Note that for mirrored cross-sections it will only move the mirrored half. The correct inputs to the move dialog are z displacement, y displacement.

Advanced

For load formulation Morison submerged:

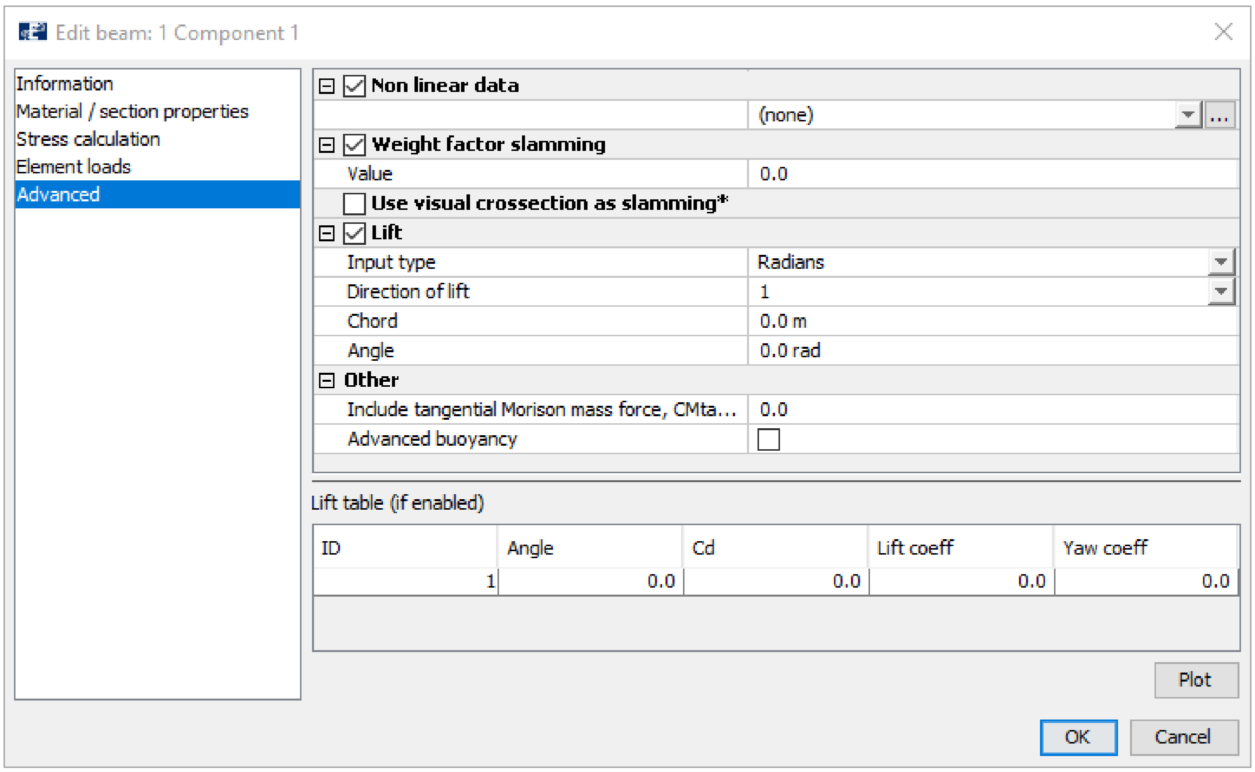
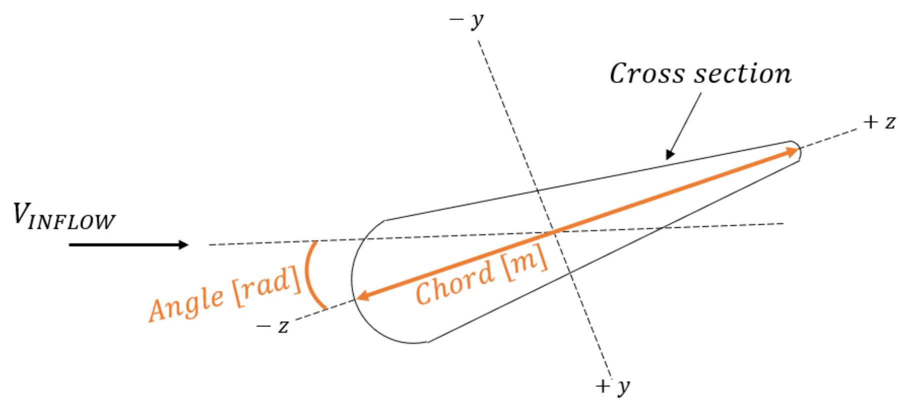


Figure 26 - Advanced panel, for Morison load model

Properties	Description
Non linear data	Non linear data can be added to the beam. For more information about non linear data, see chapter Node2Node spring.
Weight factor for slamming	Weight factor slamming can be activated if element is filled with water when submerged when using the "With slamming" under the Advanced tab in the Export dialog. This factor is the size of the non-filled volume (typical equal to area) divided by total volume (typical volume).
Lift	Lift can be added to the beam, for more information see (Aquastructures, 2024a).
Other	- **Include tangential Morison mass force, CMtan factor:** hydrodynamic mass force in tangential direction (local x-direction) can be applied to the beam. - **Advanced buoyancy:** removes pressure difference on partly- and fully submerged elements. Useful in cases where elements are e.g. fixed to a seabed, removing unrealistic excess buoyancy.

Lift

Properties	Description
Input type	This indicates if data is given in radians or degrees
Direction of lift	Positive direction of element (corresponding to 0 degrees angle of incident flow) is along the local z-axis. 1: The lift direction is in accordance with a normed space. -1: The inverse. -2: The rotation direction is opposite, but not the defined lift direction. -3: The lift direction is defined opposite, but not the angle.
Chord	The coordinate length the drag and lift coefficients are multiplied with. They are also multiplied with the length of the element. See illustration in Figure 27.
Angle	Angle between the in-flow and the local z-axis of the beam. If angle is 0, the local z-axis defines the 0-angle flow direction to the beam in the cross-flow plane. It is recommended to use angle 0 and direct the local z-axis to be in accordance with this. This is to avoid any miss on positive/negative angle.



a

Figure 27 – Definition of Chord and Angle

To define the lift for different angles it is necessary to add a table defining the different drag, lift and yaw coefficient for the incoming angle. Add rows by pressing ENTER inside the table.

Properties	**Description**
Id	Data number.
Angle	In-flow angle in the cross-flow plane relative to the local z-axis.
Cd	Drag coefficient in the flow direction.
Lift coefficient	Lift coefficient. Factor for force 90 degrees to the flow direction in the cross-flow plane.
Yaw coefficient	Coefficient for rotation around the beam of the lift object.
Plot	Provide a plot of Cd, Lift coeff and Yaw coeff as a function of Angle. Opened in a separate window.

For Hydrodynamic load model:

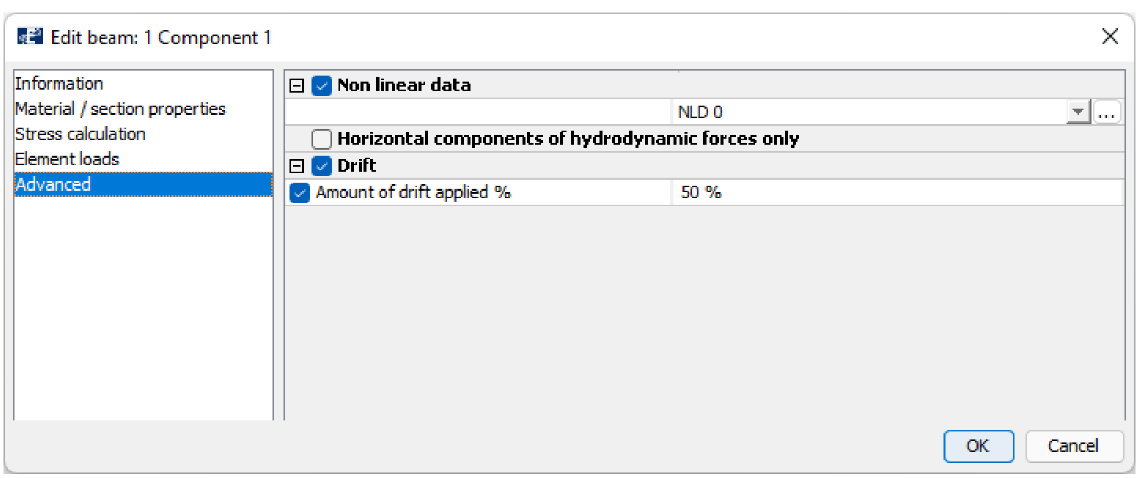


Figure 28 - Advanced panel, for Hydrodynamic load model

Properties	Description
Non linear data	See chapter Node2Node spring.
Horizontal component of hydrodynamic forces only	Only horizontal components of the hydrostatic and hydrodynamic forces are accounted for. I.e. drift and drag forces. Meaning that the hydrostatic forces and vertical hydrodynamic forces are set to 0.
Drift	Toggle on to include calculation of wave drift forces. Wave drift forces are calculated by the default AquaSim methodology. Amount of drift applied: wave drift forces are reduced manually by reducing the amplitude of the incident wave. 1% corresponds to 1% of the incident wave, 50% corresponds to 50% of the incident wave, and so on.

2.14.5 Membrane

Information

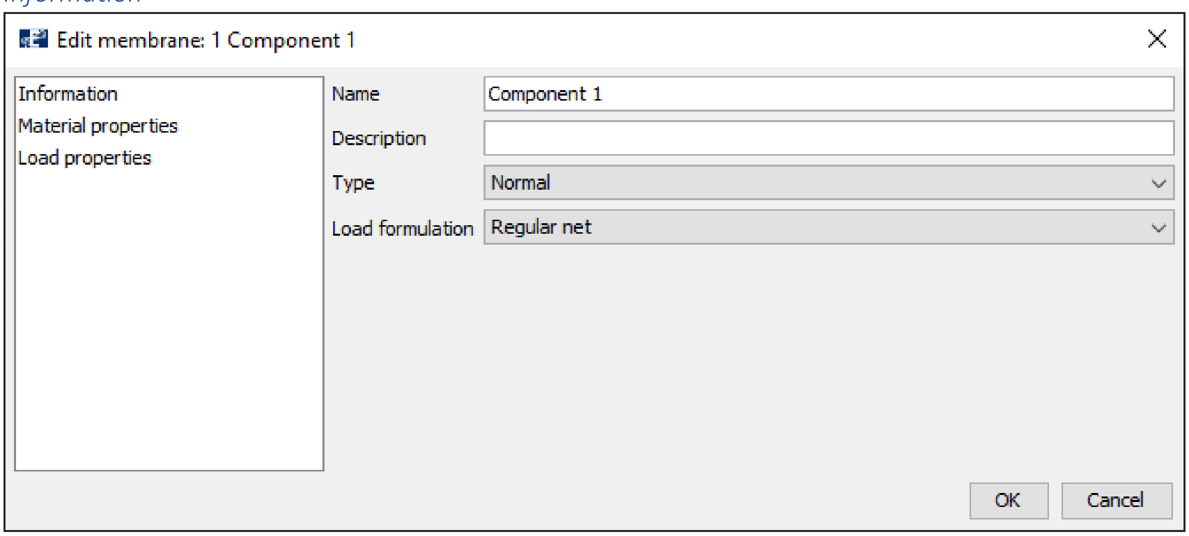


Figure 29 - Information panel

The first panel of the property window, see Figure 29, specifies the name and the description of the membrane. The type of membrane and load formulation can also be edited here. There are 3 membrane types: Normal, Normal with bending stiffness and Shell. Based on what is chosen, the tab "Material properties" will show the input needed for said type. If type "Normal with bending stiffness" is chosen, then a tab named "Bending stiffness" will appear with the input needed.

There are 5 Load formulations that can be used: Normal, Morison free plate, Lice skirt, Closed compartment and Surface tarpaulin. Based on which Load formulation is chosen, the tab "Material properties" will show the input needed for the Load formulation in question. For membrane type Shell, the Load formulations Lice skirt, Closed compartment, Surface tarpaulin and Morison free plate are available.

Material properties for Type Normal and Normal with bending stiffness

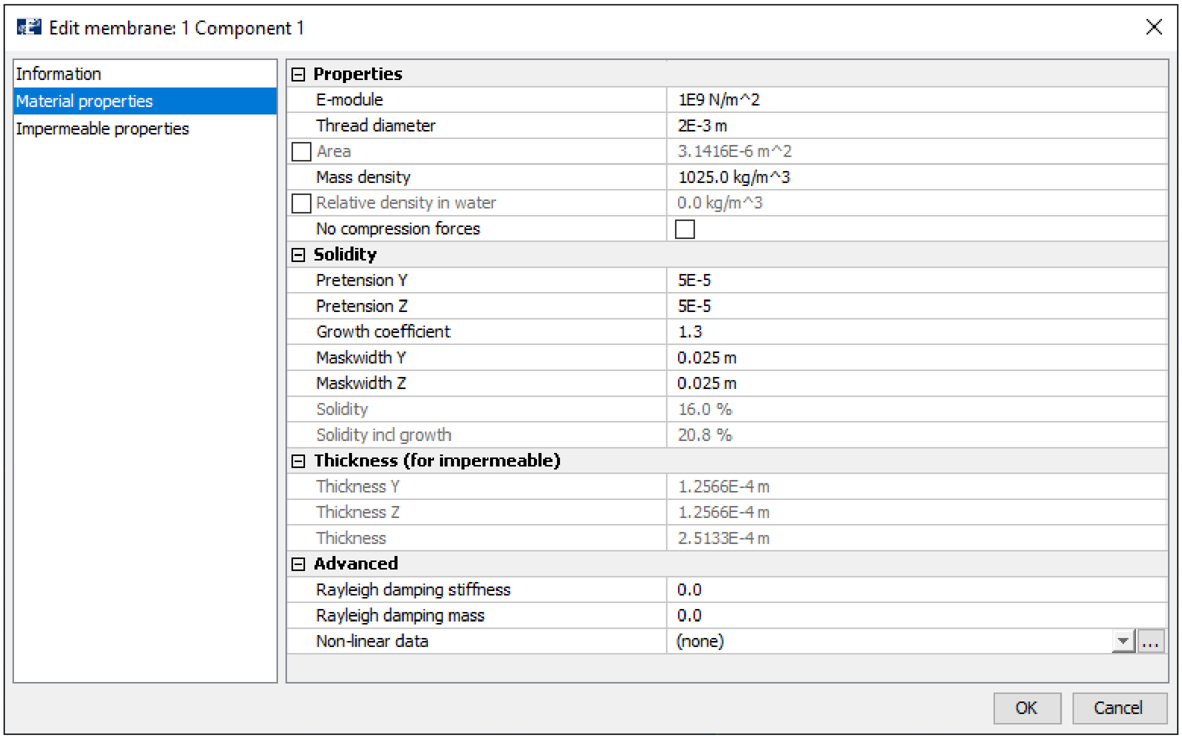


Figure 30 - Material properties for Type Normal and Normal with bending stiffness

Figure 30 shows the input for types Normal and Normal with bending stiffness.

Properties	**Description**
E-module	Young's modulus.
Thread diameter	Diameter of the thread.
Area	Cross-sectional area of the thread.
Mass density	Mass density for the thread.
Relative density in water	Relative weight in water. Density of material subtracted density of salt water (1025 kg/m^3
No compression forces	Forces in the net twines will be 0 if compressed.
Pretension Y	Elongation divided by length for threads in local z-direction.
Pretension Z	Elongation divided by length for threads in local y-direction.
Growth coefficient	Parameter to account for increased diameter of twines from fouling.
Maskwidth Y	The distance between the threads in horizontal direction.
Maskwidth Z	The distance between the threads in vertical direction.
Solidity	The solidity of the net.
Solidity incl growth	The solidity of the net, included fouling.
Rayleigh damping stiffness	See Material/section properties for beam.
Rayleigh damping mass	See Material/section properties for beam
Non-linear data	Non-linear relation between thread strain and axial force. Read more about this below.

Non-linear data:

It is possible to establish non-linear relation between strain and axial force in threads. From the drop-down menu, select (new). The non-linear data table is presented below. The user can choose between isotropic and anisotropic behavior:

- Isotropic: relation between strain and axial force in local z-direction only.
- Anisotropic: relation between strain and axial force in both local z- and y-direction.

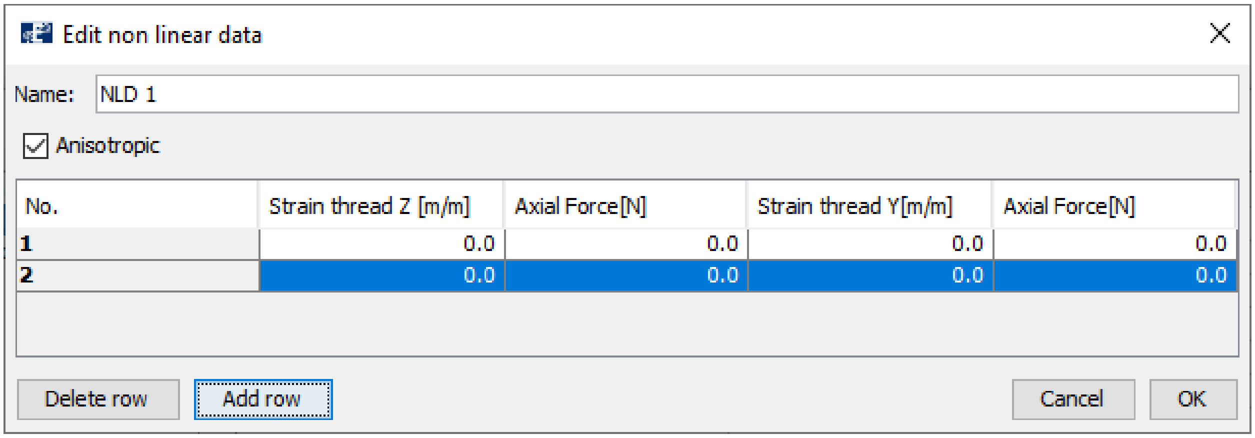


Figure 31 Non-linear data table for membrane Type Normal and Normal with bending stiffness

Bending stiffness

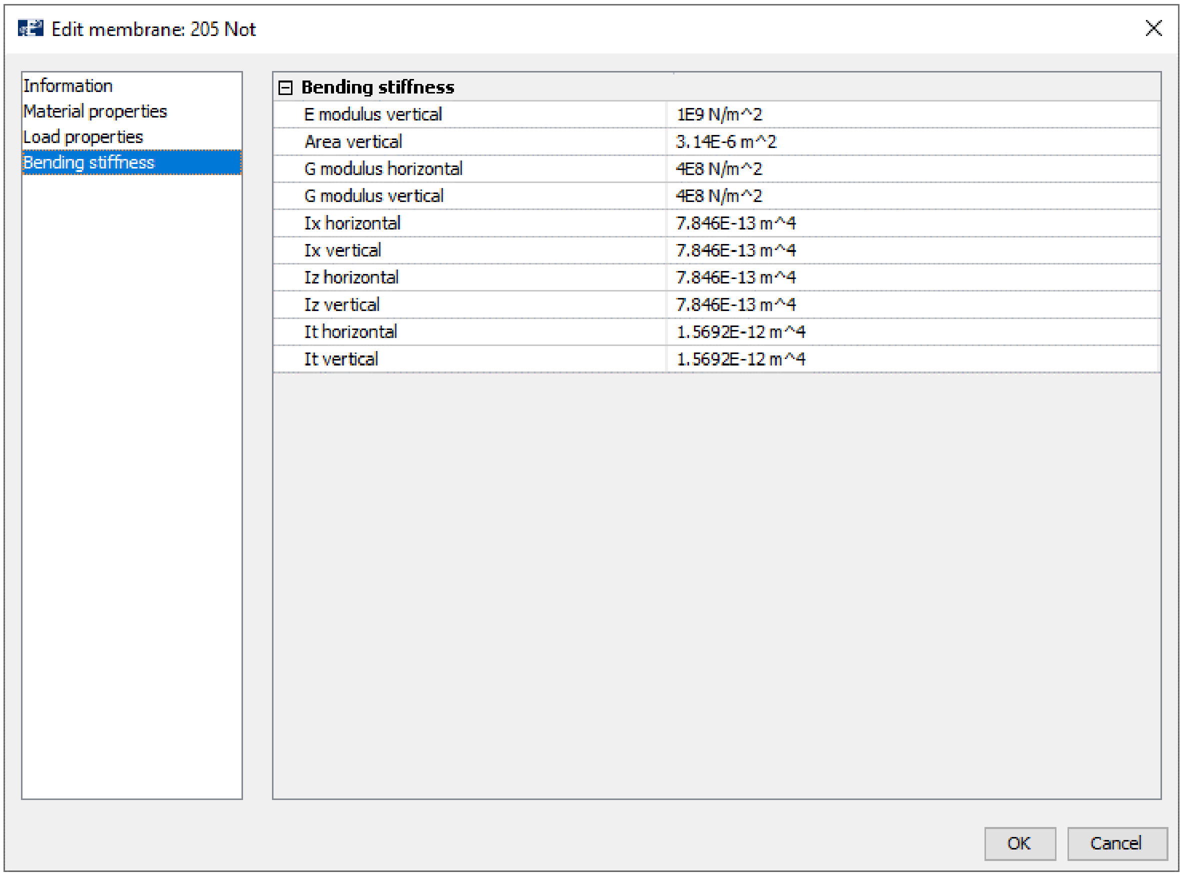


Figure 32 - Bending stiffness panel

Properties	Description
E-modulus vertical	Young's modulus for vertical threads (Young's modulus in the information tab is for horizontal threads).
Area vertical	Cross-sectional area for vertical threads (Area in the information tab is for horizontal threads).
G-modulus horizontal	Shear modulus for horizontal threads.
G-modulus vertical	Shear modulus for vertical threads.
I_x horizontal	Moment of area around local x-axis for horizontal threads.
I_x vertical	Moment of area around local x-axis for vertical threads.
I_z horizontal	Moment of area around local z-axis for horizontal threads.
I_z vertical	Moment of area around local z-axis for vertical threads.
I_t horizontal	Torsional stiffness for horizontal threads.
I_t vertical	Torsional stiffness for vertical threads.

Material properties for Type Shell

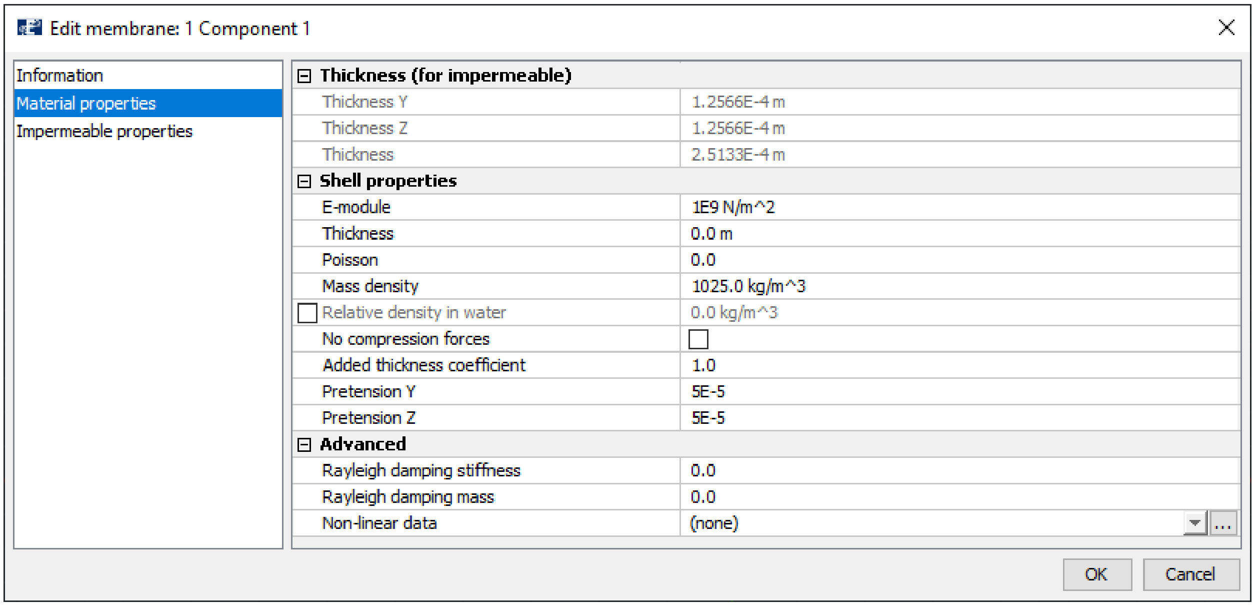


Figure 33 - Material properties for Type Shell

Figure 33 shows the input for type Shell.

Properties	**Description**
E-module	Young's modulus.
Thickness	Thickness of the shell plate.
Poisson	Poisson modulus
Mass density	Mass density of the shell plate.
No compression forces	Forces in the shell plate be 0 if compressed.
Added thickness coefficient	Coefficient to increase/decrease thickness, used for finding volume for the Froude Kriloff part of force in Morison.
Pretension Y	Elongation divided by length of shell plate in local z-direction.
Pretension Z	Elongation divided by length of shell plate in local y-direction.
Rayleigh damping stiffness	See Material/section properties for beam.
Rayleigh damping mass	See Material/section properties for beam
Non-linear data	Non-linear relation between thread strain and axial force. Read more about this below.

Non-linear data:

It is possible to establish non-linear relation between strain and axial force in threads. From the drop-down menu, select (new). The non-linear data table is presented below. The user can choose between isotropic and anisotropic behavior:

- Isotropic: relation between strain and axial force in local z-direction only.
- Anisotropic: relation between strain and axial force in both local z- and y-direction.

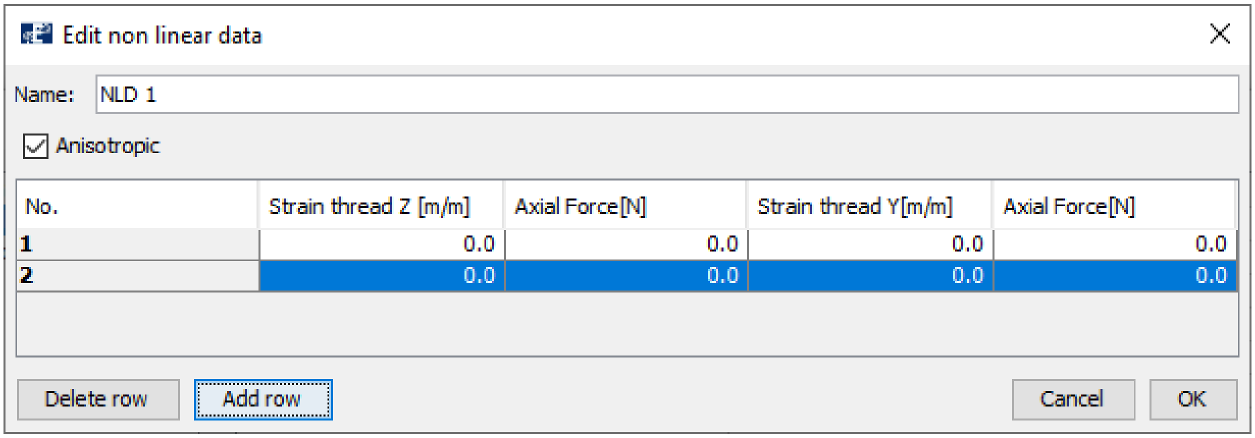


Figure 34 Non-linear data table for membrane Type Shell

Load properties for Regular net

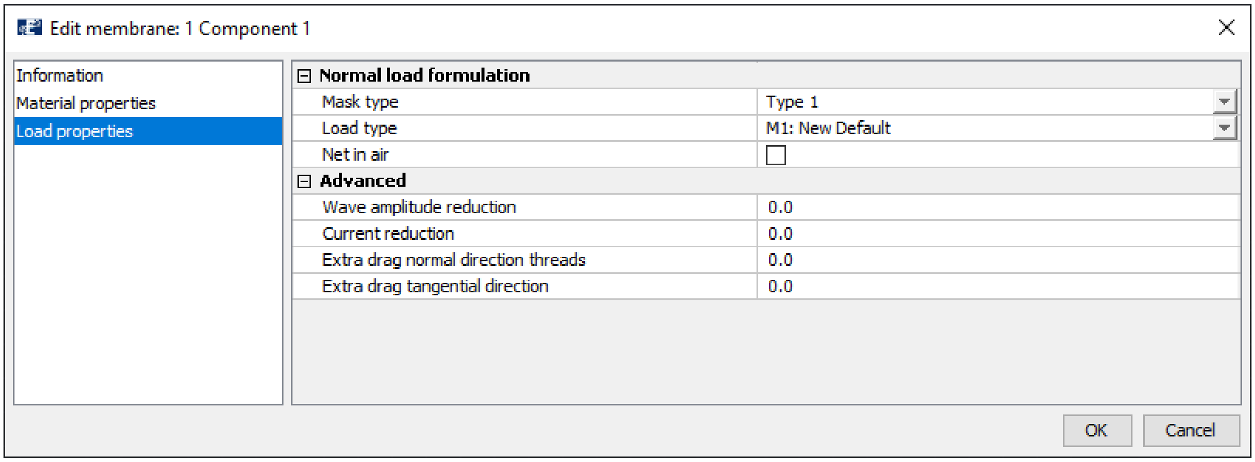


Figure 35 - Load properties, Regular net

Figure 35 shows the load properties for Regular net for types Normal and Normal with bending stiffness.

Properties	Description
Mask type	1 is square mask, 2 is diamond mask
Load type	M1: Drag force is calculated depending on the element's rotation relative to the current direction. M2: Same as M1, but drag coefficient is calculated based on the Reynolds number.
Net in air	Instead of drag from waves and current, drag will be calculated based on wind speed and density of air.
Wave amplitude reduction	Between 0 and 1, 0 is full wave and 1 is no wave. E.g. 0.5 means half wave amplitude.
Current reduction	Between 0 and 1, 0 is full current and 1 is no current. E.g. 0.5 means half current.
Extra drag normal direction threads	Factor that adds drag load on threads that are located "on the wheel" in normal direction.
Extra drag tangential direction	Factor that adds drag load on threads that are located "on the wheel" in tangential direction.

Load properties for Closed compartment

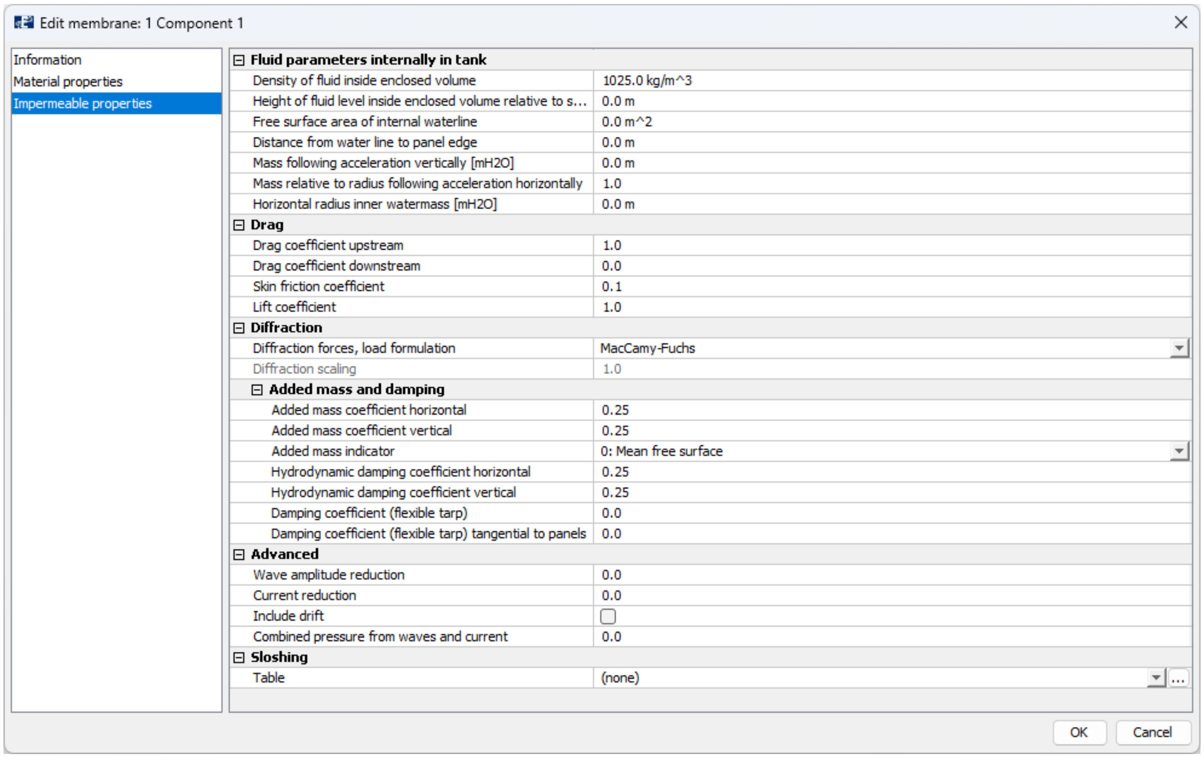


Figure 36 - Load properties, Closed compartment

Properties	**Description**
Fluid parameters internally in tank	
Density of fluid inside enclosed volume	If the bottom factor is 0.0 (e.g., lice skirt), this parameter is a dummy. In this case the internal static pressure is set equal to the external pressure. If the bottom factor is 1.0, then this parameter is the density of the fluid inside the tank. The external pressure is calculated based on the density of seawater, 1025 kg/m^3
Height of fluid level inside enclosed volume relative to sea level	This parameter is a dummy if the bottom factor is 0.0. If the bottom factor is 1.0 then this value is the static water level of the fluid inside the tank, relative to the outside water level. Positive value means that the inside water level is higher than the outside water level. This and inner waterplane area gives the extra volume.

Free surface area of internal waterline	This parameter is a dummy if the bottom factor is 0.0. When an impermeable net, where the fluid on the inside is separated from the fluid on the outside, is deformed it will change volume. This area is then the area the fluid is assumed to be pushed through when volume is changed.
Distance from water line to panel edge	Height of inside net, if 0.0 infinite height is set. If higher than 0.0 then the fluid inside can splash over the edge if it is compressed enough that it reaches over this set height.
Mass following acceleration vertically [mH2O]	mH2O of mass to be used as internal mass vibrating vertically.
Mass relative to radius following acceleration horizontally	Part of inner water mass inside tank to be used as added mass horizontally in the analysis. If this parameter is: 1, then 100% of the enclosed water volume vibrates as added mass horizontally, 0.2, then 20% of the enclosed water volume vibrates as added mass horizontally, 0, then 0% of the enclosed water volume vibrates as added mass horizontally, 1.2, then 120% of the enclosed water volume vibrates as added mass horizontally. More information is found in (Aquastructures, 2024a).
Horizontal radius inner watermass [mH2O]	AquaSim automatically detects the radius of the cylinder. If 0, then the automatically calculated radius is applied. If any other number, this is interpreted as the radius, in meters.
Drag	
Drag coefficient upstream	Gives the form drag to the object for the upstream part. Note that the drag load formulation is revised in the 2.20 version. For details, see (Aquastructures, 2024b).

Drag coefficient downstream	Gives the form drag to the object for the downstream part. Note that the drag load formulation is revised in the 2.20 version. For details, see (Aquastructures, 2024b).
Skin friction coefficient	Drag along the membrane elements.
Lift coefficient	Gives the lift coefficient on the membrane panel. Note that the drag load formulation is revised in the 2.20 version. For details, see (Aquastructures, 2024b).
Diffraction	
Diffraction forces, load formulation	Methods for calculating diffraction part of the pressure caused by waves. These are further explained in (Aquastructures, 2024b). **Flexible tarp**: load formulation adapted to flexible tarps and woven textiles, and the added mass and hydrodynamic damping factors are related to the volume of the tank. **MacCamy-Fuchs**: Diffraction loads are calculated from the MacCamy-Fuchs theory, and the added mass and hydrodynamic damping factors are related to the volume of the tank. **Numerical diffraction**: Diffraction loads are based on numerical calculation(“sink-source”). Added mass and hydrodynamic damping coefficients are in this case scaling factors used on the added mass and hydrodynamic damping found from the numerical diffraction theory. **Hybrid flexible tarp/numerical diffraction**: Combination of the Flexible tarp- and Numerical diffraction-load formulations. Added mass and hydrodynamic damping is calculated using the Numerical diffraction-theory. The user weights the methods through the “Diffraction scaling” option. **Hybrid flexible tarp/MacCamy-Fuchs**: Combination of the Flexible tarp- and MacCamy-Fuchs load formulations. The added mass and hydrodynamic damping factors are related to the volume of the tank. The user weights the methods through the “Diffraction scaling” option.

Diffraction scaling	Weight factor for the Hybrid flexible tarp/ numerical diffraction and Hybrid flexible tarp/ MacCamy-Fuchs formulations. Applying a value of 0.99 means that 99% of the diffraction force is calculated from either Numerical diffraction, or MacCamy-Fuchs. 1% is taken from the Flexible tarp-method.
Added mass and damping	
Added mass coefficient horizontal	**Flexible tarp:** Added mass for horizontal motion is calculated automatically for each element in the component group based on the 2D volume of the component group at the given depth, the element's horizontal distance from the centerline of the 2D volume at the given depth, the fluid density and scaled with the coefficient. **MacCamy-Fuchs**: Same as for Flexible tarp. **Numerical diffraction**: Added mass for horizontal motion is calculated automatically based on the Numerical diffraction-theory ("sink-source") and scaled with the coefficient. **Hybrid flexible tarp/numerical diffraction**: Same as for Numerical diffraction. **Hybrid flexible tarp/MacCamy-Fuchs:** Same as for Flexible tarp and MacCamy-Fuchs.

Added mass coefficient vertical	**Flexible tarp:** Added mass for vertical motion is calculated automatically for each element in the component group based on the 2D volume of the component group at the given depth, the element's horizontal distance from the centerline of the 2D volume at the given depth, the fluid density and scaled with the coefficient. **MacCamy-Fuchs**: Same as for Flexible tarp. **Numerical diffraction**: Added mass for vertical motion is calculated automatically based on the Numerical diffraction-theory ("sink-source") and scaled with the coefficient. **Hybrid flexible tarp/numerical diffraction**: Same as for Numerical diffraction. **Hybrid flexible tarp/MacCamy-Fuchs:** Same as for Flexible tarp and MacCamy-Fuchs.
Added mass indicator	Indicator of added mass in and out of water. **0**: Added mass both of fluid inside and outside are assumed distributed to the mean free surface and mass is distributed consistent (no mass in rotational DOFs). The mean free surface is the surface at the steady state condition where approximately 2/3 of the current velocity has been added to the system. **1**: Same as 0, but the mass is lumped to the translational DOFs of the nodes. **2**: In this case added mass for both inside and outside fluid is calculated to the actual water line during the simulation. This option can be suitable for a stiff cylinder going in and out of water in a rather wall sided manner. 2 means combining with consistent mass. **3**: Same as 2 but combining with lumped mass.

Hydrodynamic damping coefficient horizontal	**Flexible tarp:** Hydrodynamic damping for horizontal motion is calculated automatically for each element in the component group based on the 2D volume of the component group at the given depth, the element's horizontal distance from the centerline of the 2D volume at the given depth, the fluid density and scaled with the coefficient. **MacCamy-Fuchs**: Same as for Flexible tarp. **Numerical diffraction**: Hydrodynamic damping for horizontal motion is calculated automatically based on the Numerical diffraction-theory ("sink-source") and scaled with the coefficient. **Hybrid flexible tarp/numerical diffraction**: Same as for Numerical diffraction. **Hybrid flexible tarp/MacCamy-Fuchs:** Same as for Flexible tarp and MacCamy-Fuchs.
Hydrodynamic damping coefficient vertical	**Flexible tarp:** Hydrodynamic damping for vertical motion is calculated automatically for each element in the component group based on the 2D volume of the component group at the given depth, the element's horizontal distance from the centerline of the 2D volume at the given depth, the fluid density and scaled with the coefficient. **MacCamy-Fuchs**: Same as for Flexible tarp. **Numerical diffraction**: Hydrodynamic damping for vertical motion is calculated automatically based on the Numerical diffraction-theory ("sink-source") and scaled with the coefficient. **Hybrid flexible tarp/numerical diffraction**: Same as for Numerical diffraction. **Hybrid flexible tarp/MacCamy-Fuchs:** Same as for Flexible tarp and MacCamy-Fuchs.

Damping coefficient (flexible tarp)	Damping forces normal to membrane panel. Calculated based on the Flexible tarp load formulation and scaled with the coefficient. For details, see (Aquastructures, 2024b).
Damping coefficient (flexible tarp) tangential to panels	Damping forces tangential to membrane panel. The same damping as calculated for the normal direction based on the Flexible tarp load formulation, is applied in the tangential direction and scaled with the coefficient. For details, see (Aquastructures, 2024b).
Advanced	
Wave amplitude reduction	Reduction of wave amplitude on element, due to e.g., shadow effects. Input is a number between 0.0 and 1.0. 0 corresponds to 0% reduction, 1.0 corresponds to 100% reduction
Current reduction	Reduction of current on element due to e.g., shadow effects. Input is a number between 0.0 and 1.0. 0 corresponds to 0% reduction, 1.0 corresponds to 100% reduction.
Include drift	To include drift forces to the loads, “With slamming” under the Advanced tab in the Export dialog must be used. This element is calculated with forces to the actual water line, meaning that parts of drift forces are included even if this is not checked.
Combined pressure from waves and currents	The way current and wave loads are combined for calculating pressure from current. 0 means that the “raw” velocity at each element is used as basis such that the pressure is found from the pressure coefficient multiplied with relative velocity at the element. 1 means that the relative velocity is averaged over the elements at similar vertical location. In between means that the effects are weighted.
Sloshing	Edit/Add a sloshing table to the net.

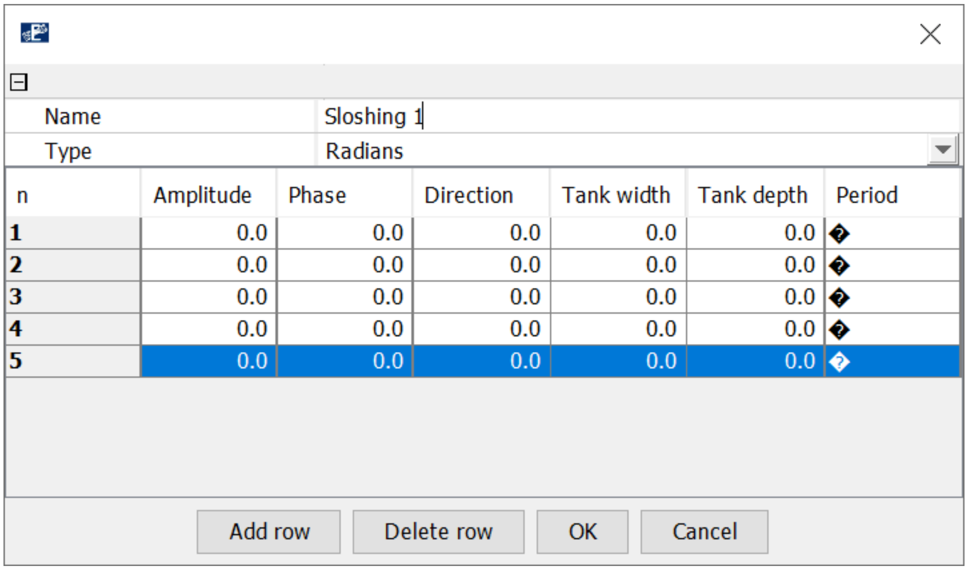


Figure 37 – Sloshing

Properties	**Description**
Type	Type of angles connected to the wave phase and direction: choose between radians or degrees.
n	The n-th sloshing wave. Each row represent a regular sloshing wave.
Amplitude	Amplitude of sloshing wave.
Phase	Wave phase in time relative to Sinus time. If set to -10 or less, then sloshing load is put on statically.
Direction	Wave direction relative to x-axis.
Tank width	Effective tank width.
Tank depth	Effective tank depth.
Period	Period of the sloshing wave [s]. Auto-calculated based on wave amplitude, tank width and tank depth.

For a more detailed description of membrane parameters and its theory, see the report (Aquastructures, 2024a).

Load properties for Lice skirt

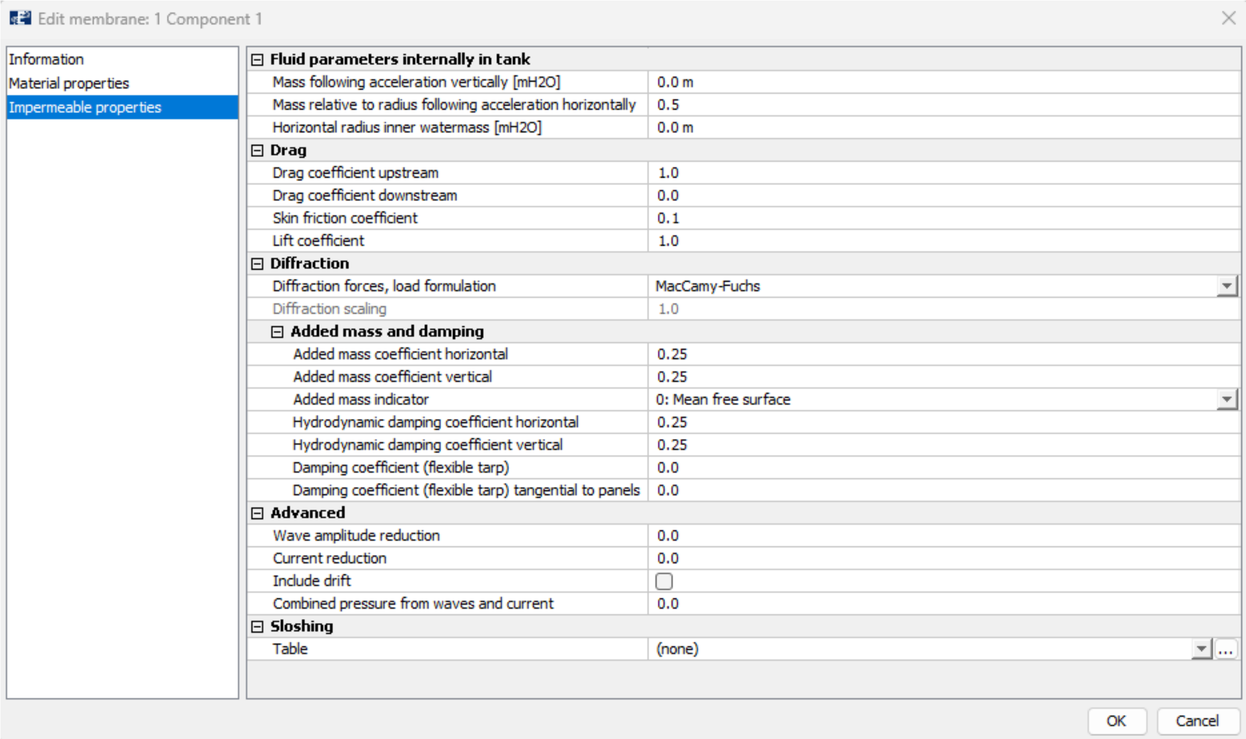


Figure 38 – Load properties, Lice skirt

Properties	**Description**
Fluid parameters internally in tank	
Mass following acceleration vertically [mH2O]	See description in section “Load properties for Closed compartment.
Mass relative to radius following acceleration horizontally	See description in section “Load properties for Closed compartment.
Horizontal radius inner watermass [mH2O]	See description in section “Load properties for Closed compartment.
Drag	
Drag coefficient upstream	See description in section “Load properties for Closed compartment.
Drag coefficient downstream	See description in section “Load properties for Closed compartment.
Skin friction coefficient	See description in section “Load properties for Closed compartment.
Lift coefficient	See description in section “Load properties for Closed compartment.
Diffraction	See description in section “Load properties for Closed compartment.
Diffraction forces, load formulation	See description in section “Load properties for Closed compartment.
Diffraction scaling	See description in section “Load properties for Closed compartment.

Added mass and damping	
Added mass coefficient horizontal	See description in section “Load properties for Closed compartment.
Added mass coefficient vertical	See description in section “Load properties for Closed compartment.
Added mass indicator	See description in section “Load properties for Closed compartment.
Hydrodynamic damping coefficient horizontal	See description in section “Load properties for Closed compartment.
Hydrodynamic damping coefficient vertical	See description in section “Load properties for Closed compartment.
Damping coefficient (flexible tarp)	See description in section “Load properties for Closed compartment.
Damping coefficient (flexible tarp) tangential to panels	See description in section “Load properties for Closed compartment.
Advanced	
Wave amplitude reduction	See description in section “Load properties for Closed compartment.
Current reduction	See description in section “Load properties for Closed compartment.
Include drift	See description in section “Load properties for Closed compartment.
Combined pressure from waves and currents	See description in section “Load properties for Closed compartment.
Sloshing	See description in section “Load properties for Closed compartment.

Load properties for Morison free plate

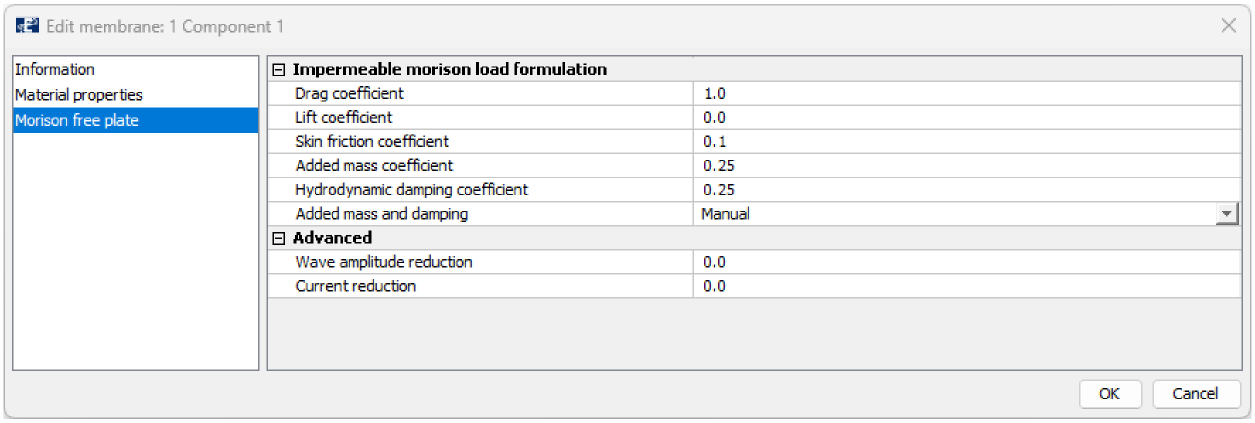


Figure 39 – Load properties, Morison free plate

Morison free plate uses the Morison load formulation to calculate the load applied to the membrane elements.

Properties	Description
Drag coefficient	Drag coefficient on membrane, normal direction. Used to calculate force normal to membrane due to flow normal to membrane.
Lift coefficient	Lift coefficient on membrane, normal direction. Used to calculate force normal to membrane due to flow tangential to membrane.
Skin friction coefficient	Skin friction coefficient on membrane, tangential direction. Used to calculate force tangential to membrane due to flow tangential to membrane. Skin friction force is due to viscous drag in the boundary layer on the membrane.
Added mass coefficient	Added mass coefficient for the membrane, normal direction. Note that the unit is in [m]. If “Added mass and damping” in AquaEdit is set to “Automatic”, then this parameter is a unitless scaling factor. For details, see (Aquastructures AS, 2024d).
Hydrodynamic damping coefficient	Hydrodynamic damping coefficient for the membrane, normal direction. Note that the unit is in [m/s]. If “Added mass and damping” in AquaEdit is set to “Automatic”, then this parameter is a unitless scaling factor. For details, see (Aquastructures AS, 2024d).

Added mass and damping	Automatic: added mass and hydrodynamic damping forces are calculated automatically based on the Numerical diffraction-theory (“sink-source”). The “Added mass coefficient” and the “Hydrodynamic damping coefficient” are in this case used as unitless scaling factors. Manual: added mass and hydrodynamic damping are calculated manually based on the “Added mass coefficient” and the “Hydrodynamic damping coefficient” respectively and multiplied with the density of seawater and the membrane area.
Advanced	
Wave amplitude reduction	See description in section “Load properties for Closed compartment.
Current reduction	See description in section “Load properties for Closed compartment.

Load properties for Surface tarpaulin

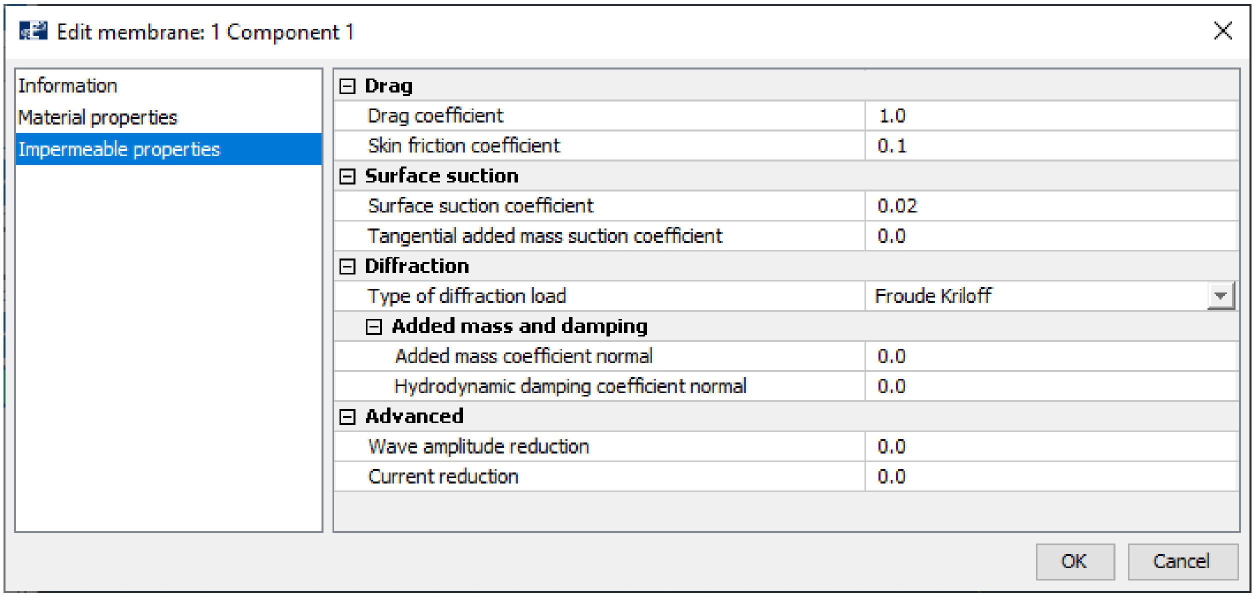


Figure 40 – Load properties, Surface tarpaulin

Properties	Description
Drag	
Drag coefficient	See description in section "Load properties for Morison free plate"
Skin friction coefficient	See description in section "Load properties for Morison free plate"
Surface suction	
Surface suction coefficient	Damping coefficient, acting in the tangential direction, representing how strongly the tarpaulin sticks to the below water particles parallel to the tarpaulin. For details, see (Berstad & Grøn, Model basin testing of a floating solar system compared to analysis for establishment of design verification culture, 2023).
Tangential added mass suction coefficient	Added mass coefficient, acting in the tangential direction, representing the mass of the below water particles moving with the tarpaulin. For details, see (Berstad & Grøn, Model basin testing of a floating solar system compared to analysis for establishment of design verification culture, 2023).

Diffraction	
Type of diffraction load	Methods for calculating diffraction part of the pressure caused by waves. These are further explained in (Aquastructures, 2024b). **Froude Kriloff:** when forces are calculated by “Froude-Kriloff” this means that pressure to the surface tarpaulin is calculated from the undisturbed incident wave, and it is assumed that the tarpaulin causes negligible diffraction. Added mass and hydrodynamic damping are calculated manually based on the “Added mass coefficient normal” and the “Hydrodynamic damping coefficient normal” respectively and multiplied with the density of seawater and the membrane area. **Numerical diffraction**: Diffraction loads are based on Numerical diffraction-theory (“sink-source”). The “Added mass coefficient normal” and the “Hydrodynamic damping coefficient normal” are in this case used as unitless scaling factors.
Added mass and damping	
Added mass coefficient normal	Added mass coefficient for the membrane, normal direction. Note that the unit is in [m], when “Type of diffraction load” is set to “Froude Kriloff”. If “Type of diffraction load” is set to “Numerical diffraction”, then this parameter is a unitless scaling factor.
Hydrodynamic damping coefficient normal	Hydrodynamic damping coefficient for the membrane, normal direction. Note that the unit is in [m/s], when “Type of diffraction load” is set to “Froude Kriloff”. If “Type of diffraction load” is set to “Numerical diffraction”, then this parameter is a unitless scaling factor.
Advanced	
Wave amplitude reduction	See description in section “Load properties for Closed compartment.
Current reduction	See description in section “Load properties for Closed compartment.

2.14.6 Explicit membrane

The Membrane X component type (explicit membrane) is a specialized type of membrane that does not automatically detect its membrane elements based on surrounding elements. Each quad is explicitly drawn. It also allows for drawing triangles.

Membrane X can be accessed by right click a new component in the Components window > Type > Membrane X.

It is possible to convert a normal membrane to an explicit one. All pre-existing elements will be converted to explicit membrane elements.

Other advantages of using explicit membranes are to avoid collision with other components, leading to erroneous membrane element generation. This can happen with e.g., ropes attached to membranes or attachments to floating elements.

Explicit membranes are also much faster when dealing with larger models.

See chapters 3.18, 3.20 and 5.4 for more information on how to use the explicit membrane type.

2.14.7 Hexagonal/ 6-sided membrane

Note: Membrane 6 components are still under development and testing, any functionality may change in future versions.

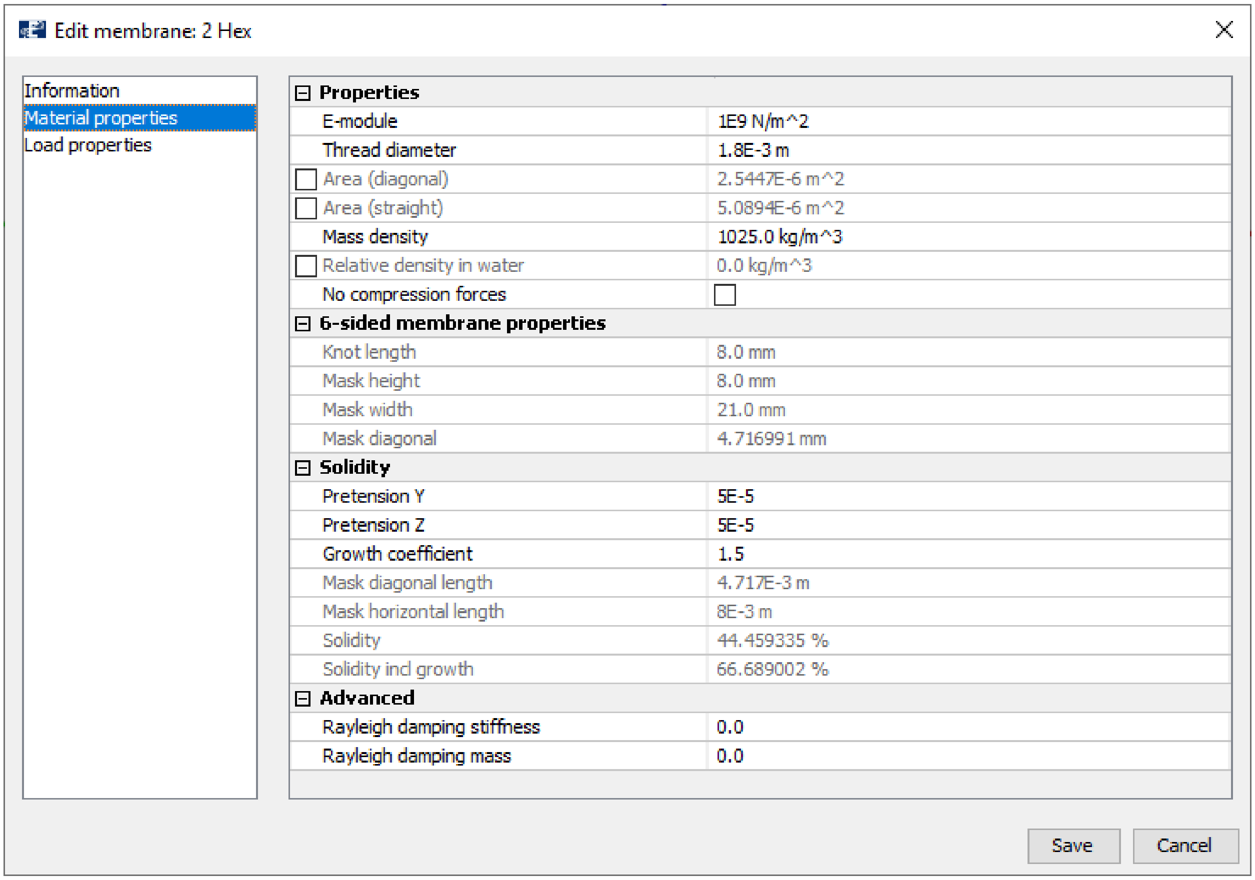


Figure 41 - Extra properties for Membrane 6 components

Hex nets are nets with hexagonal (6) sides. They are modelled with 6 sides where each side will represent several small hexes.

Note: When modelling hexes every modelled hex must have the same form. Small deviances can be tolerated, but the more skewed the worse the results will be.

See chapter Generate hex to learn more about 6-sided membranes and how to generate them.

2.14.8 Truss

Truss has a lot in common to beam, the main difference is that a truss element does not have bending or torsional resistance.

Information

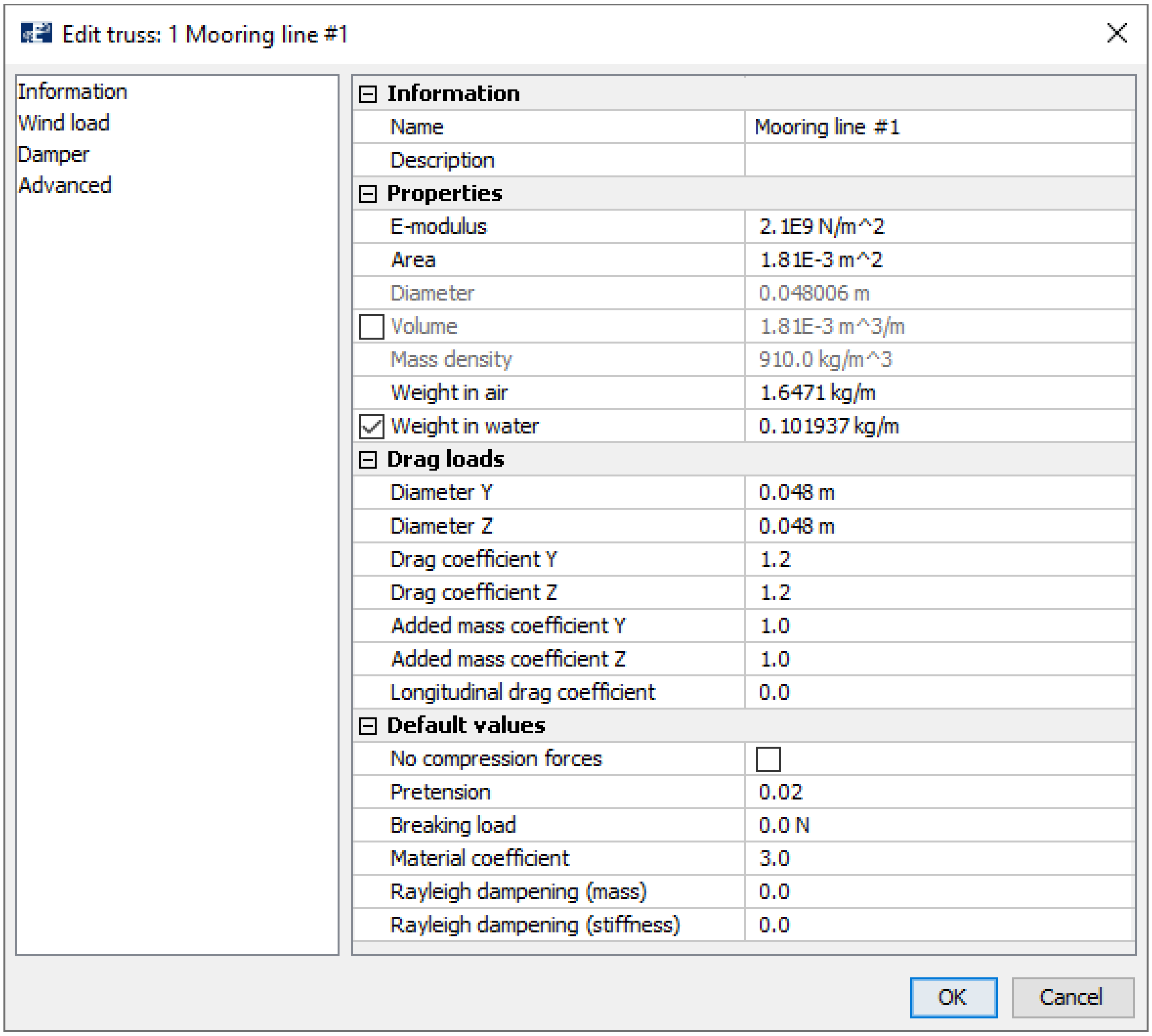


Figure 42 - Information panel

Properties	Description
Name	Name of the component.
Description	Description of the component.
E-modulus	Young’s modulus.
Area	Cross-sectional area.
Diameter	Cross-sectional diameter. Calculated based on the input area.
Volume	Volume of truss, default value equals area.
Mass density	Mass density.
Weight in air	Weight when not submerged.

Weight in water	Weight when submerged, default value Weight in air – (volume * density of water).
Diameter Y/Z	Drag area is calculated from diameter * length of element.
Drag coefficient Y/Z	Drag coefficients for local y- and z-axis
Added mass coefficient Y/Z	Added mass coefficients for local y- and z-axis
No compression forces	Use this if truss elements should not take compression forces.
Pretension	Pre-strain, elongation divided by the length of the element. Positive value will shorten the element, giving stretch forces.
Breaking load	Breaking load of truss for calculating utility factor.
Material coefficient	Material factor for truss for calculating utility factor.
Rayleigh damping (mass/stiffness)	Same as for beam, see chapter Material/section properties.
Longitudinal drag coefficient	Friction coefficient for drag in local x-axis.

Wind load is the same as for beam, see chapter Wind load for more details.

Damper

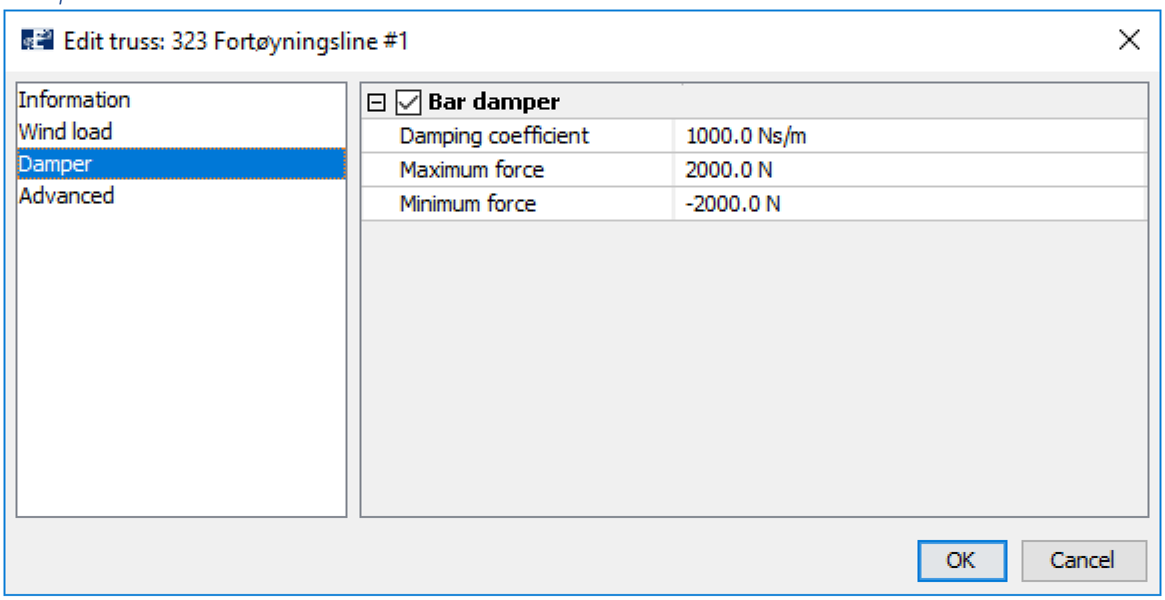


Figure 43 - Damper panel

Activate this if component is a bar damper. A bar damper is an element where the internal force is proportional to the relative velocity between the elements’ endpoints instead of the relative position. The maximum and minimum force values put a “cap” on the internal forces in the bar such that the internal forces stay within the limit also if forces reach above these values.

Advanced

It is possible to add a non-linear table to the truss. More information about non-linear tables, see chapter Node2Node spring. Note: Only Strain and Axial Force is available for non-linear data on trusses.

2.14.9 Node2Node spring

Node2Node is a component type. Two types of springs are available: “Node to node spring” and “Node to node damping”. The “Node to node spring” is a spring connected between two nodes with translatory stiffness and rotational stiffness.

“Node to node damping”, provides translatory damping and rotational damping.

For linear springs the relation between force and stiffness is given as F = k*r, which is Hooke’s Law. The springs can also be non-linear, where the user needs to add a non-linear table instead in the field “Non linear data” (see below). All non-linear tables can be found in **`Tools > Show non linear data tables`**.

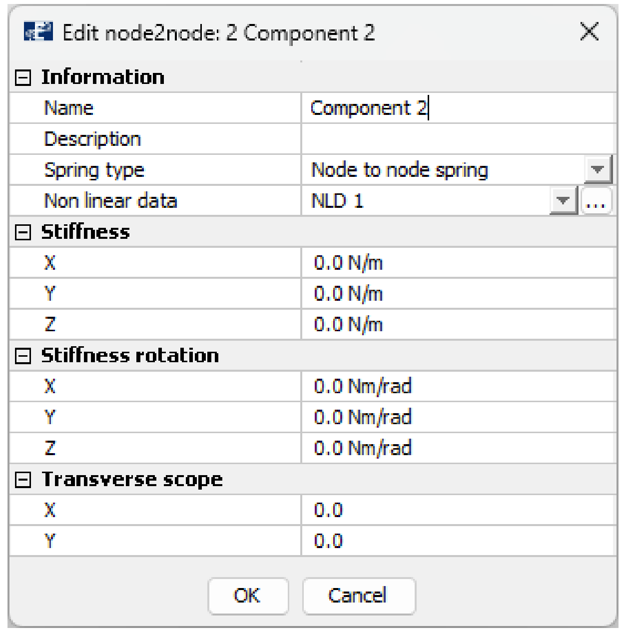


Figure 44 - Properties for node2node spring component

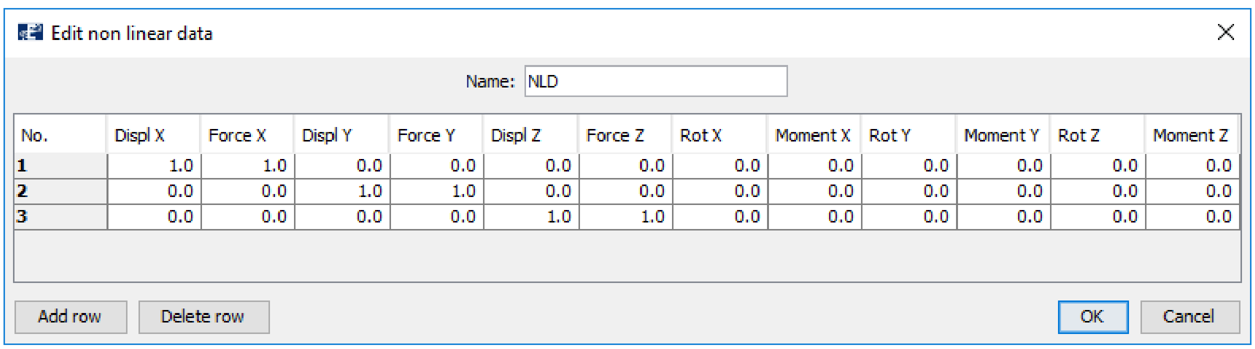


Figure 45 - Properties for non-linear data

The input here is a table where the user can input a relation between displacement and force, and relation between moment and rotation.

3. Functions and intents

As mentioned in chapter Terminology, an intent is a function in AquaEdit e.g. Draw line, Split line, Measure angle etc.

3.1 Common functionality

3.1.1 Node snapping

The snap function is toggled by pressing the Snap button on the toolbar. If this button is highlighted, most drawing functions will attempt to snap to a nearby node when drawing. If the distance to the closest node is too large, the attempt will fail, and nothing will happen.

If the user holds down ALT whilst clicking the snap function is inversed.

3.1.2 Position snapping

Snapping can also be done against the nearest corner of the grid by holding down SHIFT, snap must be disabled for this to work.

3.1.3 Keyboard input

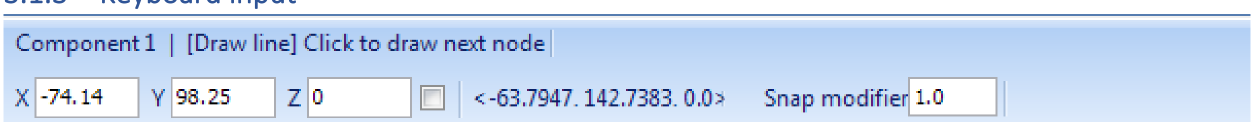


Figure 46 - Keyboard input fields

By filling in the X, Y and Z fields and then pressing ENTER the keyboard input will be used instead of a mouse click. Keyboard input is not affected by node snapping or position snapping.

When using the keyboard input the X, Y and Z fields can also be made relative from the last used position by holding SHIFT as the ENTER key is pressed, or by enabling the relative position indicator.

3.1.4 Calculator in input fields

Some input fields can use a calculator to calculate values, usually valid in the fields for beam, truss, membrane and node2node components where the input values are of floating numbers. To enable the calculator, write “=” at the start of the input. Figure 47 shows an example in the field for E-modulus for a truss component. At the right end of the field there will be a preview of what the final calculation will be. The functions available now are:

+, -, *, /, ^, sin(radians), cos(radians), sindeg(degrees), cosdeg(degrees), pi, abs(number), max(number1 number2), min(number1 number2). There is also round(number) which will round the number in the parenthesis to the lowest integer, i.e. round(2*cosdeg(45)) will return 1 since the number within the parenthesis will be 1.4142.

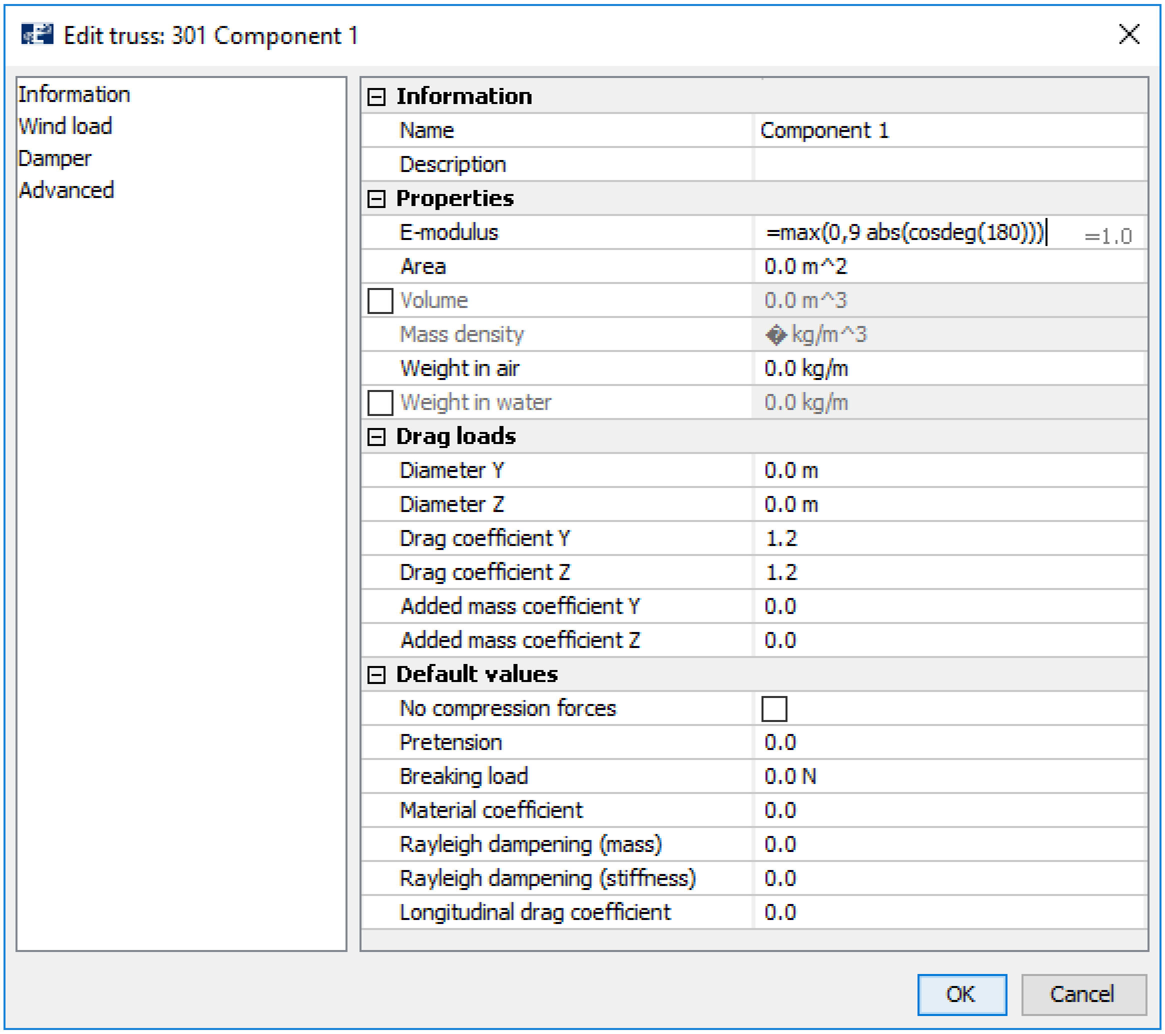


Figure 47 - Example of calculator in input field

3.2 Draw line

The most basic function draws a line between two new or existing nodes. By default, the line will be drawn continuous.

Single line mode: This will automatically start a new non-connected line after each line is drawn.

Input heading: After the first node is drawn the user can input a heading (anti-clockwise from the x-axis) and a length to automatically create a line based on this.

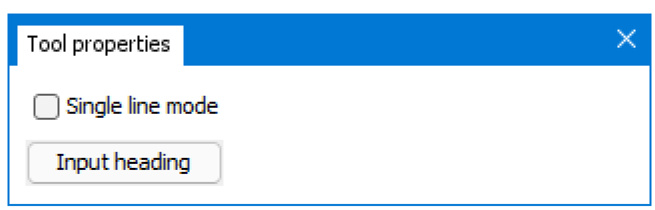


Figure 48 - Draw line properties

A line can be cancelled by pressing ESC (pressing ESC while before the first node is created will cancel the intent and revert to the Pointer intent). This will cancel the current line and let the user select a new starting node for the line.

By holding CTRL when clicking (while snap is off) the intent will split the closest line (if there is any) and create a new node her, joining the three lines.

3.3 Draw rectangle

Draw rectangle takes 3 nodes and creates a rectangle. These nodes can be either drawn or snapped using the snap functions available.

Holding down CTRL when dragging/selecting the third node will project the rectangle so an equilateral rectangle is created instead of (potentially) a rhombus, see Figure 49.

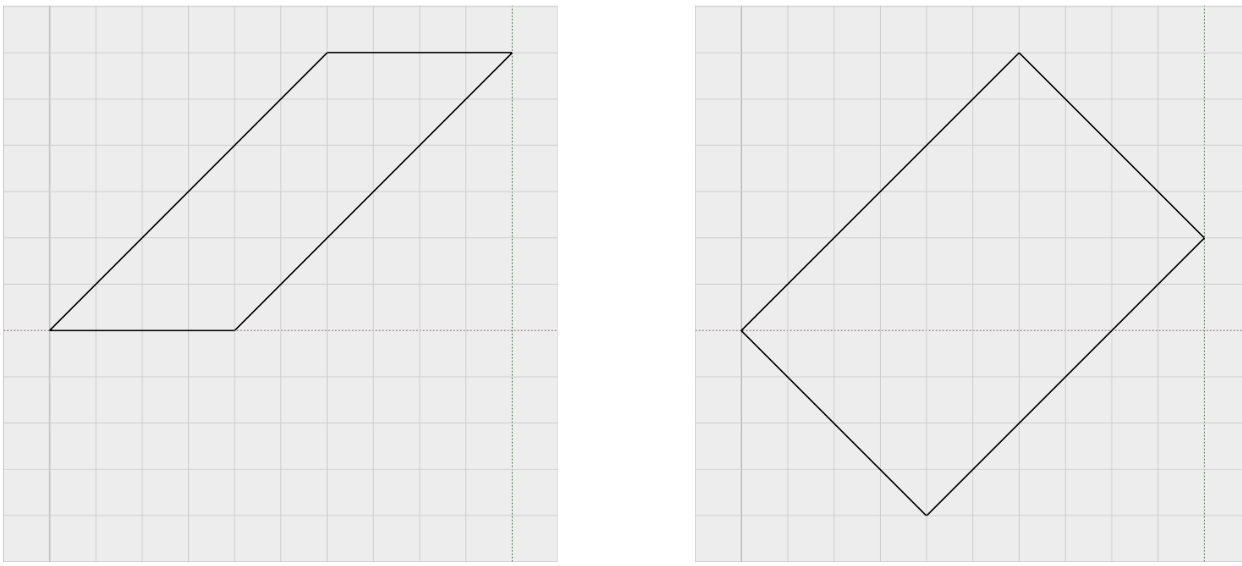


Figure 49 - Rectangle drawn as a rhombus on the left, rectangle drawn as an equilateral rectangle on the right

3.4 Draw grid

A rectangle can be drawn pre-spliced into smaller rectangles – a grid. This is especially useful for square or odd-shaped components. Compatible for all component types.

The number of slices in the two directions is given as input in the tool properties window. Any lines parallel to the outside lines of the rectangle will be split to match the intersections of the segments inside the rectangle.

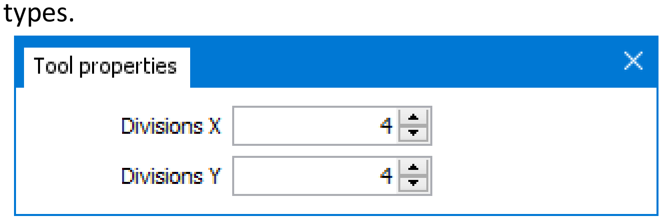


Figure 50 - Setting divisions of a membrane rectangle

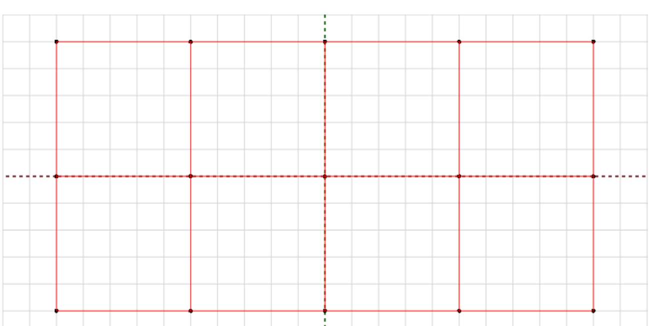


Figure 51 – A 2 x 4 grid

3.5 Draw circle

Circles in AquaEdit will be divided into elements with equal lengths. Using the draw intent, the user can input the radius and the number of segments the user wants the circle divided into. The circumference will update automatically based on the radius, the user can also input the circumference directly instead of the radius.

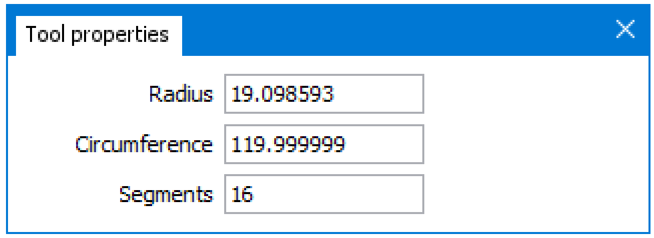


Figure 52 - Properties for drawing circles

The circle will be drawn with the given input when the user left clicks on the canvas or using keyboard input described in chapter Keyboard input. The circle will be drawn in the view that is currently active, view ISO will draw the circle in the x-y-plane.

Using the mouse, the user can also drag the size of the circle by holding down the left mouse button on the canvas. This will snap to nodes if node snapping is enabled. The radius can be made to match the distance between the center of the circle and a previously drawn node if snap is one when the user releases the mouse button. A live preview will be visible when dragging the mouse.

3.6 Draw tube

Drawing intent that creates tube for explicit membranes (Membrane X) and truss elements. This intent is similar as Draw circle. In addition, the user inputs the length of the tube (i.e., the height), number of segments in the length-direction. An option for different radius at the bottom of the tube may be activated by toggle the Bottom radius.

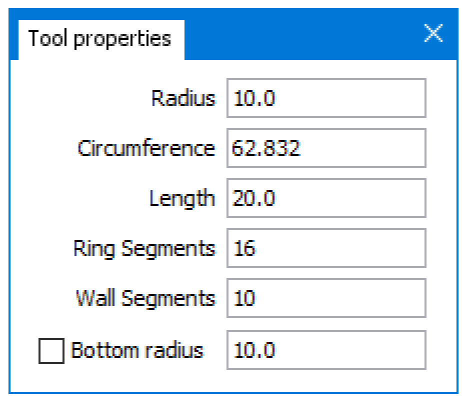


Figure 53 - Properties for drawing tube

The tube will be drawn with the given input when the user left clicks in the canvas (3D view) or using keyboard input as described in chapter Keyboard input. The tube will be drawn in the view that is currently active, with the Length-axis pointing normal to the drawing view. For ISO- and XY-view the length axis is along the Z-axis.

3.7 Move

Selected nodes, elements and quads can be moved by using the move intent. To enable moving, either nodes or elements must be selected, then the user selects a “source” node for the move operation. Selecting a “source” node is done by clicking on or near a node that is part of the selection that is to be moved. If the selection is moved while snapping is enabled the “source” node will be the part that is snapped to another node.

Elements will be disconnected from the original position, while moving nodes will keep any line connections they may have had (see Figure 54).

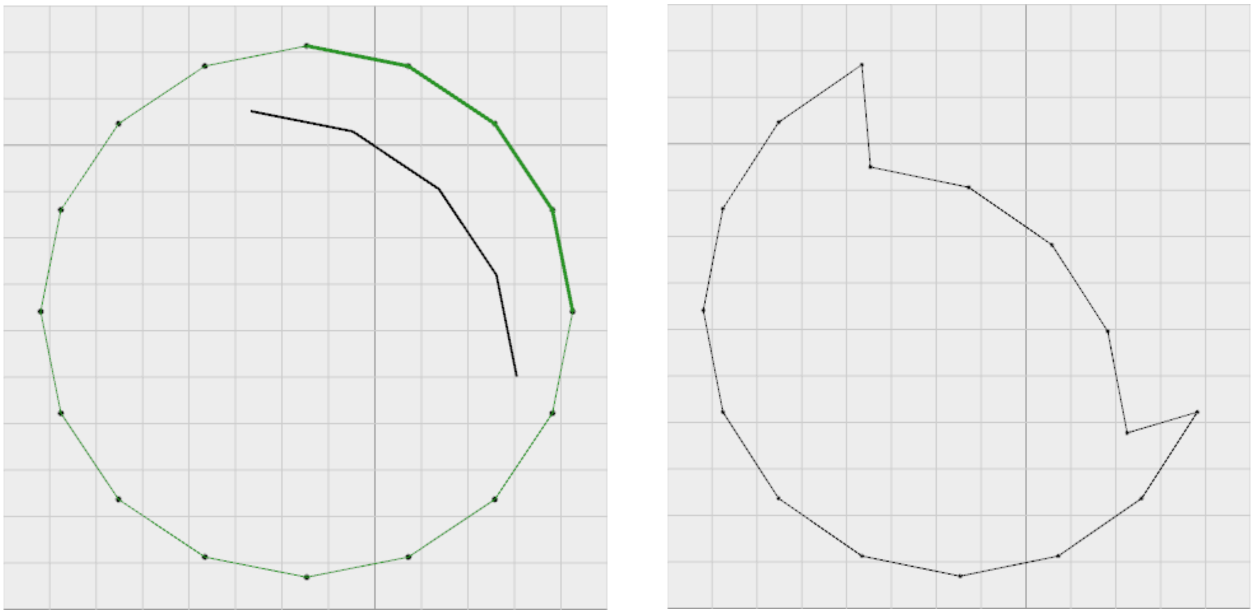


Figure 54 - Moved elements on the left, moved nodes on the right

Keyboard input can also be used to move the selection.

Selected elements can be copied instead of moved; this is done by holding down CTRL when clicking the selections into place.

3.8 Rotate

To rotate a selection a pivot point (the point of rotation) and the rotation in degrees must be given as input. When the rotate intent is activated, the pivot point will be set to the center of the selection by default.

The pivot point can be changed by either entering the point manually in the properties windows or by mouse click. If mouse click is used, then the current snap status will be used. Use mouse click with snap activated if it is required to rotate around an existing node.

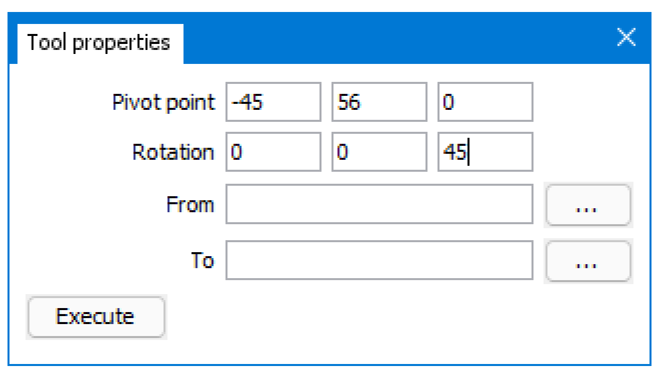
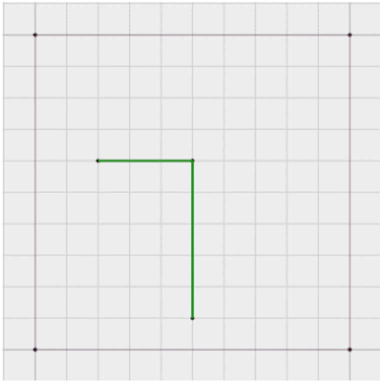


Figure 55 - Properties for rotate



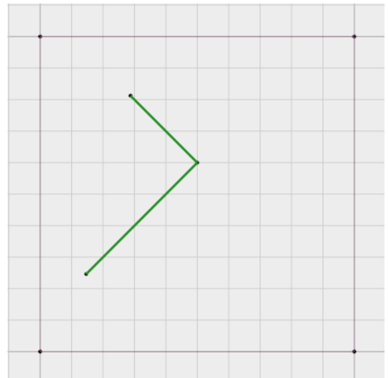


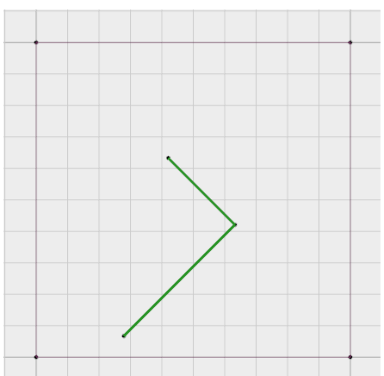
Figure 56 – Left: Before rotation. Middle: Rotation around center of selection. Right: Rotation around node

Note that rotation is done from left to right, though it is best to use multiple rotations if more than one axis is needed for the rotation. The rotation input is for the opposite axis of the view, i.e. when in top view (x, y) a rotation of 0, 0, 45 will rotate the selection 45 degrees clockwise when looking straight down (seeing "into" the z-axis).

Together with Pivot point the selections "From" and "To" creates a local rotation plane that can be applied if not the default global planes are appropriate.

3.9 Scale

When using the scale intent, the mouse can be dragged to scale the selection in the current view (i.e. the x- and y-axis when in top view). In the scale intent the pivot point will be set to the center of the selection by default.



Alternatively, the keyboard input can be used to scale the selection in all three axes. If scaling in an axis is unwanted, use 1 for no scaling.

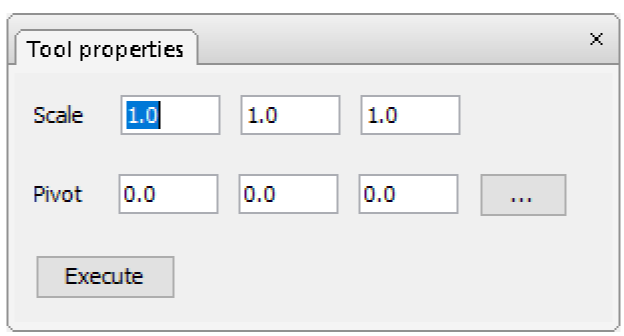


Figure 57 - Properties for scaling

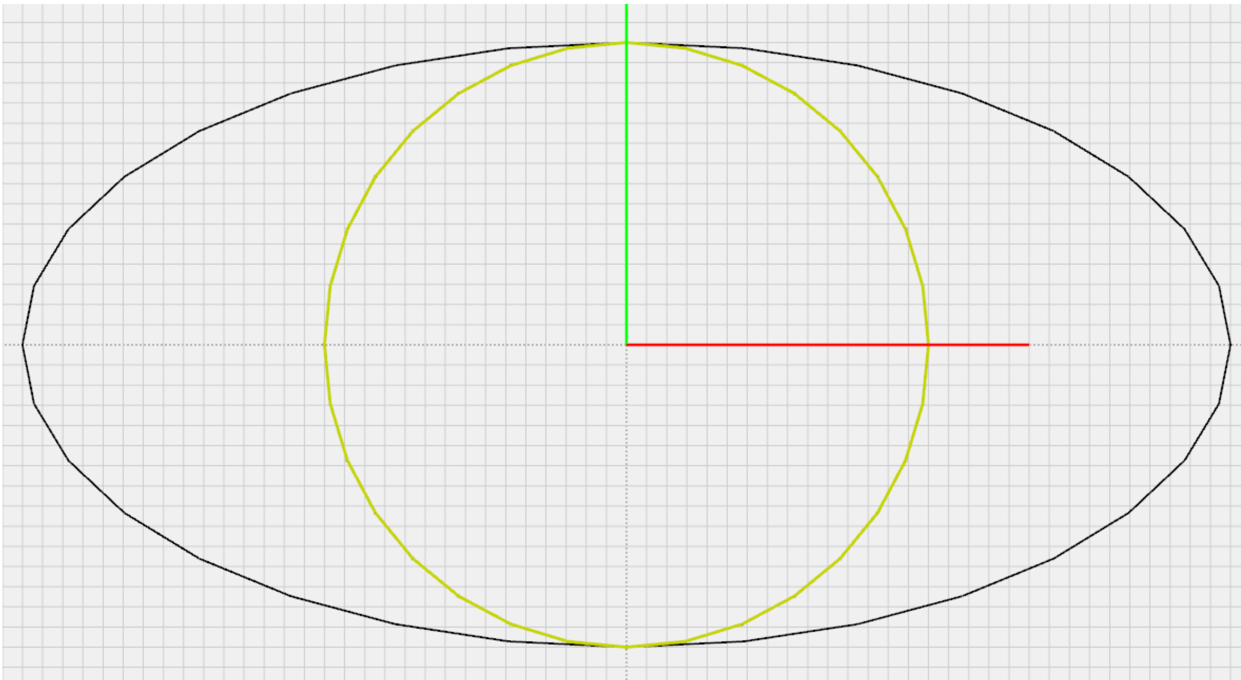


Figure 58 - Scaling of a circle

Figure 58 shows how scaling of a circle will look like if pivot point is in the center of the circle and the scaling is 1 for y- and z-axis while it is 2 for x-axis.

3.10 Duplicate

A selection can be duplicated and transformed using the duplicate intent. Note: Extrude intent works in the same way, the difference being that duplicated elements will become connected with new lines between them.

Segments

How many duplicates to make.

Vector

The vector distance (translation) of each duplicated segment.

Scale

Optionally scale the selection for each segment.

Rotate

Optionally rotate the selection for each segment, input in degrees.

Alternatively rotate around a pivot point.

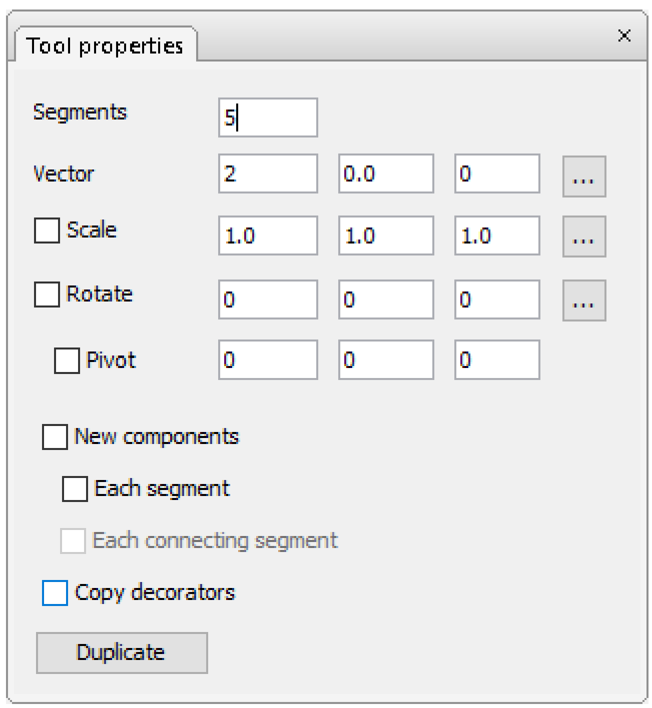


Figure 59 - Properties for duplicate

New components
This option will put the duplicated segments in its own component.

Each segment
This option will put each duplicated segment in its own component.

Copy decorators
Copy decorators from the original selection to the duplicated selection.

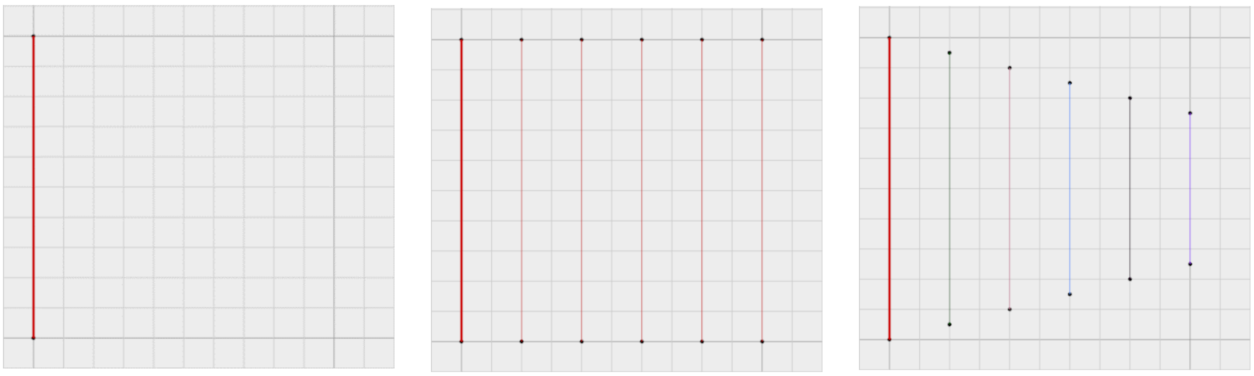


Figure 60 - Left: Before duplicate. Middle: With vector translation. Right: With vector translation, scaling and new components

3.11 Extrude
For the functionality, see chapter about duplicate above. Note: Vector is here based on the origin of the segment before.

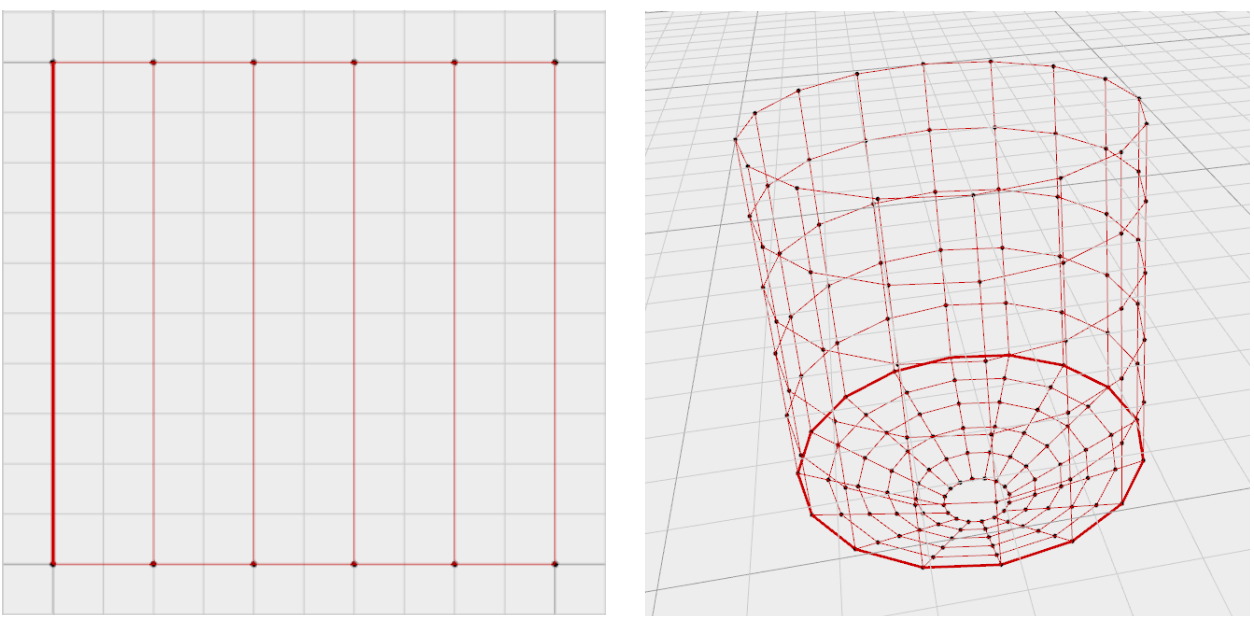


Figure 61 - Left: Exruded segments with connecting segments. Right: Two extrude operations, the second with scale 0.1, 0.1, 1

Each connecting segment
Only available when using extrude. Each connecting segment will be put in its own component.

3.12 Mirror

When mirror a selection a mirror line must be given as input. This can be done by entering the points manually or by using the mouse. When using the mouse; first click gives the first point and second click gives second point. A mirror line will be made from these points when executing this intent.

The mirror line can be snapped by either using node snapping or position snapping.

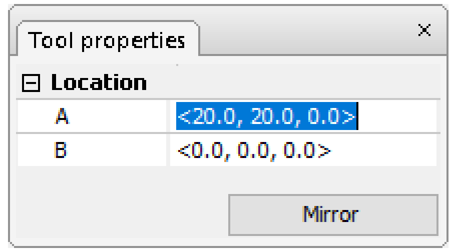


Figure 62 - Properties for mirror

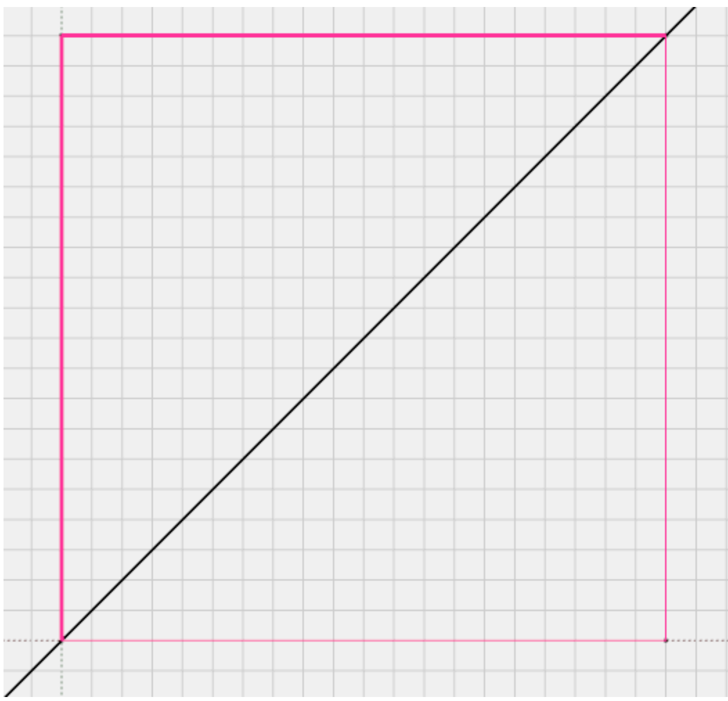


Figure 63 - Mirroring two lines to make a square. Mirror line given by using snap on the two end nodes on the existing line

3.13 Split line

Existing lines can be split using the split line intent. Input is the number of segments or a distance from an existing node.

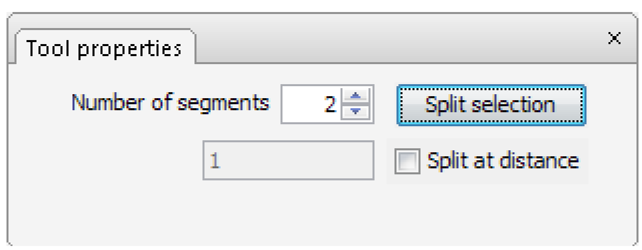


Figure 64 - Properties for split line

There are four ways to split lines.

1. Using "Split selection" to split the selected elements.
2. Clicking on an element will split the line in the segments given in the input, provided that "Split at distance" is not checked.
3. Clicking on an element with "Split at distance" checked will split the element in two with the new line having a distance equal to input. The new line will be created to the closest node of the mouse click
4. Holding CTRL and using the mouse will create new nodes where the mouse pointer is.

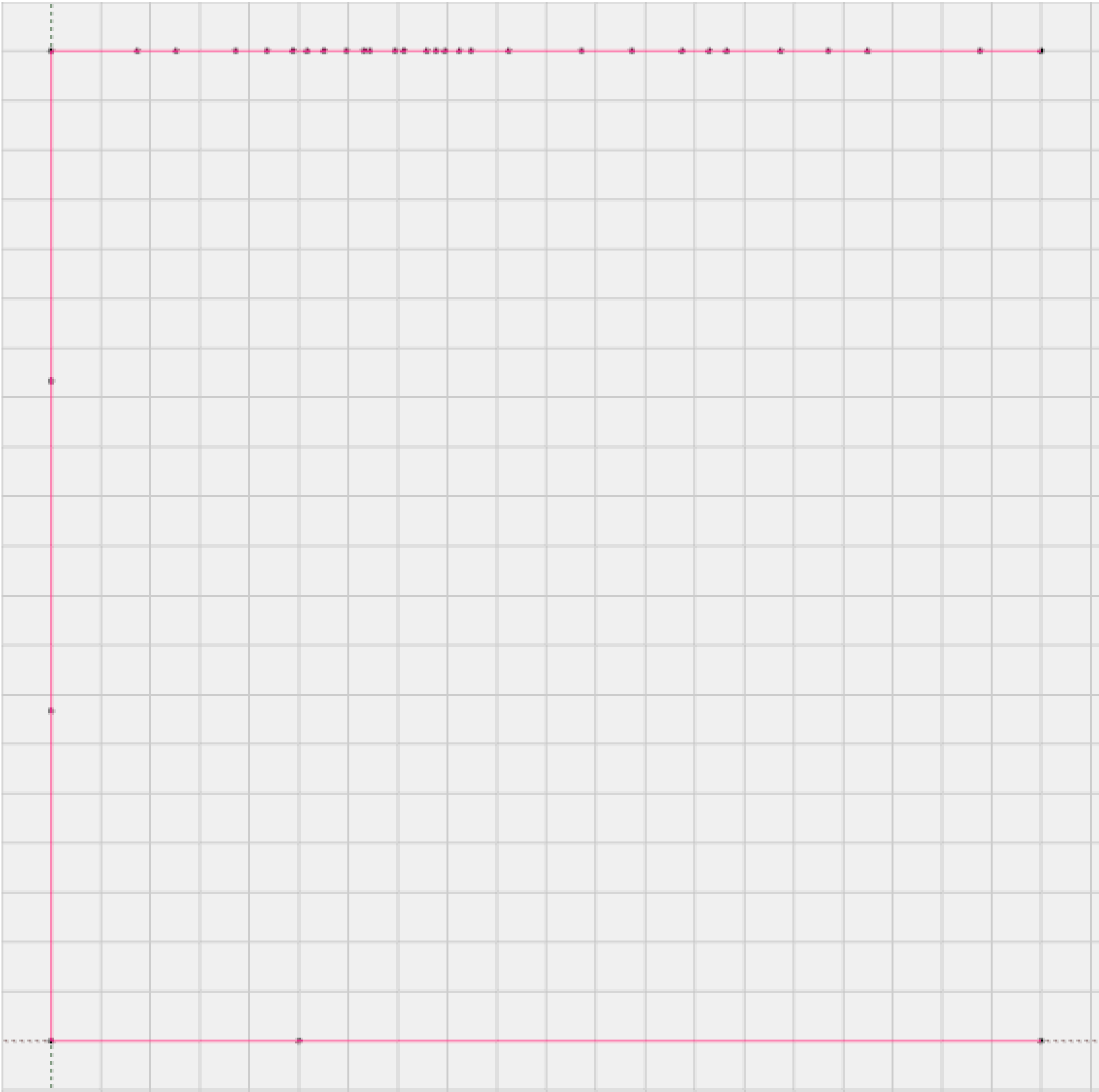


Figure 65 - Three different ways of splitting

Figure 65 shows three different ways of splitting a line:

Vertical line on the left is split using method 1 or 2. Horizontal line on bottom is split using method 3. Horizontal line on top is split using method 4.

3.14 Intersect lines

The intersect lines intent can be used to split and join lines that are drawn. Every intersection found in the current selection will be attempted split and each line will be joined into that split.

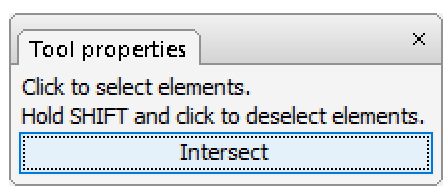


Figure 66 - Properties for intersect lines

Figure 67 shows two lines that intersect each other. A new node will be created in that intersection and we are left remaining with four connecting elements.

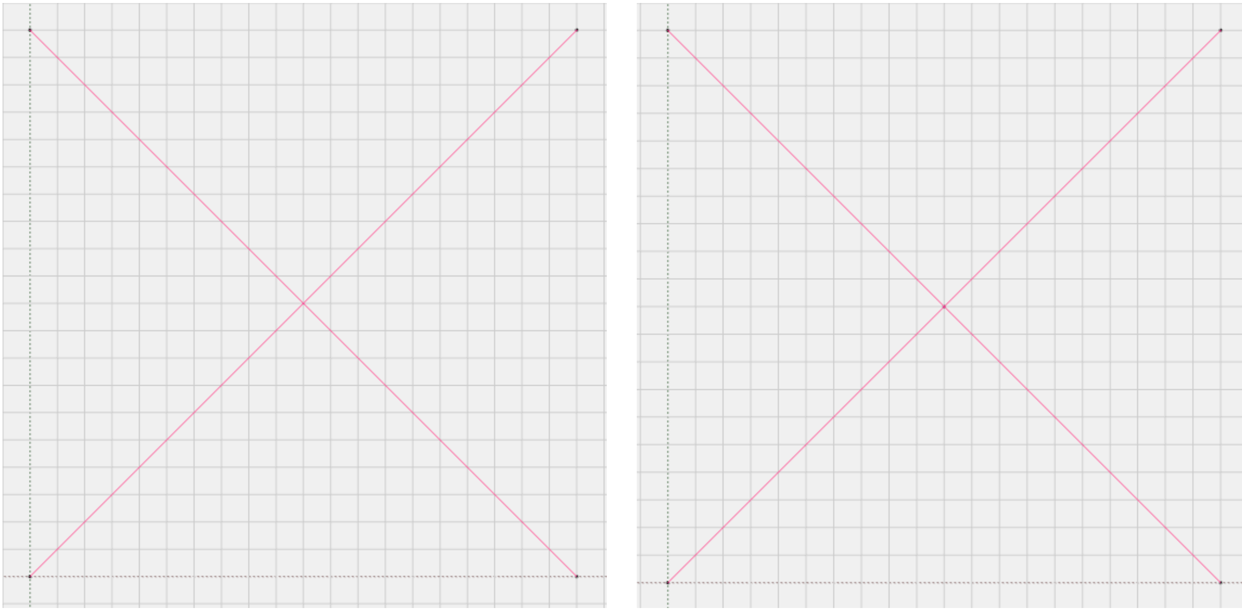


Figure 67 - Left: Two lines with intersection. Right: Lines split into four new connecting lines

3.15 Merge lines

Lines that are laid out in the same direction can be merged together to a single line using the merge lines intent. The lines need to have the same angle, meaning they must lay directly after another.

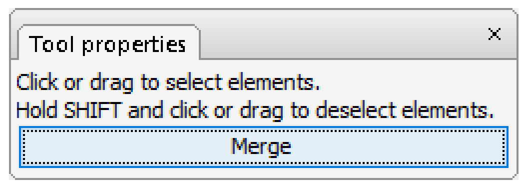


Figure 68 - Properties for merge lines

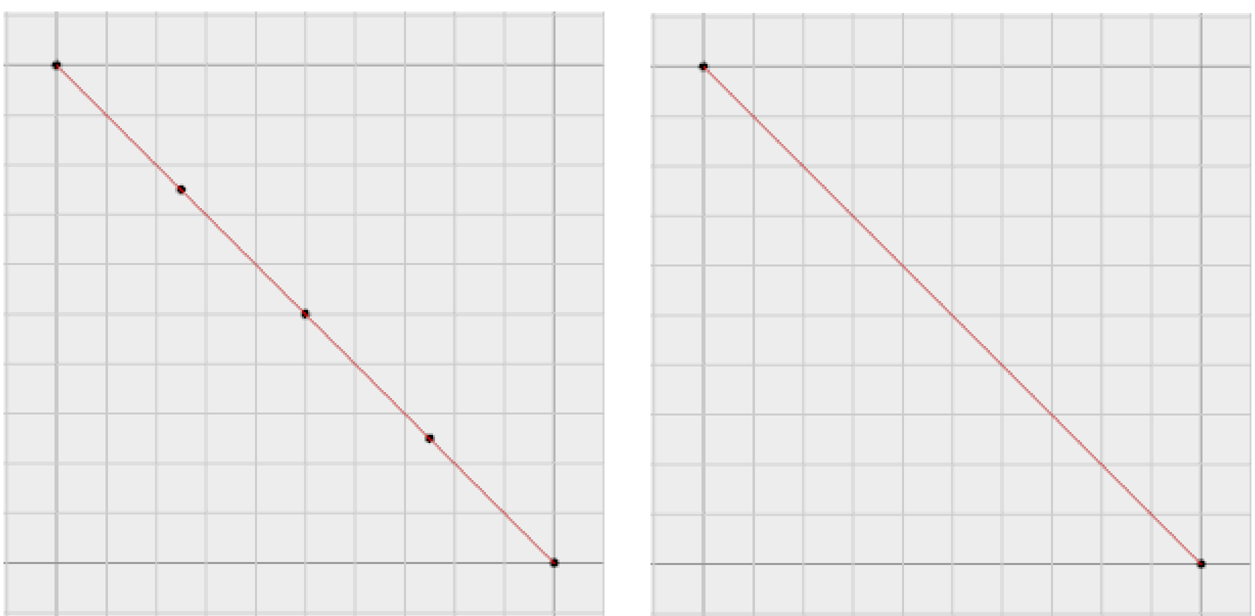


Figure 69 - Left: Elements in the same direction. Right: After merging the lines

3.16 Measure distance

The distance between two nodes can be calculated using the measure distance intent.

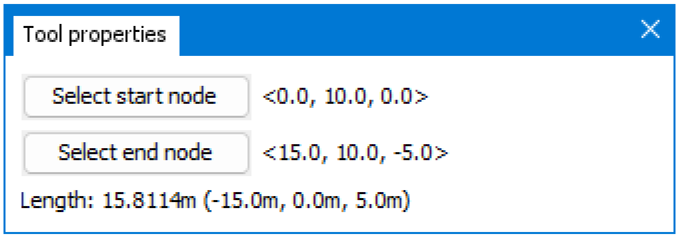


Figure 70 - Properties for measure distance

The nodes to measure distance between is activated by selecting two nodes in the 3D window. The absolute distance, and distance in x-, y- and z-direction is calculated and showed in the Tool properties window.

The user can change either start- or end node by applying the **Select start node** or **Select end node** in the Tool properties window.

3.17 Measure angle

Measuring angles of an element and between elements can be done using the measure angle intent.

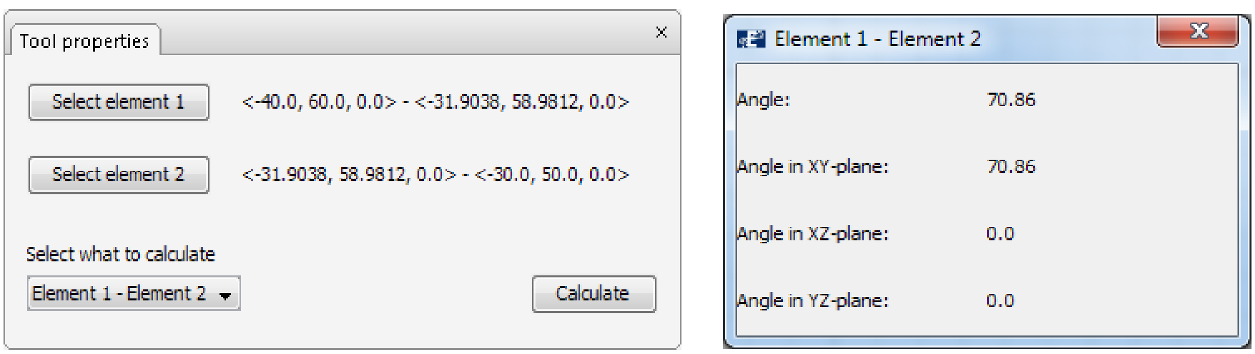


Figure 71 - Left: Properties for measure angle. Right: Results of measure angle

The input in the dialog will be filled in by the first two selected elements if the user has a selection of elements on the canvas.

3.18 Select quad(s)

Explicit membranes have its own selecting tool, Select quad element. This tool allows the user to select, delete or modify quad elements. Figure 73 shows a quad element that is selected.

The following selection options are available:

- Left click inside a quad: the quad is selected. If it is left clicked again, the quad is de-selected.
- Left click + hold + drag to the left: any quad elements that have a node inside the box area is selected.
- Left click + hold + drag to the right: any quad elements that have all nodes inside the box area is selected.

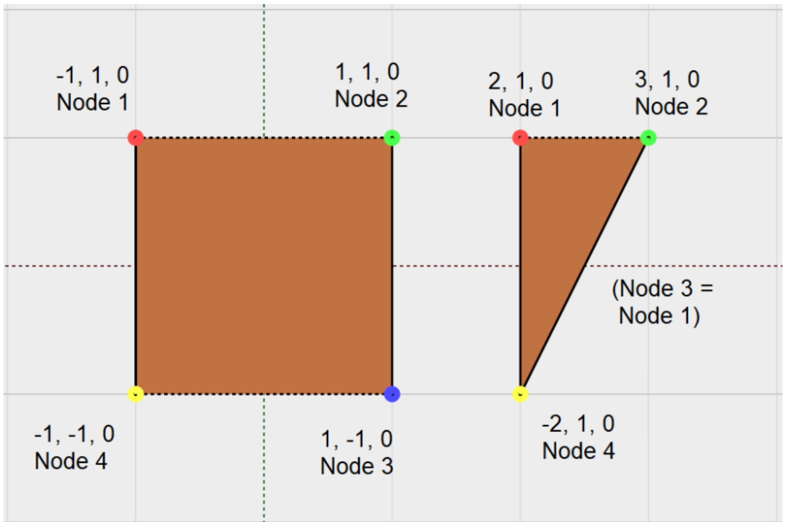


Figure 72 – Quad elements selected

When selecting an explicit quad element you will be able to see the horizontal and vertical edges by the color of the nodes (be careful when selecting more than one element if you are interested in the edges, since the color of the nodes can be overwritten by nearby elements).

The nodes are numbered in the order of red, green, blue and yellow. The red to green edge is the first horizontal edge, green to blue the first vertical edge. Blue to yellow is the second horizontal edge, yellow to red the second vertical edge. The horizontal edges also shown as a stippled line. Note that these must be drawn correctly for net forces to be calculated correctly.

In the case of triangular membrane elements, the internal representation looks slightly different, with node 3 equal to node 1 to indicate that it isn't rectangular.

3.19 Subdivide quad(s)

Explicit membranes (Membrane_X) has its own tool for subdividing elements. This tool allows the user to subdivide in several directions:

- Subdivide: subdivide the selected quad in horizontal and vertical direction in one operation.
- Subdivide vertical: subdivide the selected quad about the quad's vertical axis.
- Subdivide horizontal: subdivide the selected quad about the quad's horizontal axis.
- Triangulate: subdivide the selected quad along the quad's diagonal, in such that two triangles are generated.

These options are shown in Figure 73.

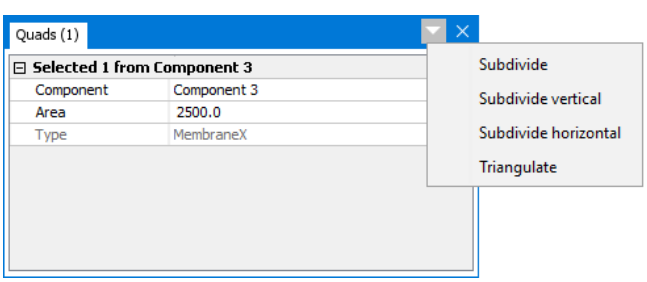


Figure 73 - Subdivide options for quad(s)

In order to access the subdivide tool, the user must first select the relevant quad(s) by applying e.g., the Select quad(s) tool.

3.20 Draw membrane

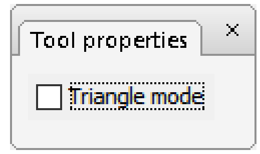


Figure 74 – Properties for draw membrane

Explicit membranes have its own drawing tool. It has the option to draw quads or triangles. When using explicit membranes, it is important to get the vertical and horizontal edges correct, as the tool does not do these corrections for the user.

The tool will prompt the user on which axis to draw, see Figure 76. It will be the vertical edge first, then the horizontal edge, back to the second vertical edge and, in the case of quads, the second horizontal edge last.

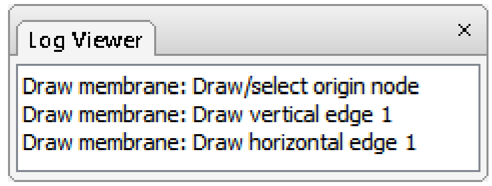


Figure 75 - Prompting user for edge input

3.20.1 Twine direction in explicit membranes

In quad elements, the twines are organized in the same manner as Mask Type 1 for conventional membranes. The twines will cross perpendicular to the horizontal edge of the quad element. This is illustrated to the left in Figure 76, where the green grid representing the twines, and the blue lines are the quad element. The dashed line is the horizontal edge of the quad element, and the solid line is the vertical edge.

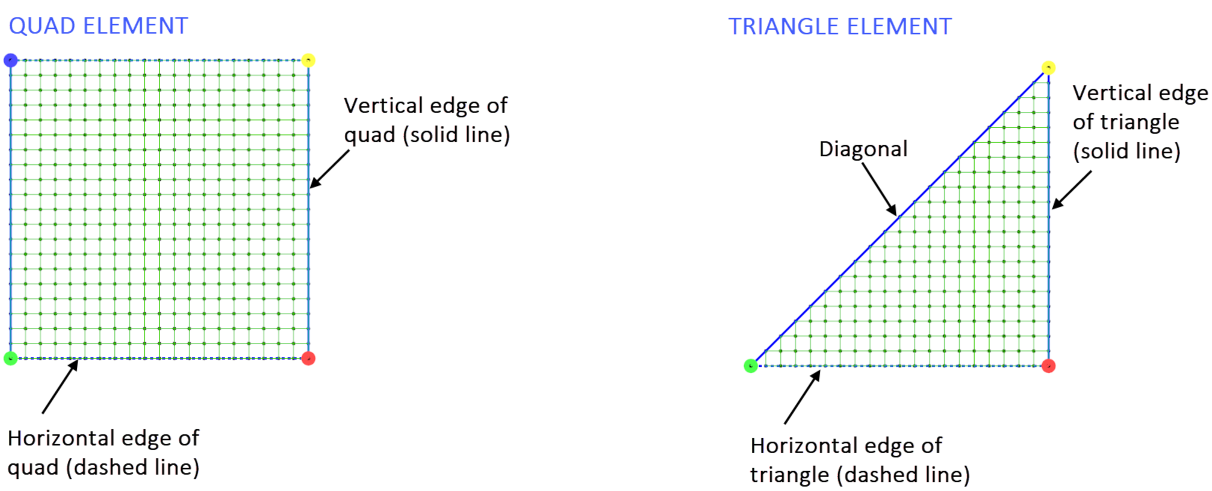


Figure 76 - Twine direction in quad- and triangle elements

Triangle elements are similar (right in Figure 76), the twines cross perpendicular to the horizontal edge of the triangle element. The diagonal (i.e., solid line between green and yellow node) will cross the twine’s diagonal.

3.21 Selecting

There are several tools for selecting in AquaEdit. The main selection tool is the pointer which covers the most usage, see chapter Interacting with the 3D model. The pointer is activated by default when opening AquaEdit. It may be selected pressing ESC (sometime a couple times) or selecting it from the toolbar.

Select all/deselect all

To use select all press CTRL+A when on the canvas. It will automatically select all elements, or all nodes based on what is selected on the canvas. If nothing is selected the function will select all elements. Select all is also available in EDIT->Select All.

To deselect everything press CTRL+U, use the menu `Edit > Select none` or press ESC when the pointer is activated.

Pointer

The Pointer intent provides several selection modes. Having selected the Pointer, the Tool properties window will provide access to the selection modes, see Figure 70.

- Mixed: selects only line elements (i.e., excluding quads) and nodes.
- Elements: selects only line elements.
- Quads: selects only quad elements (associated with membraneX).
- Elements and Quads: selects both elements and quads in one operation.
- Nodes: selects only nodes.

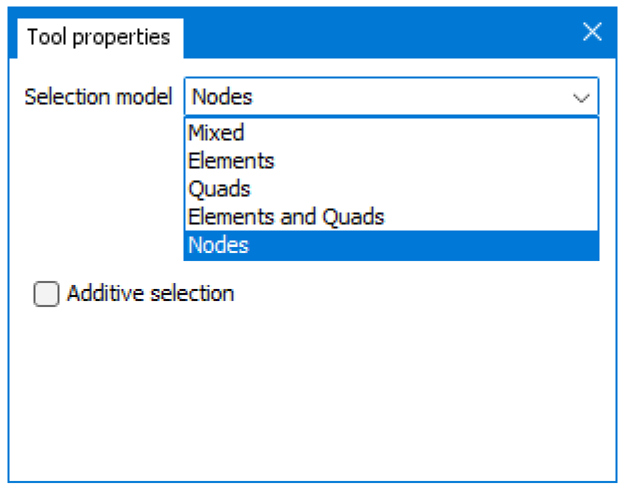


Figure 77 - Pointer selection in Tool properties window

Additive selection means that new selections are added for each time the mouse pointer is released and pressed again in the 3D-view.

By clicking in the 3D-view, the nearest line or node will be selected.

A box selection can be used by clicking and dragging the left mouse button across the canvas. The drag direction matters when deciding what lines to include. Dragging from left to right will select lines that are completely covered by the quad, dragging from right to left will select all lines that has one or both nodes within the quad.

If you wish to select nodes instead of lines when dragging, hold SHIFT while dragging.

To add or remove to the current selection, hold CTRL when clicking. You can also add to the selection by holding CTRL while dragging, but dragging cannot be done to remove from the selection.

Holding ALT while making a box selection will zoom in on the elements within the selection.

On models that are especially large or small it may become necessary to tweak the values controlling whether nodes or lines are selected with the pointer. In these cases, the snap modifier can be set to i.e., 0.1 to allow easier selection of elements.

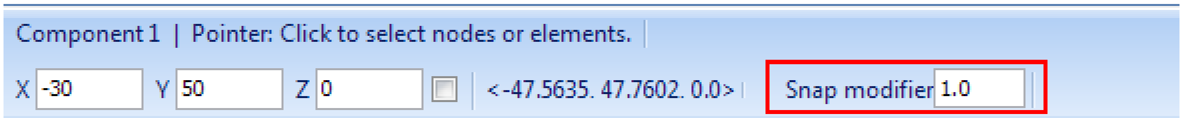


Figure 78 - Snap modifier

Select node(s)
Select node will limit all selection to just nodes.

Select element(s)
Select element will limit all selection to just lines.

Select component
Select component will select all lines belonging to the same component as the lines selected with this tool.

Box select
Selects elements that are inside the box that is created by either click + drag to the left or, click + drag to the right on the canvas.

Select quad
Selects all within a quad. Same as with pointer, but cannot use CTRL to add to selection or ALT to zoom.

3.22 Rotate view
This gives left mouse button the ability to rotate around on the canvas; it gives left mouse button same function as the middle mouse button.

4. Property editing

AquaEdit allows editing of all aspects of properties of nodes, elements, hinges, point loads etc. that are possible in AquaSim.

4.1 Nodes

Displays name, global coordinates and degrees of freedom (DOF) for selected node(s). These can also be edited directly in the panel. If a terrain is imported, then the GPS-position is shown here too. The …-button in the GPS-field opens the location in Google Maps in your browser. The GPS-position can also be set to a different type, see chapter Settings.

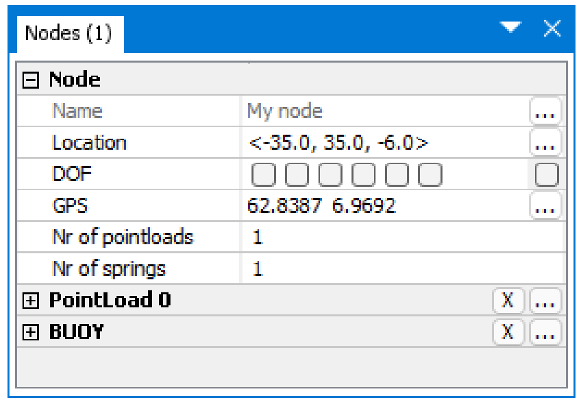


Figure 79 - The node property panel

The global coordinates can be edited by clicking the …-button on the Location-property.

Selection and de-selection of all DOFs in one click can be done by selecting the right-most checkbox. The panel also shows all decorators added to the node. The decorators can be edited by clicking the …-button and removed by clicking the X-button.

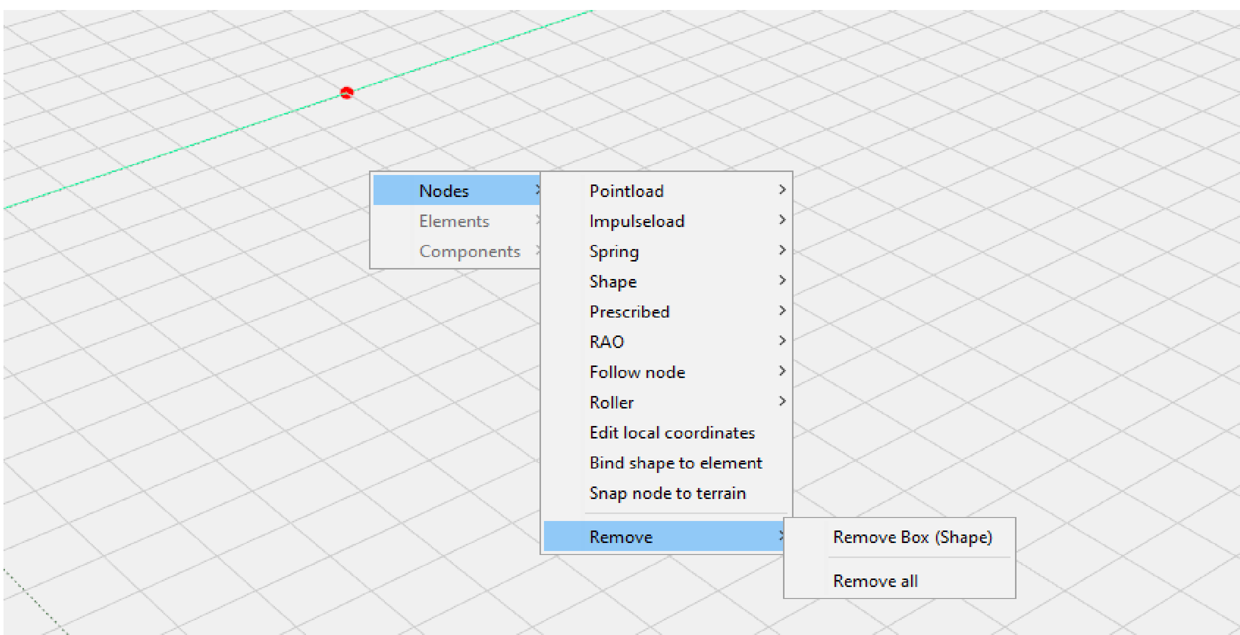


Figure 80 - Context menu for nodes

Figure 80 shows the menu when the mouse is right clicked in the draw window. This menu gives the possibility to add new or existing decorator to nodes. The same menu, excluding Remove-menu, is found in the popup-menu for the Node property panel shown in Figure 80 and the menu **Commands > Nodes**. The menu “Bind shape to element” will only appear if the node has a Shape added to it. The menu “Snap node to terrain” will only appear if the model has a terrain imported.

4.1.1 Degrees of freedom

DOF can be set by selecting the nodes(s) you want to edit and checking on/off the boxes in the Node property panel, shown in Figure 64. The first three boxes correspond to translation in x-, y- and z-direction. The last three boxes correspond to rotation about the x-, y-, and z-axis.

4.1.2 Pointloads

Point loads and momentums work according to the global coordinate system. These are static loads and will be added to the model in the first step.

Three translational forces in x-, y- and z-direction can be assigned, as well as three rotational moments in x-, y- and z-direction.

For information about the different load types, see (Aquastructures, 2024a).

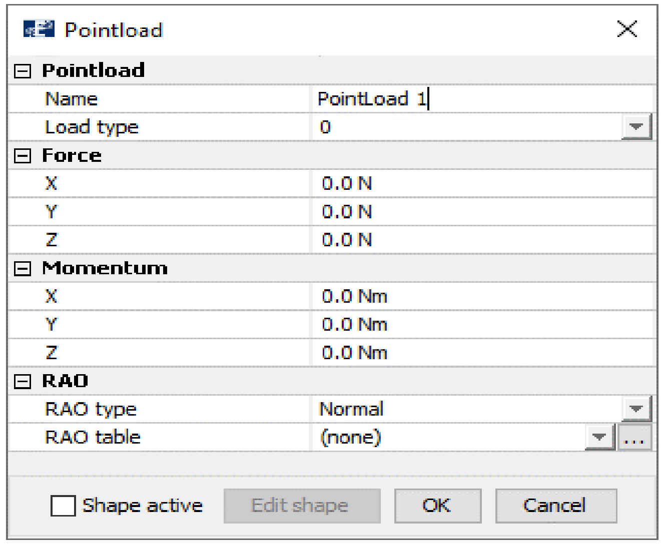


Figure 81 - Properties for pointload

4.1.3 RAO on Pointloads

It is also possible to connect a RAO to the Pointload. Available RAO types are:

- **Normal**
- **Drift polar**
- **Drift cartesian**

To include RAO, a RAO-table must be established. These can be added using the dropdown menu and selecting "(new)". Settings for the RAO table is described below, these are similar for Normal, Drift cartesian and Drift Polar. The RAO-tables can also be edited in the menu `Tools > Tables > Show RAO tables`. For information about the different RAO types, see (Aquastructures, 2024a).

Properties	**Description**
Name	Name of the table.
Symmetric	Activates symmetry in the RAO tables. If RAO tables are defined for 0-180 degrees, selecting "Symmetric" checkbox AquaSim will apply equivalent tables for 180-360 degrees.
Angle of 0 direction	The angle of 0 direction in the RAO.
Type	Indicator as to the RAO shall be given in terms of Wave frequency (WF), Hertz (Hz), Wave period (WP) or Time.
Indicator of wave angle	Indicator as to whether angles is given in radians or degrees.
Indicator of phase angle	Indicator as to phase angles in RAO shall be added or subtracted.
Indicator of rotations	Indicator as to how the operator for rotations is given: Radians per unit wave elevation.

	Degrees per unit wave elevation. Relative to the steepness of the wave.

RAO Type: Normal

For RAO type Normal, RAOs are defined in terms of force, moment and direction. When RAO-table Type WF, Hz or WP is chosen, the RAO-table will look like Figure 82. Parameters are elaborated in the succeeding table.

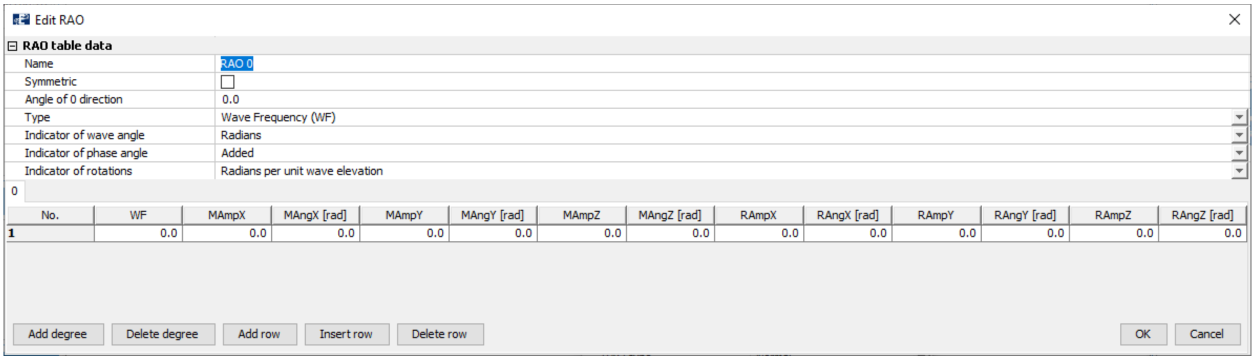


Figure 82 RAO-table for type WF, Hz and WP

Properties	**Description**
WF/ WP	Wave frequency/ Wave period.
MAmpX	Amplitude of force in x-direction.
MAngX	Angle of force in x-direction.
MAmpY	Amplitude of force in y-direction.
MAngY	Angle of force in y-direction.
MAmpZ	Amplitude of force in z-direction.
MAngZ	Angle of force in z-direction.
RAmpX	Amplitude of moment about the x-axis.
RAngX	Angle of moment about the x-axis.
RAmpY	Amplitude of moment about the y-axis.
RAngY	Angle of moment about the y-axis.
RAmpZ	Amplitude of moment about the z-axis.
RAngZ	Angle of moment about the z-axis.

The input values of the amplitudes (MAmpX, -Y, -Z and RAmpX, -Y, -Z) are positive values. The angles (MAngX, -Y, -Z and RAngX, -Y, -Z) will correct for change in signs of amplitude according to the right-hand rule.

When RAO-table Type Time is chosen, the RAO-table will look like Figure 83. Parameters are elaborated in the succeeding table.

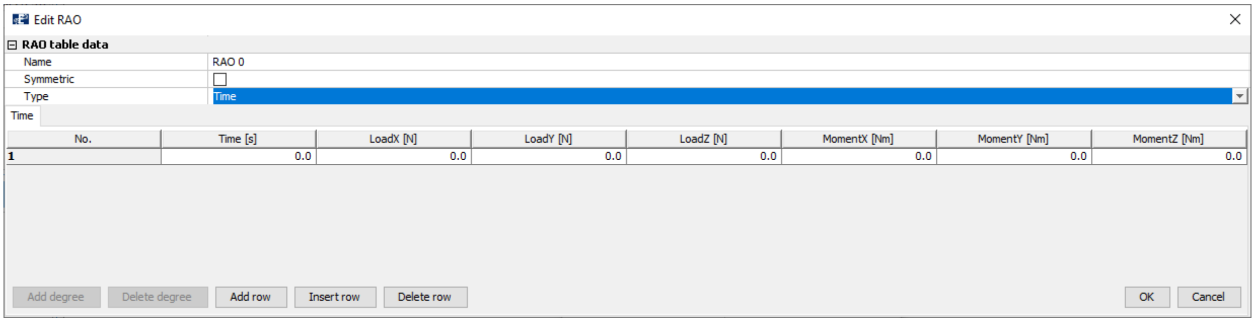


Figure 83 RAO-table for type Time

Properties	Description
Time	The time instant for when the load and/or moment is to occur.
LoadX [N]	Magnitude of force in x-direction.
LoadY [N]	Magnitude of force in y-direction.
LoadZ [N]	Magnitude of force in z-direction.
MomentX [Nm]	Magnitude of moment about x-axis.
MomentY [Nm]	Magnitude of moment about y-axis.
MomentZ [Nm]	Magnitude of moment about z-axis.

RAO Type: Drift polar

Selecting RAO type Drift polar, RAOs are defined in terms of drift-coefficient in polar coordinates.

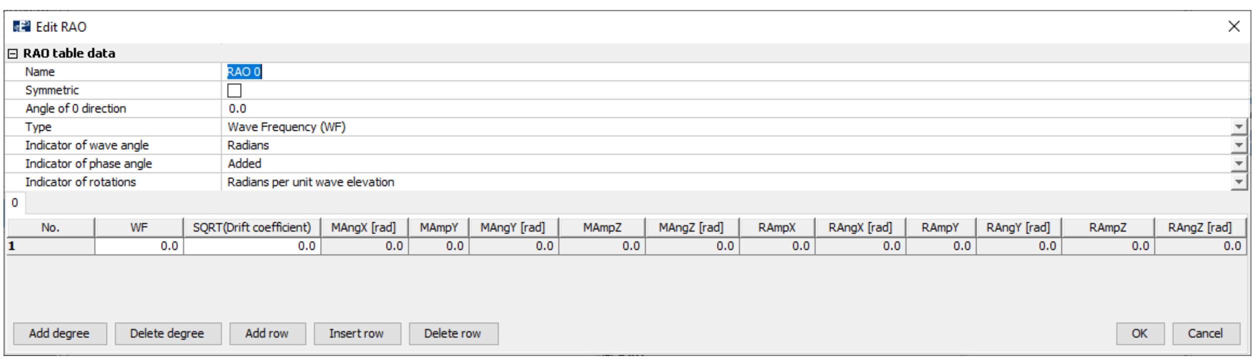


Figure 84 RAO table for RAO type Drift polar

Properties	Description
WF/ WP	Wave frequency/ Wave period.
SQRT(Drift coefficient)	Magnitude of drift-coefficient (sqrt(T)).

The direction of the drift-coefficient is regulated by the ‘Add degree’.

When RAO-table Type Time is chosen, the RAO-table will look like Figure 85. Parameters are elaborated in the succeeding table.

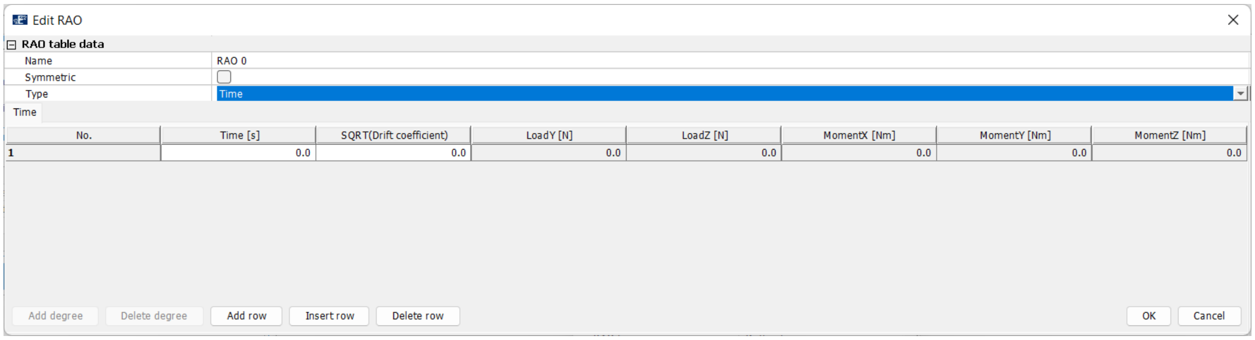


Figure 85 – RAO table for type Time

Properties	**Description**
Time	The time instant for when the drift-coefficient is to act.
SQRT(Drift coefficient)	Magnitude of the drift-coefficient (sqrt(T)).

RAO Type: Drift cartesian

Selecting RAO type Drift cartesian, RAOs are defined in terms of a drift-coefficient in cartesian coordinates (x, y, and z).

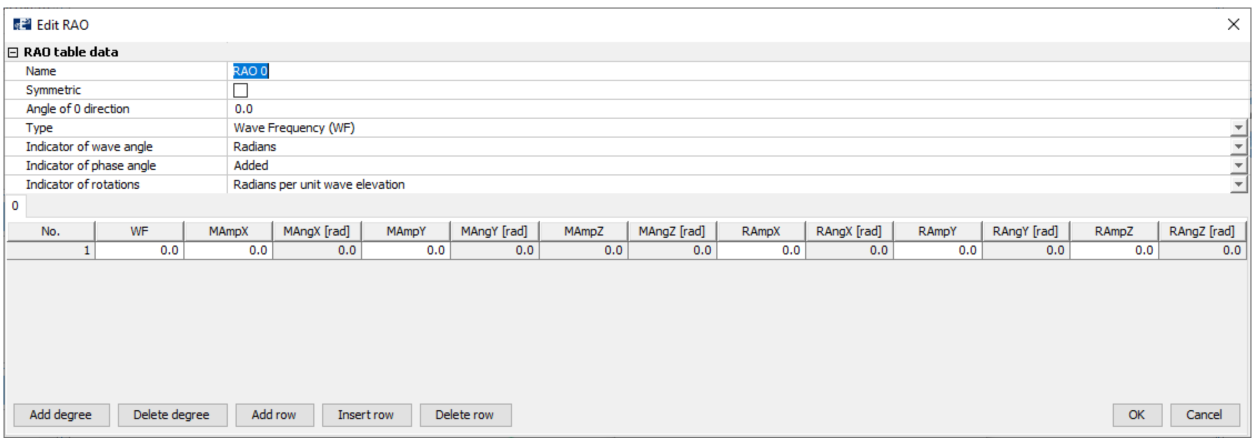


Figure 86 RAO table for RAO type Drift cartesian

Properties	**Description**
WF/ WP	Wave frequency/ Wave period.
MAmpX	Magnitude of drift-coefficient (sqrt(T)) in x-direction.
MAmpY	Magnitude of drift-coefficient (sqrt(T)) in y-direction.
RAmpX	Magnitude of drift-coefficient (sqrt(T)) about the x-axis.
RAmpY	Magnitude of drift-coefficient (sqrt(T)) about the y-axis.
RAmpZ	Magnitude of drift-coefficient (sqrt(T)) about the z-axis.

The input values of MAngX, MAngY, MAmpZ, MAngZ, RAngX, RAngY and RAngZ is inactive because drift forces are not included for these directions. The direction of the drift-coefficient is regulated by the 'Add degree'.

4.1.4 Impulse loads

Impulse loads can be used to simulate the effect of e.g. collisions or other impulse loads. This is a dynamic calculation and will be added to the model after the initial steps. To get an accurate result, it is important to input correct mass and speed; dynamic forces acting on the node is a result of accelerations.

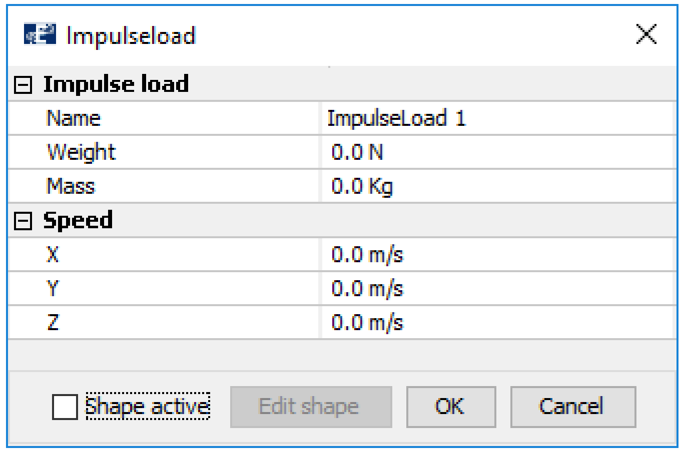


Figure 87 - Properties for impulse load

4.1.5 Springs

Springs works according to the global coordinate system. There are 8 different springs that can be added to nodes: Normal, Initial, Dampner, Displaced, Buoy, Offloaded, Mass and Offloaded type 2. For more information about the different springs, see (Aquastructures, 2024c).

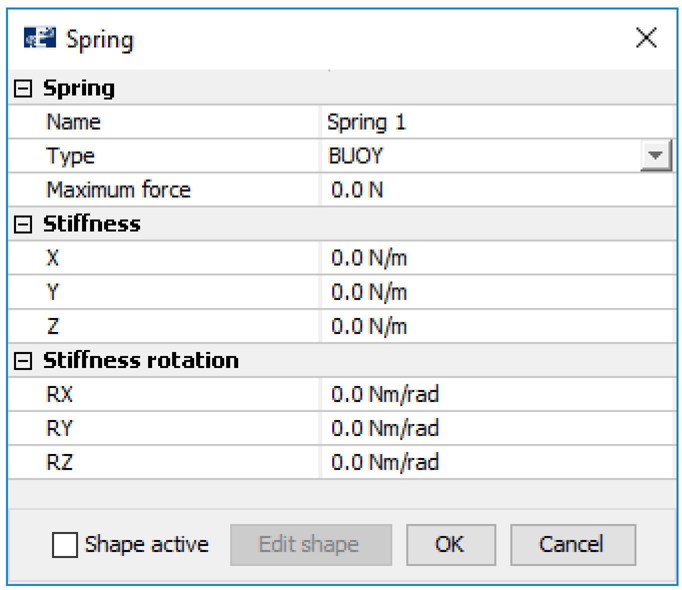


Figure 88 - Properties for spring, here represented by a buoy

Buoy and Displaced springs will have an extra line of information. Buoy can have a maximum force; the spring force cannot exceed this upper limit. Displaced have an input of meters the spring can move in the negative z-value.

The input will change depending on what kind of spring is being added:

Mass – Mass and Inertia

Displaced – Stiffness and Damping

Dampner – Damping translation and damping rotation

Rest – Stiffness and Stiffness rotationea

4.1.6 Shapes

Shape properties

Name	The name of the shape. Visible in the node decorators window and in AquaView.
Shape	The 3D model file for this shape
Derived from nodes	Normal rotation only works for beam elements, ticking this option allows two degrees of rotation to be extracted from truss and other non-beam elements.
Swap direction	Applies for rotation that are derived from nodes (option above), tick this if your shape rotates in the wrong direction in the result.
Translation	Optional offset from the center of the model.
Scale	Optional scale of the model.
Rotation	Optional rotation of the model
Color	Optional color of the model. Each color (Red, Green, and Blue) can be individually set between 0 and 1. 1, 1, 1 is white. 1, 0, 0 is bright red, etc.

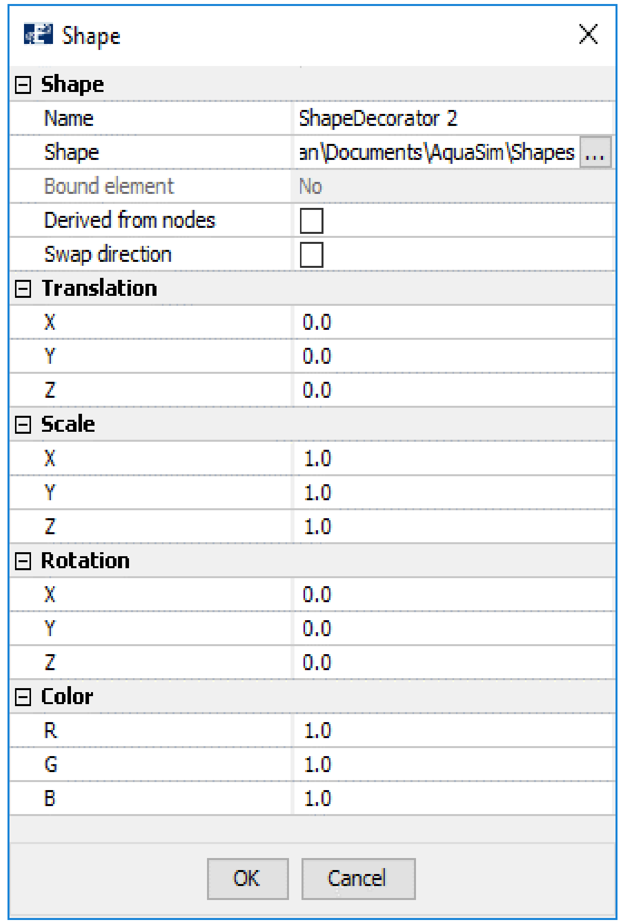


Figure 89 - Properties for shapes

Bind shape to element

Shapes can additionally be bound to an element. First use the command “Bind shape to element”, see Figure 80, then left click on an element attached to the node to bind it. A shape bound to an element will follow the rotation of the element.

If the shape is bound to an element the user must enable Rotation in the result output in the Environment dialog.

Shape active

AquaEdit automatically adds shapes to pointloads, springs, hinges etc. on export. These default shapes can be overridden by checking the box “Shape active” when editing decorators. Then the button “Edit shape” will become active and the user can add/edit a shape for the decorator.

4.1.7 Prescribed displacement

Adding a prescribed displacement will force a node to move to the location – or rotate, in the initial steps. Input is the absolute coordinate (position) or radians (rotation).

For any position or rotation to be active, they must be activated in the check boxes.

The input for position can be changed to relative position to the node by checking the box "Relative displacements".

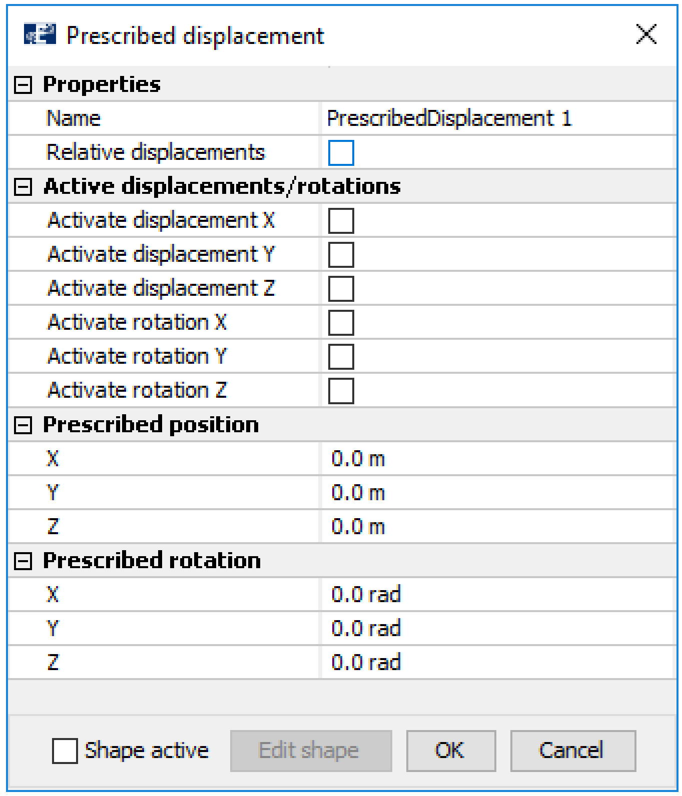


Figure 90 - Properties for prescribed displacement

4.1.8 RAO

Response Amplitude Operators are used to move nodes in the dynamic steps. As prescribed displacement one must activate which DOF the RAO should apply to using the check boxes.

Fixed translation gives the x-, y- and z-translation, where prescribed rotation specify rotation about the same axes. In the initial steps the nodes are prescribed in linear increments. Then the end point becomes the origin for the RAO motion.

Position of RAO data defines the origin of the RAO data. If (X, Y, Z) = (0m, 0m, 0m) then RAO data will have its source in the origin of the global coordinate system.

One must also specify an RAO table. These can be added using the dropdown menu and selecting “(new)”. All the RAO tables in the model are also shown in the dropdown menu. The RAO tables can also be edited in the menu `Tools` > `Show RAO tables`.

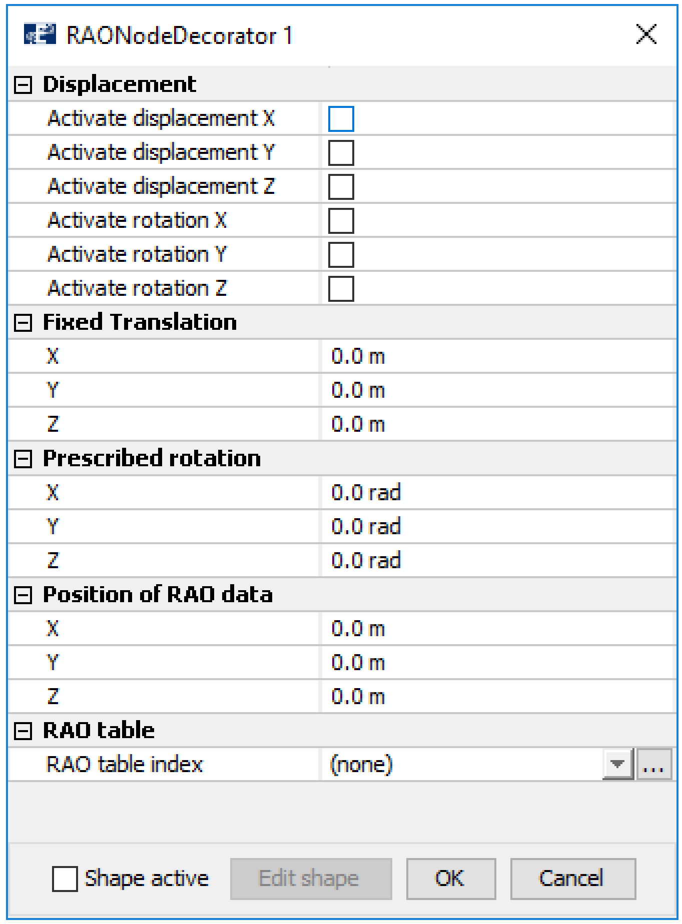


Figure 91 - Properties for RAO on nodes

RAO table

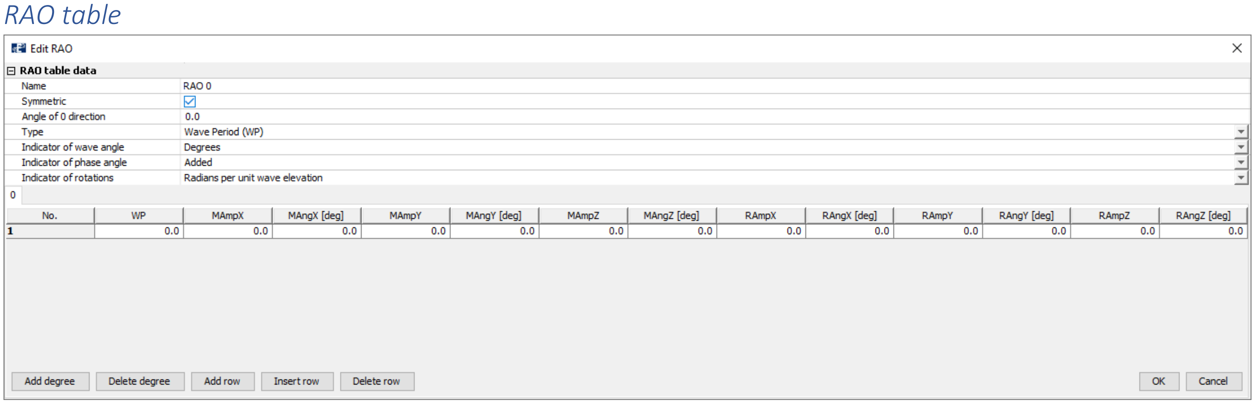


Figure 92 - Properties for RAO table

Properties	Description
Name	Name of the table.
Symmetric	Activates symmetry in the RAO tables. If RAO tables are defined for 0-180 degrees, selecting “Symmetric” checkbox AquaSim will apply equivalent tables for 180-360 degrees.
Angle of 0 direction	The angle of 0 direction in the RAO.
Type	Indicator as to the RAO shall be given in terms of Wave frequency (WF), Hertz (Hz), Wave period (WP) or Time.
Indicator of wave angle	Indicator as to whether angles is given in radians or degrees.
Indicator of phase angle	Indicator as to phase angles in RAO shall be added or subtracted.
Indicator of rotations	Indicator as to how the operator for rotations is given: Radians per unit wave elevation. Degrees per unit wave elevation. Relative to the steepness of the wave.

Properties	Description
WF/ WP	Wave frequency/ Wave period.
MAmpX	Amplitude for motion in x-direction.
MAngX	Angle for motion in x-direction.
MAmpY	Amplitude for motion in y-direction.
MAngY	Angle for motion in y-direction.
MAmpZ	Amplitude for motion in z-direction.
MAngZ	Angle for motion in z-direction.
RAmpX	Amplitude for rotation about the x-axis with reference to wave height.
RAngX	Angle for rotation about the x-axis.
RAmpY	Amplitude for rotation about the y-axis with reference to wave height.
RAngY	Angle for rotation about the y-axis.
RAmpZ	Amplitude for rotation about the z-axis with reference to wave height.
RAngZ	Angle for rotation about the z-axis.

The input values of the amplitudes (MAmpX, -Y, -Z and RAmpX, -Y, -Z) are positive values. The angles (MAngX, -Y, -Z and RAngX, -Y, -Z) will correct for change in signs of amplitude according to the right-hand rule.

If Time is chosen as type, the RAO-amplitudes will operate as a function of time. The amplitudes are displacement and rotations, see Figure 93.

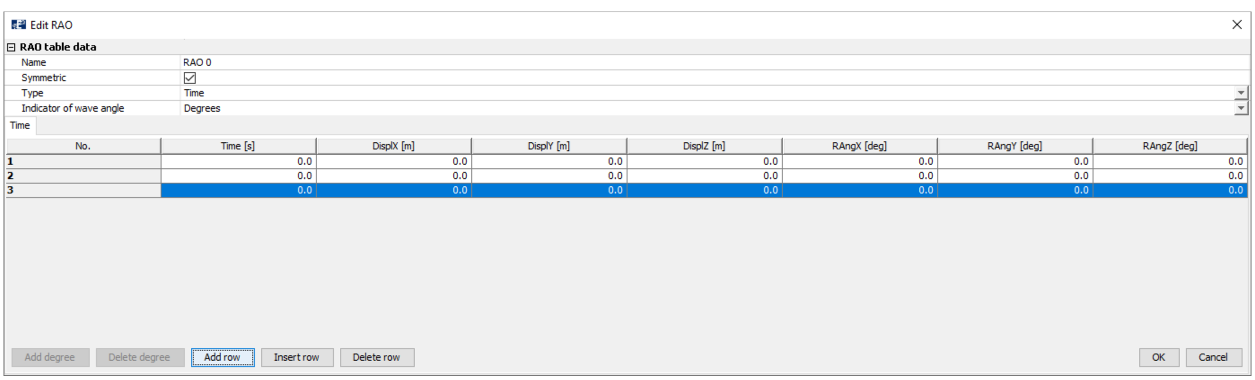


Figure 93 - Properties for RAO table, type Time

4.1.9 Follow node

Follow node are used to allow nodes to follow the translation and/or the rotation of other nodes.

Activate the checkboxes to allow the node to follow the selected node, which can be changed by clicking "Select node" and selecting a node in the drawing window.

Pressing the …-buttons will give the user the option of selecting a node to follow for that DOF, so in theory a node can follow up to 6 different nodes.

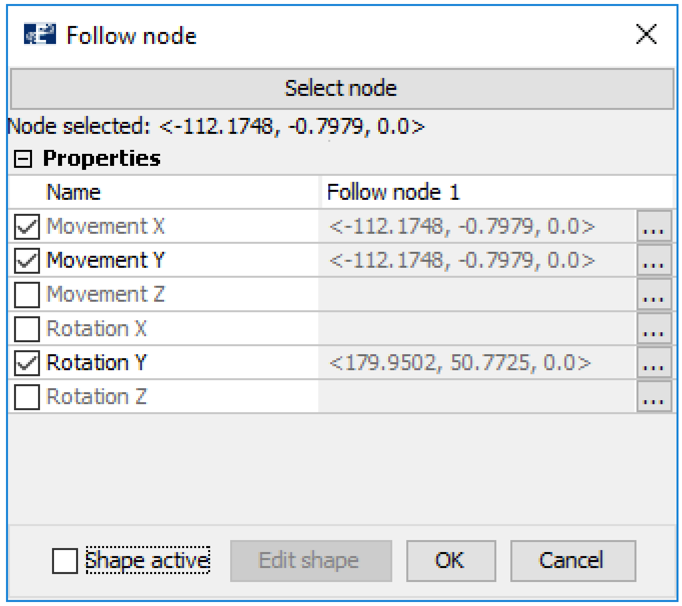


Figure 94 - Properties for follow node

4.1.10 Roller

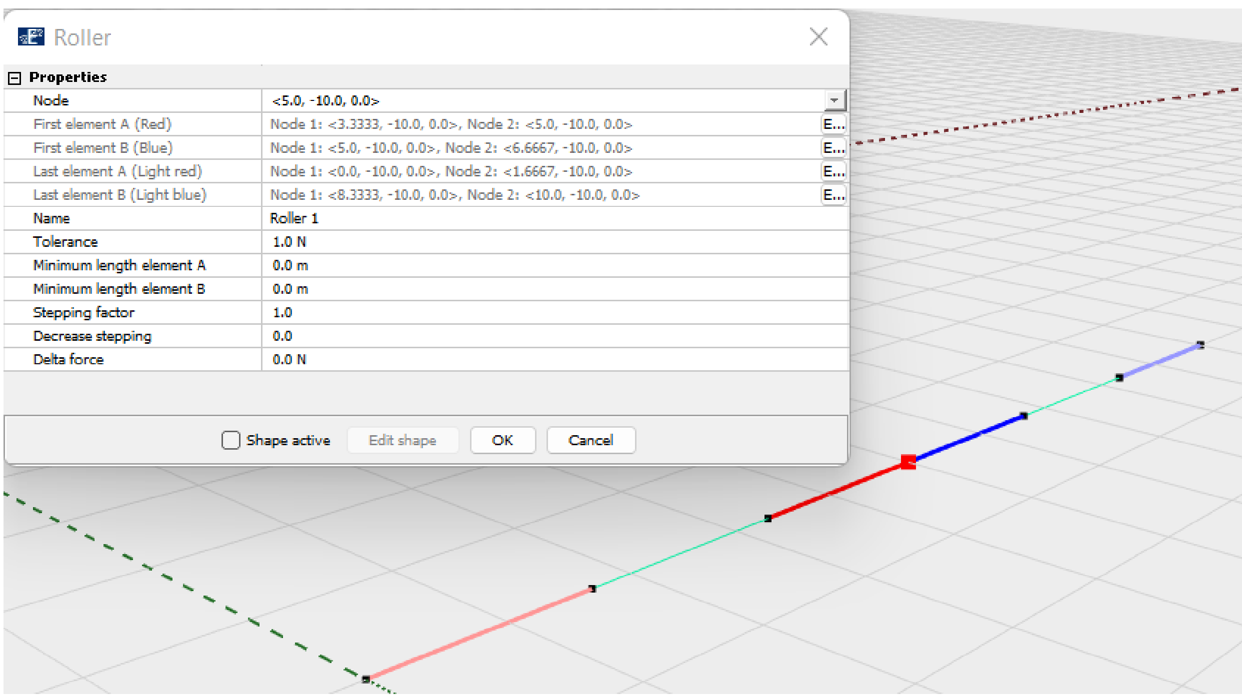


Figure 95 - Properties for roller

A roller will roll along the elements as mentioned in the input. It can roll along the elements attached to it all the way to the last elements specified. In Figure 95 the roller can roll along the elements marked in red (first element A) and light red (last element A) on the left side of the node, and the elements marked in blue (first element B) and light blue (last element B) on the right side of the node.

The elements that the roller can roll over last is set as default the same as the first elements; the elements attached to the node. This can be changed by the **…**-button and has three options:

- Set same as first element (the default).
- Set to end of component (the roller can roll along the entire component that the first element has as parent.
- Choose with pointer (click on the element wanted with the mouse pointer, used in Figure 95).

Minimum length element A / B:
The minimum lengths give the length the roller can roll towards the end points. E.g., minimum length of 0.5 will make the roller stop when it is 0.5 meters from the end of the cable (given as last element A and B in the dialog). Stepping factor and Decrease stepping are numerical parameters applied when AquaSim iterates to find the roller position.

Tolerance and Delta force:
When the analysis is run, the axial forces in element A and B are compared and the time step will not converge until the forces are within the "Tolerance" of the defined "Delta force". If "Delta force" is 0, the difference in axial force in element A and B need to be lower than the "Tolerance".

Delta force:
Positive values means one element is larger than the other, while negative values mean the opposite element is largest.

Several rollers can be edited at the same time after creation by selecting nodes with roller on them, right clicking with the mouse and using `Roller > Edit`.

4.1.11 Edit local coordinates

A node can be assigned a local coordinate system. The x-, y- and z-values of the local x-axis and local z-axis are the parameters that determines this.

If the decorators on the node are dependent on the orientation and the element the decorator is connected to, a local coordinate system on the node is useful. This will enable the decorator to change its own axis during the analysis to rotate along its connected element. A local coordinate system will affect all decorators attached to the node.

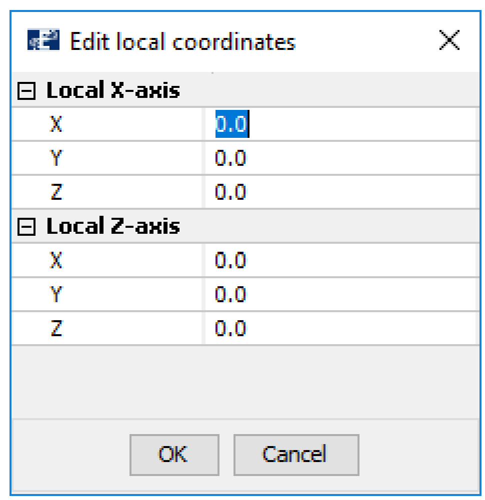


Figure 96 - Properties for local coordinates

4.1.12 Snap node to terrain

If the model has an imported terrain, this will make all selected nodes snap to the location of the terrain directly below the nodes.

4.2 Elements

When one or more elements are selected the Object Property Window will display the properties of the element(s).

This show the element length, component name that the element belongs to, element name, global coordinates of the start and end nodes as well as the point 3 for the element.

The panel also shows all decorators added to the element. The decorators can be edited by clicking the …-button and removed by clicking the X-button.

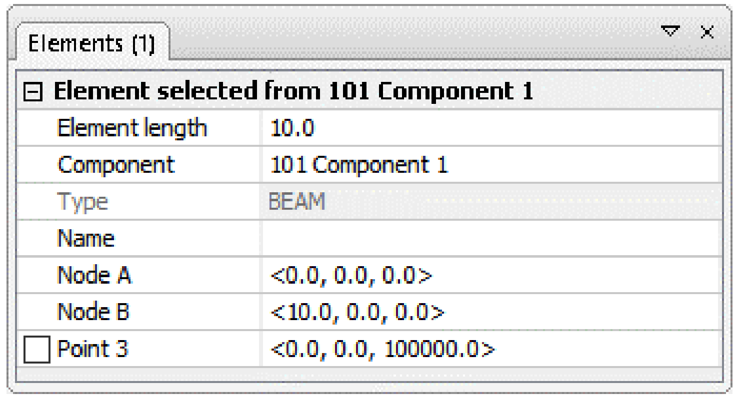


Figure 97 – The element property panel

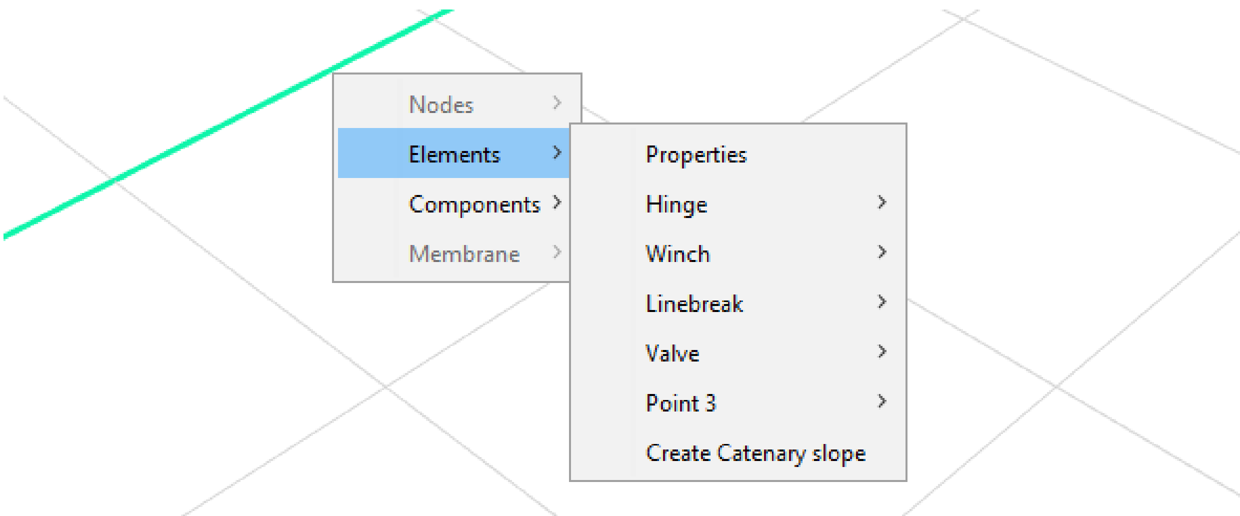


Figure 98 - Context menu for elements

Figure 98 shows the menu when the mouse is right-clicked in the draw window. This menu gives the possibility to add new or existing decorator to elements. The same menu, excluding Remove-menu, is found in the popup-menu for the element property panel shown in Figure 98 and the menu **`Commands`** –> **`Elements.`** This menu also has the option **`To active component`**, this is explained in chapter To active component, and the option **`Selection to elements`**, that turns selected nodes into selected elements.

4.2.1 Hinge

A hinge is applied on one end of an element of a beam. The hinge will by default be added to node B of the element and will be visualized by a green cone, see Figure 100. The hinge can be added to node A by checking the box "Reversed".

Check the boxes in the "DOF"-field to lock the hinge in one or more of the six Degrees Of Freedom of the node. Default is locked in all degrees.

To make the hinges follow the rotation of the hinged elements during the analysis, check the box "Co-rotated" and here the user can define the local coordinate system of the hinge. The checkbox "Local" will calculate the local coordinate system of the hinge based on the position of the nodes of the element.

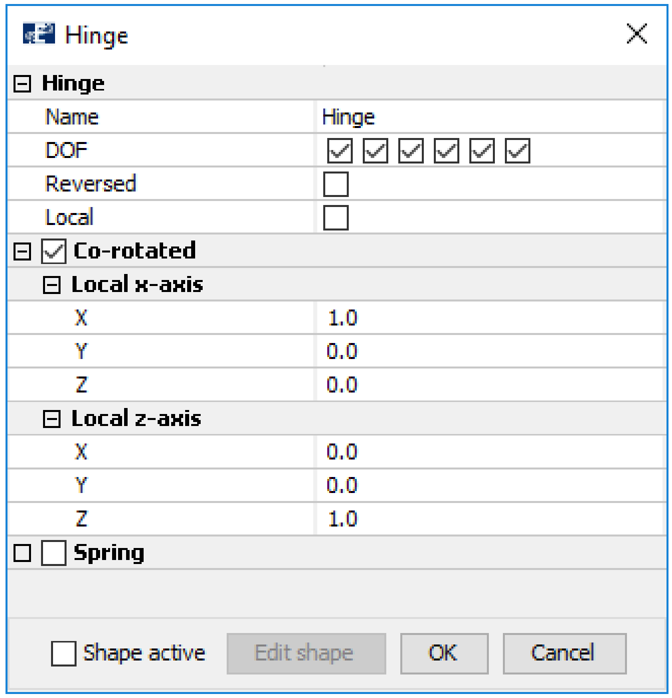


Figure 99 - Properties for hinge

It is possible to add a spring to the hinge by checking the box "Spring". Input is almost the same as shown in chapter Springs. But, the type of spring allowed is *Node to node spring* and *Node to Node damping*, see chapter Node2Node spring.

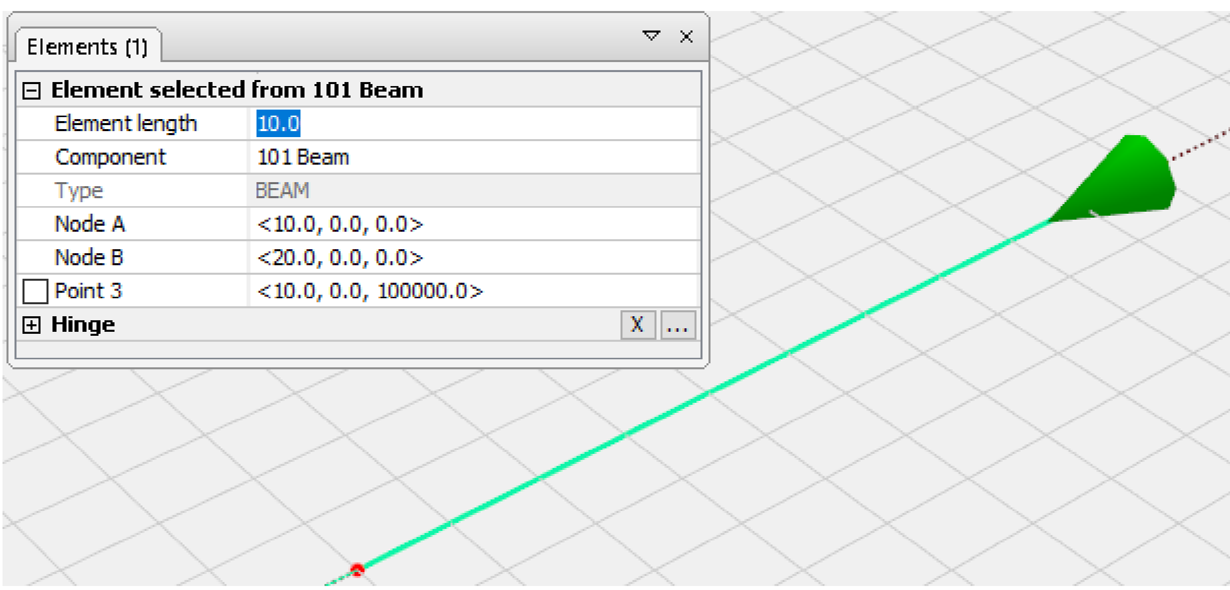


Figure 100 - Visualization of hinge on beam element

4.2.2 Winch

A winch is applied on one end of an element of a beam or a truss. The winch will by default be added to node B of the element and will be visualized by a green puck, see Figure 102. The winch can be added to node A by checking the box “Reversed”.

Type In means that the winch can winch in the entire length of the element it is attached to. “Maximum force” is the maximum allowed force in the winch.

Type Out means the winch can winch out elements lying behind the element it is attached to, input “Segments” will tell AquaSim how many dummy elements it can create for the winch. E.G. segments 5 and element length of 10 meters will allow the winch to winch out 50 meters. Note: If there are several out winches in the model, the one with the highest “Segments” will apply to all.

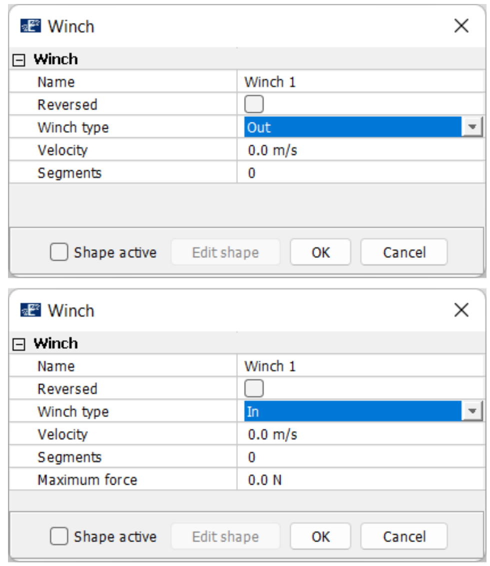


Figure 101 - Properties for winch

The winches are winched with the velocity given. Unit is m/s.

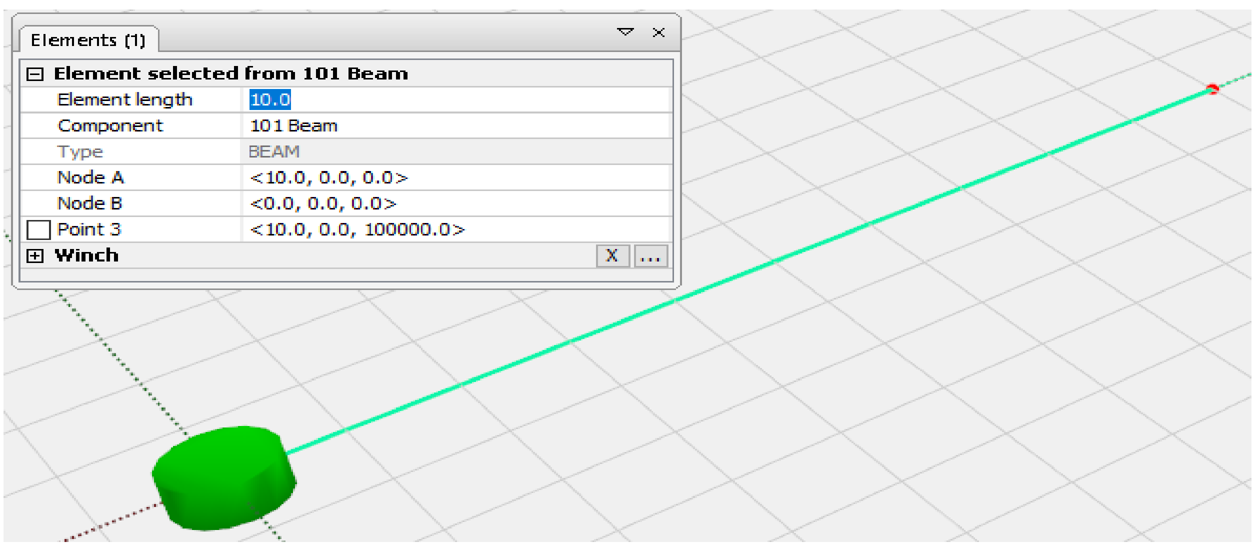


Figure 102 - Visualization of winch on beam element

4.2.3 Linebreak

It is possible to simulate a break in an element during the analysis by adding a Linebreak to it. One may choose to break the line at a specific time-step in the analysis, or as a function of specified max force. Either selection, AquaSim will remove the entire element at the given criterion.

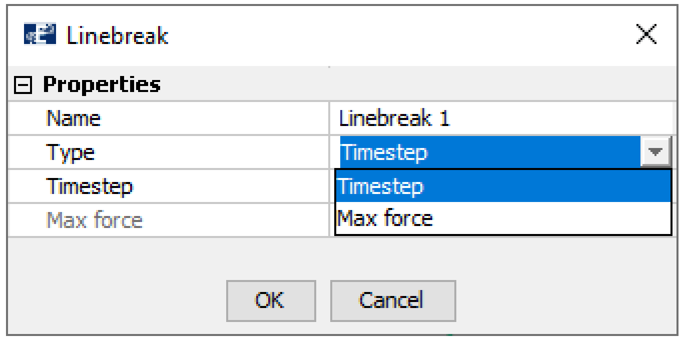


Figure 103 - Properties for linebreak

4.2.4 Valve

A valve is applied on one end of an element of a beam. The valve will by default be added to node B of the element and will be visualized by a green puck atop a green disc, see Figure 105. The valve will by default be added to node B of the selected element. It can be added to node A by checking the box “Reversed”.

For theory on valves and the input parameters, read (Aquastructures, 2024a).

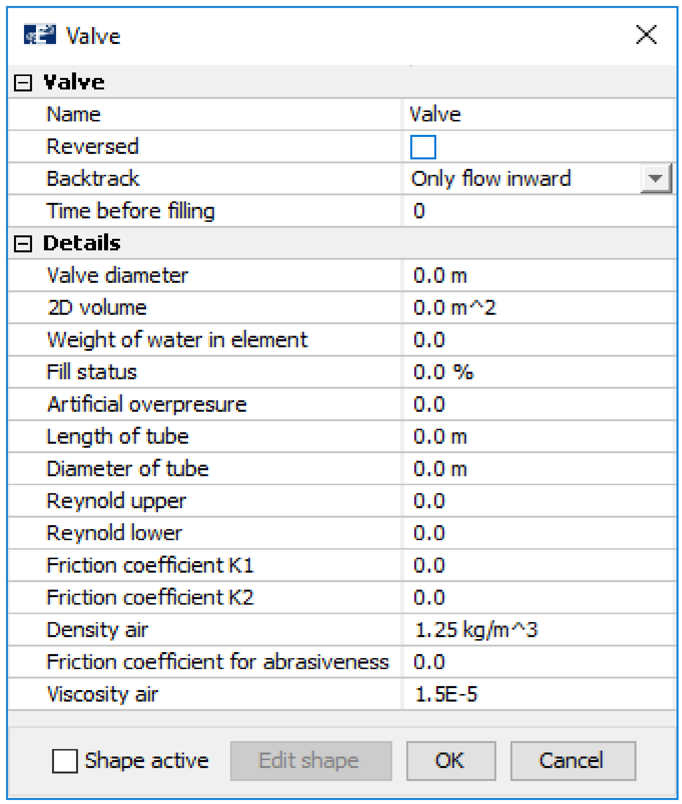


Figure 104 - Properties for valve

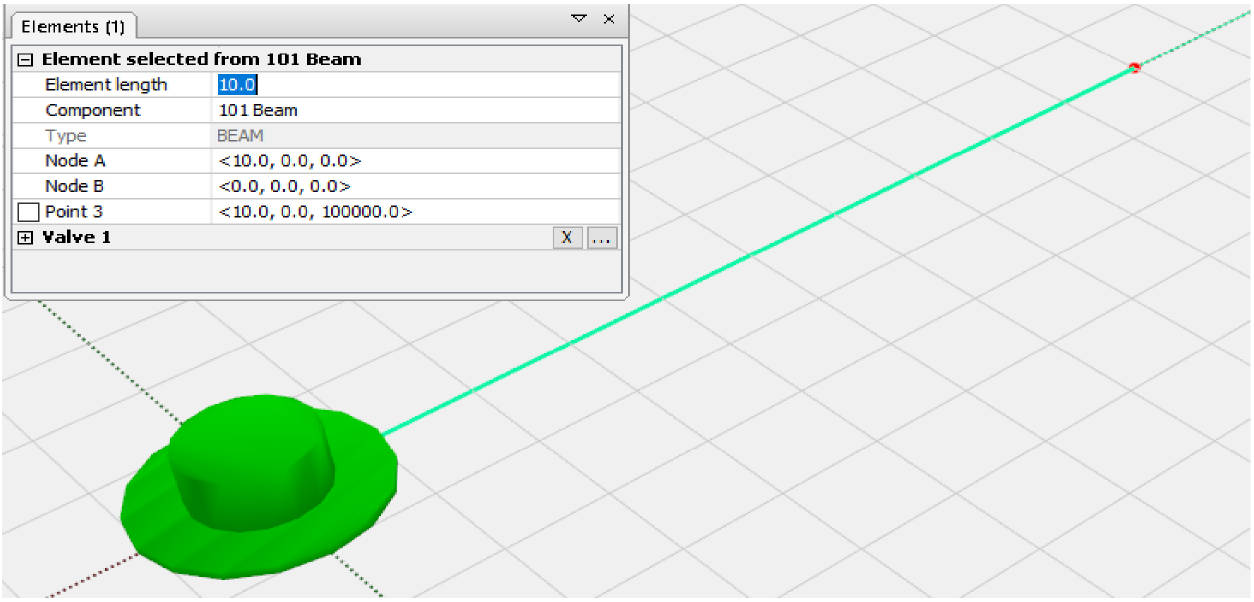
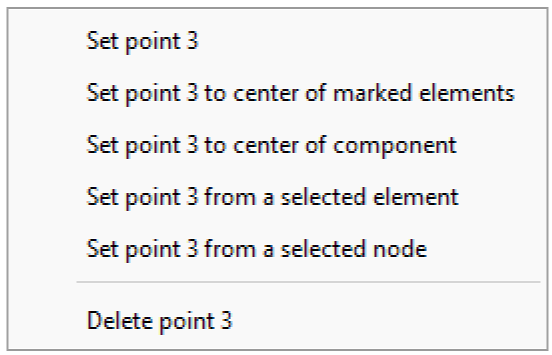


Figure 105 - Visualization of valve on beam element

4.2.5 Point 3



Point 3 is the point in which trusses and beams will have their z-direction orientated, this coupled with the local x-axis that goes from node A to node B gives the local y-axis and the cross-section of beams is orientated after this.

AquaEdit will automatically calculate this number. For all elements, but vertical ones, point 3 will be based on the position of node A and added a large z-value so that the point 3 will lie high above the element and thus gives a correct local y-axis. For vertical elements this will not suffice and is a bit trickier. So AquaEdit sets point 3 for vertical elements equal to 0,0,0 for all elements not lying in the global z-axis, then the point 3 is set as 1,0,0.

The user can edit this in three ways.

Set point 3
Edit point 3 manually.

Set point 3 to center of marked elements
Sets point 3 to the center of all the currently marked elements.

Set point 3 to center of component
Sets point 3 to the center of the parent component on the marked elements.

If point 3 has been edited, it can also be deleted by using the menu “Delete point 3”, this will return to the default that AquaEdit calculates point 3 automatically.

Set point 3 from a selected element

Copy point 3 coordinates from an element that is selected in the model.

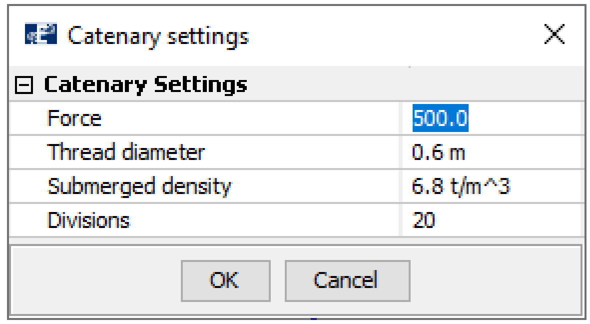
Set point 3 from a selected node

Sets point 3 of selected elements to point towards a selected node in the model.

Delete point 3

Deletes the Point 3 of the selected element.

4.2.6 Create Catenary slope



A Catenary slope allows the user to create catenary mooring system. The slope is calculated and generated based on the input as illustrated in the figure to the left.

The user must first draw a straight line in XZ- or YZ-plane (works as a guideline). Select the line **`Elements > Create Catenary slope`**. For information about theory, reference is made to (Aquastructures, 2024a).

4.2.7 Write wave reflection information

Writes data for reflected wave on an element for each timestep in the analysis. Useful when investigating drift forces and the effect of reflected waves on structures. Selecting this option, an element decorator is made available in the Decorators window.

When analysis is conducted, a file <analysis name>drift.txt is generated where the data is found. Note that this option is only available for one element at a time. If selecting a new element, the existing node decorator will be overridden.

4.2.8 Flip selected elements

This will cause the nodes to swap places, node A will move to node B and vice versa. Useful in cases where the placement of node A and B is significant, such as for hinge or roller. The user can then flip the elements with the wrong node to make up for this. This option is found in **`Commands > Elements`**.

4.2.9 Create turbine on element

The user can create a turbine on one of the element nodes by selecting **`Elements > Others > Create turbine on element`**, as seen in Figure 106.

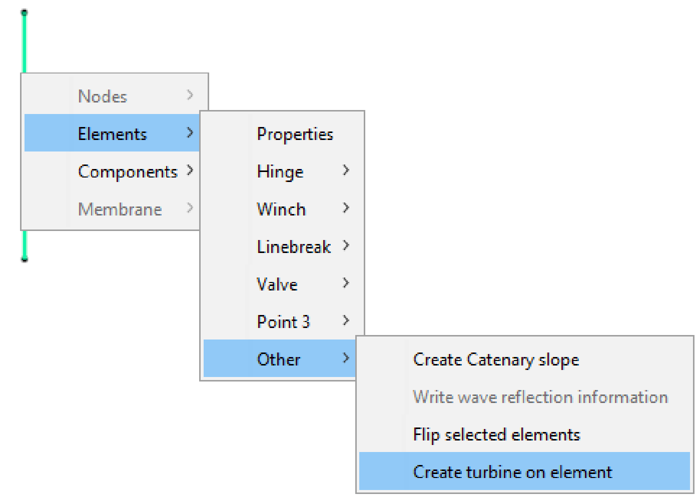


Figure 106 – Generate turbine on element

Having selected Create turbine on element, the Generate turbine tool is loaded, as illustrated in Figure 107.

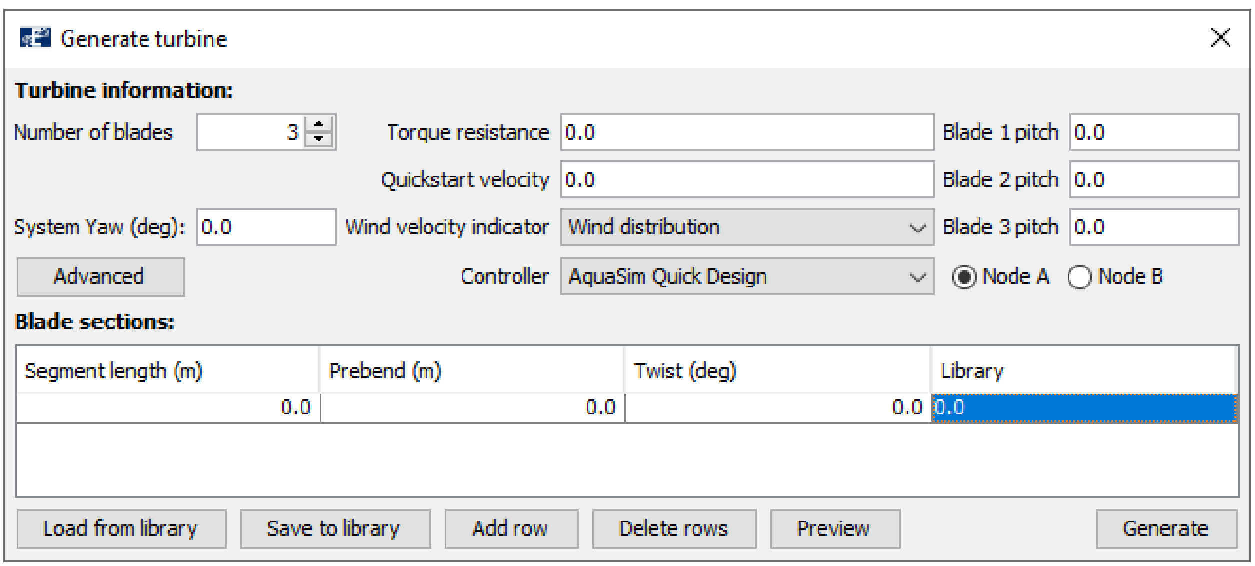


Figure 107 – Generate turbine tool

The input is described in detail in the table below.

Option	Description
Number of blades	The number of blades that is distributed around the nave. Usually, three blades are applied.
System Yaw (deg)	Orientation of the nave and system at the start of the analysis. 0 degrees means that the nave is pointing along the global x-axis (positive direction). 90 degrees mean the nave is pointing along the global y-axis (positive direction). a

Torque resistance	This is a resistance that is proportional with the rotational velocity of the nave. This parameter interacts with the Controller. Unit is Nm/(rad/s).
Quick start velocity	Velocity chosen for quick start in the analysis. When the dynamic part of the AquaSim analysis starts, the turbine blades will have this rotational velocity. Unit is revolutions per minute.
Wind velocity indicator	Options for describing how the wind flow should be distributed. Two options are available: - Wind distribution: wind flow is predicted a wind distribution (i.e. a wind profile). - Nominal constant wind: wind flow is flat (constant) and fixed in time and space.
Controller	The type of controller that should be applied. The controller regulates torque resistance, system yaw and blade pitch as the blades rotate. Three options are available: - AquaSim Quick Design: simplified controller where torque resistance, system yaw and blade pitch are constant through the analysis. - Internal: a PI (Proportional-Integral) controller. Appropriate PI-controller parameters in the Advanced menu must be defined: *Rated power*, *Rated generator speed PI kp*, *PI kI* and *PI kI, pow* must be defined. - ROSCO (Python integration): a PI (Proportional-Integral) controller. This option offers to Python and NREL ROSCO. More information about controllers is found in (Aquastructures, 2024c).
Blade 1-3 pitch	Regulates the pitch of the blades.
Node A / Node B	Which node on the selected element the turbine should be attached to. Node A is the first node that is defined when the element is generated, Node B is the second.
Advanced	Gives you access to advanced setting for the wind turbine. More information is found in the succeeding table.
Blade sections	- Segment length (m): radial length of each blade segment. Unit in meters. - Prebend (m): how much the element is directed towards the wind (if positive values) over the element length. That is, how much more upwards to the wind the outer node of the segment is relative to the inner node. - Twist (deg): Twist of each blade segment. This can be viewed as local pitch of each segment in the blade. - Library: the path to where the cross-section properties of each blade segment are located. This is coded to point to AquaSim\Library\Beam-folder.
Load from library	Enables loading turbine data from a library.
Save to library	Enables saving a generated turbine to library.
Add row	Adds a new row in the Blade sections-part.
Delete rows	Deletes row in the Blade sections-part.
Preview	Preview the turbine in the 3D window with the current settings.
Generate	Generates the turbine in the 3D window, with associated component groups and node decorators.

Selecting Advanced, options as shown in Figure 108 will be available.

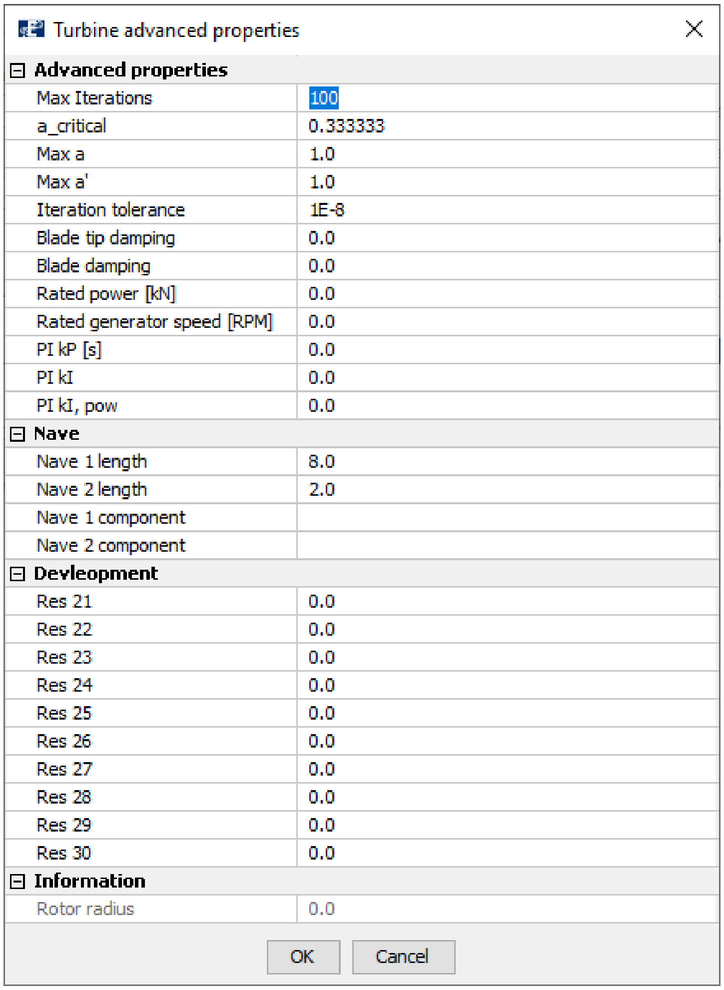


Figure 108 – Advanced options in the Generate turbine tool

Option	Description
Max iterations	Maximum iterations used in the process of finding *a* and *a'*.
a_critical	The critical value for *a*. Normally, this is 1/3.
Max a	Upper bound for *a*. Normally, this is 1.2 and the lower bound is 0.
Max a'	Upper bound for *a'*. Normally, this is 1 and the lower bound is 0.
Iteration tolerance	Tolerance in the iteration process to find *a* and *a'*.
Blade tip damping	A type of damping that scales quadratically with radius relative to the tip radius. Works in the axial direction of the blade elements.
Blade damping	Not implemented as per version 2.19.0.
Rated power [kN]	Is the maximum power output that a wind turbine is designed to generate under standard or rated operating conditions.

Rated generator speed [RPM]	The rotational velocity at which the generator of a wind turbine is designed to operate most efficiently to produce the rated power output.
PI kP [s]	Factor for proportional term in PI-controller. Responsible for providing an immediate response to the current error, which is the difference between the desired set-point and the actual output or process variable. The proportional action is directly proportional to the magnitude of the error.
PI kI	Factor for integral part of PI-controller. It addresses accumulated or persistent error over time. It integrates the error over time, providing a control action that is proportional to the cumulative sum of past errors. This part relates to spin velocity.
PI kI, pow	Factor for integral part of PI-controller. It addresses accumulated or persistent error over time. It integrates the error over time, providing a control action that is proportional to the cumulative sum of past errors. This part relates to power.
Nave 1 length	Length of the nave-part that is fixed. Will correspond to *NaveInner* in the components window when having generated the turbine.
Nave 2 length	Length of nave-part that is rotating. Will correspond to *NaveOuter* in the components window when having generated the turbine.
Nave 1 component	Library path to where the cross-sectional properties of *NavenInner* is stored.
Nave 2 component	Library path to where the cross-sectional properties of *NavenOuter* is stored.
Res 21-30	Dummy parameters. Not in use as per version 2.19.0
Rotor radius	Distance from nave to blade tip.

4.3 Components

4.3.1 Hide component

Hides the parents of the selected elements.

4.3.2 Select

Selects the parents of the selected elements.

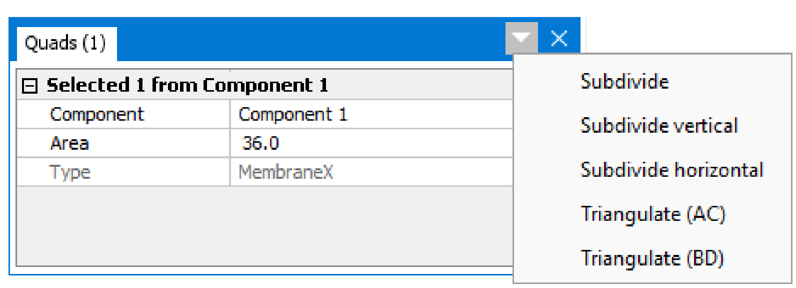
4.3.3 Edit

Brings up the property window for the component.

4.4 Membrane

When one or more quads (Membrane X) are selected the Object property Window will display the properties of the quad(s).

This shows the component name, the area and component type. The panel also enables to subdivide the quads, this is explained in section 3.19.



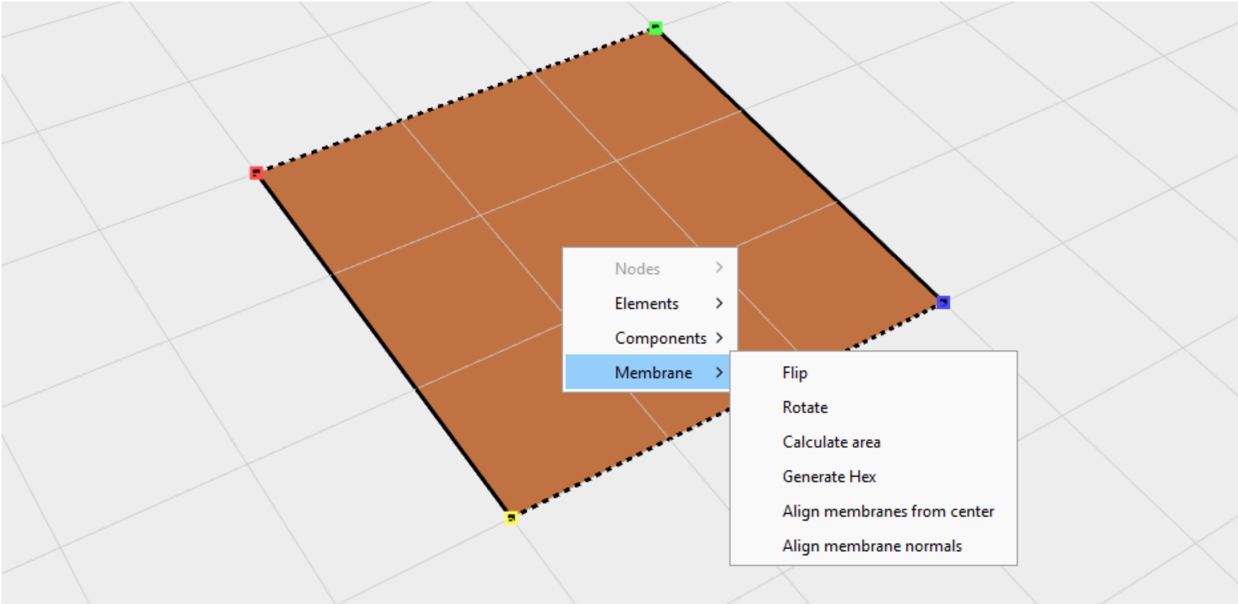


Figure 109 – Context menu for quads (Membrane X)

Figure 109 shows the menu when the quad is selected and right-clicked in the draw window. This menu gives the possibility to flip, rotate or calculate the selected quad(s). In addition, one can generate hexagonal masks and adjust the alignment of the quad Normals.

Quad Normals is displayed by selecting **`Normals`** from the Toolbar menu and click on a quad in the draw window. The green axis is the local x-direction, the red is y-direction and blue z-direction, as seen in Figure 110.

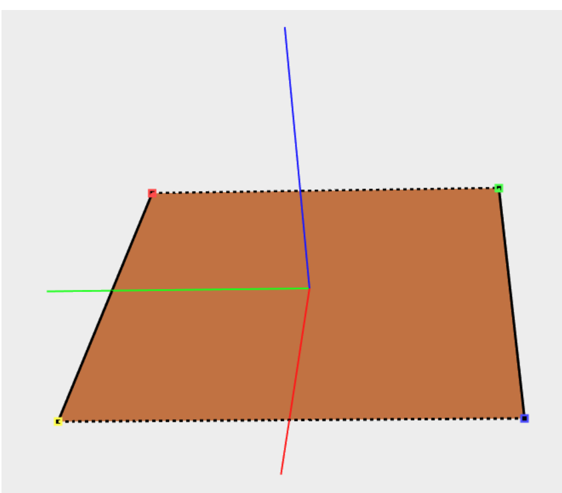


Figure 110 – Quad Normals

4.4.1 Flip

This will cause the quad to flip. Node 1 will switch place with Node 2, and Node 3 with Node 4. And vica versa. This is illustrated in Figure 112. Useful when you need to change the direction of the quad Normal.

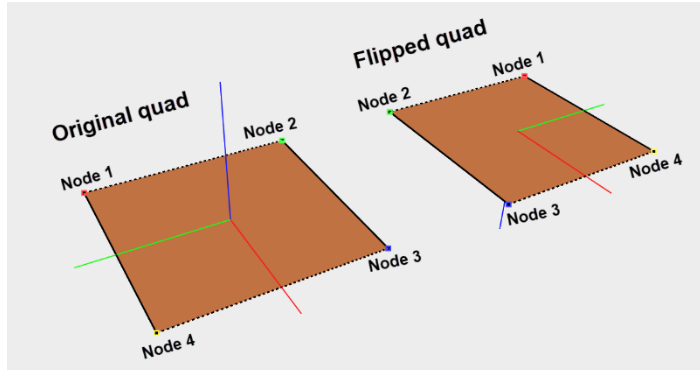


Figure 111 – The use of Flip

4.4.2 Rotate

This will cause the quad to rotate. The rotation is clockwise as illustrated in Figure 113.

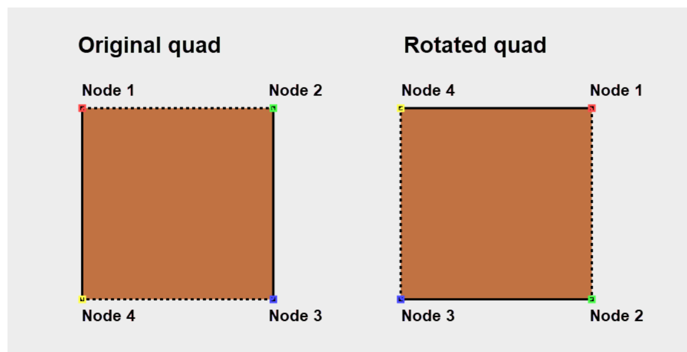


Figure 112 – The use of Rotate

4.4.3 Calculate area

Calculates the area of the selected quad(s). The unit is [m2]

4.4.4 Generate Hex

Opens the Generate Hex Tool for modeling of hexagonal shaped masks. Read more in section *Generate hex*.

4.4.5 Align membranes from center

Aligns all quads to point in the direction from center. The center is the geometrical center of the selected quads, and is the sum of the largest x, y and z in your selection divided by 2. The quad Normal (blue line) will then point in the direction away from the center. This is useful in cases where the Normal points in different directions, as illustrated in the figure below.

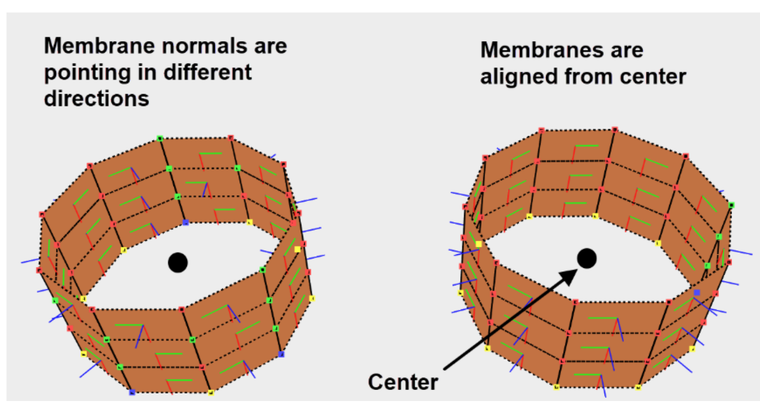


Figure 113 – The use of Align membranes from center

4.4.6 Align membrane normals

Align all Normals (within the same component group) according to the selected quad. How it works: select one quad with a Normal that points in the direction you want, right click this quad > Membrane > Align membrane normals. What happens is that the direction of the Normal (blue line) to the neighboring quad is checked and adjusted according to the selected quad. Then the next element is checked and adjusted according to the neighboring element, and so on until all quads within the component group is have been checked an adjusted. This is useful when having complex geometry and the direction of the Normals matter.

5. Menus

There are 6 main menus: File, Edit, Commands, Tools, Windows and Help.

5.1 File

New

This will create a new model. If there is an open model with changes made, the user will get a question asking to save the model.

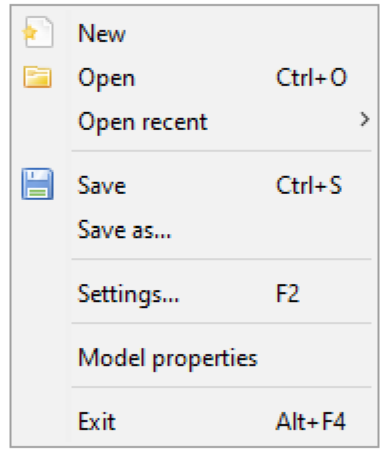


Figure 114 - File menu

Open

Opens a dialog where the user can choose which saved model to open.

Open recent

This will contain a list of the ten most recent models worked on. This list can be cleared by clicking “Clear recent files” under the recent models in the list.

Save

Saves the model. If the model has not been saved to file before, the user will get a dialog asking to choose a file name for the model.

Save as...

Allowing the user to save the model to a file name of the users choosing.

Settings

Open the settings window, see chapter Settings.

Model properties

Opens a window where the user can input and view some properties of the model, see Figure 115.

Exit

Closes the application. If there is a model with changes the user will get a question to save the model before exiting.

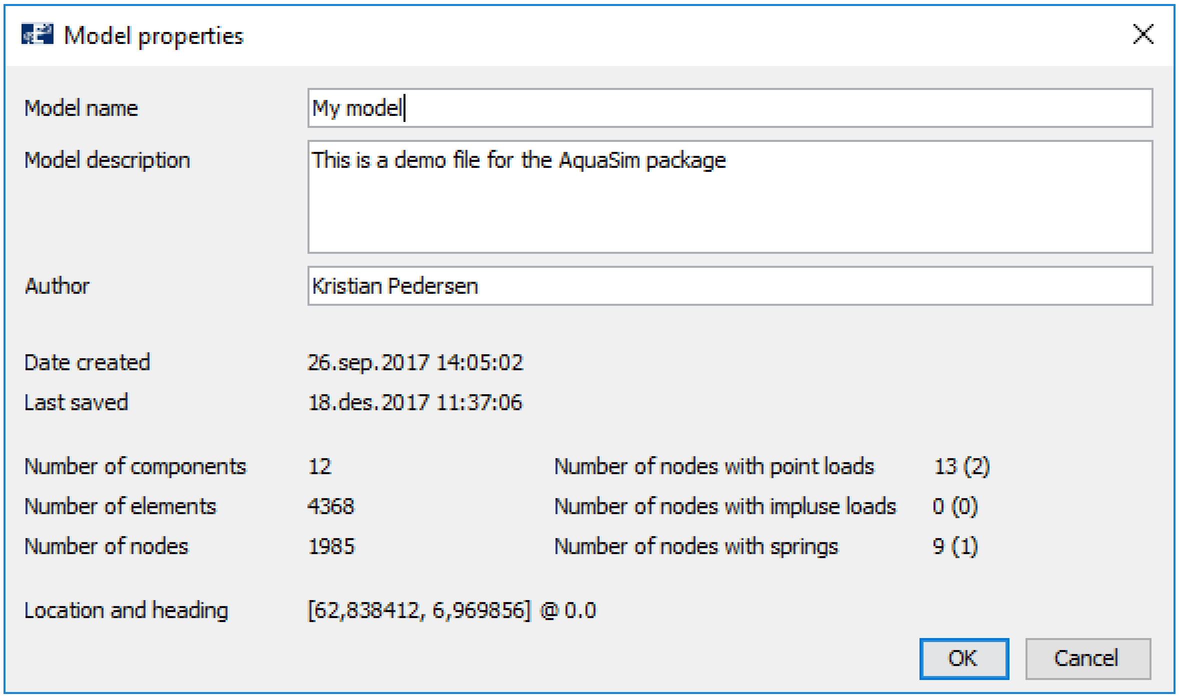


Figure 115 - Model properties window

5.2 Edit

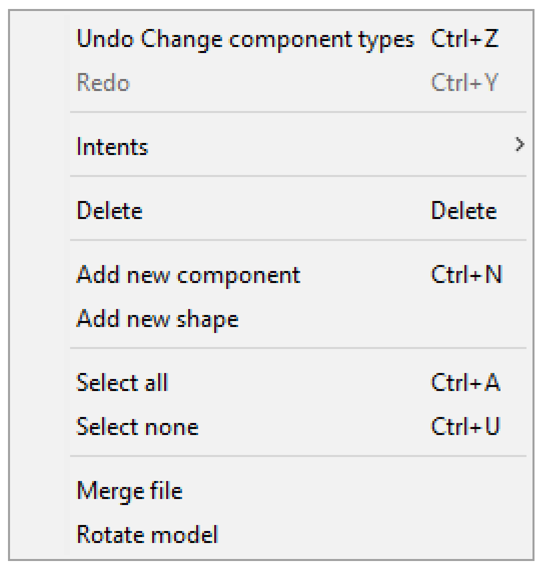


Figure 116 - Edit menu

Undo
Undoes the last action in the model.

Redo
Redoes the last undo.

Intents
Same as covered in Functions and intents.

Delete
Deletes selected nodes/elements.

Add new component
Adds a new component.

Add new shape
Adds a new shape component.

Select all
Selects all nodes/elements, based on what is already selected in the model.

Select none
De-selects all selected nodes/elements.

Merge file
Open another model and merges the current model with this.

Rotate model

Rotates the entire model clockwise based on input from user, given in degrees.

5.3 Commands

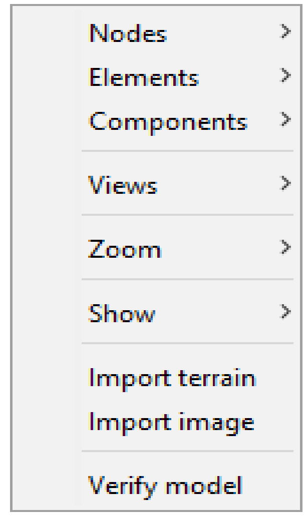


Figure 117 - Commands menu

Nodes

See chapter Nodes.

Elements

See chapter Elements.

Components

Some of the sub-menus are covered in the chapter Components.

Views

Mostly covered in chapter 2.5, but this menu gives a few more options to view the model from other viewpoints.

Zoom

Gives the user some different ways of zooming. E.g. to the largest or smallest x-values, to the middle of the model or zoom the model to fit in the current view. Keyboard shortcuts are also shown for using these menus fast via keyboard.

Import terrain

See chapter Import terrain.

Import image

Gives the user the option of importing an image file that will lay as a shape component with the center of the image in the origin.

Verify model

Checks the model for errors and warnings. How much this menu will check and how thorough it will be can be edited in **`Errors and warnings`** under **`Settings.`**

5.4 Tools

Scripts

This shows all the scrips available in the scripts-folder in the AquaSim-folder located usually in My Documents.

Generate net

Creates an optimized net – just the bottom or the net in its entirety. This tool is found in **`Tools > Generate Net`**.

When generating the bottom, the Net generator takes advantage of the explicit membrane type to allow for triangles to be generated to optimally fill the geometry of a bottom. The net sides are generated with the convention membrane component type. See Figure 119 and the following table for description of the input to the generator.

See Figure 121 for an example of a net generated with triangle membrane elements included.

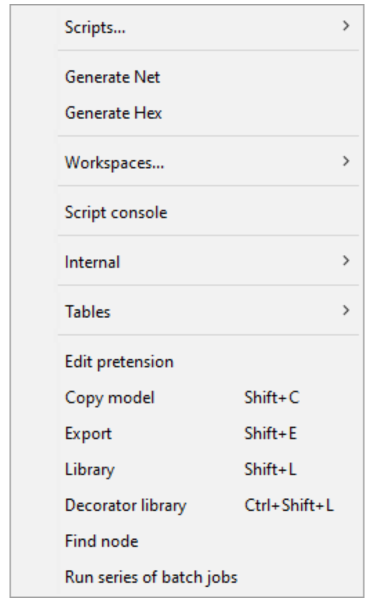


Figure 118 - Tools menu

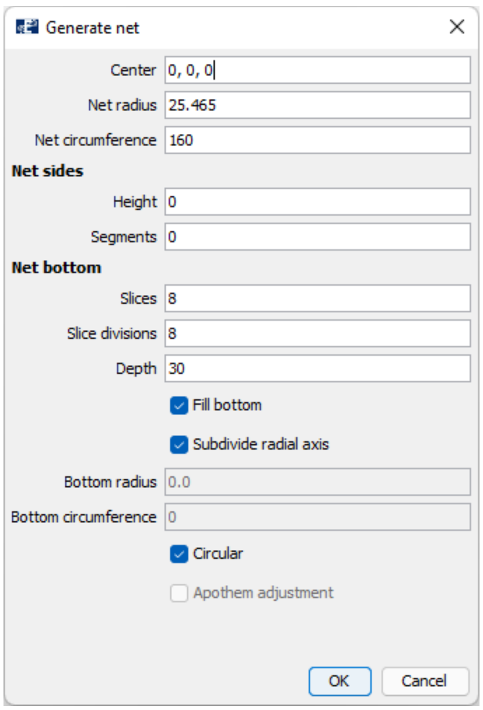


Figure 119 - Net generator input

Properties	**Description**
Center	This marks the center of the net. This is also the upper boundary of the net.
Net radius	The radius of the net.
Net circumference	The circumference of the net.
Net sides	
Height	If a straight-sided net is to be generated. This parameter defines the height of the sides.
Segments	Number of elements along the net height. See example in Figure 122.
Net bottom	
Slices	The number of “wedges” the net is cut into. Total amount of outside nodes is Slices * Slice division.

Slice divisions	The number of nodes to split each "wedge" into. Note: Amount of tangential lines depend on the Division resolution.
Depth	The depth of the imagined center point of the net.
Fill bottom	Enables filling the bottom of the net using triangles.
Subdivide radial axis	Splits each element into 2 on the tangential axis.
Bottom radius **Bottom circumference**	The net bottom can be generated with an open tip. Applying this, the user can define the radius or circumference of this opening.
Circular	Circularizes the mesh.
Apothem adjustment	Creates a non-circular net with number of sides according to the number defined in "Net division". Example: if *Net divisions* =8, then an octagon will be generated, see Figure 120.

Figure 121 provides an example of appliance of the tool where only bottom is generated.

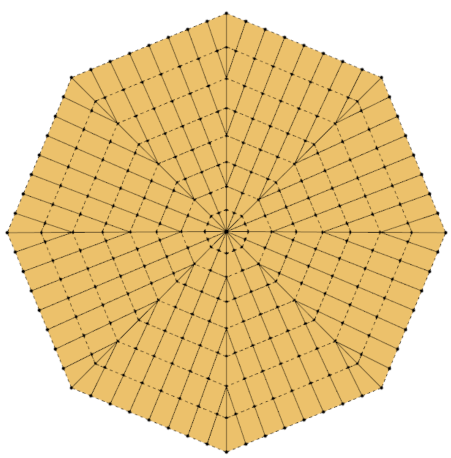


Figure 120 – Example of Apothem adjustment

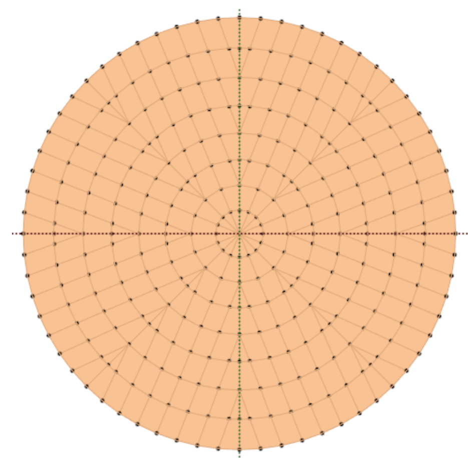


Figure 121 - Example of net bottom generated by the generator

Figure 122 provides an example of appliance of the tool with net bottom and sides.

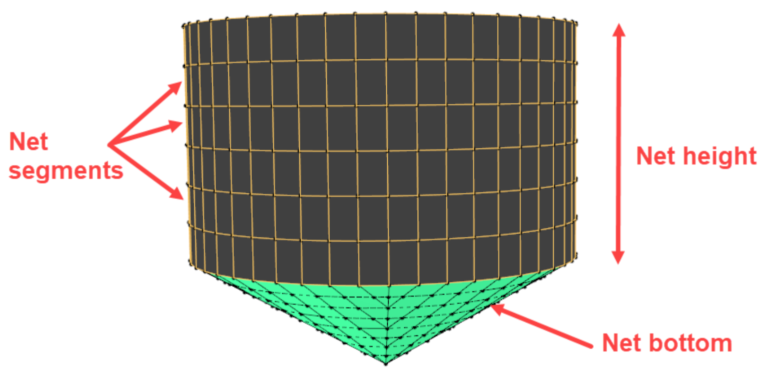


Figure 122 – Straight-sided net generated applying the net generator

Generate hex

Creating a hexagonal net can be done with the Generate Hex. This is found in **`Tools > Generate Hex`**.

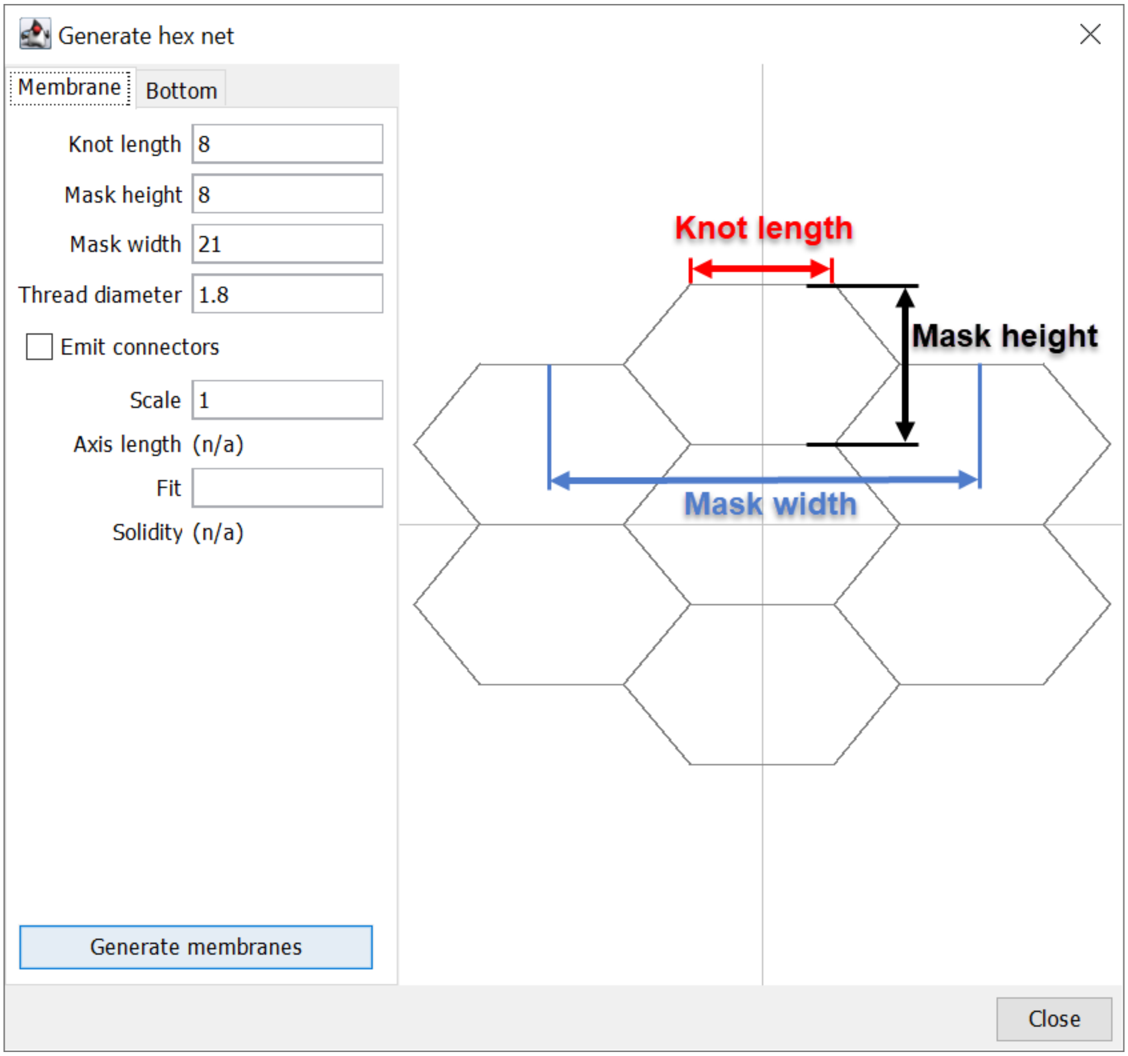
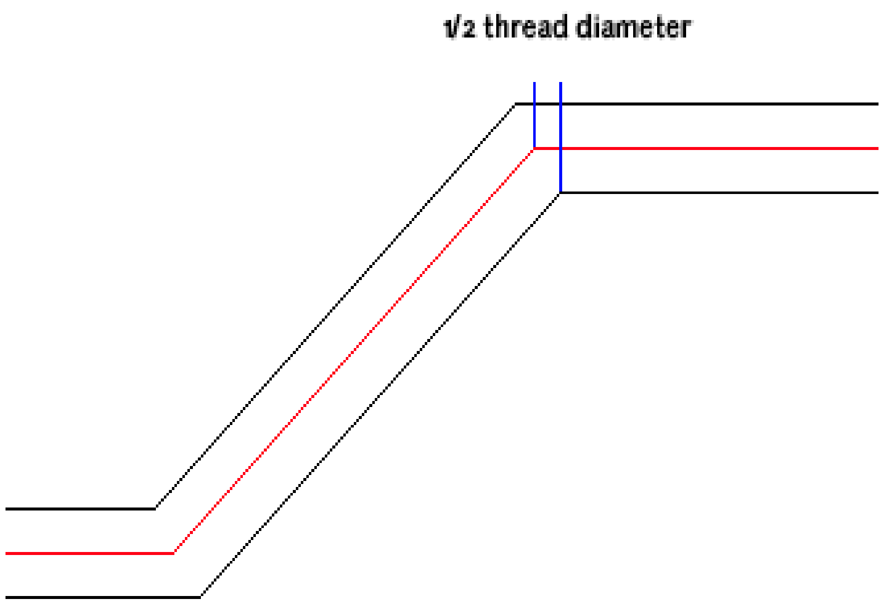


Figure 123 - Generate hex dialog

Each hex is represented by their knot length, mask height, and mask width. The relationship between these are shown in the diagram above. Noted that Mask width is the length equal to the length the middle of two masks.

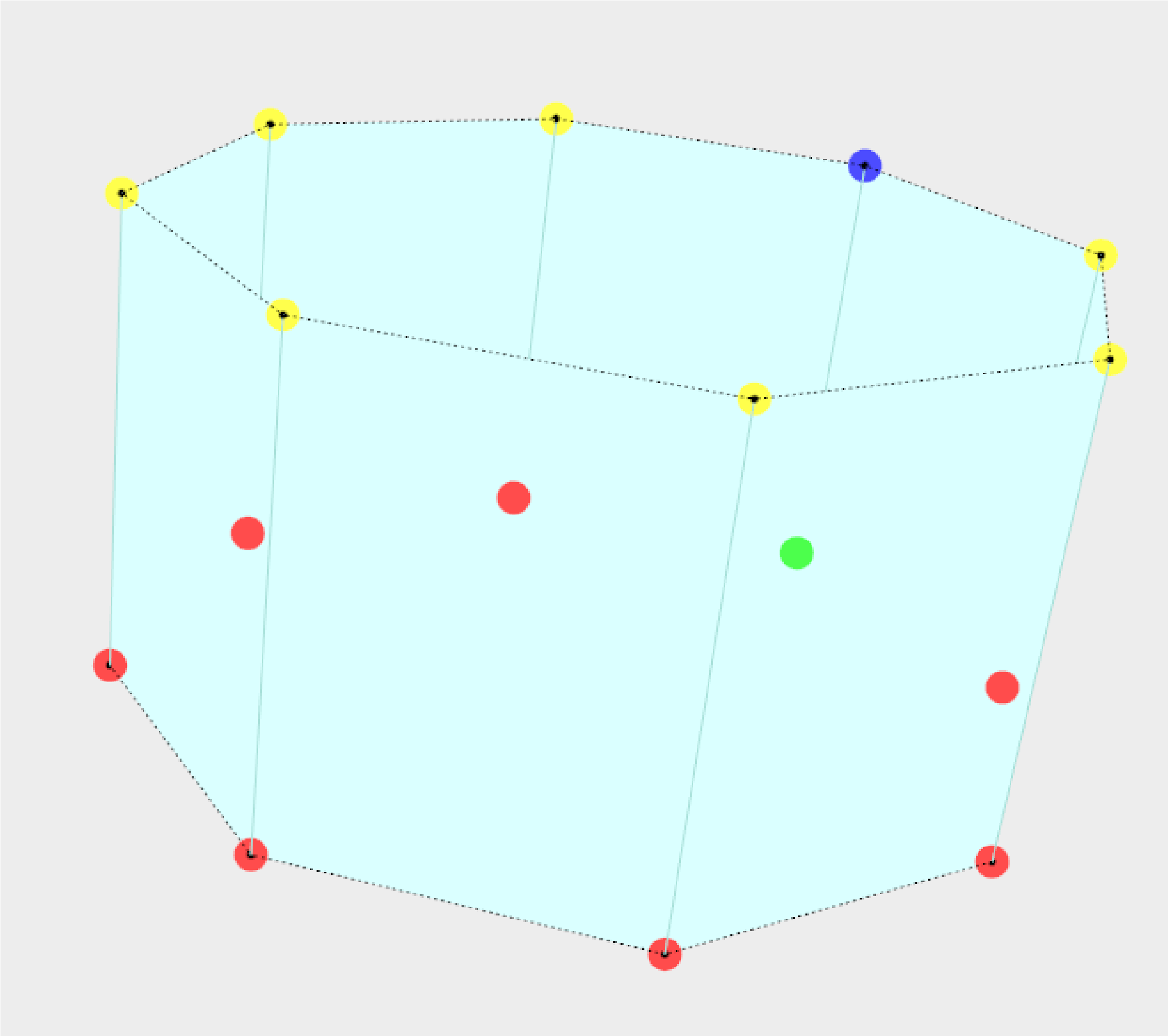


Because AquaSim describes geometry with lines with no thread thickness, thread diameter modifies the width of knot length and mask width to take into account the thickness of the thread diameter by moving the connection points by ½ thread diameter.

Generating membranes are done by first creating **Membrane X** elements that will be used for the template to generate the hex elements inside.

The simplest way of generating a net is to use Draw Circle and create the outline of the net, use Extrude to generate the height of the net, and convert the component to **Membrane**, and then to **Membrane X**. Then select all the elements (right click component, operations > select).

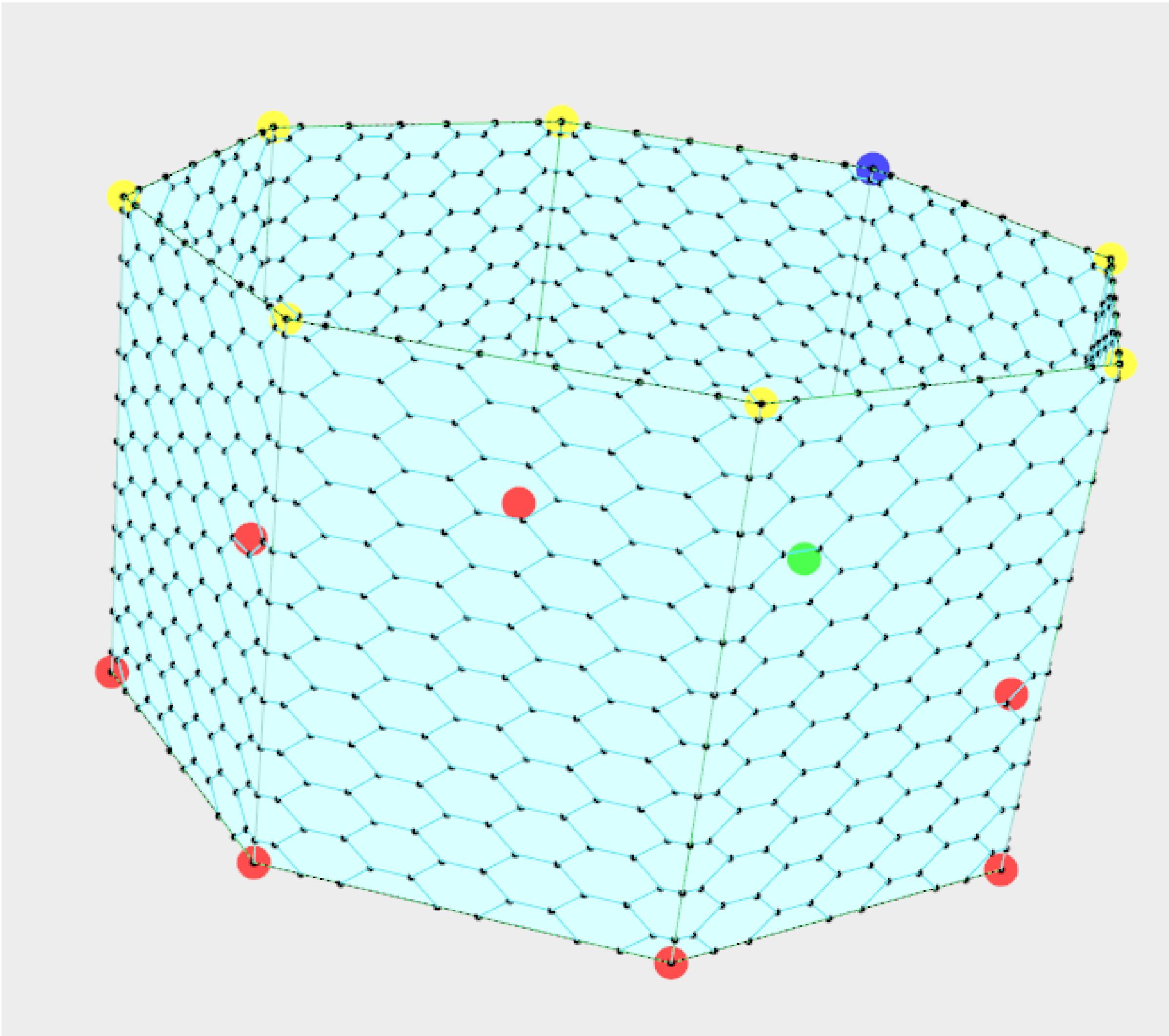
This should yield something akin to the example picture.



Bring up the hex membrane window and enter the wanted parameters.

Axis length should now be calculated from the size of the first membrane element that was selected (it is strongly advised that all Membrane X elements are the same size in this procedure). Emit connectors enables generating ‘help trusses’ in net hex net.

The Fit field can be used to set the number of hexes per axis length of the element (Note: a hex is the length of mask width, which technically makes two masks. This is done to have the edges of hexes meet up when generating nets).



After generating the hexes, we should have something similar as the picture above. The nets should have created a good fit around the edges.

Note: For generating a bottom for this, use the Bottom tab and set Fit to same as when generating the net, set the radius to the same as when generating the circle for the net, make slices the same amount as segments for the circle, and slice divisions to 0. More information on this in the Generating bottom segment under.

After generating, the Membrane X component will still be in the components list. This can either be disabled or deleted, it is no longer needed.

A scale can be entered manually if wished. Scale will be equal to (mask width * 1000 / scale). Generally using Fit is advised.

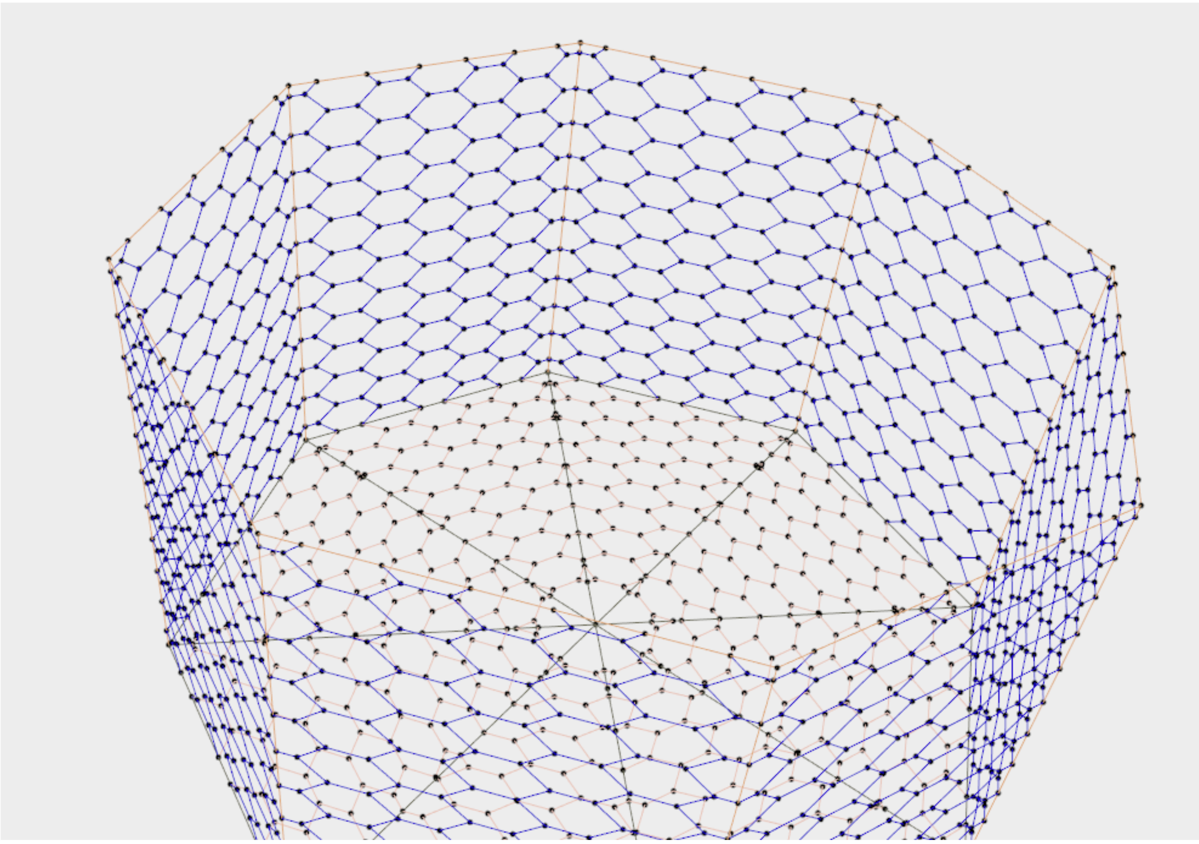
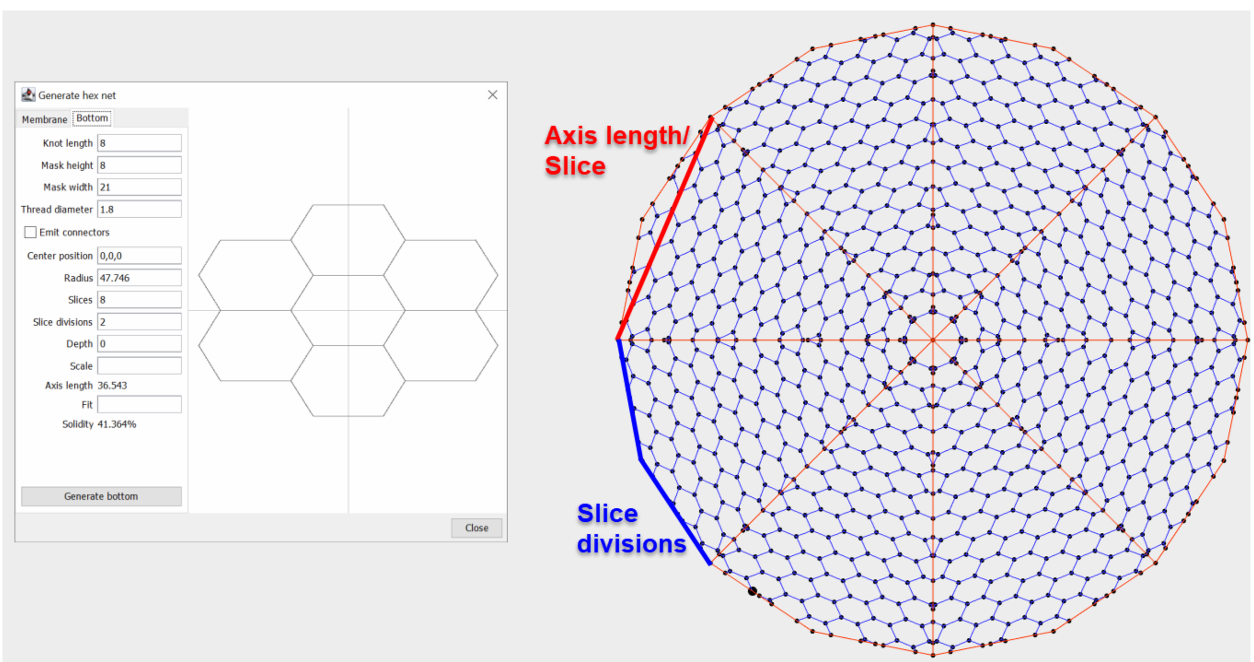


Figure 124 - Example of a completed net

For generating bottoms, the geometry settings are the same as for general hex nets.



A bottom is generated at a center position with a specific radius. These should match the center and depth of the start of the bottom, and the radius of the wanted net.

In addition, the number of slices and slice divisions can be specified. The example drawing is generated with 8 slices (represented by the dark red lines) and each of these slices has been divided into 2.

Note: In order to get the best fit with circular nets we recommend that the slice divisions are set to 0 and extra slices are generated. If slice divisions are used the net edges will not match up and will have to be sown together manually at the edges.

As with the generation of membranes, the bottom also has an axis length (which corresponds with the length of a slice, see drawing) which helps in setting the number of hexes generated per slice.

By entering a number in fit scale will automatically update to be axis length / fit, this will make exactly fit number of hexes per slice. The example drawing has specified 3 masks (*where a mask is specified by mask width*).

A scale can be entered manually if wished. Scale will be equal to (mask width * 1000 / scale).

Manual drawing or editing Membrane 6 components / elements is possible. To make the component use the generate operation as normal. All drawing tools are applicable to the lines for Membrane 6 elements. The main difference is the “Is vertical” toggle on lines that are classed as vertical. Every Membrane 6 “element” must be made up by vertical and “non” vertical lines. Vertical elements are the slanted sides, there should be 4 vertical and 2 horizontal for each hex.

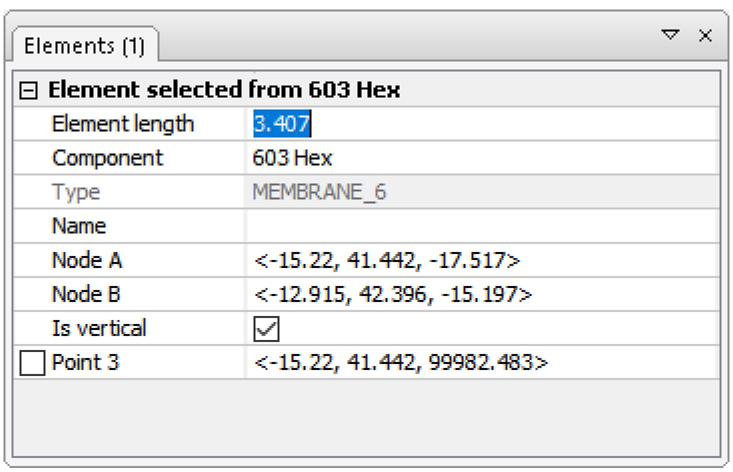


Figure 125 - Element property of membrane 6 element

Workspaces

The user can save the layout and position of all the movable windows, and the saved layouts will be shown here. The layout “Default” will always be there, it will reset the windows to the default layout on the current monitor. Workspaces can also be deleted from the menu option at the bottom. Note: Default cannot be deleted

Script console

Opens the script console. Here the user can open, save, edit and run scripts.

Tables

Tables enables the user to get overview of tables that are generated in the model.

- Show RAO tables: shows all RAO-tables in the model. The tables can be accessed by double-click or select Edit.
- Show non linear tables: shows all the non linear tables in the model.
- Show sloshing tables: shows all tables for sloshing parameters in the model.
- Show component contacts: shows all component contacts in the model.

Edit pretension

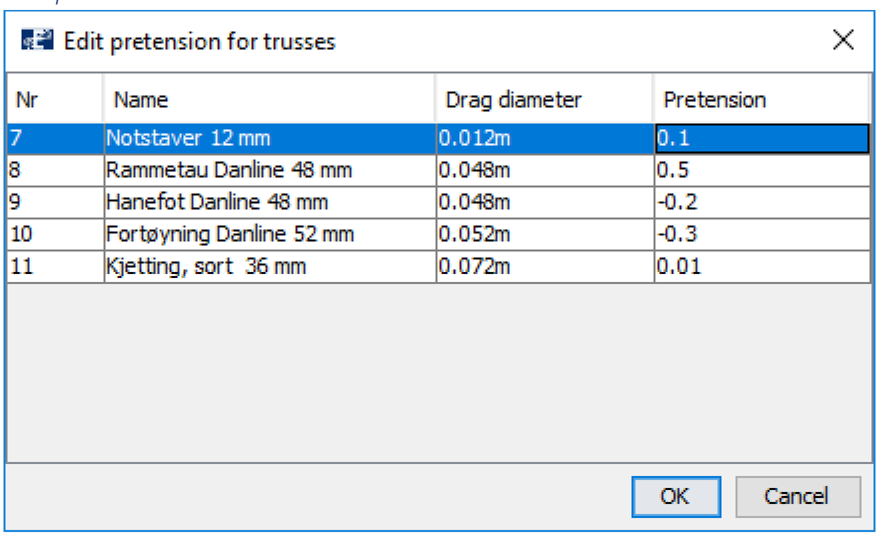


Figure 126 - Edit pretension window

This window will show all the trusses in the model. Here it is possible to easy edit the pretension for all the trusses at once instead of editing one at a time.

Copy model

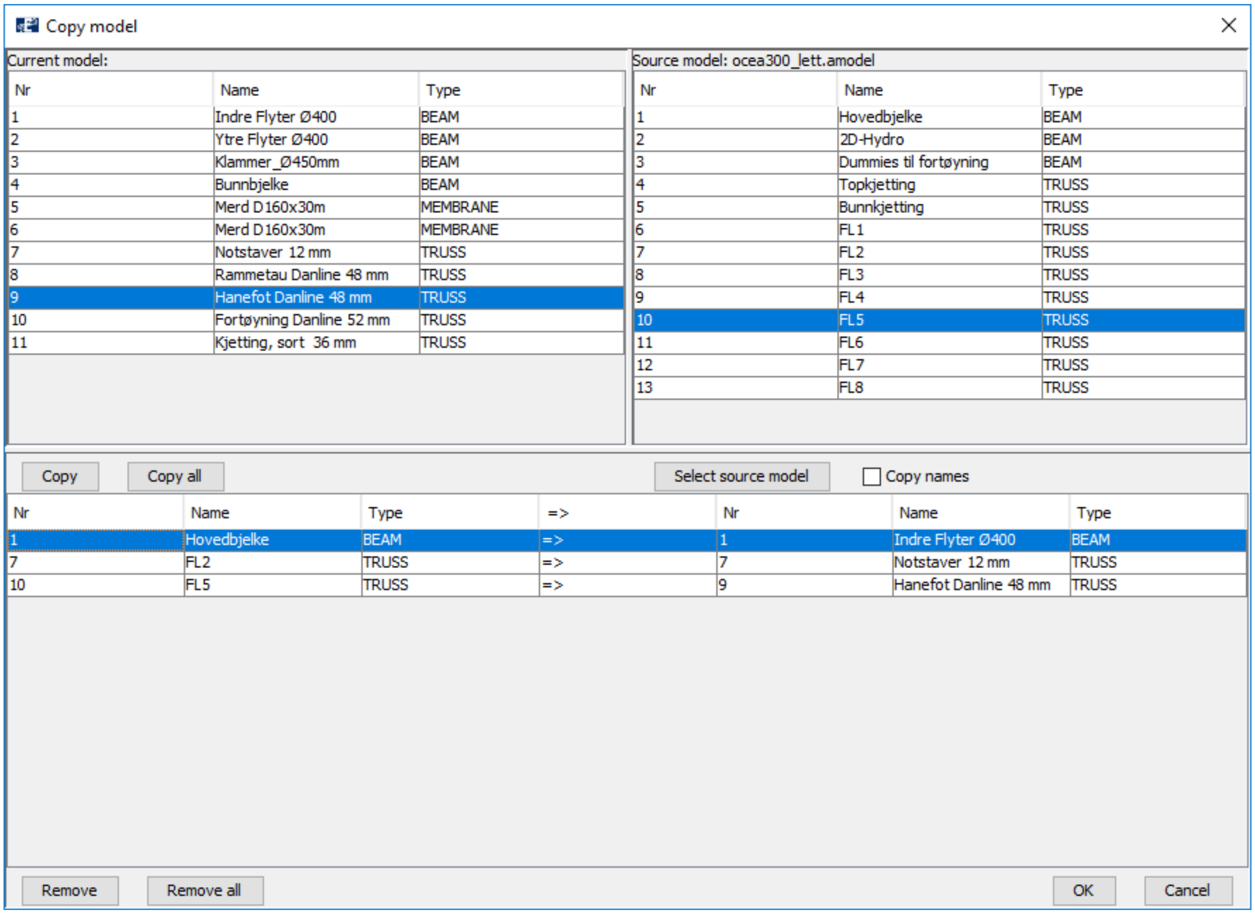


Figure 127 - Copy model window

This makes it possible to easy copy component properties to the current model from another model, or from the current model if several components should have the same properties.

The current model's components can be seen on the left, on the right is the source model one can copy properties from. As in the component list, components with invalid properties are shown in cursive.

Mark one or one and make a copy connection by clicking "Copy", the copied component lines will appear in the window below. Pressing "Copy all" will copy all components with corresponding numbers automatically.

Export Shift +E

Enters the window for environmental conditions, see chapter Export for more information.

Library Shift+L

Enters the Library window, see chapter Library for more information.

Decorator library

Enters the Decorator library, see chapter Decorator library for more information.

Find node

After exporting, a node cache with node numbers is created. It is possible to search up nodes using node numbers after export using this option.

Run series of batch jobs

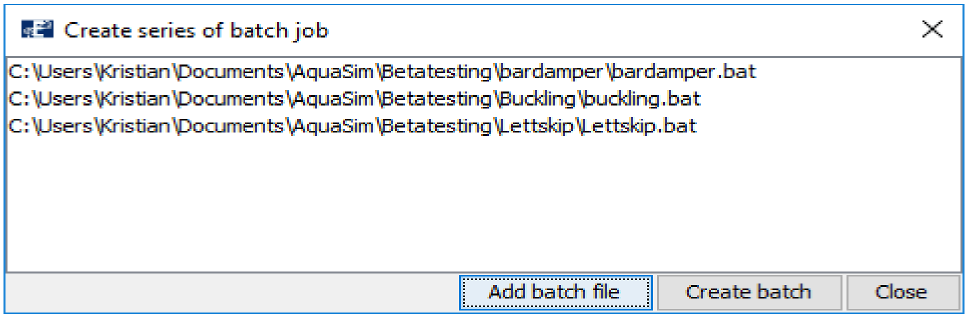


Figure 128 - Series of batch jobs window

If it is wanted to run several batch jobs after one another, then it can be set up as a series using this option. Clicking “Add batch file” will add an existing bat-file to the list and clicking “Create batch” will create a master batch-file that runs all the batch jobs in the list after another. Note: Since they are run one after another, if one analysis fails then the main batch job will stop and wait for user interaction.

Open analyse window

Loads the Analyse window, see chapter Analyze immediately after export for more information.

5.5 Windows

This will show the tool windows that are currently shown and not shown. Clicking on one of the items in the list will show/hide the corresponding tool window.

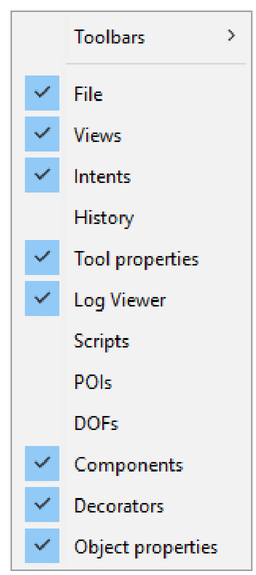


Figure 129 - Windows menu

5.6 Help

Welcome

Will show the welcome window and change log.

User manual

Opens this user manual in a pdf viewer, if installed.

Documentation center

Opens the online Documentation center for AquaSim. Tutorials, papers and reports available for the user.

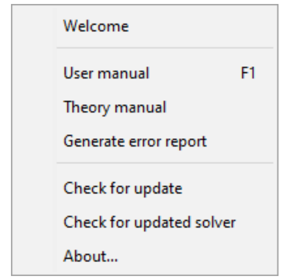


Figure 130 - Help menu

Theory manual

Opens the theory manual, (Aquastructures, 2024a), in a pdf viewer, if installed.

Generate error report

The user can send an error report by mail to the support team by using this option. Write in the issue, email-address and optionally add files to be included in the mail.

Check for update

Checks if there is an update to the AquaSim package available.

Check for updated solver

Checks if there is a new solver available for downloading.

About

Shows information about AquaEdit. And gives the option to view and enter a new license key.

6. Library

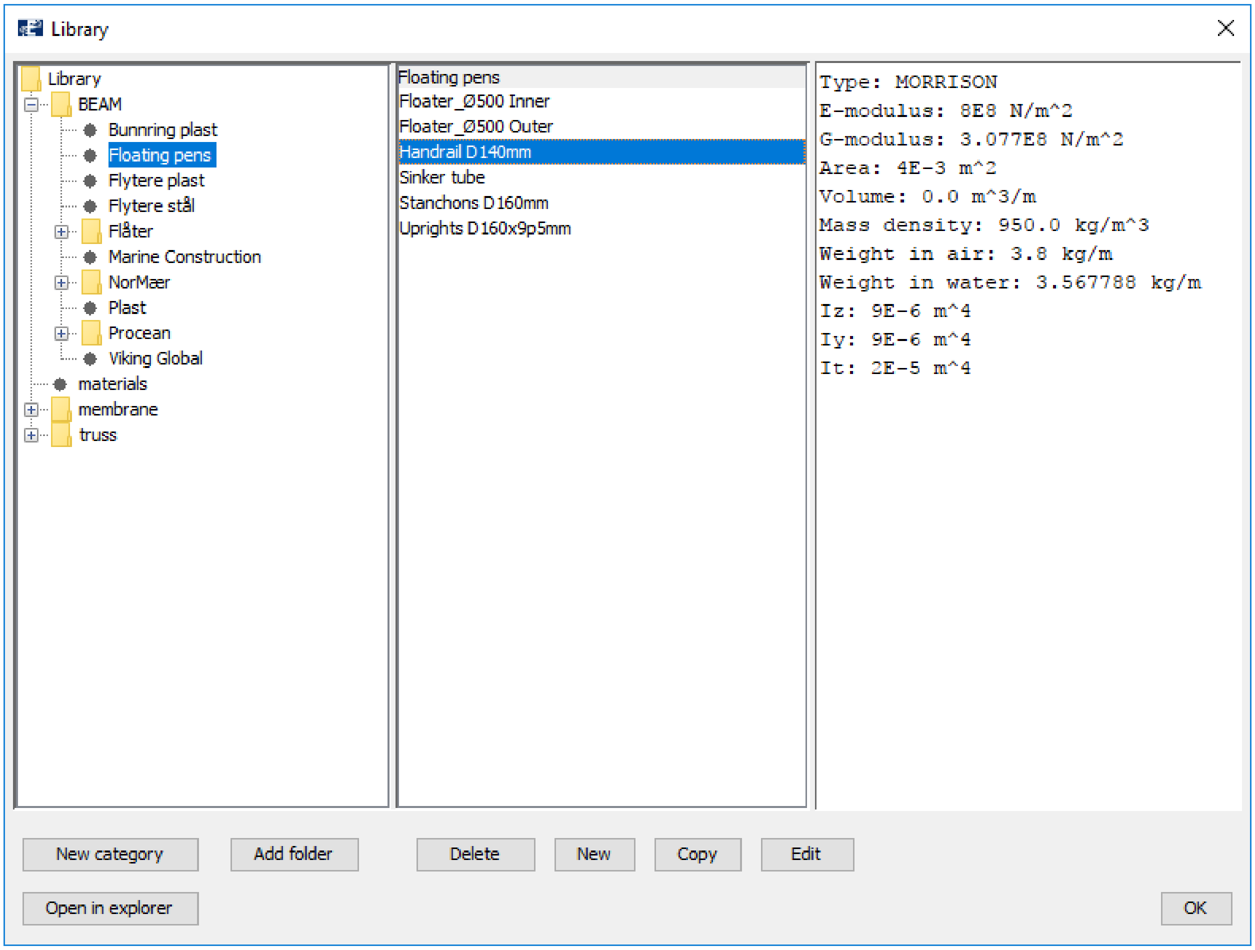


Figure 131 - The library window

The library window will show the content of the library folder, which can be edited in **`Settings > General`**. The installation of AquaSim will by default install an empty folder named library in My Documents/AquaSim, if this folder already exists then nothing will happen. This default folder contains four sub-folders; beam, materials, membrane and truss.

The left tree shows the folders and sub-folders. If the selected folder contains any library files, then these will be shown in the middle list. The right list shows some properties from the selected file in the middle list.

It is possible to drag & drop items to and from the middle list. It is possible to drag directly to the component list and to the windows explorer. It is also possible to drag from Windows Explorer and into the middle list.

New category

Will create a new sub-folder under the folder selected in the left tree.

Add folder

Will add a selected folder from the computer in the left tree.

Open in explorer

Will open the selected folder in Windows Explorer.

Delete

Will delete the last selected item. If the last selected item is a library file from the middle, then only this file is removed. If the last selected item is a folder from the left tree, then this folder and all files in the folder will be deleted. Root folders cannot be deleted.

New

Will create a new library file. If one of the four default folders or their sub-folders are selected, then a new preset of that type will be created. If the root folder is another then the user will be asked what kind of library file to be created; beam, truss, membrane or material.

Copy

Will make a copy of the selected library item into a new name selected by the user. It is possible to edit the properties of the copied item before saving.

Edit

Edits the selected library item.

If the library is opened from the component list by right-clicking on an item and selecting “Library”, then the options are “Export” and “Apply”. There are no drag & drop options here.

“Export” will export the selected component properties into the library, in the chosen folder.

“Apply” will apply properties to the component from the selected library item.

7. Decorator library

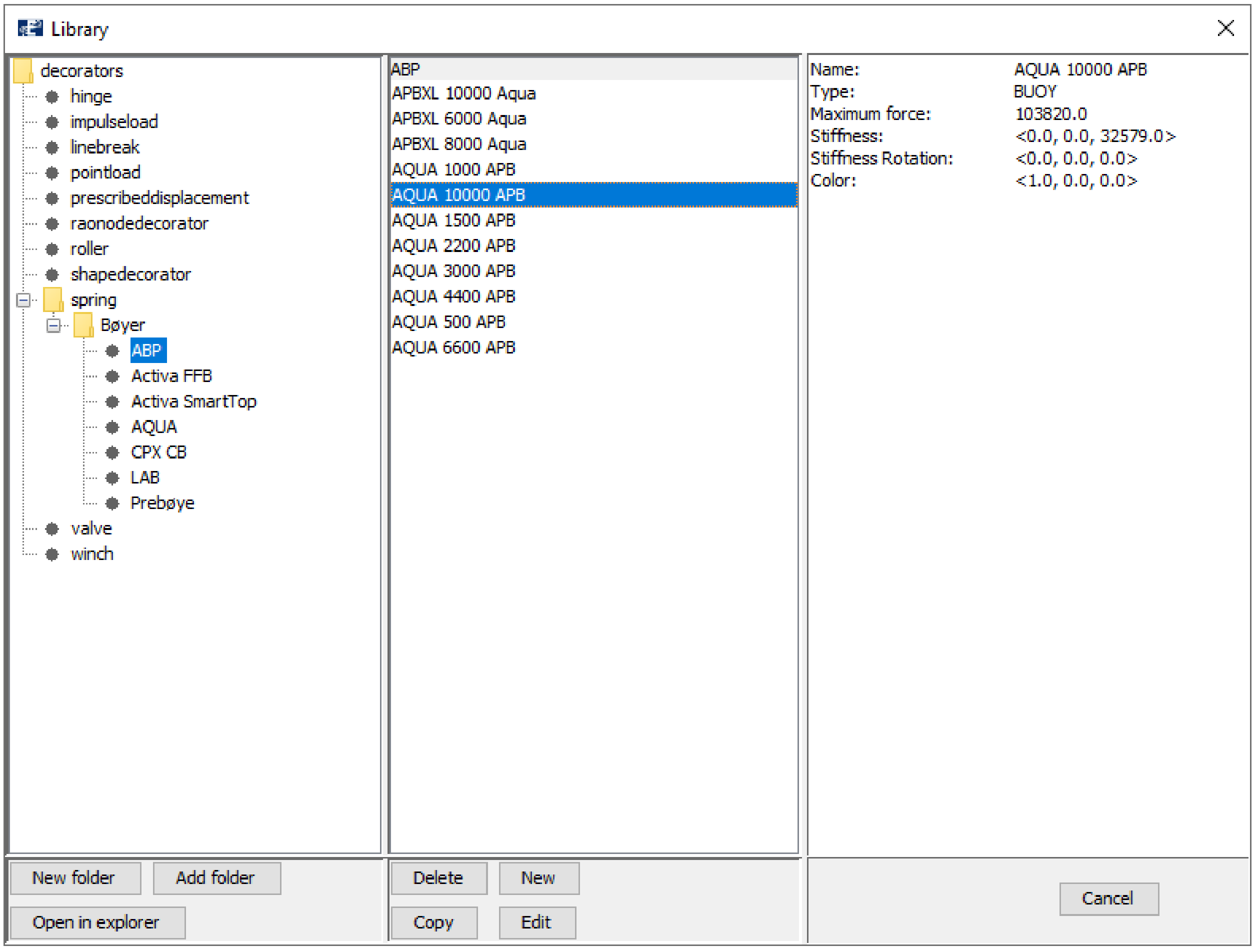


Figure 132 - The decorator library

The decorator library window will show the content of the decorator library folder, which is a folder within the library folder named "decorators". The installation of AquaSim will by default install empty folders with all the decorators having its own folder; hinge, impulseload, linebreak, pointload, presctibeddisplacement, raonodedecorator, roller, shapedecorator, spring, valve, winch.

The left tree shows all folders and sub-folders. If the selected folder contains any library files, then these will be shown in the middle list as shown in Figure 132. The right list shows some properties from the selected library file.

New folder
Will create a new sub-folder under the folder selected in the left tree.

Add folder
Will add a selected folder from the computer in the left tree.

Open in explorer
Will open the selected folder in Windows Explorer.

Delete

Will delete the last selected item. If the last selected item is a library file from the middle list, then only this file is removed. If the last selected item is a folder from the left tree, then this folder and all files in the folder will be deleted. Root folder cannot be deleted.

New

Will create a new library file.

Copy

Will make a copy of the selected library file into a new name selected by the user.

Edit

Edits the selected library file.

If the decorator library is opened from the Decorators list by right-clicking on an item and selected "Library", then the options are "Export" and "Apply".

"Export" will export the selected decorator into the library, in the chosen folder.

"Apply" will apply properties to the decorator from the selected library item.

8. Settings

8.1 General

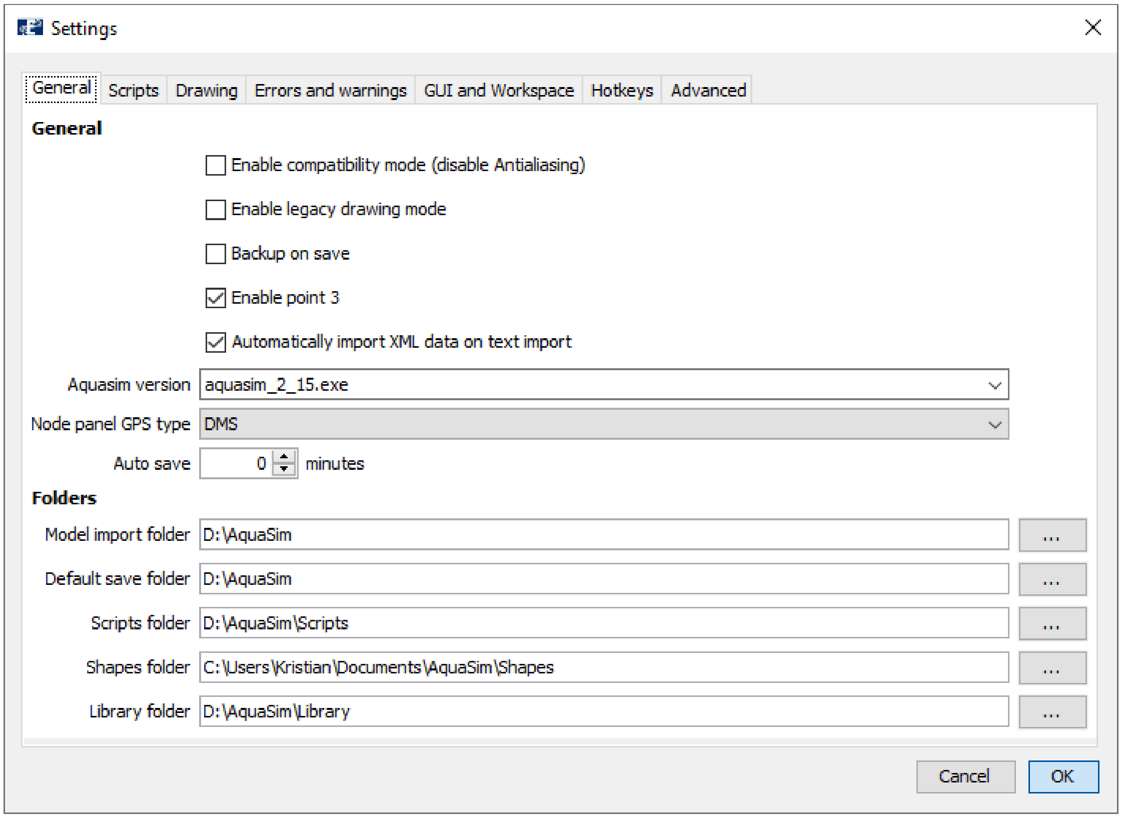


Figure 133 - General tab

Enable combability mode

This is an optional setting for some older/incompatible graphics card (esp. older Intel integrated cards). If the screen is black, try enabling this and restart AquaEdit.

Enable legacy drawing mode

The default render engine used in AquaEdit is based on shaders. Older graphic cards and some integrated graphic cards from Intel may not be powerful enough to leverage this render mode. Toggle this to go back to the old render mode. Restart AquaEdit to apply this change.

Backup on save

When enabled, a copy of any overwritten file will be made. Should something prevent the new model from saving then you will be able to find a backup of the last save file named something like: “mymodel-2018-01-15-12-39-06.bak” in your model folder.

Enable point 3

This will let AquaEdit handle point 3 information. It is recommended to keep this on.

Automatically import XML data on text import

When this is enabled, importing an AquaSim text file automatically import the companion XML file.

AquaSim version

Specifies which AquaSim executable should be used when exporting and running an analysis.

Node panel GPS type

In the properties for nodes, a GPS property will be shown if a terrain or olex plot has been imported. How the GPS value is shown can be edited by this. The values are DD, DM, DMS & UTM.

Auto save

If this is set higher than 0 AquaEdit will attempt to save the model every x minutes into a separate file, named “mymodel.amodel.autosave”. If something goes wrong, then it is easy to load the autosave file and not lose any data.

Folders

The paths to the different folders used by AquaEdit.

8.2 Scripts

Not currently in use.

8.3 Drawing

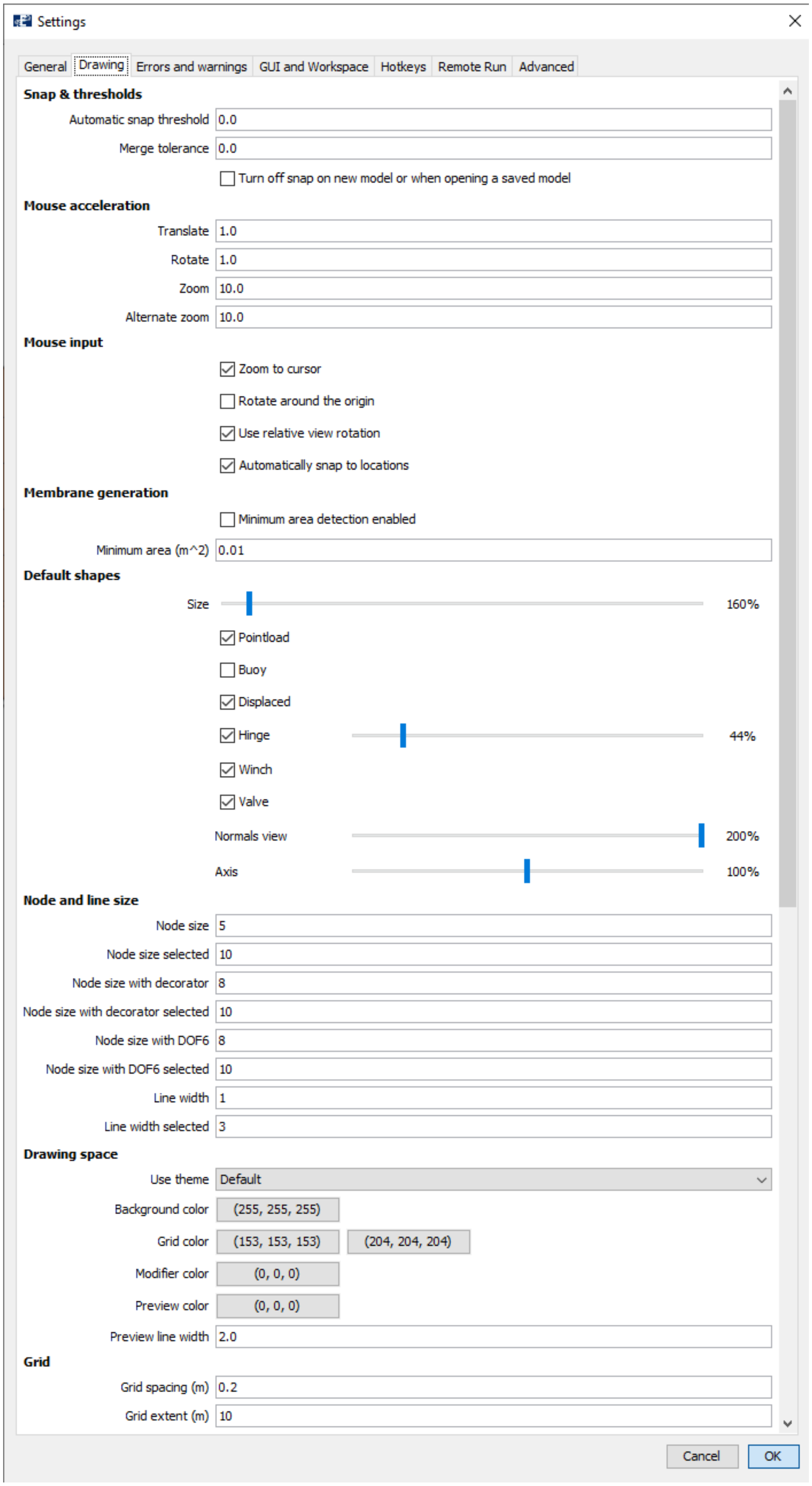

Settings
General
Drawing
Errors and warnings
GUI and Workspace
Hotkeys
Remote Run
Advanced
Snap & thresholds
Automatic snap threshold 0.0
Merge tolerance 0.0
Turn off snap on new model or when opening a saved model
Mouse acceleration
Translate 1.0
Rotate 1.0
Zoom 10.0
Alternate zoom 10.0
Mouse input
Zoom to cursor
Rotate around the origin
Use relative view rotation
Automatically snap to locations
Membrane generation
Minimum area detection enabled
Minimum area (m^2) 0.01
Default shapes
Size 160%
Pointload
Buoy
Displaced
Hinge 44%
Winch
Valve
Normals view 200%
Axis 100%
Node and line size
Node size 5
Node size selected 10
Node size with decorator 8
Node size with decorator selected 10
Node size with DOF6 8
Node size with DOF6 selected 10
Line width 1
Line width selected 3
Drawing space
Use theme Default
Background color (255, 255, 255)
Grid color (153, 153, 153) (204, 204, 204)
Modifier color (0, 0, 0)
Preview color (0, 0, 0)
Preview line width 2.0
Grid
Grid spacing (m) 0.2
Grid extent (m) 10
Cancel
OK

Figure 134 - Drawing tab

Snap and thresholds
"Automatic snap threshold" is the minimum distance that AquaEdit will allow. Anything drawn closer than this value to another node will automatically snap to that node.

"Merge tolerance" is the threshold used in Merge lines. If the angle between two elements being merged is larger than this threshold then the lines will not be merged together.

"Turn off snap…." will set node snapping to off if enabled, this will come in affect when opening a model or creating a new model.

Mouse acceleration
These specify how fast AquaEdit should respond to different mouse inputs. The "Alternate zoom" will be used when **SHIFT** is held down when zooming with the mouse wheel. By default, this zooms faster. Use a smaller value than the normal zoom value if the wanted zoom speed should be lower than the normal zoom. To alternate the zoom, negative values can be typed in "Zoom".

Mouse input
"Zoom to cursor" enables to always zoom towards the location of the mouse pointer. If this is disabled, then the zooming will be towards the middle of the screen.

"Rotate around the origin" will make the mouse rotate around the point 0,0,0. If disabled, the mouse will rotate around the current pointer location, or the selected nodes/elements in the model if selection is non-empty.

"Use relative view rotation" will maintain the centered camera view of the model when applying hotkeys Arrow UP/DOWN/LEFT/RIGHT, HOME, END, PgUp/PgDwn.

"Automatically snap to locations" toggle on means that the preview of Move and Draw line will at a certain distance snap to the nearest node.

Membrane generation
Enabling this will discard membrane elements with its area smaller than the input here. This avoids some degenerated membrane elements from being generated.

Default shapes
When adding some decorators, a default shape will be added to the visual model. These can be scaled using the slider. One can also choose which of the default shapes will be visible by using the checkboxes.

Individual scaling of size of Hinge shape and the global axis system is also available.

"Normals view" scales the size of element Normals (i.e., the projections of Point 3). "Axis" scale the size of the global coordinate system indicators.

Node and line size
Specifies the visual sizes of nodes and elements.

Drawing space
Sets the colors of the drawing canvas.

Grid

"Grid spacing" is the space between each grid square. Input of 1.0 here means the grid squares is 1x1 meters.

"Grid extent" is the extent of the grid in x- and y-direction from the origin. Input of 50 here will give a grid size of 100x100 meters.

8.4 Errors and warnings

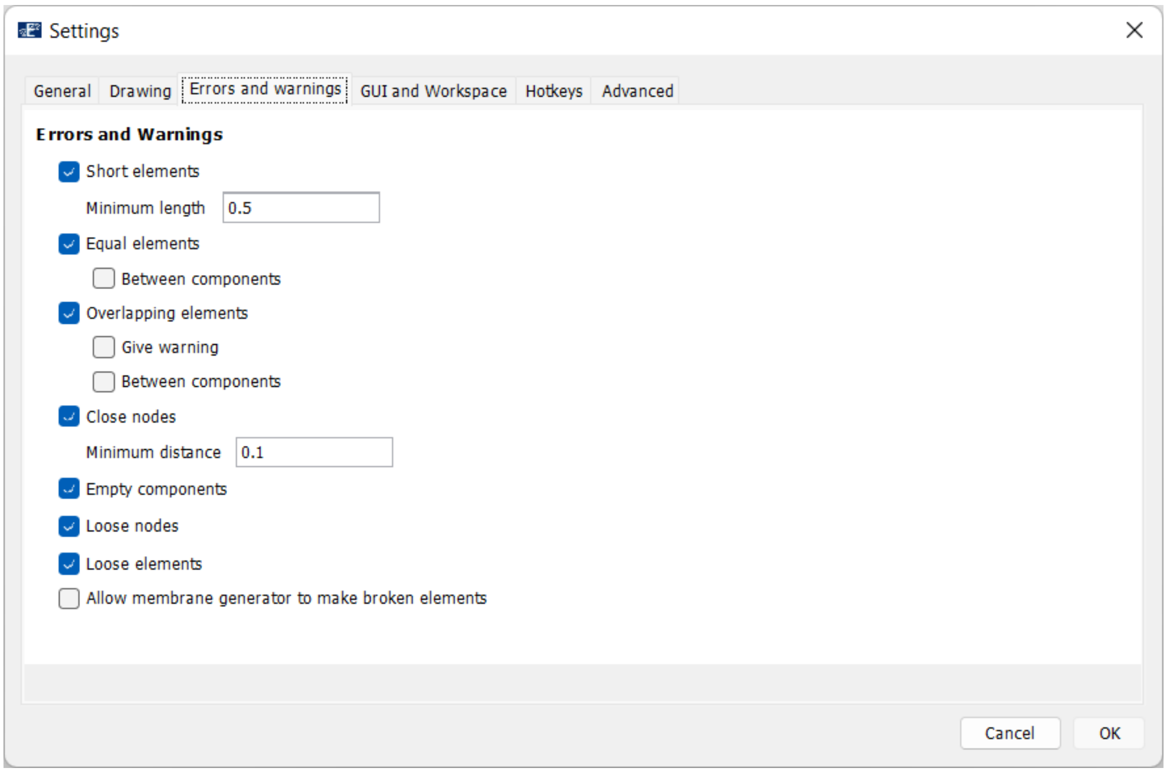


Figure 135 - Errors and warnings tab

When running Verify model there are some checks that might be unwanted.

Short elements

This enables the detection of elements that are shorter than the minimum length.

Equal elements

Elements that are equal are elements going to/from the same nodes. Enabling this will give an error if there are any equal elements on the same component.

Enabling "Between components" will give a warning if there are equal elements that does not belong to the same component.

Overlapping elements

Overlapping elements are elements going to/from one equal node and one element overlapping the other. E.g. one element being <0,0,0> -> <1,0,0> and the other being <0,0,0> -> <2,0,0>. Enabling this will give an error if there are any overlapping elements on the same component.

Enabling "Give warning" will give a warning instead of an error for the problem of overlapping elements on the same component.

Enabling "Between components" will give a warning if there are overlapping elements that does not belong to the same component.

Close nodes
Checks to see if nodes are closer together than the minimum distance (usually indicates they should have been merged).

Empty components
Allow components that have no elements in them.

Loose nodes
Enabling this will give a warning if there are nodes with only one element attached to them. If the node has checked off DOF for translation on all three axes, then this will give no warning.

Loose elements
Enabling this will give an error if there are elements that are not connected to any other elements and both nodes are loose.

If there is a component that has invalid properties (e.g. negative e-modulus on a beam) then Verify model will produce an error, this cannot be turned off since an invalid component will cause the analysis to crash.

Allow membrane generator to make broken elements
Enables membrane elements that has an aspect ratio close to a triangle to be accepted and allowed. Having this option deselected, AquaEdit will not allow panels of component type Membrane that resembles too much of a triangle to be included in the analysis.

Component has questionable properties (Low emodulus)
Component E-modulus lower than 100 N/m2 will trigger this warning. This to make the user aware that the properties may be incorrectly specified.

Component has invalid properties (Lift table is out of order)
This is connected to the Lift table for beam elements found in the Advanced-tab. The column Angle must be defined in increased order. To correct this, order the data in Lift table with increasing values for Angle.

8.4.1 Fix / Fix problem type

Having detected findings according to specifications in the "Errors and warnings"-tab, AquaEdit provides solution proposals to correct, or fix, the model. Choosing Fix will solve the isolated problem, and Fix problem type will solve all problems within the same problem-category. To view and highlight the problem area in the model, the user may select the Zoom to problem-button. Description of actions AquaEdit do upon selecting Fix or Zoom to problem for the different errors and warnings, is provided in the table below.

Verify actions	Description
Short elements, Warning:	Fix: cannot fix. The user must either adjust the length of the element, or the reduce value defined in "Errors and warnings"-tab.
	Zoom to problem: select and zooms to the short element.
Equal elements	Fix: deletes all but one of the equal elements.
	Zoom to problem: zooms to the element and selects the element that will remain after fixing the problem.
Overlapping elements	Fix: deletes all but one of the overlapping elements.
	Zoom to problem: selects the element that will remain after fixing the problem.
Close nodes	Fix: merges the two nodes.
	Zoom to problem: zooms to the problem node, selects both nodes.
Empty components	Fix: deletes the component without drawn elements.
	Zoom to problem: cannot zoom, since there are no drawn elements.
Loose nodes	Fix: sets the DOF to limit translation/rotation.
	Zoom to problem: zooms to the node, selects the node.
Loose elements	Fix: sets DOF on node A of the element to limit translation/rotation.
	Zoom to problem: zooms to the element and selects the element.
Broken elements	Description: broken elements are elements that have duplicate nodes. Usually caused by improper merging or automatically generated models.
	Fix: deletes the broken elements.
	Zoom to problem: selects the broken element and zooms to it.
Node/element decorator	Description: error has been found in a node or element with a decorator.
	Fix: deletes the decorator.
	Zoom to problem: zooms to the element/node with the problem-decorator in question.
Invalid component properties	Description: an error has been found in the properties of a component. Usually due to missing input or improper values.
	Fix: opens the editor for the component.
	Zoom to problem: cannot zoom.
Point 3 on line	Description: point 3 coincides with the element length-axis.
	Fix: resets point 3 back to default values.
	Zoom to problem: selects and zooms to the element in question.
Roller tolerance	Description: roller tolerance must be 0.1 or higher.
	Fix: cannot fix. The user must correct the values in the roller properties-window.
	Zoom to problem: zooms to the node with the invalid properties, opens the roller properties-window.
Hinge cannot exist on non-beam elements	Description: a hinge is only supported on beam elements.
	Zoom to problem: selects the problem-element and plate the camera looking at the element.
	Fix: deletes the invalid hinge from the element.
	Fix problem type: deletes all invalid hinges from elements they are attached to.
Membrane too triangular, consider using	Description: the quad element (i.e., a 4-noded membraneX) resembles a triangle too much.
	Fix: the user must convert the quad element to a triangle element.

triangular membrane x triangle elements	Zoom to problem: centers the 3D view to the node that is in-line with the two other nodes in the quad element.
Varying current are in the wrong order on load 1 in the Directional group	Description: the order of Loc in the Varying current-tab (Environment-window) is not in descending order. Fix: cannot fix. The user must correct the input values in the Edit load-window from Environment. Zoom to problem: loads the Edit load-window with inappropriate input data for Loc in the Varying current-tab. The user can now input Loc-values in correct descending order.
Zero length element	Description: an element with length of 0 meters is detected. Fix: cannot fix. The user must manually correct this by defining the element length to be greater than zero. Zoom to problem: centers the 3D-view to the element with length of 0 meters.
… is unsupported in the selected solver	Description: a feature that is not supported by the solver version that is selected. This applies for the features: - Current reduction method: Energy Method - Advanced buoyancy - Line break, type Max force Fix: cannot fix. The user must change to a solver version where these are implemented.

8.5 GUI and Workspace

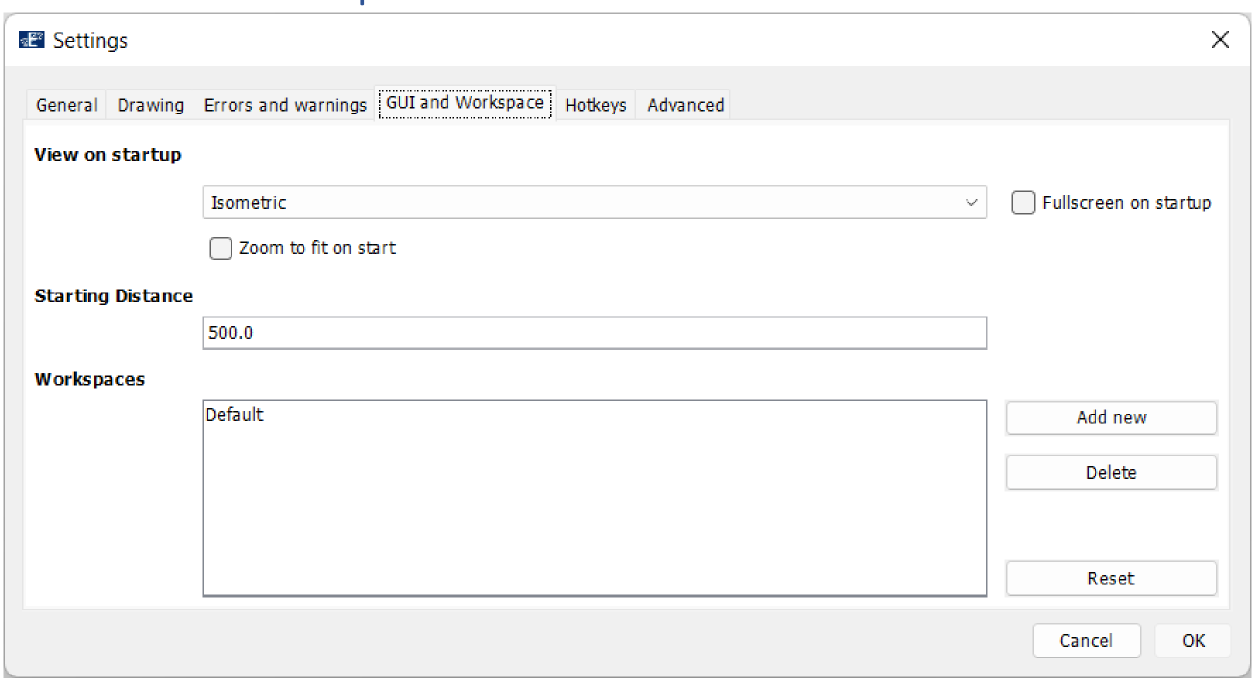


Figure 136 - GUI and Workspace tab

Properties	Description
View on startup	Sets a default view of the 3D model upon loading AquaEdit. Choose between Isometric, Front-, Top-, Left-, Right-, Back- and Bottom View.

Zoom to fit on start	Zooms in or out so you can see the entire model upon start of AquaEdit.
Fullscreen on startup	Selection for always load AquaEdit in fullscreen mode.
Starting distance	Sets a default distance to the model upon loading AquaEdit. Unit in meters.
Workspace	All saved workspaces are shown here, as well as Workspaces. - Add new: saves the current workspace layout. - Delete: deletes a highlighted workspace from the list. - Reset: sets the saved workspace back to default. Only "Default" will be shown.

8.6 Hotkeys

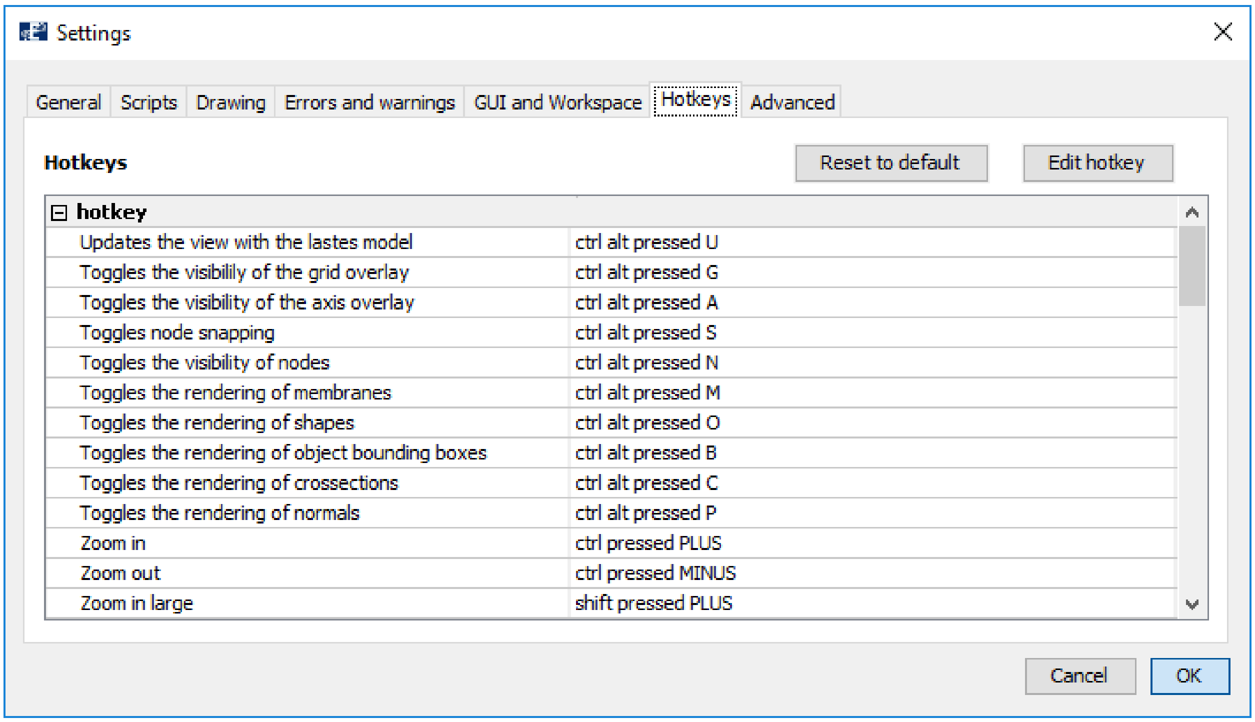


Figure 137 - Hotkeys tab

There are several functions in AquaEdit that can be used with hotkeys on the keyboard. All the hotkeys are stored and shown here. These can be reset to default by clicking the "Reset to default" button. The hotkeys can also be edited by clicking on a hotkey in the list and then clicking the "Edit hotkey" button.

8.7 Advanced

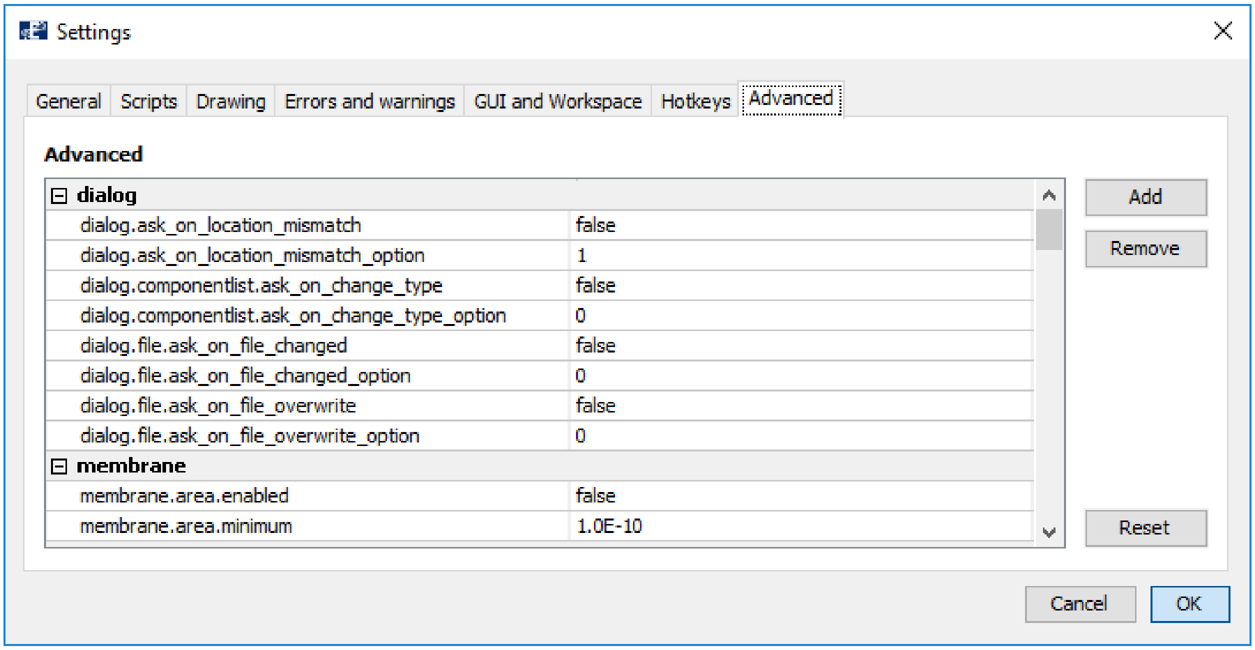


Figure 138 - Advanced tab

Advanced settings show all settings that are set in AquaEdit and allows adding and removing settings.

In some cases, you may be sent advanced settings by Aquastructures support employees to fix or alter AquaEdit behavior. These settings are added by selecting "Add" and filling in the custom setting dialog.

All settings can be reset to default by clicking the "Reset" button.

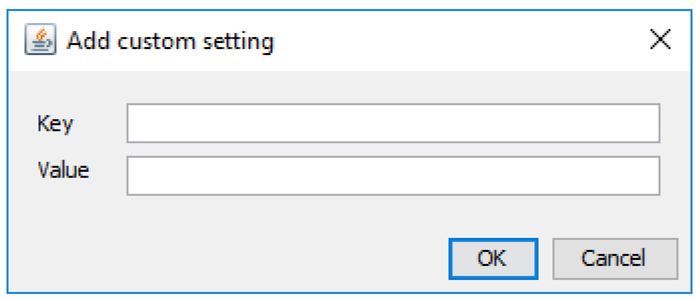


Figure 139 - Add custom setting

9. Export

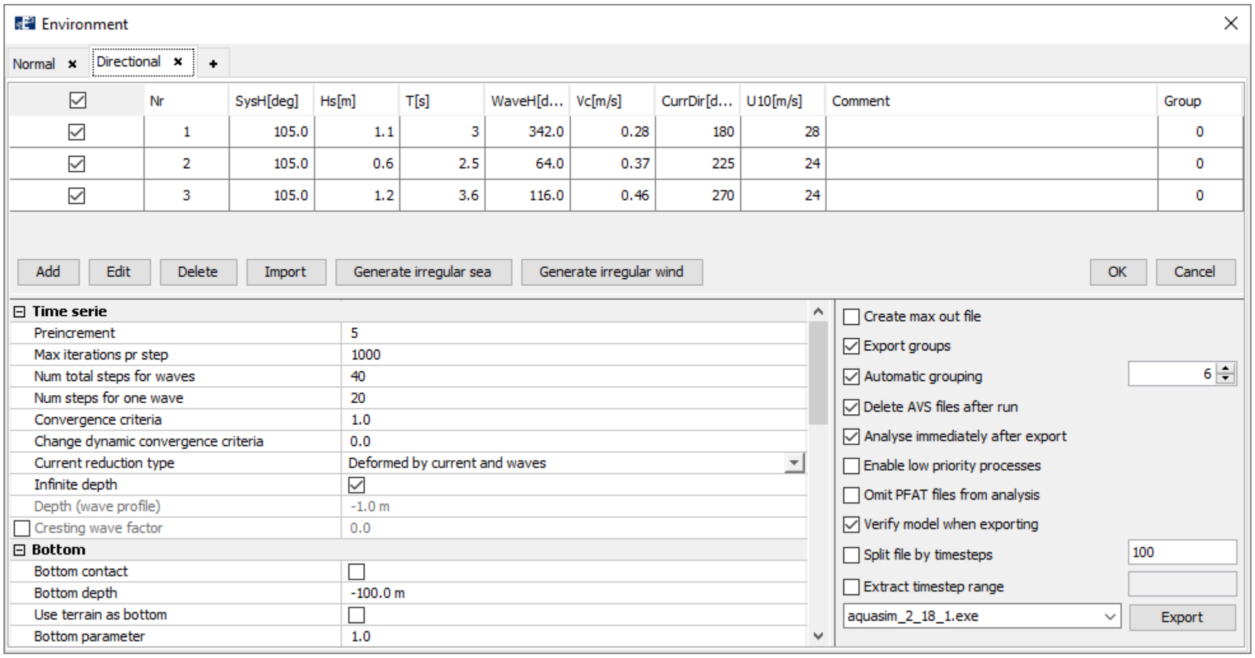


Figure 140 - Export window

There are two different environment loads that can be added. A set of vectors (Normal) or calculated based on a direction (Directional). By default, there are two tabs in the export window named Normal and Directional. The user can add more tabs by clicking the "+" button on top, this gives the option of adding another tab and will be given a name and a type. To remove a tab, click the "x" button on the tab on top.

Right click on a tab will give the option to rename a tab, duplicate the tab into a new one, close the tab or close all other tabs.

To add a row of environment load to the list, click the "Add" button or Enter key on the keyboard while the last row is selected. Clicking "Delete" will delete an environment load from the list.

Normal (directions are based on the global coordinate system):

Properties	Description
Nr	Order of load condition.
Amp	Wave amplitude (0.95 * significant wave height) in meters.
T	Wave period in seconds. Note: For irregular sea state, T correspond to the zero-crossing period.
V	Wave direction from global positive x-direction, counter clockwise in degrees.
cX	Current speed in global x-direction in meters per second.
cY	Current speed in global y-direction in meters per second.
wX	Wind speed in global x-direction in meters per second at 10 meters height.
wY	Wind speed in global y-direction in meters per second at 10 meters height.
Comment	Description of load condition.
Group	Grouping of load conditions when running several simultaneously analyses.

Directional:

Properties	Description
Nr	Order of load condition
SysH	Heading of the system from north to global x-axis, clockwise, in degrees.
Hs	Significant wave height (Amp / 0.95) in meters
T	Wave period in seconds. Note: For irregular sea state, T correspond to the zero-crossing period.
WaveH	Wave heading (from). Note: Wind heading is the same as wave heading, in degrees.
Vc	Current speed in meters per second
CurrDir	Current direction (towards) in degrees.
U10	Wind speed (at altitude 10 meters) in meters per second.
Comment	Description of load condition
Group	Grouping of load conditions when running several simultaneously analyses.

Values in the tables can be imported/exported by using Ctrl+c / Ctrl+v when working inside the table. Note: Only the selected rows will be exported to the clipboard using Ctrl+c.

Create max out file

Generate a max-file from multiple environment data.

Export groups

Do several simultaneously analyses. Each process of AquaSim uses one thread on the processor, so by using more threads at the same time can reduce analysis time significantly. Grouping is done by using group numbers or by using “Automatic grouping” with the number of simultaneously analyses as input.

Automatic grouping

Automatic groups the analyses that is run. This is an alternative to manually group the analyses. The number of groups to be generated is typed in the field to the right of “Automatic grouping”.

Delete AVS files after run

Automatically delete AVS result files after the analysis is finished.

Analyze immediately after export

Will open the window shown in Figure 141 when exporting (given that Export groups is disabled). This shows all the analyses that can be run. Clicking on “Start” will start the analysis, and you can start as many or few as you want. When one analysis is done, it will jump to the next not-started analysis in the list and start that.

“Show messages” opens another window that shows the output coming from the AquaSim analysis.

When an analysis is done the button “Open” will become available and this will open the finished analysis in AquaView. “Cancel” will stop the running analysis.

The label on the left will show the status of the analysis. Dark grey means it has not started. Green shows when running and everything is running okay. Yellow means a step has not converged. Red means an error has occurred during the analysis or the analysis has been cancelled.

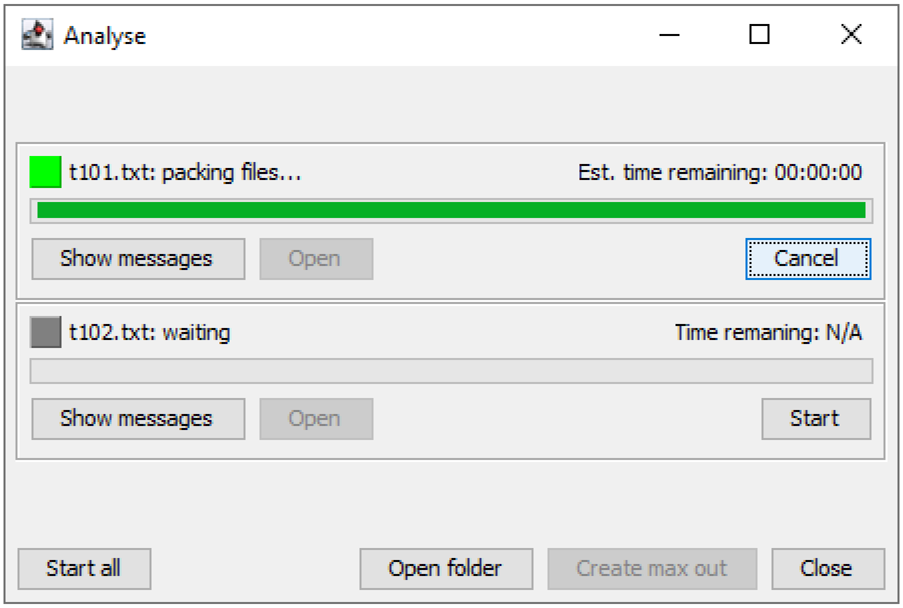


Figure 141 - Analyse window

“Open folder” will open the folder where the analyses files are in Windows Explorer.

After an analysis is done the “Create max out” button will be enabled and will create a max-file on the finished analyses.

Enable low priority processes

Starts the analyses using IDLE priority class. This means that the running analyses will lose processor time if a higher priority class needs processor time. More information on the different priority classes can be read about here: https://technet.microsoft.com/en-us/library/cc940378.aspx

Omit PFAT files from the analysis

Will tell AquaSim to not write out PFAT result files.

Verify model before exporting

Will run Verify model to check for any problems before exporting.

Split file by timesteps

Will split the result file into several files using ppfilter.

Extract timestep range

Will extract a range of steps from the main analysis using ppfilter.

Solver version and solver check

Different solver versions may be selected from the dropdown menu. If the latest solver version is not selected, a warning symbol appear; “Not using the latest solver”. The user may check for updated in Help > Check for updated solver.

9.1 Edit

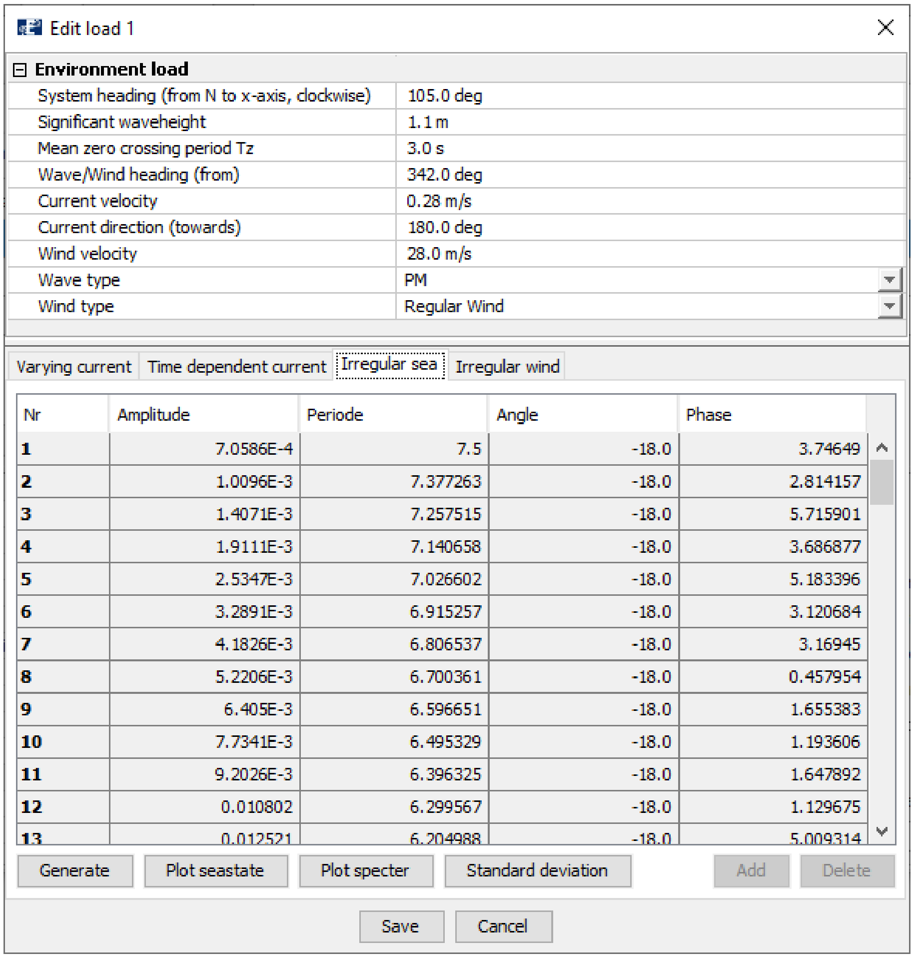


Figure 142 - Edit environment load

Clicking the "Edit" button when an environment load is chosen, will open the window shown in Figure 142. The same values can be edited here as in the table itself. Here it is also possible to add/edit varying current, time dependent current and irregular sea.

9.1.1 Varying current

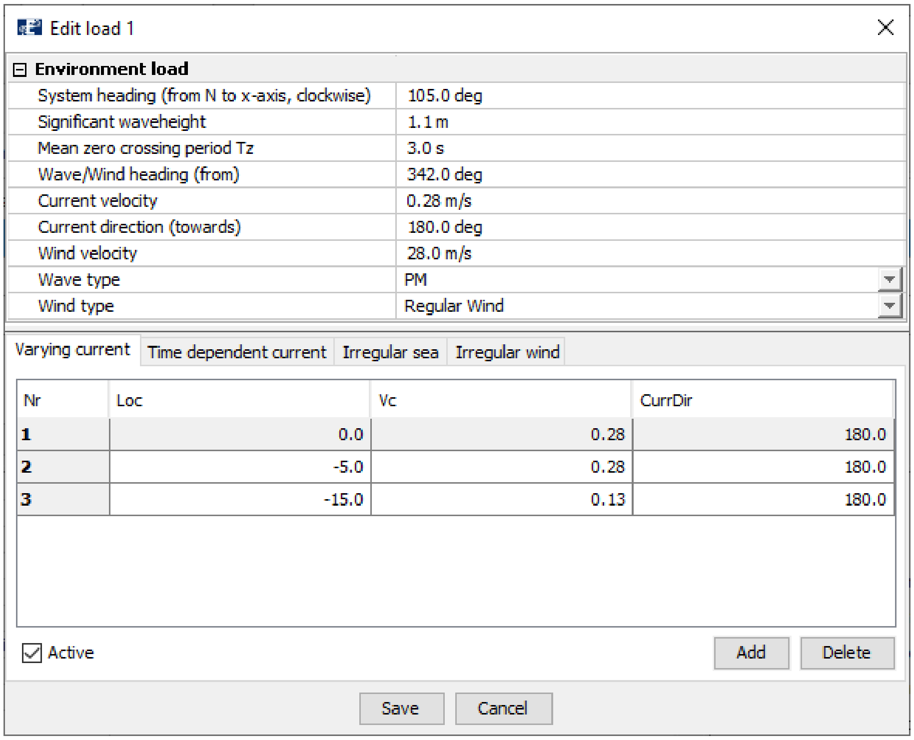


Figure 143 - Varying current

To activate varying current, go into the “Varying current” tab and enable the “Active” box. Rows can be added and removed by using the “Add” and “Delete” buttons.

Each row in the table has a depth, current velocity in x-direction and y-direction for type Normal.

Each row in the table has a depth, current speed and direction for type Directional.

When a load condition has varying current the current cells in the environment table will be marked in grey.

9.1.2 Time dependent current

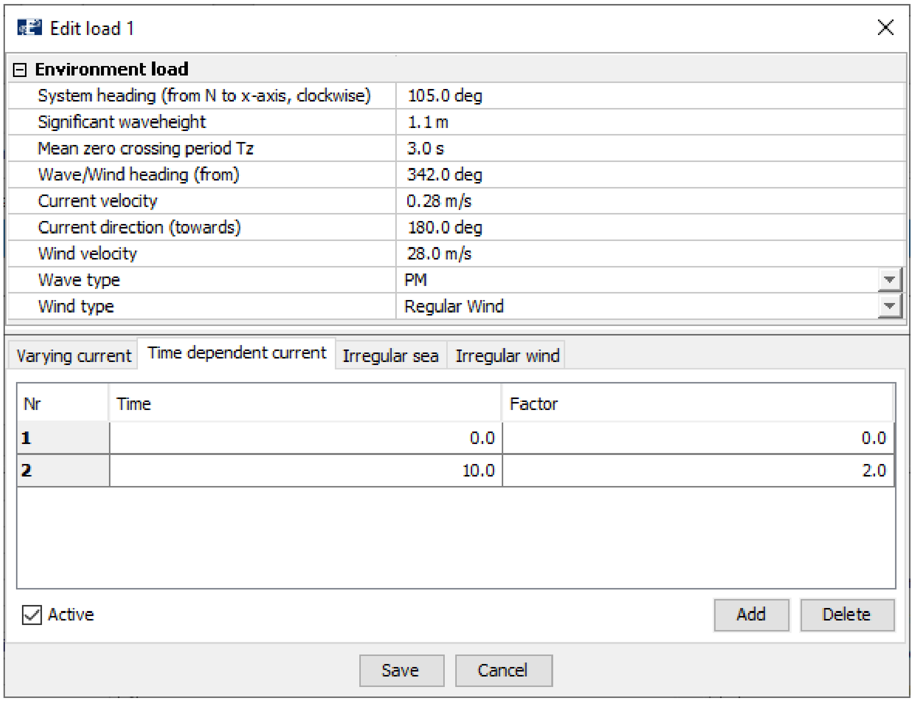


Figure 144 - Time dependent current

To activate time dependent current, go into the “Time dependent current” tab and enable the “Active” box. Rows can be added and removed by using the “Add” and “Delete” buttons.

Each row has a Time (in seconds) and Factor. This will add the factor with the current at the set time. This will only work during the dynamic steps of the analysis, during the static steps it will be the same as before.

9.1.3 Irregular sea

To generate irregular sea there are a couple options. You can choose "Irregular" from the "Wave type" dropdown list and make a table of waves self or generate automatically using a spectrum. To generate from a spectrum, click "Generate" when type is "Irregular" or choose spectrum from the "Wave type" dropdown list. There are PM spectrum, Jonswap spectrum and ITTC spectrum.

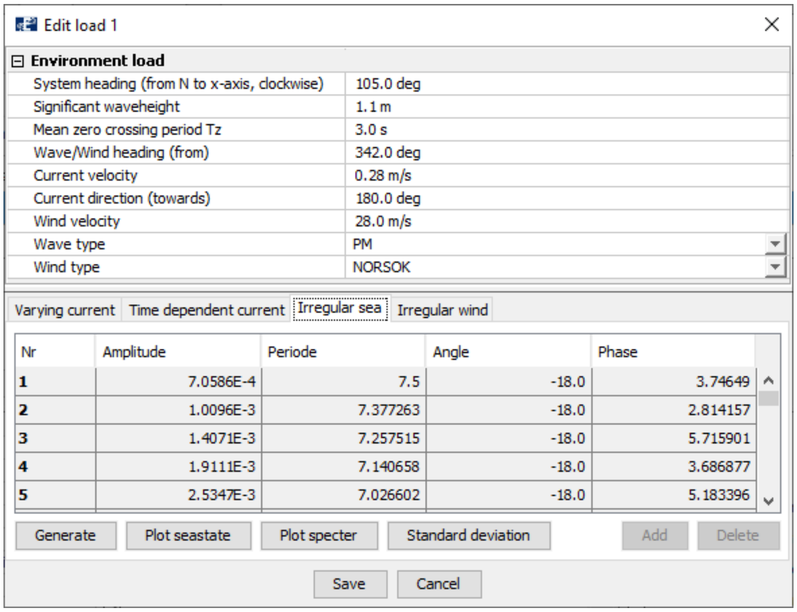


Figure 145 – irregular sea

For a better understanding of the irregular spectrums, read (Berstad A. J., Irregular wave calculation and validation, 2013).

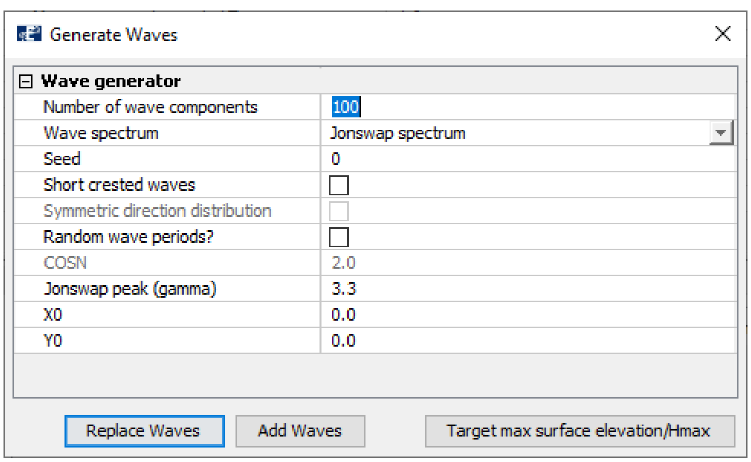


Figure 146 - Generate waves window

Figure 146 shows the windows that will appear when the user wants to generate waves automatically.

Enter a random value into the Seed field in order to specify the random seed for the generation of waves. Each specific Seed will produce the same wave profile.

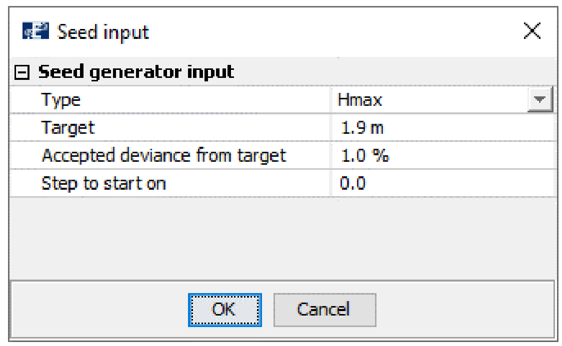
Targeting specific wave profiles

It is possible to target a given Hmax or surface elevation using the “Target max surface elevation/Hmax” button.

“Hmax” will by default set “Target” to 1.9 * Amplitude. This will attempt to generate a wave spectrum and find the difference between the top and bottom of the graph generated and see if this difference is within the wanted target. Note that the smaller allowed deviation the harder it will be to find a suitable wave spectrum.

The generation of the wave spectrum is random and will (in most cases) give a unique profile each time it is ran.

Equation 3 in (Berstad A. J., Irregular wave calculation and validation, 2013) gives the wave elevation based on N number of wave components.



An example of targeting a given Hmax to occur after the first steps, with target 1.9m and accepted deviance at 1%. The main wave can be seen on Figure 142, total steps is 1000. This will search for a seed that gives Hmax higher than 1.881 and lower than 1.92, which in this case is 12.

“Add waves” will add the number of waves to the wave table, “Replace waves” will remove the old waves and add the new created ones. After waves have been added to the wave table it is possible to click in the “Plot” button and see the wave height as a function of time, see Figure 147.

If the wave profile contains an Hmax that exceeds the given target it will be rejected during generation.

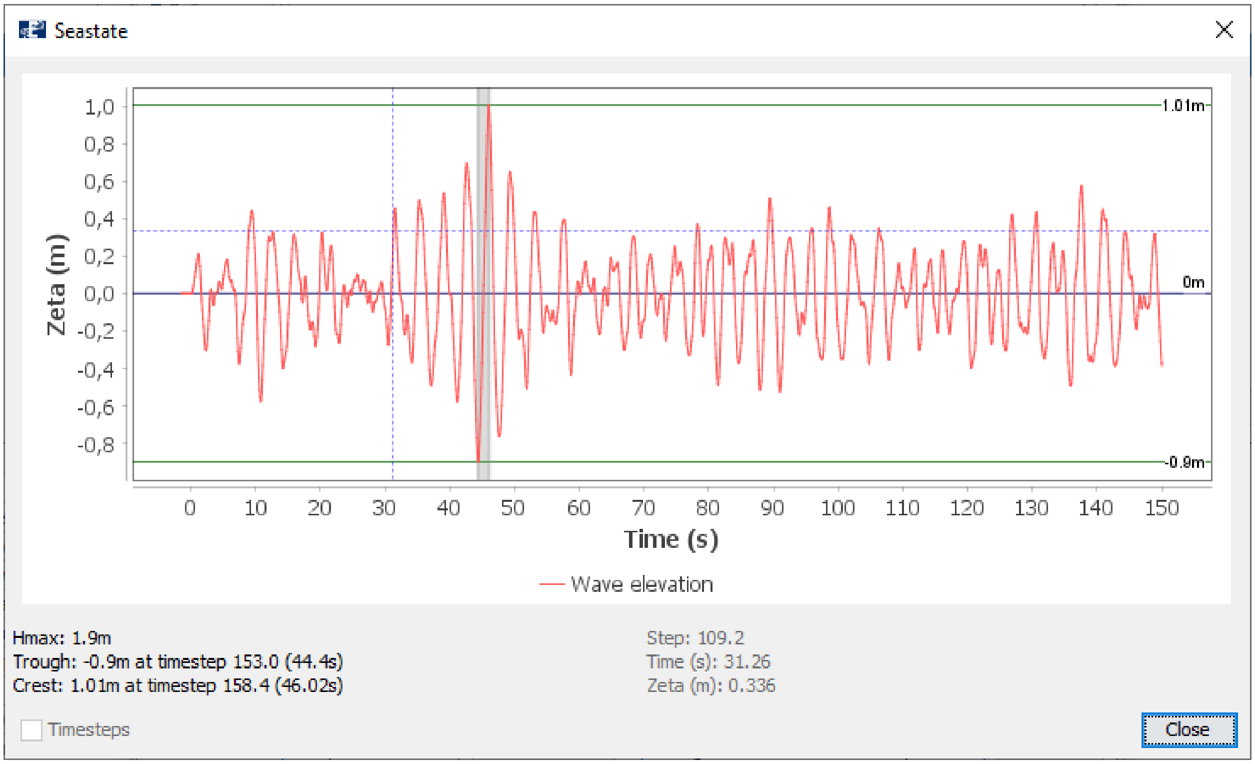


Figure 147 - Wave elevation over time

Using our example from above it is easy to see that the highest crest is 1.01 and the trough is -0.9, which gives Hmax of 1.91 which is within our accepted deviance.

It is important to use the plot function to verify that the given wave profile is acceptable.

When a load has irregular sea the wave cells in the environment table will be marked in grey.

It is also possible to plot the spectral values over period with using the “Plot specter button”. Using the same irregular sea as before we get the result as seen in Figure 148.

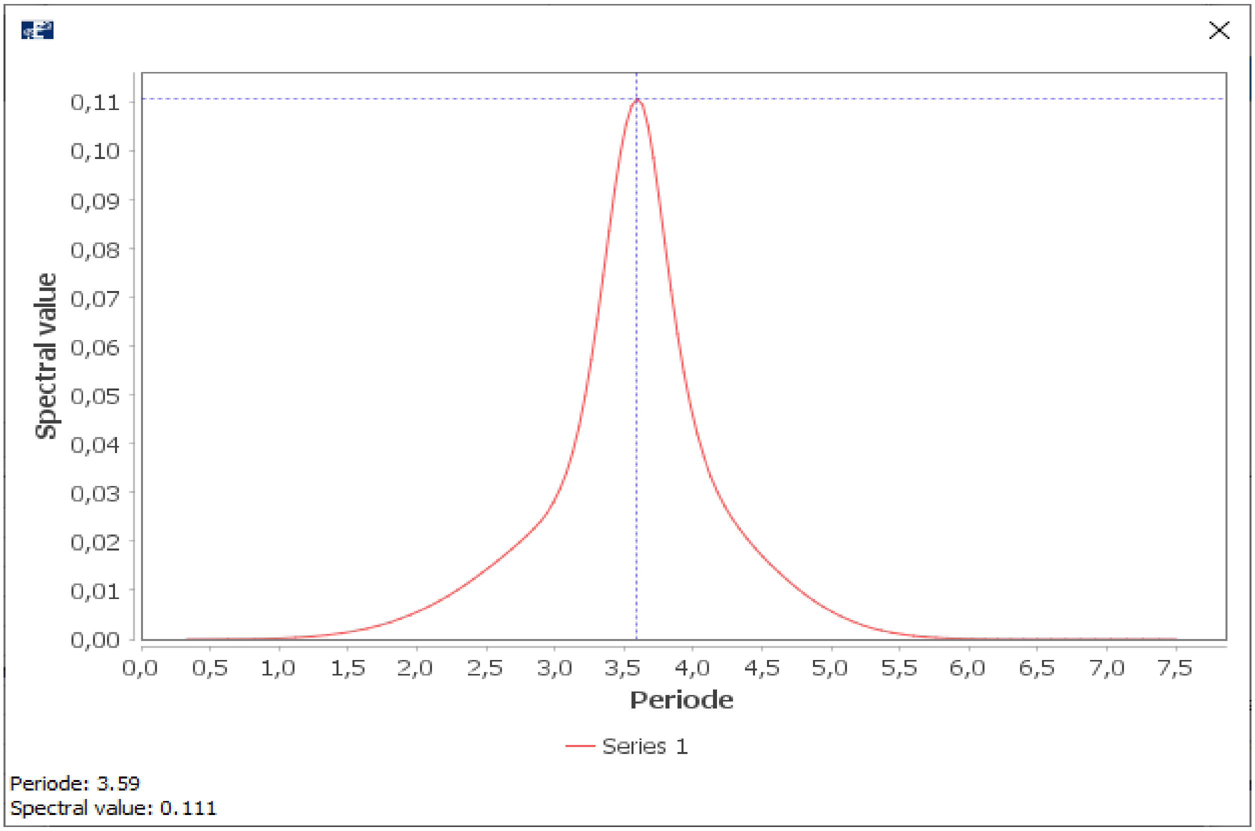
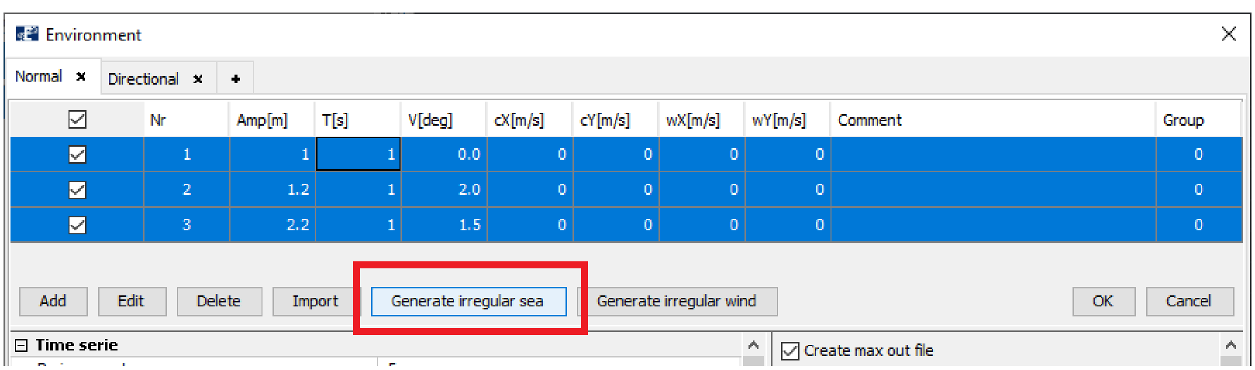


Figure 148 - Spectral value over periode

It is also possible to view the standard deviation of the spectrum with the “σ”-button. In the example used above we get a standard deviation of 0.25.

Generating multiple wave profiles

Generating irregular waves targeting an Hmax or surface elevation can also be done to multiple load cases at once. On the “Environment” screen multi-select the load cases and click on “Generate irregular sea”, a similar dialog to the previous chapter is shown, but now it also lists the load cases that will be generated. Choose “Replace waves” to use the new wave profile, or “Add waves” to add the generated waves to the existing wave profile.



Note that the generated waves will not be visible in this dialog and the user will have to go to each load case to verify the wave profile.

9.1.4 Irregular wind

Irregular wind, or wind gust, is possible to generate from the “Wind type”-drop down menu.

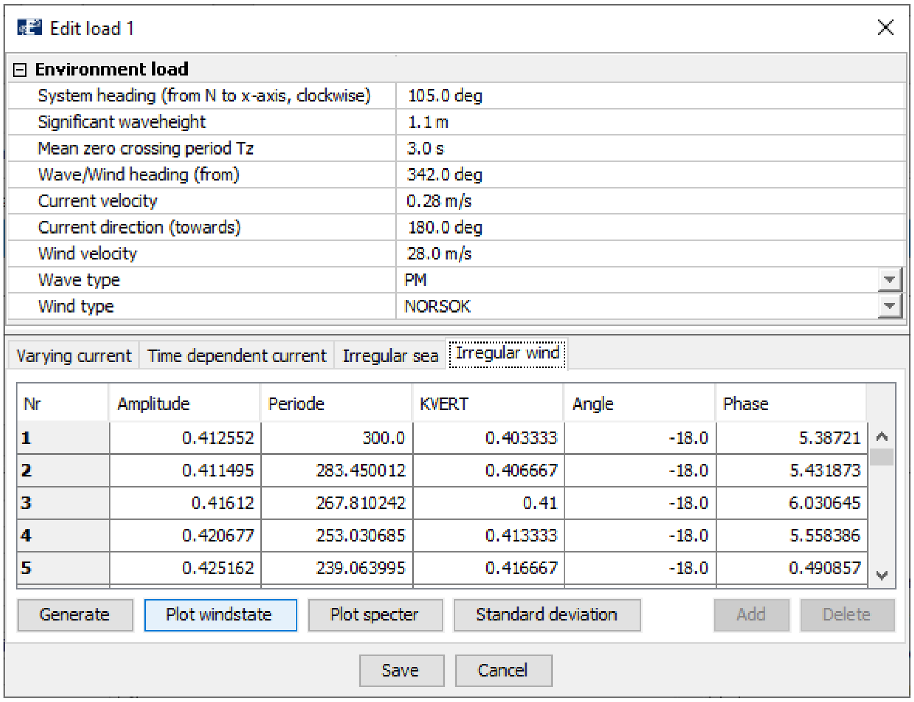


Figure 149 – Irregular wind

To generate irregular wind there are currently two options. You can choose “Irregular” from the “Wind type” dropdown list or choose NORSOK from the dropdown list to generate irregular wind automatically.

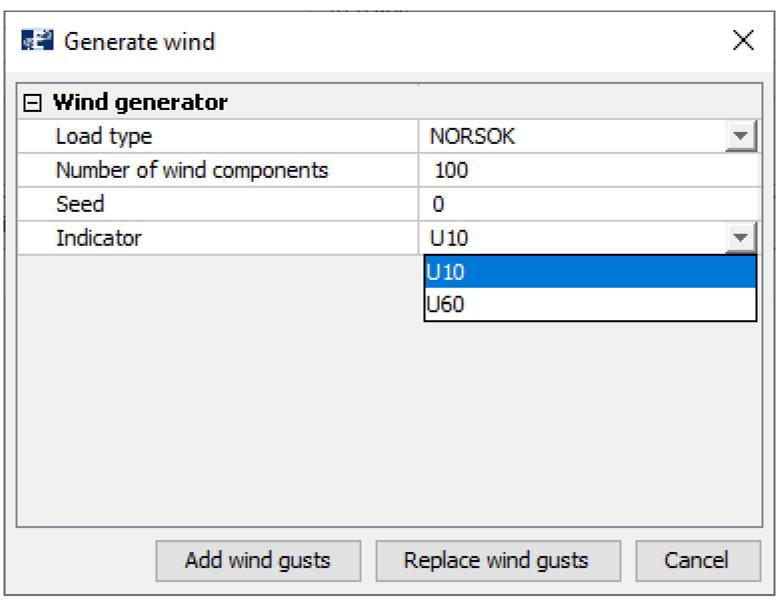


Figure 150 - Generate wind window

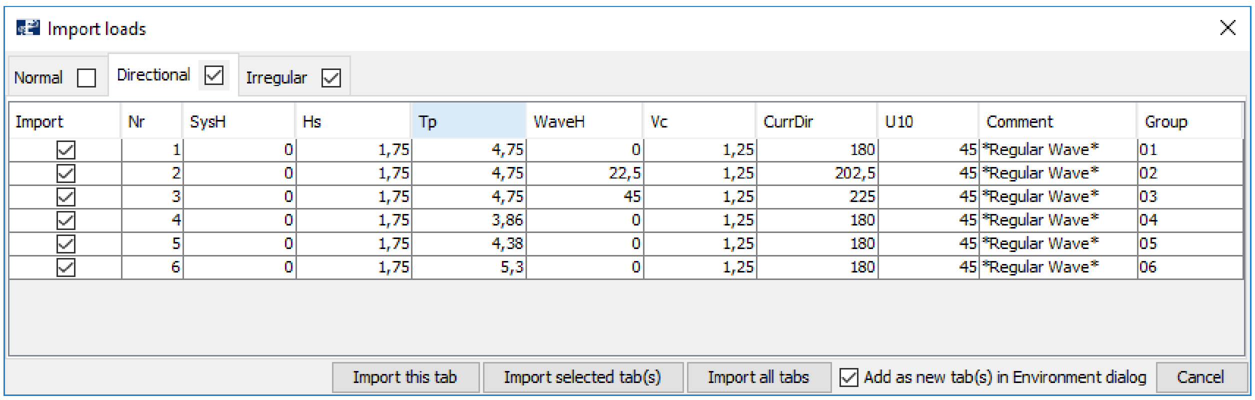
Figure 150 shows the window that will appear when the user wants to generate winds automatically. "Load type" is the type of wind spectra that can be applied. AquaSim supports NORSOK wind spectra. "Number of wind components" correspond to the number of sinusoidal wind components. "Seed" is a number to initialize the generation of the wind components. "Indicator" is the reference wind, the user can choose between 10-minute mean wind (U10) or 1-hour mean wind (U60).

For a better understanding of the NORSOK spectrum, see (Berstad A. , 2019).

Plotting and viewing standard deviation is the same as it is for irregular sea.

9.2 Import

Clicking the "Import" button will import environment loads from a selected model. Models made before version 2.13 will add all loads from that model, with the same type as the current selected tab. Models made after 2.13 with customized tabs will open an import window, see Figure 151.



Import	Nr	SysH	Hs	Tp	WaveH	Vc	CurrDir	U10	Comment	Group
☑	1	0	1,75	4,75	0	1,25	180	45	*Regular Wave*	01
☑	2	0	1,75	4,75	22,5	1,25	202,5	45	*Regular Wave*	02
☑	3	0	1,75	4,75	45	1,25	225	45	*Regular Wave*	03
☑	4	0	1,75	3,86	0	1,25	180	45	*Regular Wave*	04
☑	5	0	1,75	4,38	0	1,25	180	45	*Regular Wave*	05
☑	6	0	1,75	5,3	0	1,25	180	45	*Regular Wave*	06

Figure 151 - Import window

Here, all the tabs from the selected model will be shown. Tabs can be selected to import loads from, and it is possible to check off only the loads wanted too.

"Import this tab" will import the current tab.

"Import selected tabs" will import all the selected tabs.

"Import all tabs" will import all tabs.

"Add new tabs…." enabled will add the imported tabs as new tabs into the export window. This disabled will import the loads into the current tab in the export window. Note: It is not possible to import loads of type Normal into a tab that supports tab Directional, and vice versa.

9.3 Generate irregular sea for several load conditions

It is possible to generate irregular sea for several load conditions at the same time using this option.

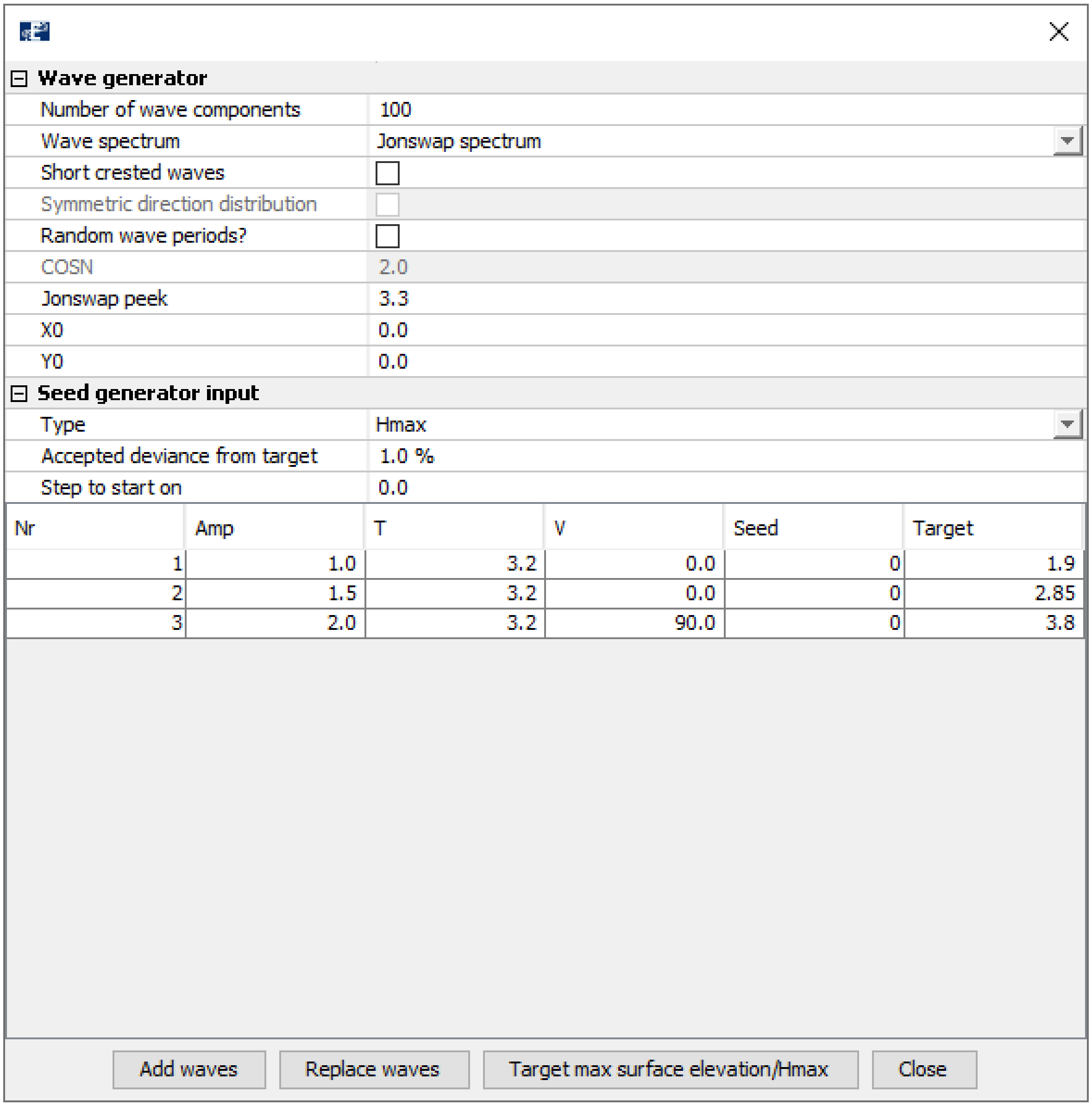


Figure 152 - Generate several irregular seas

This will open the window shown in Figure 152 with a table with the selected loads in a table sharing the same input for generating irregular sea. The table can be edited manually, and waves can be generated by entering in a seed for the loads in the table and using “Add waves” or “Replace waves”. It is also possible to find the seed automatically for all loads at the same time using “Target max surface elevation/Hmax”. The target is in a separate column in the table and all loads will target the same Accepted deviance and start at the same step.

If we use the example above, we will find the seed for the load as shown in Figure 153.

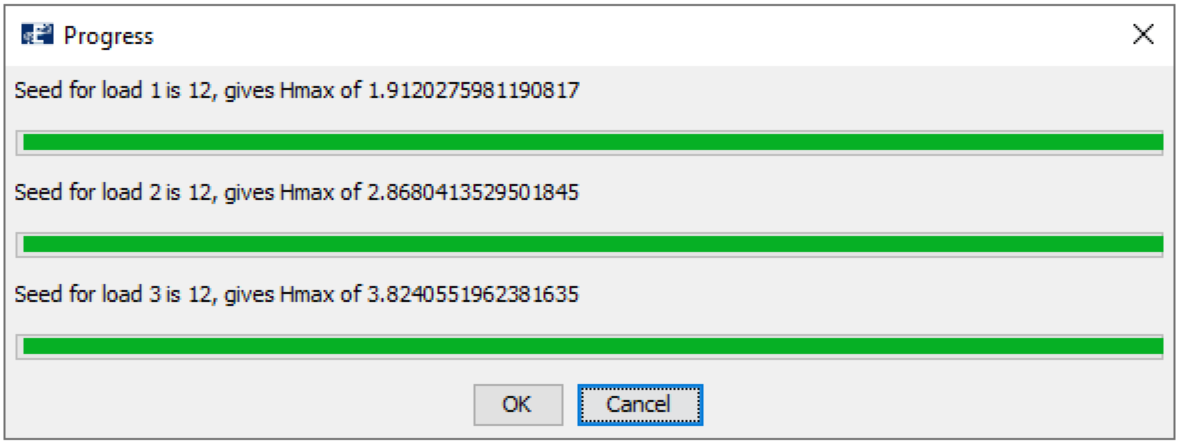


Figure 153 - Seeds found

Clicking OK will add the seeds to the table, and we can use these to add or replace irregular waves for all the load conditions.

9.4 Properties of time domain simulation

Flowing description of settings for time domain simulations in AquaSim, and other options.

Properties	Description
Time series	
Preincrement	Number of initial steps where the model obtains equilibrium and current and wind are applied.
Max iterations per step	Maximum iterations before moving to the next step.
Num total steps for waves	Total steps for the waves
Num steps for one wave	Number of steps per wave. 40 total steps and 20 steps per wave analyse two wave periods
Convergence criteria	The convergence limit before moving to the next step. The lower this is, the more correct the results will be.
Change dynamic convergence criteria	Larger than 0 if convergence criteria shall be changed between the static steps and the dynamic steps. 10 means the convergence limit is 10 times larger for the dynamic steps, 0.1 means the convergence limit is 10 times stricter for the dynamic steps.
Current reduction type	**No reduction:** Current reduction for nets succeeding other nets is not accounted for. **From initial shape:** Current reduction from succeeding nets is accounted for according to formulae from (Løland, 1991). The reduction factor is found in the initial configuration and kept during the analysis. **Deformed by current:** Current reduction from succeeding nets is accounted for according to formulae from (Løland, 1991). The reduction factor is found in the initial configuration and updated for the deformed configuration as current is put on. **Deformed by current and waves:** Same as deformed by current, but the current reduction factor is updated at each wave step.
Infinite depth	This enabled will apply linear Airy wave theory, see (Aquastructures, 2024a).
Depth (wave profile)	If “Infinite depth” is disabled, this is depth of the velocity field below a wave.
Cresting wave factor	Enable non-linear wave profile, see (Aquastructures, 2024a).
Bottom	
Bottom contact	Simulation of ocean floor by adding spring stiffness (“Bottom parameter”) to nodes moving beneath “Bottom depth”. If there is an imported terrain then this will be used to check for distance between nodes and bottom floor, instead of “Bottom depth”.
Bottom depth	Depth to ocean floor, in meters (< 0).
Bottom parameter	Define the spring on the bottom, needs to be higher than 0. Higher the value, softer the spring becomes. Meaning larger possibility for convergence.
Bottom friction	Friction coefficient between elements and ocean floor.

Dynamic vertical friction	Provides friction relative to velocity to the seabed in z-direction. This friction can be interpreted as damping between the model and seabed. The higher this value, the more damping is introduced. As an example this parameter regulates how much an object can bounce on the seabed.
Dynamic horizontal friction	Provides friction relative to velocity to the seabed in x- and y-direction. This friction can be interpreted as damping between the model and seabed. The higher this value, the more damping is introduced.
Advanced	
Water volume correction	**None:** Run without any water volume correction for hydrodynamic elements. **Normal:** Gives a maximal buoyancy and turns off water plane stiffness when a hydrodynamic element is submerged and the total of submerged volume of the element is equal to the component buoyancy. **With slamming:** Removes buoyancy, drag and added mass when elements move out of water. Also calculates slamming forces from waves. All input properties for Morison elements should be as if submerged in water.
Reported steps	How many steps should be reported to the avz-files. 1 is every step, 2 is every second step etc. If 0 then AquaSim decides how many steps are reported based on number of elements and steps. Note: Every step is always reported to the PFAT files.
Convergence accelerator	Parameter where one may slack on demands for the dynamic damping.
Newmark damping	Parameter in the Newmark beta distribution. 0.5 means constant average acceleration.
Analysis type	**Static:** Only static loads – internal forces, current and regular wind – are applied from step number 1 in the analysis. **Normal:** Static loads – internal forces, current and regular wind – are applied in the initial steps. The initial steps are defined by Preincrement. Dynamic loads – waves and wind gust – are applied on top of static loads in succeeding dynamic steps. The dynamic steps are defined by Num total steps for waves and Num steps for one wave. **Dynamic:** Static- and dynamic loads are applied from step number 1 in the analysis.
Type of mass	**Lumped mass:** Use lumped mass for beams and trusses. **Consistent mass:** Consistent mass formulation for beams and trusses. **Consistent mass on beams:** Consistent mass formulation for beams, lumped mass on trusses.
Buckling/eigen period analysis	See chapter 9.4.1.
Non linear density field	See chapter 9.4.2.
Membrane normals are verified	When checked on, the user confirm that the direction of the membrane Normals has been checked and is correct

	according to user intentions. The Normal directions are then 'locked' and used 'as is' when the model is exported. More on membrane Normals are found in chapter 4.4. When checked off, there is a chance that AquaSim will changes the direction of the Normals when exporting the model. This happens because AquaSim will seek to orient the Normals towards the geometric center of all membrane panels in the component group.
Number of threads	Defines the number of threads AquaSim will use per analysis.
Hydrodynamic properties	
Wave headings	Results are interpolated between the wave headings analyzed.
Segments on hull	Discretization along the hull.
Segments on water surface	Discretization along the surface before matched with far field condition.
Results	What kind of results should be written to the result files.

9.4.1 Buckling/eigen period analysis

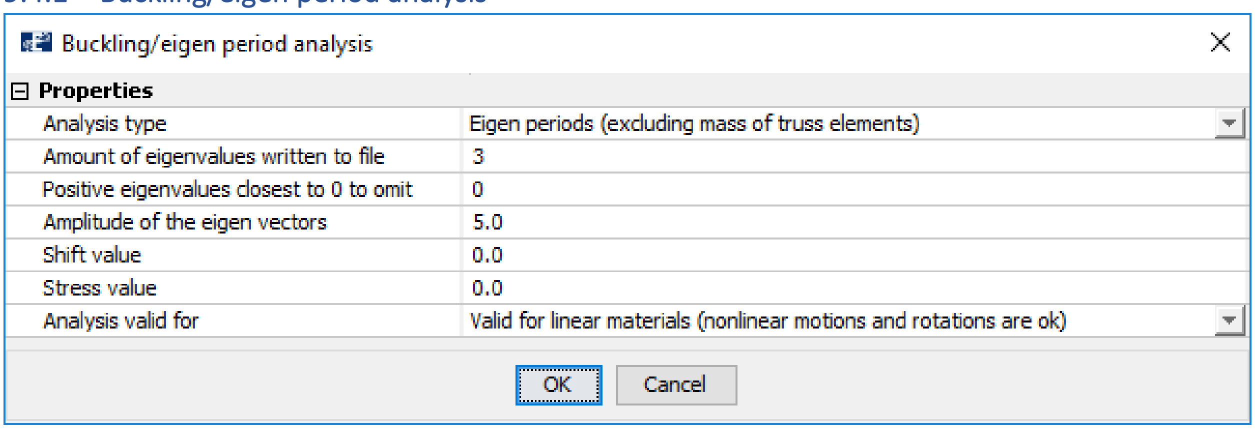


Figure 154 - Bucking/eigen period analysis window

It is possible to run an own buckling/eigen period analysis of the model. This is activated by the "Buckling/eigen period analysis" under Advanced for the time series. To edit the analysis run, click the …-button and the window shown in Figure 154 will show.

Analysis type

The type of analysis that should be conducted. Four options are available:

- Buckling: calculation of buckling factor. Buckling analysis is available for BEAM (Hydrodynamic and Morsion submerged) and MEMBRANE/ MEMBRANEX (Shell and Normal with bending stiffness)
- Eigen periods (excluding mass of truss elements): eigen period analysis, where mass of truss is omitted from the analysis.

- Eigen period analysis (including mass of truss elements): eigen period analysis, where mass of truss is included in the analysis.
- Eigen period analysis (excluding mass of truss elements and mass caused by node loads): eigen period analysis, where mass of both truss and Pointload is omitted from the analysis.

Amount of eigenvalues written to file

Must be higher than 0 and lower than 7.

Positive eigenvalues closest to 0 to omit

How many positive eigenvalues closest to 0 that shall be omitted in the eigenvalues written to file.

Amplitude of the eigen vectors

The amplitude the eigen vectors are multiplied with in the results. Makes it easier to evaluate results.

Shift value

Value where found eigenvalues are centered around.

Stress value

Target stress value. Currently not used, but all stress values are reported.

Analysis valid for

Eigenvalue analysis valid for linear material relation (nonlinear motions and rotations are ok) and general, but noisier buckling response.

9.4.2 Non linear density field

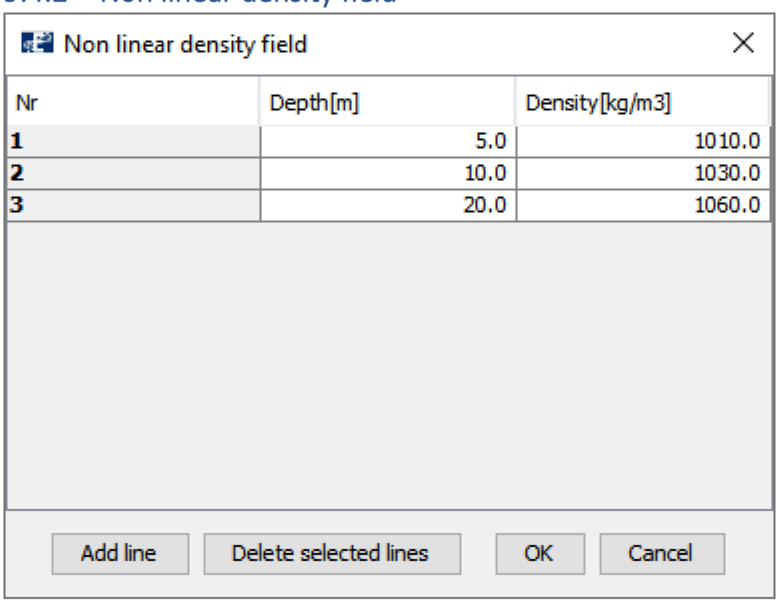


Figure 155 - Non linear density field window

It is possible to change the density of fluid outside membrane elements. This applies for these load formulations; Lice skirt and Closed compartment. This is activated by the “Non linear density field” under Advanced for the time series. To edit the density, click the …-button and the window as shown in Figure 155 appear.

One may add or delete input-rows. Values between input-rows are interpolated. The table in Figure 155 will for instance give a density of 1045kg/m^3 at 15 meters depth. Values above the highest depth and below the lowest depth will be constant. In this example, the density at depth 0-5 meters is 1010 kg/m^3 and depth below 20 meters is 1060 kg/m^3.

9.5 Export to file

Clicking the “Export” button in Figure 140 will trigger an analysis. First a file name is chosen and files for the AquaSim engine is created. If “Analyse immediately after export” is enabled, then the analysis will start immediately. If grouping is enabled the corresponding bat-files are started in the cmd-window in Windows.

10. Import terrain

Importing terrain is done from **`Commands > Import terrain.`**

10.1.1 Center location

For AquaEdit to be able to use the terrain data it needs to map the origin of the model to a known GPS coordinate in the terrain data.

Input type

Selects the type of GPS coordinates. Supported types are Decimal (DD), Hour Minute (DM), Hour Minute Second (DMS) and UTM.

When UTM is selected the field “Zone” will be active and this must have an input for the UTM coordinates to make sense.

Longitude and latitude

The longitude and latitude coordinates that will be mapped to the origin.

Heading

If the terrain should be rotated, input here in degrees. This can be edited after import is done as well.

North axis

Tells where the maps north axis is pointing towards, by default this is along the positive x-axis.

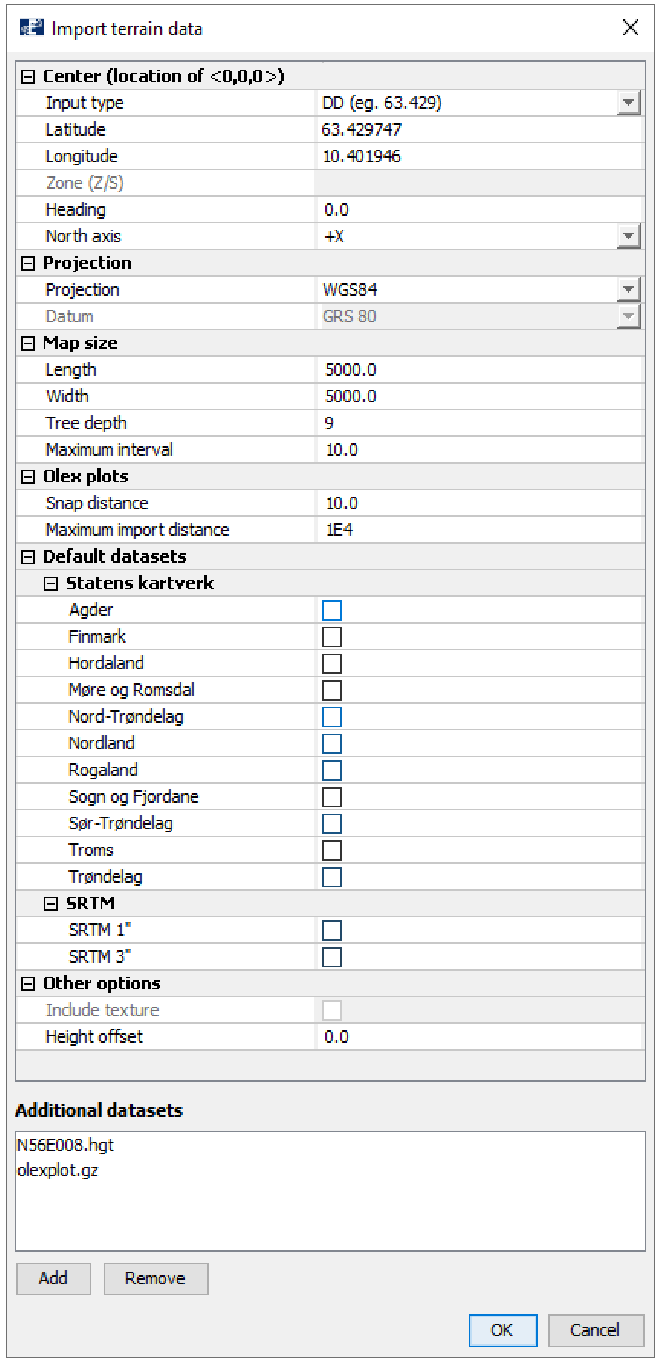


Figure 156 - Import terrain window

10.1.2 Projection

Projection

Two types of projections can be used; WGS84 and UTM. The UTM projection is recommended to only be used along with input type of UTM coordinates.

Datum

The only datum currently supported is GRS80.

10.1.3 Map size

Length and width

The size of the imported terrain in meters.

Tree depth

The terrain data will be split into rectangles and further subdivided to add detail. The higher the tree depth the more subdivisions will be made. The number of subdivisions will depend in the detail of the map data. Very low resolution may need to be tweaked down to 6-8, very high-resolution data can go up 20. If the data is not of adequate resolution, there will be gaps in the terrain model.

Maximum interval

Maximum interval is the maximum slope that is allowed in the model. If the model is very jagged and of high density this can go all the way down to 0.5-1. If the data is of poor quality, especially with very high height differences this value might have to be tweaked up towards 20-50, but usually this means the data is too poor for a good result.

10.1.4 Olex plots

It is possible to import geometry exported from Olex as well as a terrain here. Under “Additional datasets” it is possible to add Olex geometry files that will be read. The geometry within these files will be added as nodes and elements to the model. In order for the terrain importer to understand that a Olex geometry is imported, rather than Olex terrain, the user must specify this in the File type drop down field at the bottom of the “Open files” dialog.

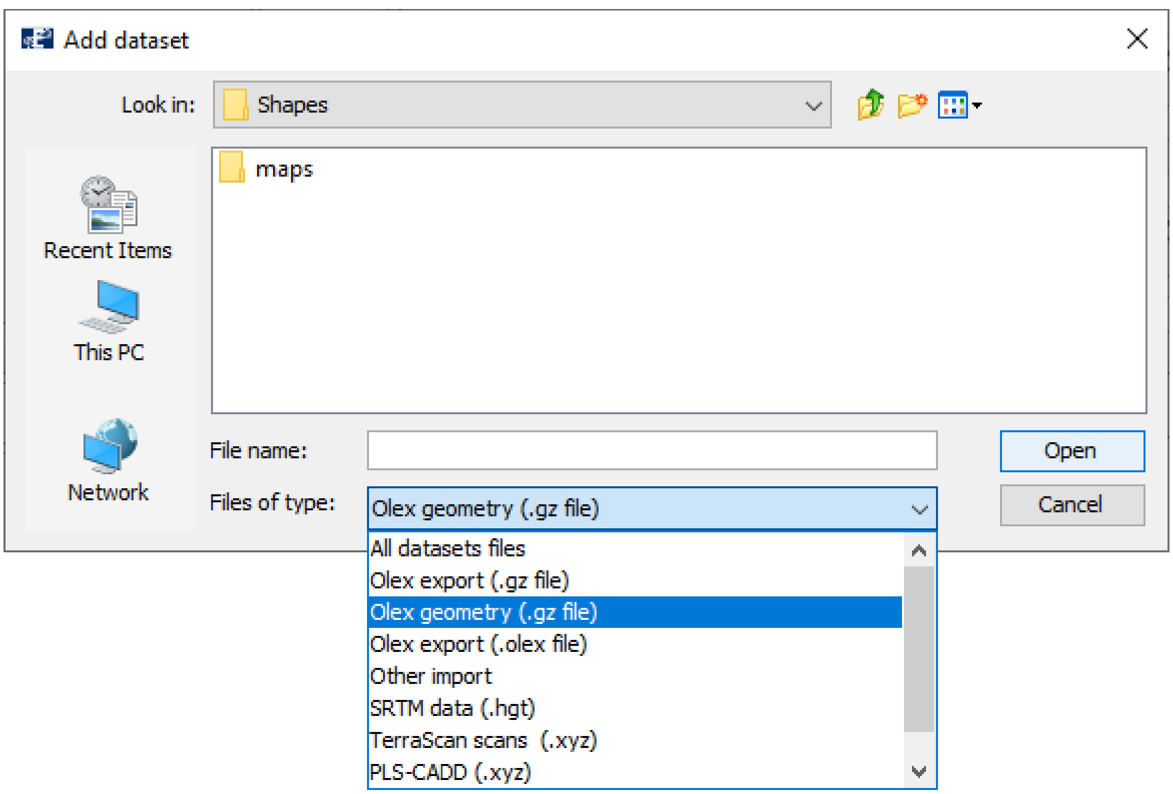


Figure 157 Selecting the correct file type for importing olex geometry

Snap distance

How close nodes can be before they snap together.

Maximum import distance

Points that are further away from the center than this distance will not be imported.

10.1.5 Default datasets

If AquaEdit detects known datasets in the shapes\map folder these will be available to include here. To use data from Statens Kartverk the file names in the shapes\map folder must be named after the abbreviation of the county. For instance, Nord-Trøndelag must be named "NT.txt" for it to be found. Here are the abbreviations of the counties in Norway:

Østfold: ØF

Akershus: AK

Oslo: OS

Hedmark: HE

Oppland: OP

Buskerud: BU

Vestfold: VF

Telemark: TE

Agder: AGD

Aust-Agder: AA

Vest-Agder: VA

Rogaland: RO

Hordaland: HO

Sogn og Fjordane: SF

Møre og Romsdal: MR

Trøndelag: TRL

Sør-Trøndelag: ST

Nord-Trøndelag: NT

Nordland: NO

Troms: TR

Finmark: FI

To use SRTM data one must download these data and put them in the respected folders named "hgt1" and hgt3" in the shapes\map folder. The link to download this is http://viewfinderpanoramas.org/dem3.html and one can use the interactive maps on this site to download data for 1" and 3". This is downloaded as a .zip-file and the user must unzip these and copy the files into the folders. It is not recommended to include both 1" and 3" data at the same time.

10.1.6 Other options

Include texture

This will download google earth satellite data for the area and apply that to the terrain. Not available at the current moment.

Height offset

Applies the given offset to the Z axis of the imported terrain, effectively raising or lowering the terrain.

10.1.7 Additional datasets

Other data sources can be added by clicking the "Add" button, all other datasets will be shown here. Supported sources are Olex files, generic txt-files, STRM data, XYZ files and OBJ 3D-models.

11. TPE file format

In order to facilitate easy import og geometry generated or otherwise obtained from other programs, scripts, or manual modelling, AquaEdit can also import a file format called .TPE or Point-To-Point files. These files follow a very simple format based on two pairs of xyz coordinates the create a line segment. It also has some functionality to create components, set names of components, and to load files from the library with component properties.

11.1 Description of .tpe file format

See 0 for example of file.

11.1.1 Elements

The basics of a .tpe file is two sets of coordinates represented by 6 numbers. X, Y, Z of the first node and Z, Y, Z of the 2nd node. At the most basic level, this is all a .tpe file needs to contain. Two sets of coordinates always represents one line, and the number can be placed after each other with spaces in between.

Example:

```
0 0 0 1 1 1
```

Creates one line from (0, 0, 0) to (1, 1, 1).

Example:

```
0 0 0 1 1 1 2 2 2 3 3 3
```

Creates two lines. One from (0, 0, 0) to (1, 1, 1) and one from (2, 2, 2) to (3, 3, 3).

In these cases line splitting does not matter.

11.1.2 Components

The lines can be divided into components, either using “***” which creates a new, unnamed truss component named “Imported data (number in the sequence)”. This command can be interleaved anywhere.

Example:

```
0 0 0 1 1 1 *** 2 2 2 3 3 3
```

Creates two components, one implicit called “Imported data” for the first element, and “Imported data (1)” for the 2nd element.

In addition to this command there is also a command to insert a named component using “###”. This command requires a line break after the name of the component.

Example:

```
0 0 0 -10 -10 -10 ### Test1
1 1 1 10 10 10
```

Creates two components, one called “Imported data” for the first element, and one called Test1 with a line from (1, 1, 1) to (10, 10, 10) in it.

Note how this requires a line break after the component name in order to function correctly, also note that “(“ and “)” are reserved letters and cannot be used in the component name.

11.1.3 Component type

When using ### to create components one can also specify the type of the component, this is done by putting the type of the component in parentheses behind it, like (truss), (beam), or (membrane)

Example:

```
### Test2 (beam)
10 10 10 50 50 50
50 50 50 100 100 100
```

This creates a component named “Test2” of type beam with two lines (10, 10, 10) to (50, 50, 50) and (50, 50, 50) to (100, 100, 100).

11.1.4 Library integration

When specifying the type of the component one can also optionally choose a template from the library to copy the component properties. This is done by specifying the name of the library template after the component type, separated by a comma. Note that component type must be supplied for AquaEdit to find the correct library. The name for the library template is the name set in the template component itself, which can be different from the filename of the template.

Example:

```
### Test2 (truss,LibraryItem)
10 10 10 50 50 50
50 50 50 100 100 100
```

This creates a component called "Test2" of type truss, and applies the library template called LibraryItem, if it can find it, from the truss library on the PC.

11.1.5 DOF

Translational degrees of freedom can also be specified using underscore (_) in front of the number. There must not be any spaces between them. The last overridden DOF will always be used.

Rotational degrees of freedom can be specified using star (*). Note that there cannot be any spaces, and if both translational and rotational is specified, the translational must be specified first

Example:

```
### Test1
_1 _1 _1 10 10 10
1 1 1 -1 -1 -1
5 5 5 _*15 _*15 _*15
```

This creates the following lines, one from (1,1,1) to (10,10,10). The node at (1,1,1) will have all three translational DOFs turned on. The 2nd line from (1,1,1) to (-1,-1,-1) will have its (1,1,1) node merged with the previous element, and since there are no DOF overrides on this line the node will continue to have all three translational DOFs turned on.

A line will also be made from (5, 5, 5) to (15, 15, 15), the last node will have all its degrees of freedom locked.

11.1.6 Divide line

Using the tilde (~) a line can be set to divided into multiple segments.

Example:

```
0 0 0 ~10 100 0 0
```

This will create a line from (0, 0, 0) to (100, 0, 0) and it will be split into 10 segments. Note that the nodes in between the two endpoints cannot be assigned properties.

11.1.7 Point load

Using the percentage symbol (%) a pointload can be assigned to a node. It takes a single number following the % with no spaces. This value becomes the Z force (in Newton) of the pointload

Example:

```
%50 0 0 0 %10 100 0 0
```

The above example will create a line from (0, 0, 0) to (100, 0, 0) and a 50 N in the Z direction pointload will be added to the first node, and a 10 N in the Z direction pointload on the 2nd node.

11.1.8 Node decorator from library

Using &&& Decorator type,Name of decorator library item on its own line will make the next node get a node decorator loaded from the library.

The decorator type must be specified first (spring, impulseload, pointload, prescribeddisplacement, valve,shapedecroator) and then the name of library item. This must match the filename found in LIBRARY\decorators\type of decorator, excluding the .xml file.

Example:

```
10 10 10
&&& Spring,Test Boye
50 50 50
```

11.1.9 Continuation

Instead of having to specify every node twice when creating longer segments, the hash symbol (#) can be used to specify using the last node position instead.

Example

```
0 0 0 # 0 10 0 # 10 10 0 # 10 0 0 # 0 0 0
```

This example creates a 10m by 10m square, from (0, 0, 0) to (0, 10, 0) to (10, 10, 0) to (10, 0, 0) and back to (0, 0, 0)

11.2 Example .tpe file

```
0 0 0 1 1 1
***
0 0 0 -10 -10 -10
### Test1
_1 _1 _1
&&& Spring, Test Boye
10 10 10
### Test2 (truss,LibraryItem)
10 10 10 50 50 50
50 50 50 100 100 100
### Test3 (beam,LibraryItem2)
100 100 100 1000 1000 1000
### Verification
5 5 0 _10 _5 _0
10 5 0 *20 *5 *0
20 5 0
~10
%100
_*100 _*5 _*0
#
~5
100 100 0
#
~5
5 100 0
#
~5
5 5 0
```

Please note: This example assumes that there is a truss library item name LibraryItem, a beam library item named LibraryItem2, and a spring library item called Test Boye

11.3 Advanced settings

import.tpe.charset	The charset to use when importing files. Default is UTF-8. Acceptable values include (but not limited to): UTF-8, windows-1252

12.References

Aquastructures. (2023). *AquaSim - Verification and Benchmarking.* Tech. rep. TR-20000-1755-1. Rev 11.

Aquastructures. (2024a). *The AquaSim Package Theory User Manual.* Techn. rep. TR-20000-2049-1.

Aquastructures. (2024b). *Loads on Impermable Nets and Large Volume Objects in AquaSim.* Techn. rep. TR-FOU-2328-6. Rev9.

Aquastructures. (2024c). *Calculation of loads on turbine blades using BEM theory.* Techn. rep. TR-FOU-100004-5.

Aquastructures AS. (2022d). *AquaCross User Manual.* Trondheim: TR-20000-583-1.

Aquastructures AS. (2022e). *AquaTool User Manual.* Trondheim: TR-20000-583-4.

Aquastructures AS. (2022f). *AquaView User Manual.* Trondheim: TR-20000-583-3.

Aquastructures AS. (2022g). *Other User Manual.* Trondheim: TR-20000-583-5.

Aquastructures AS. (2024d). *Morison free plate load formulation to shell and membrane elements.* Techn. rep. TR-FOU-2328-8, Rev3.

Berstad, A. (2019). *Spectral wind loads in AquaSim.* Oslo.

Berstad, A. J. (2013). *Irregular wave calculation and validation.* Oslo.

Berstad, A. J., & Grøn, P. (2023). Model basin testing of a floating solar system compared to analysis for establishment of design verification culture. *X International Conference on Computational Methods in Marine Engineering. MARINE2023*.

Berstad, A. J., & Heimstad, L. F. (2015). Numerical formulation of sea loads to impermeable nets. *VI International Conference on Computational Methods in Marine Engineering. MARINE2015*, pp. 339-352.

Løland, G. (1991). *Current Force On and Flow Through Fish Farms. Dr.ing dissertation.* Trondheim: Division of Marine Hydrodynamics, The Norwegian Institute of Technology.