AquaEdit
AquaCross
» AquaView
AquaTool
Other

User manual

Document ID: TR-20000-583-3
Date: 01\03\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 AquaView. AquaView provides a visual presentation of the model and results from the analysis, allowing the user to see response as the model is exposed to environmental loads. It loads information about the displacement of nodes from both .AVS- and .AVZ-files. In addition, several properties of the model, such as axial force, moment, stress etc. can be displayed in the 3D-model.

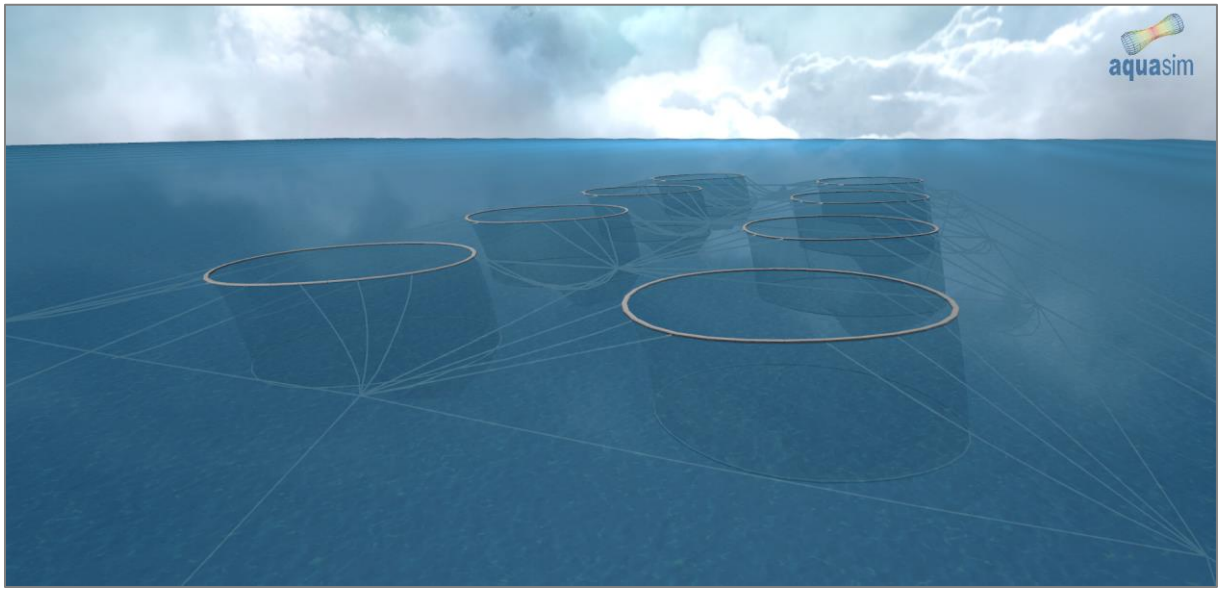

Figure 1 Analysis of a fish farm, viewed in AquaView.

2 Interface

2.1 Loading files

AquaView loads .AVS- and .AVZ-files, and may be opened by either double click an .AVZ-file or right click .AVS-file and choose **View in AquaView**. The .AVZ-files are zip-archives made by the AquaSim post-processor and contains compressed .AVS-files together with the corresponding .xml-files (containing component names) and .obj-files (containing shapes). Depending on the number of load conditions analyzed, there are up to three different types of .AVS- or .AVZ-files generated. The following example illustrates the three different file types:

Assuming a base filename *Run*, and load condition *01*, the first two .AVZ-files are named:

- *Run01.AVZ* (containing analysis results for each timestep)
- *Run01PFAT.AVZ* (containing maximum value for each node/element throughout the analyzed load condition) **Note (1)**.

If two or more load conditions are analyzed, it is convenient to obtain the maximum value from each and one of them. Upon completion of the analyzed load conditions, the file **max_run.bat** loads the different *PFAT.AVZ-files and creates a folder called **max_run_**. This folder contains a file presenting the maximum value for each node/element from all load conditions. Assuming the same bas filename, this file is named:

- *Run_max_out.AVZ*

Equivalent, the .AVS-files are named:

- *Run_01.AVS* (containing analysis results for each time step)
- *Run_01PFAT.AVS* (containing maximum values for each node/element)
- *Run_max_out.AVS* (maximum value in each node/element for all analyzed load condition)

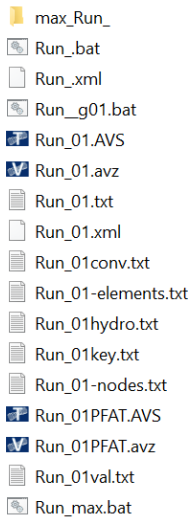

Figure 2 Generated .AVS- and .AVZ-files after completion of analysis

Note (1)

The user may experience that maximum values in PFAT.AVS differs from the values in .AVZ. In PFAT.AVS, AquaSim extracts the maximum value obtained from the last static step and all dynamic steps in the analysis. In the .AVZ files, AquaView views the maximum value obtained from all static- and dynamic steps.

2.2 Main View

Figure 3 presents the main view when starting AquaView. It contains a visual presentation of the model in 3D and is movable using the mouse cursor. On the top left side is the Top Menu bar, providing access to different options. At the bottom left side, options for controlling the playback of the analysis are found.

The transparency of the water surface is adjustable. By click and drag the blue bar on the top of the model-view, the water surface either gets dense or transparent. See red arrow in Figure 3.

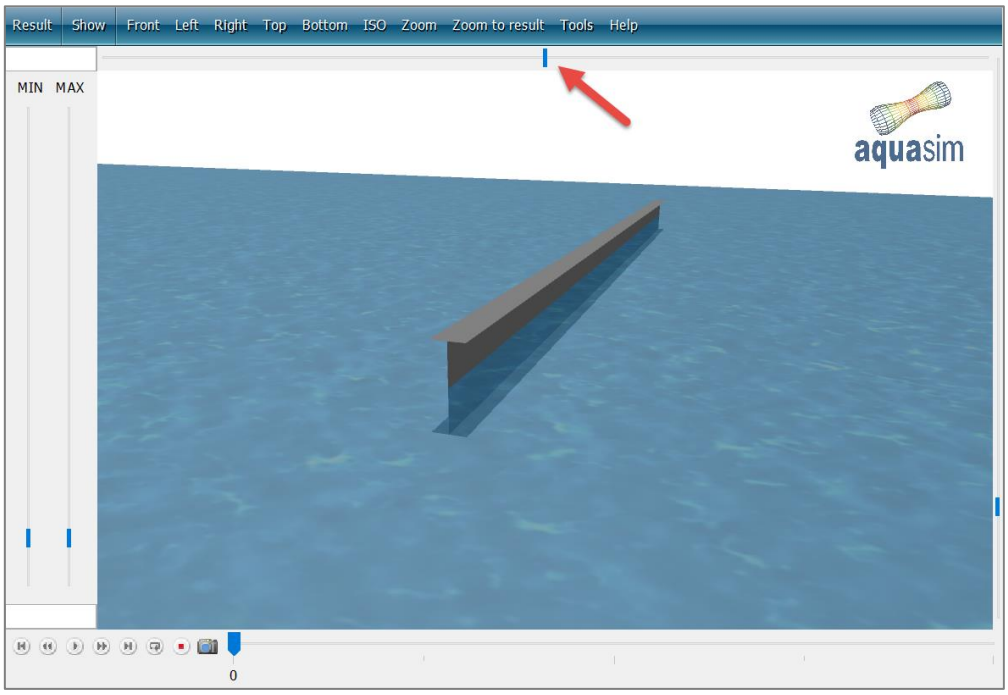

Figure 3 AquaView main view

2.3 Playback control

The playback controls are found in the lower left corner of the main view, see Figure 4 for enlarged section. It enables for continuous playback of the analysis, as well as choosing specific time steps. There are also possibilities for recording videos or taking screen shots by using the two rightmost buttons. Detailed description of the options is found in Table 1.

Figure 4 Playback control

Table 1 Options for playback control. The numbers refer to Figure 3

Option	Definition
1	Resets view to the first timestep in the analysis.
2	Resets view to the previous timestep in the analysis.
3	Continuous playback of the analysis.
4	View subsequent timestep in the analysis.
5	View the last timestep in the analysis.
6	Controls the playback speed, in seconds. By default, this is 10. This means it takes 10 seconds for playback from start to finish.

7	Record video of the analysis. Selecting this option, a window for choosing directory for saving will appear. Follow subsequent instructions.
8	Screen shot of the model for the current timestep.
9	Indicator for which timestep is displayed in the model view.

2.4 Top Menu bar

The Top Menu bar is found in the top left corner and provides access to view results and other settings. By default, the bar consists of twelve buttons, as illustrated in Figure 5. It contains a Result Menu, Show Menu, View options (Front/ Left/ Right/ Top/ Bottom/ ISO), Zoom options (Zoom and Zoom to result), Tools and Help. Detailed description of the options is presented in chapter 3 through 8.

Figure 5 The Top Menu bar

2.5 Graphic Color Palette

The Graphic Color Palette controls the color view of the results. Figure 6 exemplifies this by viewing axial forces from an analysis. By default, the maximum value is viewed as red and minimum as grey. The color palette may be customized, see **Tools** > **Settings** > **Visual** > **Stress Color palette**. By click and drag the blue bars, under **MIN** and **MAX**, the user may adjust the presented result values. It is also possible to adjust and type exact result values in the top left and lower fields.

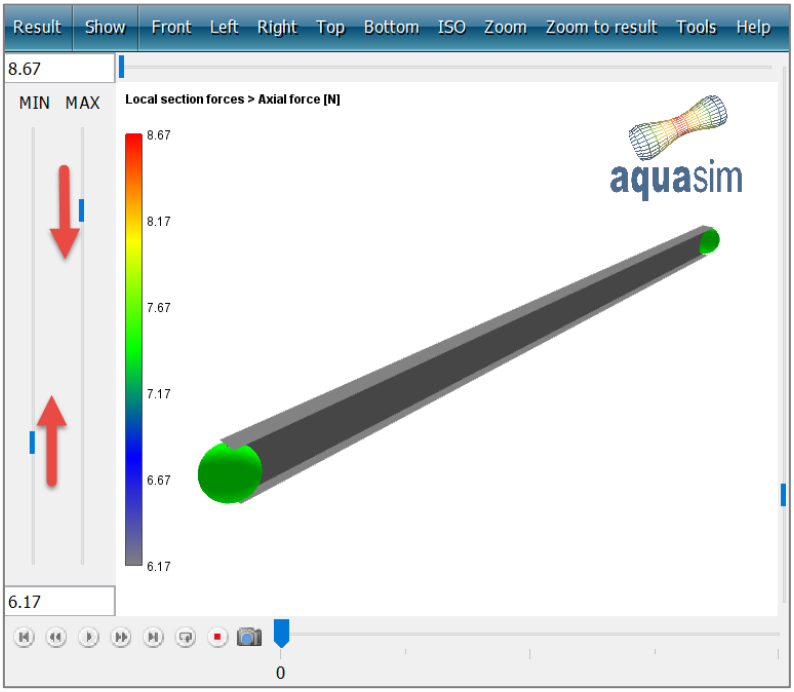

Figure 6 Graphic Color Palette. Local section force, Axial force is shown as an example

2.6 Selections in main view

Use the computer mouse for selections and navigation in the model. The available selection- and navigation options are described in Table 2.

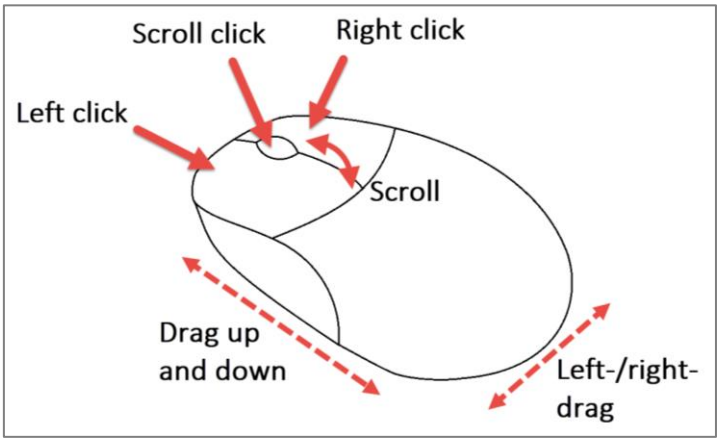

Figure 7 Use computer mouse for selections and navigation

Table 2 Selections and navigation

Description	**Selection computer mouse**
Select element	Left click
Translation	Right click + right-drag
Translation, slower	Shift + right click + drag
Zoom	Scroll
Rotation	Scroll click + drag
Rotation about global coordinate system	Shift + scroll click + drag
Rotation about selected element	Left click + scroll click
Description	**Result selections (require a selection in the Result menu prior)**
Display results for individual elements	Right click
Display maximum analysis results for component groups	J
Display max and min analysis results for component groups, see Figure 8	I
Copy results	Ctrl + C
Copy options in the “Max/ min values”-window, see Figure 9	Ctrl + Alt + C
Description	**Special selection trackpad**
Rotation	Ctrl + right-drag
Zoom	Alt + right-drag up and down

Max/min values for Local_section_forces.Axial_force_[N]

Component name	Component index	Max	Min
Collar Ø500 SDR13.6	1	31504.00	-10699.00
Ring	2	3167.90	-7815.40
Net	3	N/A	N/A
Lice skirt	4	N/A	N/A
Mooring line	5	99206.00	5144.10
Chain	6	98639.00	4973.70
Frame	7	39009.00	-4276.50

Figure 8 "Max/ min values"-window, when pressing I in AquaView

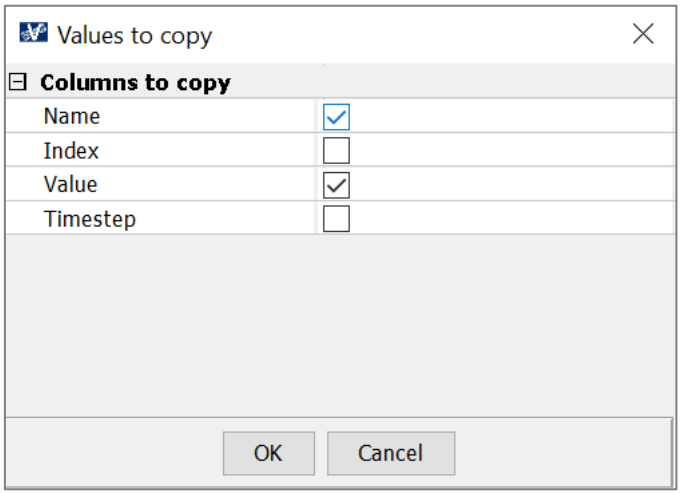

Figure 9 Copy options in the "Max/ min values"-window Ctrl + Alt + C

2.7 Adjust node indicator size

A selected element is indicated by two green spheres as shown in Figure 10. The size of these spheres can be adjusted by click and drag the blue bar to the right in the main view.

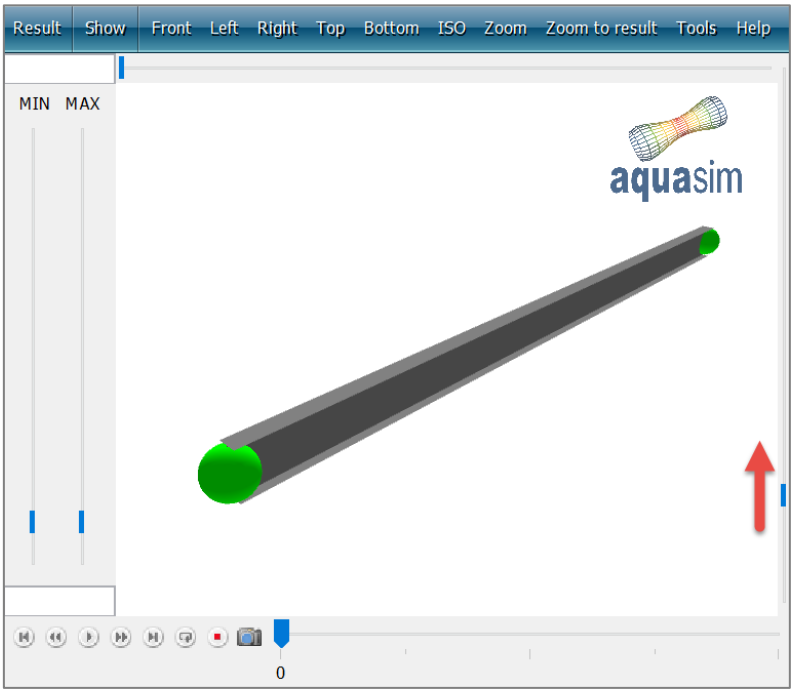

Figure 10 Node size indicator

2.8 View results for individual elements

Results may be viewed for individual elements in the model. This is done by right click on the element of interest, a new window will appear as shown in Figure 11. More information about the Element Result-window is provided in chapter 3.12.

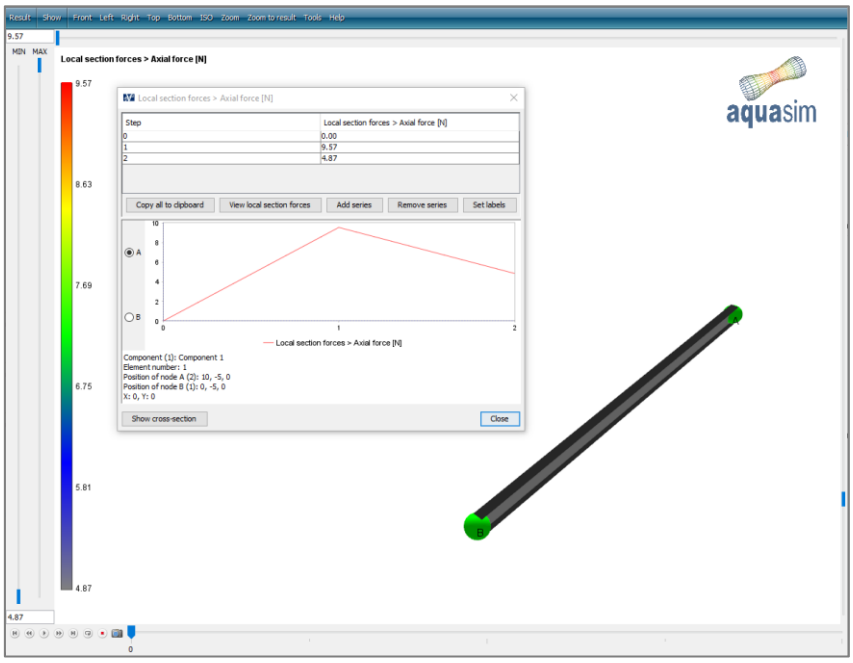

Figure 11 Element Result-window

3 Result Menu

This chapter provides a detailed description of what is found in the Result Menu. Different result options vary with the file type; e.g. .AVZ has other options than *PFAT.AVZ. It also varies with the content of the AquaSim model, if specific component groups are not modelled, AquaView does not load results for this. E.g., if membrane (net) is not a part of the model, AquaView does not present result options for this component group. This chapter describes all result options in the Result Menu, regardless of the file type (.AVS, *PFAT.AVS, *max_out.AVS, .AVZ, *PFAT.AVZ or *max_out.AVZ). Details about the options are presented in Table 3.

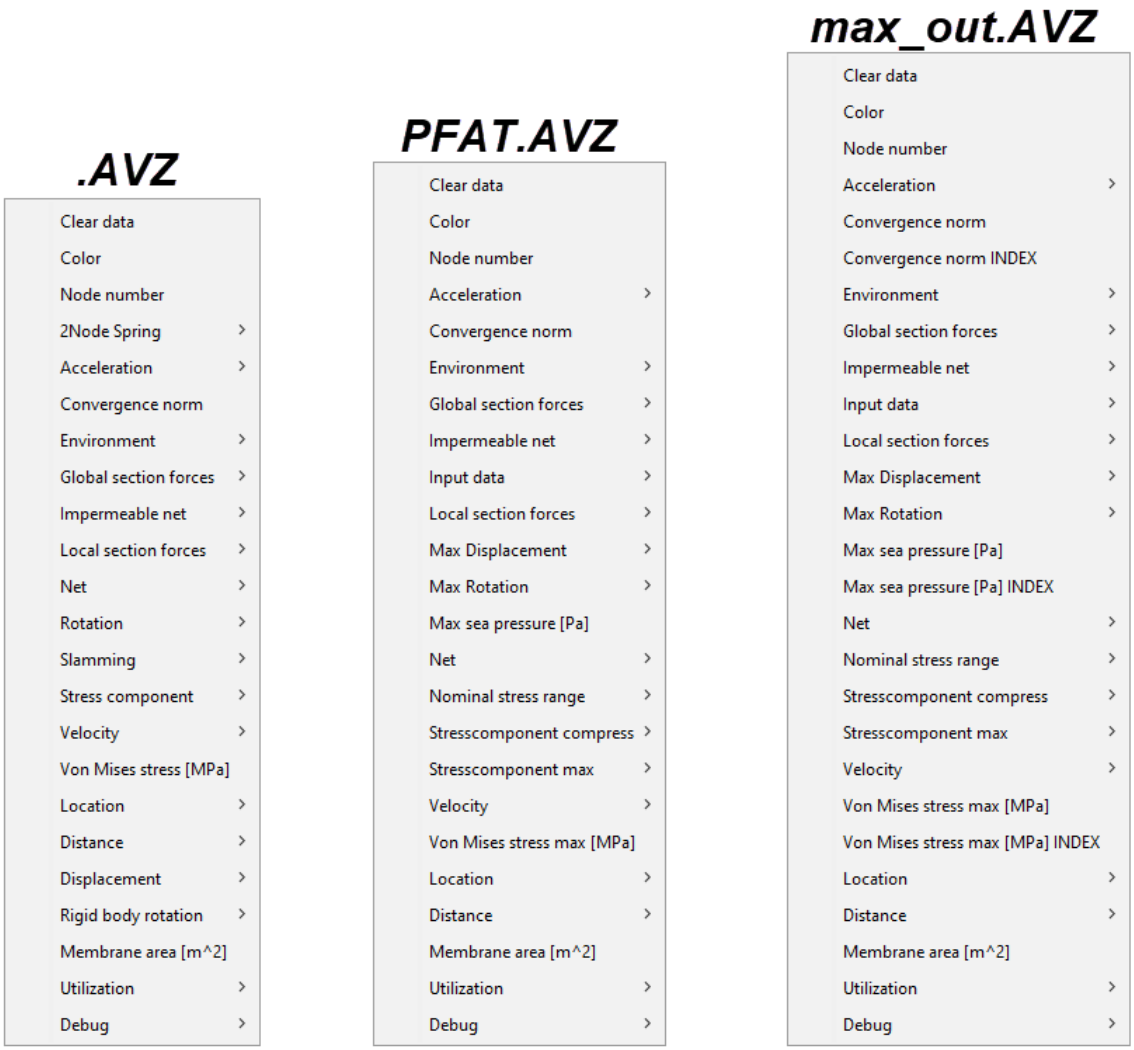

Figure 12 Result Menu options

The files containing maximum result value for several load conditions (the max_out.AVZ/ -.AVS-files) has additional option: to view which load condition causing the maximum result value, this is named INDEX. E.g. Von Mises stress this is called **Von Mises stress max [MPa] INDEX**. By pressing **J** the link between component group and INDEX is displayed. The link between INDEX and file numbering is found by pressing **Ctrl+F**.

Table 3 Options in the Result Menu

Option	Definition
Clear Data	Removes any plotted data.
Color	Color the component groups after the scheme from AquaEdit.
Node number	Plots the node numbers of all nodes.
2Node Spring	If the model contains the component type NODE2NODE, this option will be available. See chapter 3.11 for more information.
Acceleration	Plots max acceleration of nodes throughout analyzed load condition(s). Available for PFAT.AVZ, PFAT.AVS and max_out.AVZ/ -.AVS. Hinge: having selected the checkbox "Local" in AquaEdit, acceleration in the Hinge-node is with respect to element's local coordinate system. Otherwise, it is with respect to the global coordinate system.
Convergence norm / Convergence norm INDEX	- **Convergence norm:** plots the convergence norm for different parts of the model. This function is useful in the case of identifying regions with convergence difficulties. AquaSim reports the status of convergence for each time step. For more information on convergence and convergence criteria, reference is made to the AquaSim Theory Manual. - **Convergence norm INDEX:** view which load condition causing the maximum convergence norm.
Environment	- **Distance to surface incl diff est [m]**: the distance between a selected node and the water surface, measured in meters. This distance includes the effect of diffracted waves. Negative signs mean the node is below the water surface, positive signs mean the node is above the water surface. This result option will show results for load formulations that include calculation of diffracted waves (Except for General impermeable net which is found in Impermeable net > Height to water surface [MH2O]). - **Max above water surface [m]**: the maximum distance between a selected node and the water surface from achieved from an .AVZ-file. This distance includes the effect of diffracted waves. Negative signs mean the node is below the water surface, positive signs mean the node is above the water surface
Global section forces	Plots force and bending moment with respect to the global coordinate system in the model.
Impermeable net	Result options for impermeable nets. More information is found in chapter 3.1.
Input data	Enables several different options for information about the model defined in AquaEdit. E.g. the ID-number of components, elements and nodes, the E-modulus, net fouling factor, weight of elements etc. This information is useful for controlling the nature of the model and the input-parameters.
Local section forces	Plots axial- and shear forces, bending moment, torsion moment, etc. with respect to each element's local cartesian system. - **Axial force inc net [N]:** axial force in truss and membrane panels that adjoin the truss. Commonly applied to check the load distribution between truss and membrane.
Displacement / Max Displacement	- **Displacement**: plots the displacement of nodes in the model. The options are in X-, Y-, and Z-direction separately with respect to global coordinate system.

	- **Max Displacement:** reports maximum displacement of nodes for one or several load conditions.
Max Rotation	Reports maximum rotation of elements for one or several load conditions.
Max sea pressure [Pa] / Max sea pressure [Pa] INDEX	- **Max sea pressure [Pa]:** plots sea pressure on beam element. Can be applied for study of submersion caused by waves or current. The unit is Pascal [Pa]. Total pressure is $p = p_d - \rho g z$, where p_d is dynamic pressure due to waves (see e.g. (Faltinsen, 1990) pp. 16) and $\rho g z$ is hydrostatic pressure. - **Max sea pressure [Pa] INDEX:** view which load condition causing the maximum sea pressure.
Net	Available options are: - **Axial force in horizontal twines [N]** - **Axial force in vertical twines [N]** - **Elongation horizontal twines [%]** - **Elongation vertical twines [%]**
Nominal stress range	Plots the stress range i.e., the distance between the upper- and lower limit in the stress curve. Options for reporting from different positions of a cross section. More details are provided in chapter 3.7.
Stress component compress	Similar as Stress component. Only the stress component is based on compression force and bending moment about local axis, rather than axial force and bending moment. More information about stress component in chapter 3.6.
Stress component/ Stress component max	Plots stress component for different positions in a cross section. More details are provided in chapter 3.6. / Plots maximum stress component for one or several load conditions.
Rotation	Plots the rotation of elements in the model. The options are in X-, Y-, and Z-direction separately with respect to global coordinate system.
Slamming	Se chapter 3.2.
Velocity	Plots the maximum velocity of nodes throughout the analysis. Reported as radial- and vectorial for x-, y- and z-direction separately. Hinge: having selected the checkbox “Local” in AquaEdit, velocity in the Hinge-node is with respect to element’s local coordinate system. Otherwise, it is with respect to the global coordinate system.
Von Mises stress [MPa]/ Von Mises stress max [MPa]/ Von Mises stress max [MPa] INDEX	- **Von Mises stress [MPa]:** Plots Von Mises stress for elements for each time step. More information about Von Mises is found in chapter 3.8. - **Von Mises stress max [MPa]:** Plots maximum Von Mises stress for one or several load conditions. - **Von Mises stress max [MPa] INDEX:** view which load condition causing the maximum Von Mises stress.
Location	Plots the positions of nodes with respect to the Global Coordinate System for each time step. The options are in x-, y- and z-position separately.
Distance	Four options are available: - **Between components:** plots max/min distance between components with respect to the global coordinate system. To select a reference-component, double left click on a component in the 3D view. Reported for X-, Y-, Z- or All axis separately. - **Between elements:** plots max/min distance between elements with respect to the global coordinate system. To select a reference-element, double left click on an element in the 3D view. Reported for X-, Y-, Z- or All axis separately.

	- **Between nodes:** plots distance between nodes with respect to the global coordinate system. To select a reference-node, double left click on an element in the 3D view. The reference node will change to red color. Reported for X-, Y-, Z- or All axis separately. - **Distance to water surface [m]:** the distance between a selected node and the water surface. Note: this option does not include the effect of diffracted waves, if the analysis includes waves. - **To terrain:** plots distance to terrain. This option is only available if terrain is included in the model.
Rigid body rotation	Plots the change in angle about the global coordinate system X-, Y-, and Z-direction from initial condition.
Membrane area [m2]	Plots the surface area of a net. A net is a circular, or conical structure, built up by membrane elements. - By right clicking on a membrane element, the surface area of the individual net, as a function of each time step is presented. Pressing **I**, the maximum and minimum surface area throughout the analysis is presented for each membrane component group.
Utilization	This option assumes that Breaking load and Material coefficient for truss elements is defined in AquaEdit. The utilization of truss elements is presented, using load factor from NS9415:2009.
Information on springs	Plots forces, moments and displacement of springs defined in AquaEdit. An option for only displaying information of buoys is available. Useful for quickly obtain e.g. maximum forces on buoys. The results are presented together with spring number, node ID (i.e. the spring node) and Node name.

3.1 Result > Impermeable net

If the AquaSim model include components with load formulation General impermeable net, Lice skirt, Closed compartment or Surface tarpaulin, result options for Impermeable net will be available in AquaView. The result options include force components relevant for impermeable nets, see Figure 13.

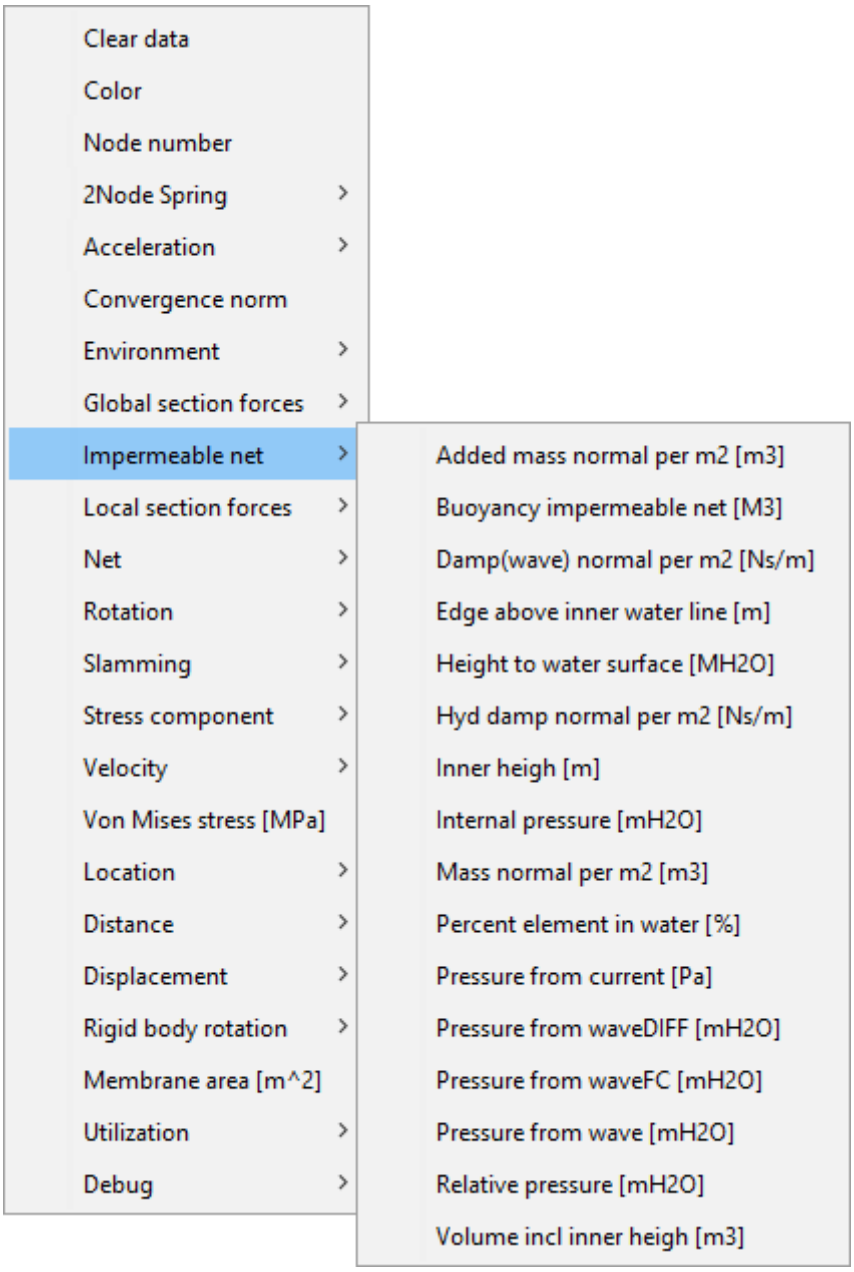

Figure 13 Result options for membrane with load formulation General impermeable net, Lice skirt, Closed compartment and Surface tarpaulin

Description of the options in **Result > Impermeable net** is provided in Table 4. The word 'tank' is being used to describe some of the options related to impermeable net. This is for simplifying reasons and is just a term to describe a closed compartment built up by impermeable net panels.

Table 4 Options in Result > Impermeable net

Option	Definition
Added mass normal per m2 [m3]	This option views the resulting added mass normal to the inside of the tank, results are volume per square meter. This option is related to the input "Added mass coefficient" for impermeable nets in AquaEdit. The added mass is calculated as follows for the different "Type of diffraction load": - MacCamy-Fuchs: the added mass is multiplied with the radius of the tank. For example: if the radius is 10m, an added mass coefficient of 1.0 will result in 10mH2O per square meter added mass of the cylinder. - Numerical diffraction: the "Added mass coefficient" is treated as a factor of what the numerical method predicts as added mass. For example: if "Added mass coefficient" = 1, all of the numerically calculated added mass is accounted for. A factor of 0.0 omits the added mass, and 2.0 will double the numerically added mass. For more information, reference is made to (Aquastructures, 2024a).
Buoyancy impermeable net [m3]	Buoyancy volume of the tank. Results require that the option "Bottom factor, 0 if water flow through bottom" = 1.0 in AquaEdit.
Damp(wave) normal per m2 [Ns/m]	Wave damping normal to the tank wall, results are [Ns/m] per square meter.
Edge above inner water line [m]	The distance between water level inside the tank and tank edge, as illustrated in Figure 14.
Height to water surface [MH2O]	The distance between a selected node on the tank wall and the water surface. This height includes the effect of diffracted waves. Reported in meters water column [MH2O], where 1.0 MH2O correspond to 9806.7 Pa.
Hyd damp normal per m2 [Ns/m]	Hydrodynamic damping normal to the tank wall. Hydrodynamic damping is due to acceleration of water particles on the outside tank wall.
Inner height [m]	The distance between outside- and inner water level in the 'tank', see Figure 14.
Internal pressure [mH2O]	Static internal pressure on the tank wall. This pressure includes contribution from "Height of fluid level inside tank relative to sea level" if this is defined in AquaEdit.
Mass normal per m2 [m3]	Amount of water normal to the inside tank wall that is accelerated due to tank motions, results are volume per square meter. This is a type of added damping and is related to the "Inner fluid mass scaling" for impermeable nets in AquaEdit.
Percent element in water [%]	Plots the percentage amount of the element that is submerged in water.
Pressure from current [Pa]	Resulting pressure on the tank wall based on relative velocity between the fluid flow (current and waves) and tank. Results are reported in Pascal.
Pressure from waveDIFF [mH2O]	Diffraction pressure on tank wall, due to incoming waves. Reported in mH2O, where 1.0 mH2O corresponds to 9806.7 Pascal.
Pressure from waveFC [mH2O]	Froude-Kriloff pressure on tank wall, due to incoming waves. Reported in mH2O, where 1.0 mH2O corresponds to 9806.7 Pascal.
Pressure from wave [mH2O]	Total pressure from incoming waves (i.e. the sum of diffraction pressure and Froude-Kriloff-pressure). Reported in mH2O, where 1.0 mH2O corresponds to 9806.7 Pascal.
Relative pressure [mH2O]	Relative pressure between inside- and outside the tank wall. Reported in mH2O, where 1.0 mH2O corresponds to 9806.7 Pascal.

Volume incl inner heigh [m3]	Total volume of water inside the tank, inclusive the inner heigh of water extending above the waterline outside the tank.

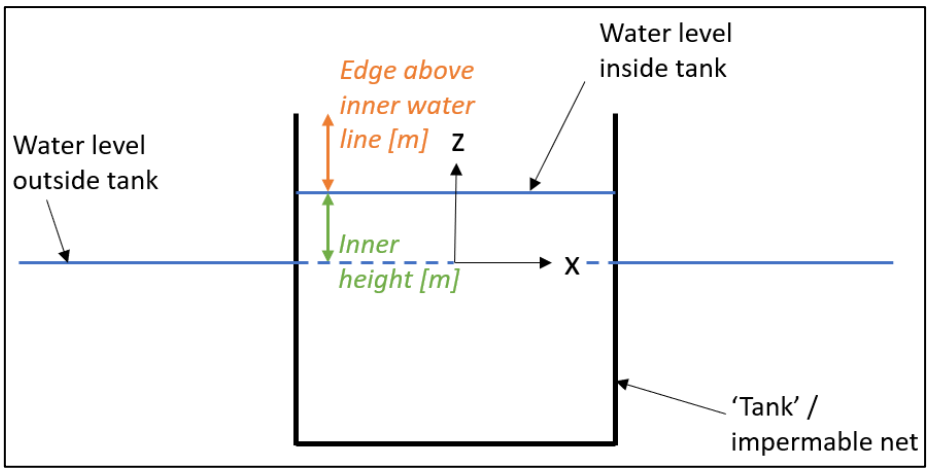

Figure 14 Result options for membrane with load formulation General impermeable net, Lice skirt, Closed compartment and Surface tarpaulin

The force (i.e. pressure) on impermeable nets are calculated by Morison equation, with the following force contributors:

$$F = \underbrace{\rho\, V\, \dot{u}}_{\mathbf{1}} + \underbrace{\rho\, C_a\, V\,(\dot{u} - \dot{v})}_{\mathbf{2}} + \underbrace{\frac{1}{2}\,\rho\, C_d\, A\,(u - v)\,|u - v|}_{\mathbf{3}}$$

Where:

- **1:** Froude-Kriloff and diffraction forces
- **2:** Added mass and hydrodynamic damping forces
- **3:** Drag forces

For more information about Morison equation, see (Wikipedia, 2020) or (Aquastructures, 2024a) chapter "Load formulation membrane: impermeable net".

3.2 Result > Slamming

This result option is available if the model contains beam elements with the load formulation *Morison submerged*, and the water volume correction *With slamming* is applied. The available results are shown in Figure 15. Description of the options are provided in Table 5.

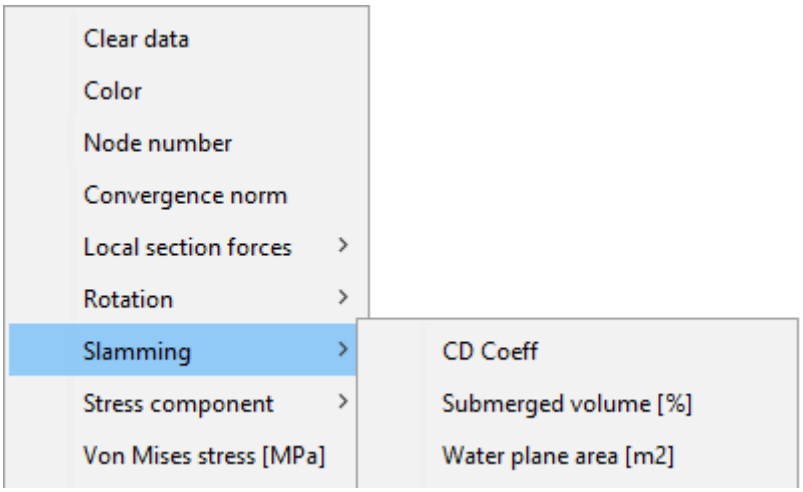

Figure 15 Result options for components with load formulation Morison submerged

Table 5 Options in Result > Slamming

Option	Definition
CD Coeff	Plots the instantaneous pressure coefficient, according to (Faltinsen, 1990) Fig. 9.11.
Submerged volume [%]	Plots the percentage amount of the element that is submerged in water.
Water plane area [m2]	Plots the area of the element intersecting with the water line.

3.3 Result > Impermeable free tarp

If the AquaSim model include membrane components with the Morison free plate load formulation, result options for Impermeable free tarp will be available in AquaView. The result options include force components relevant for tarp structures, see Figure 16.

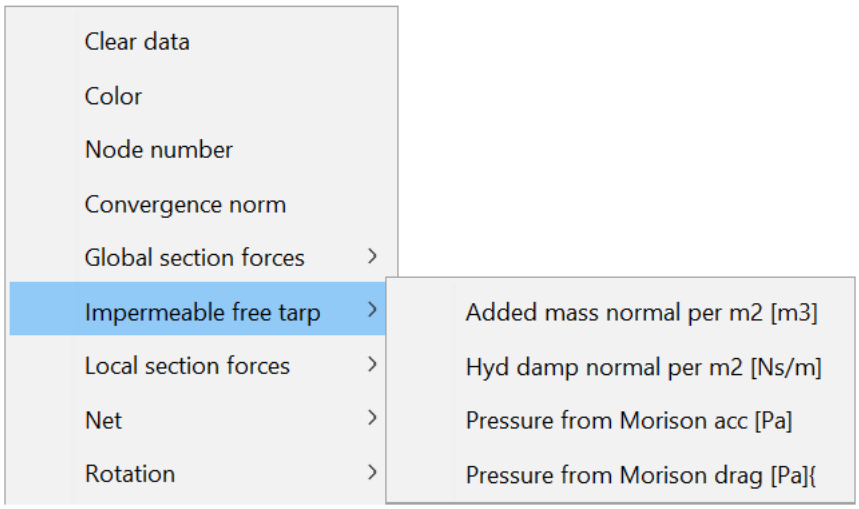

Figure 16 Result options for membrane with load formulation Morison free plate

Description of the options in **Result > Impermeable free** tarp is provided in Table 6.

Table 6 Options in Result > Impermeable free tarp

Option	Definition
Added mass normal per m2 [m3]	The amount of added mass per square meter normal to the impermeable free tarp panel.
Hyd damp normal per m2 [Ns/m]	The hydrodynamic damping force component per square meter normal to the impermeable free tarp panel.
Pressure from Morison acc [Pa]	Pressure on the impermeable free tarp panel due to fluid acceleration. Unit is Pascal.
Pressure from Morison drag [Pa]	Pressure on the impermeable free tarp panel due to drag force. Unit is Pascal.

The forces (i.e., pressure) on impermeable free tarps are calculated by Morison equation. For more information about the theoretical formulation of the Morison free plate, reference is made to (Aquastructures, 2024a) in chapter “Load formulation membrane: Morison free plate”.

3.4 Result > Shell and Shell stress

If the AquaSim model include membrane components with type Shell, options for Shell and Shell stress will be available in AquaView. The result options include force components and stresses relevant for shell structures, see Figure 17.

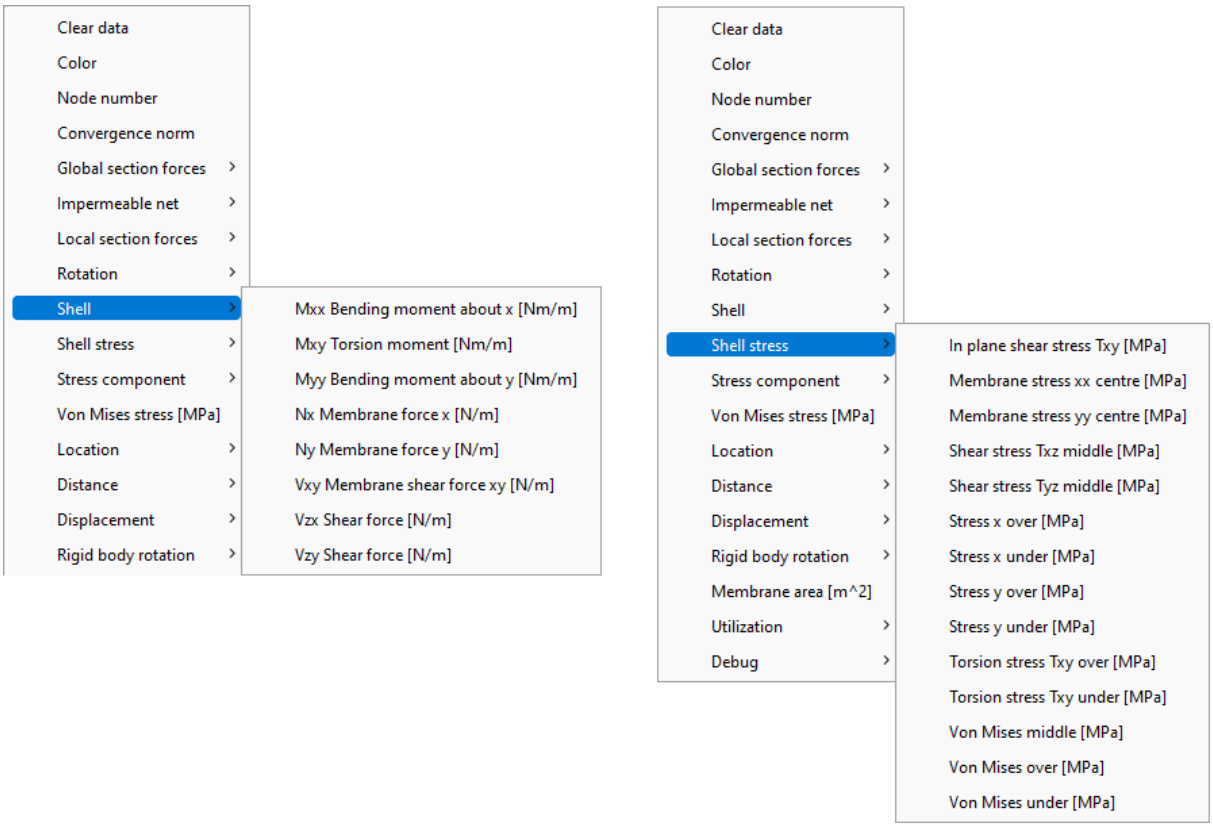

Figure 17 Result options for membrane of type Shell

The plane and local coordinate system of shell is presented in Figure 18. The local coordinates x and y is in the plane of the shell and z pointing outwards from the plane. Local x-direction is from node A to B, and y-direction is from node B to C.

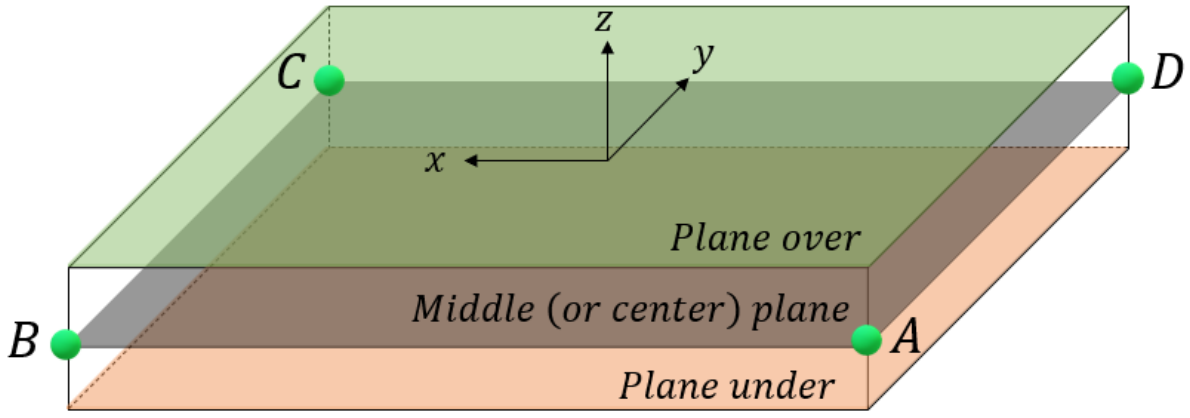

Figure 18 Shell element

Table 7 Options in Result > Shell

Option	Definition
Mxx Bending moment about x [Nm/m]	Bending moment about local x-axis of shell panel.
Mxy Torsion moment [Nm/m]	Torsion moment about local z-axis of shell panel.
Myy Bending moment about y [Nm/m]	Bending moment about local y-axis of shell panel.
Nx Membrane force x [N/m]	Axial force in local x-direction of shell panel.
Ny Membrane force y [N/m]	Axial force in local y-direction of shell panel.
Vxy Membrane shear force xy [N/m]	Shear force in shell panel in local xy-plane.
Vzx Shear force [N/m]	Shear force in shell panel in local zx-plane.
Vzy Shear force [N/m]	Shear force in shell panel in local zy-plane.

Table 8 Options in Result > Shell stress

Option	Definition
In plane shear stress Txy [MPa]	Shear stress in local xy-plane of shell.
Membrane stress xx centre [MPa]	Axial stress in x-direction, middle plane
Membrane stress yy centre [MPa]	Axial stress in y-direction, middle plane
Shear stress Txz middle [MPa]	Shear stress, in middle plane
Shear stress Tyz middle [MPa]	Shear stress, in middle plane
Stress x over [MPa]	Axial stress, in x-direction plane over
Stress x under [MPa]	Axial stress, in x-direction plane under
Stress y over [MPa]	Axial stress, in y-direction plane over
Stress y under [MPa]	Axial stress, in y-direction plan under
Torsion stress Txy over [MPa]	Torsional stress, in plane over
Torsion stress Txy under [MPa]	Torsional stress, in plane under
Von Mises middle [MPa]	Von Mises stress, in middle plane
Von Mises over [MPa]	Von Mises stress, in plane over
Von Mises under [MPa]	Von Mises stress, in plane under

3.5 Result > Net (6-sided masks)

If the AquaSim model include 6-sided masks (or hexagonal masks/hex-mesh), some additional options will be available in the Net-section in AquaView. The additional options are shown in Figure 19.

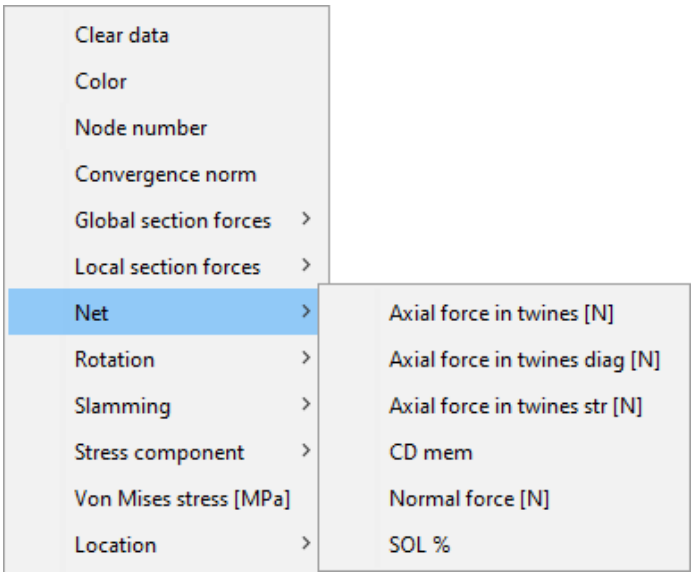

Figure 19 Result options for hexagonal nets (hex-mesh)

The options are further described in Table 9. The definitions for hexagonal shaped masks are given in Figure 20.

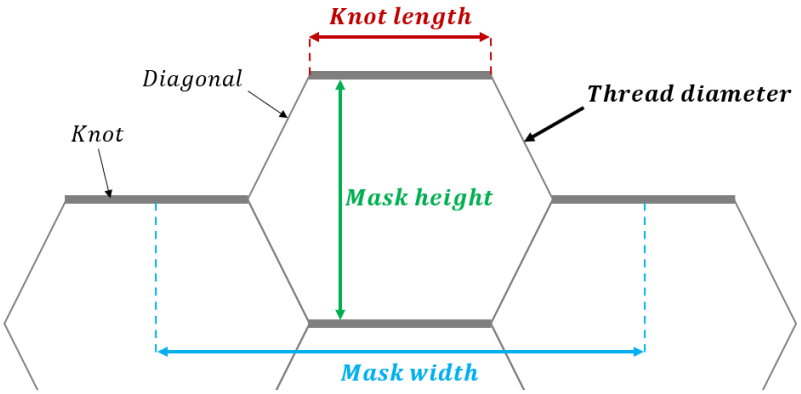

Figure 20 Definitions hexagonal shaped masks

Table 9 Options in Result > Net

Option	Definition
Axial force in twines [N]/ PFAT-file: Max axial force in twines [N]	Axial force in both Diagonal and Knot./ Maximum axial force from the entire time series.
Axial force in twines diag [N]	Axial force in individual Diagonal of the hexagonal mask.
Axial force in twines str [N]	Axial force in individual Knot of the hexagonal mask.
CD mem	Membrane drag coefficient for the hexagonal membrane panel.
Normal force [N]	The force in the 'visible' thread on the membrane panel. That is, the force summed over the number of threads it represents.
SOL %	Solidity of the hexagonal membrane panel.

3.6 Result > Stress component

Consider a 2D cross section as illustrated in Figure 21 (green rectangle). AquaView views stress components for four positions on a 2D cross section; **Upper flange**, **Lower flange**, **Left web**, and **Right web**. The stress component is based on the normal force (i.e., axial force) about the local coordinate system (y- and z-axis in Figure 21). Here, **a** is the distance from the y-axis to the position **Lower flange**. For beam elements, AquaView views stress component at four positions in the cross section. For truss elements, the stress component is found by selecting **Right web**. Stress due to shear force and torsion is not included in Stress component, only normal stress is included.

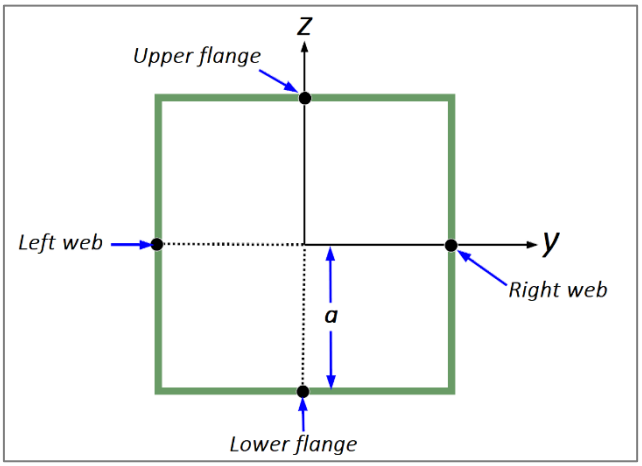

Figure 21 Stress components in a 2D cross section

3.7 Result > Nominal stress range

Nominal stress range is defined as the distance between the upper- and lower limit in the stress curve, illustrated in Figure 22. As for **Stress component**, **Nominal stress range** also provides options for viewing the nominal stress range at four positions in the 2D cross section, as illustrated in Figure 21.

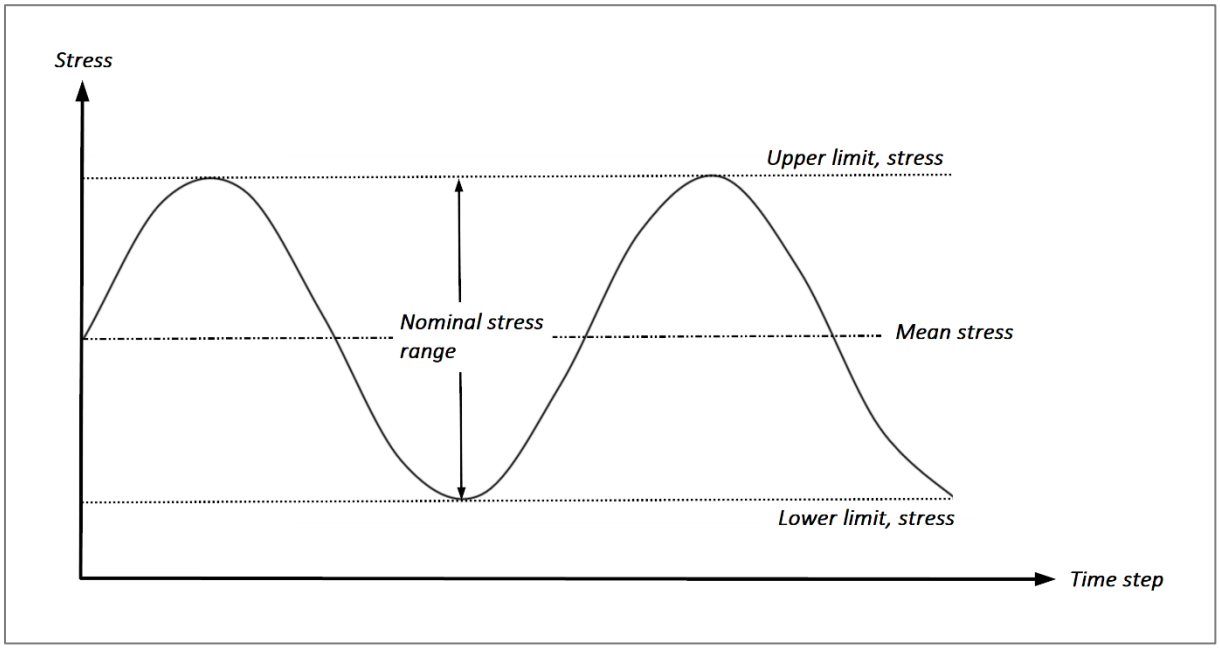

Figure 22 Nominal stress range in Aquaview

3.8 Result > Von Mises stresses

Von Mises stress is commonly considered when deciding if a ductile material is within the yield limit. The von Mises stress is expressed as:

$$\sigma_v = \sqrt{\sigma_x^2 + 3\left(\tau_y^2 + \tau_z^2 + \tau_t^2\right)}$$

where

- σ_v is the von Mises stress,
- $\sigma_x = \sigma_{xN}^2 + \sigma_{xMy}^2 + \sigma_{xMz}^2$ is the normal stress,
- τ is shear stress.

Note that the von Mises stress becomes a non-negative scalar. The von Mises stress found in Result > von Mises stress is calculated based on the input parameters from 'Stress calculation' found in the Edit beam window in AquaEdit, see Figure 23.

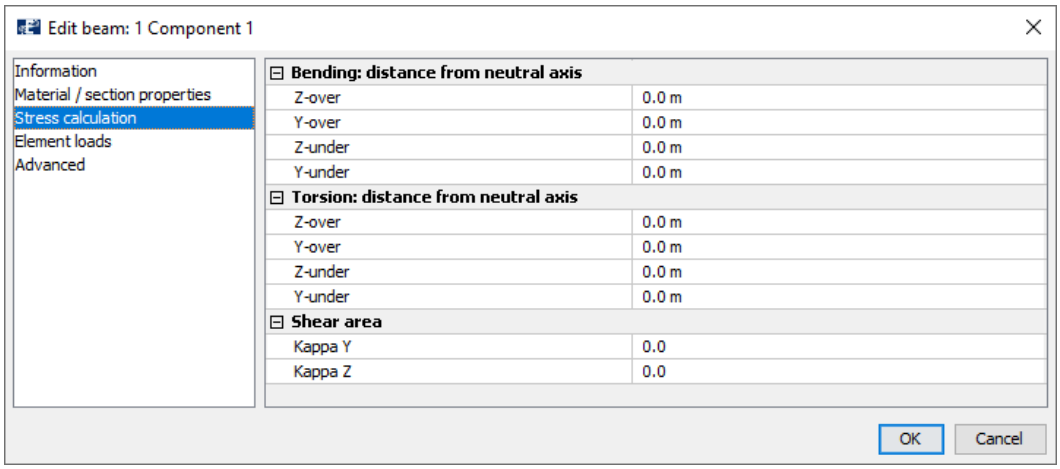

Figure 23 Input parameters from 'Stress calculation' found in the Edit beam window in AquaEdit

Consider a simplified 2D cross section with four points (black dots), as illustrated in Figure 24. AquaSim calculates the von Mises stress based on some simplifying assumptions:

- The stresses are calculated in the four points only and is not related to any location of local shear- or axes of gravity.
- The distanced from the four points to the cross-section's neutral axis y and z is decided from input parameters Z-over, Y-over, Z-under and Y-under.

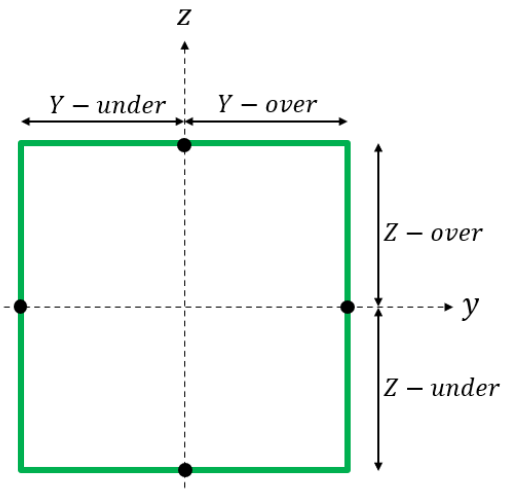

Figure 24 Simplified 2D cross section illustrating the four calculation points

Normal stress due to axial force:

$$\sigma_{xN} = \frac{F_N}{A}$$

where

- F_N is the force acting normal to the cross section [N] (equivalent to axial force),
- A is the cross-section area [m2].

Normal stress due to bending moment about y-axis:

$$\sigma_{xMy} = \frac{M_y \cdot (Z - over)}{I_y}$$

where

- M_y is bending moment about y-axis [Nm],
- $Z - over$ is distance to neutral axis (as defined in Figure 24) [m],
- I_y is moment of area about local y-axis [m4].

Normal stress due to bending about z-axis:

$$\sigma_{xMz} = \frac{M_z \cdot (Y - over)}{I_z}$$

where

- M_z is bending moment about z-axis [Nm],
- $Y - over$ is distance to neutral axis (as defined in Figure 24) [m],
- I_z is moment of area about local z-axis [m4].

Shear stress due to shear force in y-direction:

$$\tau_y = \frac{V_y}{A} \cdot \kappa_y$$

where

- V_y is shear force in y-direction [N],
- κ_y is the shear area factor.

Shear stress due to shear force in z-direction:

$$\tau_z = \frac{V_z}{A} \cdot \kappa_z$$

where

- V_z is shear force in z-direction [N],
- κ_z is the shear area factor.

Shear stress due to torsion:

$$\tau_t = \frac{T \cdot (Z - over)}{I_t}$$

where

- T is torsion moment [Nm],
- I_t is torsional moment of area [m4].

3.9 Result > Contact

If the AquaSim model contain Component contact (that is: contact between component groups), options for Contact will be available in AquaView. The result options include reporting of distances and forces relevant when contact is established, see Figure 25.

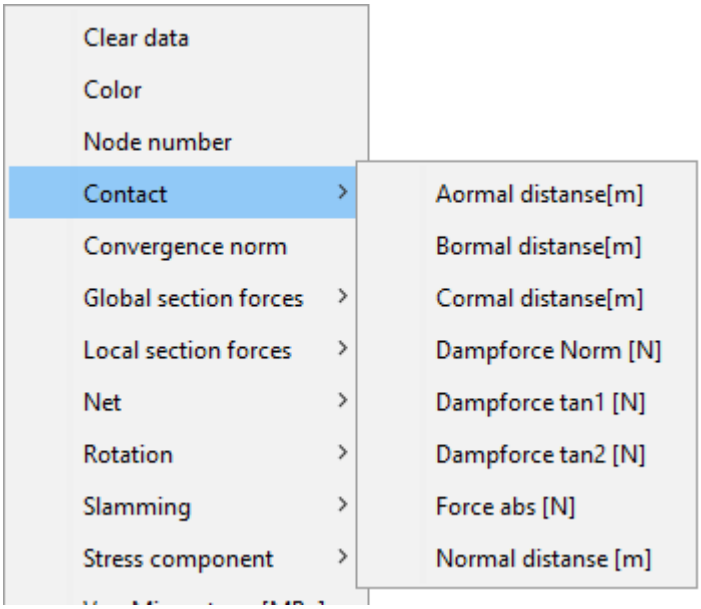

Figure 25 Result options for components with contact

When Component contact is added in AquaSim a spring is established between First part and Second part. This is referred to as “contact-spring”.

Table 10 Result > Contact

Option	Definition
Aormal distanse[m]	Distance between a node on First part to node A on Second part. This description assumes contact has been established between a beam/truss as First part and membrane as the Second part, see Figure 26.
Bormal distanse[m]	Distance between a node on First part to node B on Second part. This description assumes contact has been established between a beam/truss as First part and membrane as Second part, see Figure 26.
Cormal distanse[m]	Distance between a node on First part to node C on Second part. This description assumes contact has been established between a beam/truss as First part and membrane as Second part, see Figure 26.
Dampforce Norm [N]	Damping force in the contact-spring, in normal direction. See Figure 27.
Dampforce tan1 [N]	Damping force in the contact-spring, in tangential direction no. 1, see Figure 27.
Dampforce tan2 [N]	Damping force in the contact-spring, in tangential direction no. 2, see Figure 27.
Force abs [N]	Absolute value of the force in the contact-spring
Normal distanse [m]	Shortest distance between the node in First part and the surface of Second part.

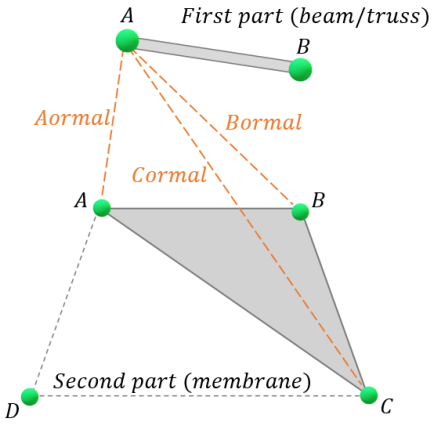

Figure 26 Contact > Aormal, Bormal and Cormal

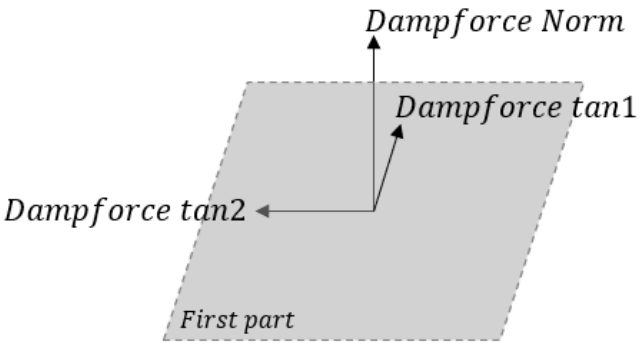

Figure 27 Contact > Damping force

3.10 Result > Turbine environment and Turbine results

If the AquaSim model contain a turbine, options for Turbine environment and Turbine results will be available.

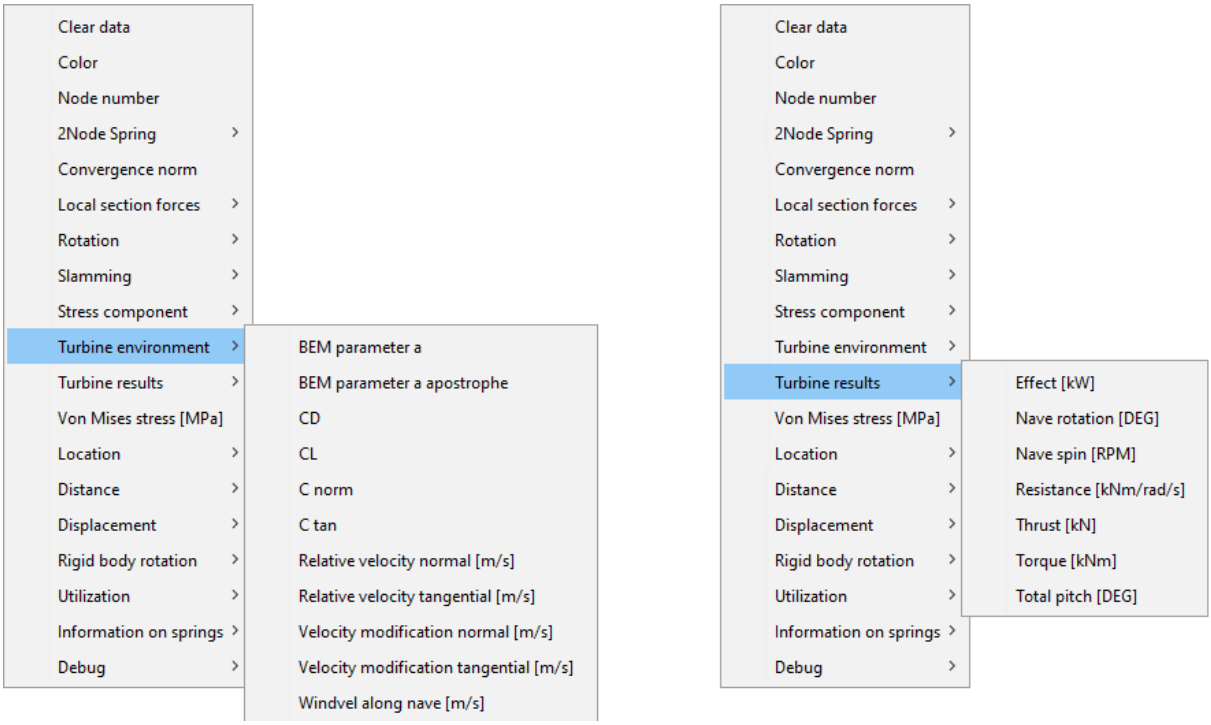

Figure 28 Results > Turbine environment and Turbine results

Turbine environment contain key results relevant for the turbine environment, more information is provided in Table 11. Turbine results includes specific turbine results, more information is provided in

Table 11 Options in Result > Turbine environment

Option	Definition
BEM parameter a	Factor for modification (reduction) of the inflow velocity, in axial direction according to the BEM (Blade Element Momentum) theory (Aquastructures, 2024b).
BEM parameter a apostrophe	Factor for modification (increase) of velocity in tangential direction according to the BEM theory (Aquastructures, 2024b).
CD	Drag coefficient, from the drag/ lift -table in AquaEdit.
CL	The lift coefficient, from the drag/ lift -table in AquaEdit.
C norm	Coefficient C_n according to (Aquastructures, 2024b).
C tan	Coefficient C_t according to (Aquastructures, 2024b).
Relative velocity normal [m/s]	The relative velocity (between inflow wind and blade velocity) in direction normal to the blade.
Relative velocity tangential [m/s]	The relative velocity (between inflow wind and blade velocity) in direction tangential to the blade.
Velocity modification normal [m/s]	How much the normal component of the velocity is reduced according to BEM. This is the inflow wind velocity multiplied with *BEM parameter a*.
Velocity modification tangential [m/s]	How much the tangential component of the velocity is increased according to BEM.
Windvel along nave [m/s]	Wind velocity parallel to the nave. Any deformations of the nave are not included.

Table 12 Options in Result > Turbine results

Option	Definition
Effect [kW]	The turbine effect. Unit is kilo watt.
Nave rotation [DEG]	Rotation of the nave.
Nave spin [RPM]	Spin velocity of the nave. Unit is revolutions per minute. Hinge: having selected the checkbox "Local" in AquaEdit, velocity in the Hinge-node is with respect to element's local coordinate system. Otherwise, it is with respect to the global coordinate system.
Resistance [kNm/rad/s]	Torque resistance in the generator.
Thrust [kN]	Axial force in the nave.
Torque [kNm]	Torque force in the generator.
Total pitch [DEG]	This is the total pitch of the turbine blades. Blade pitch minus twist. Potential deformations of blades due to external loads are also included in the total pitch.

3.11 2 Node spring

This result option is available if your model contains the component type Node2Node (Spring type: Node2Node or Type14). Available result options are described in the table below.

Table 13 Options in Result > 2 Node spring

Option	Definition
Effect X, Y and Z [W]	Available if the spring type is Type14. Translatory effect (watt) in X-, Y- and Z-direction for translation. More information is found in (Aquastructures, 2024a).
Effect RX, RY and RZ [W]	Available of the spring type is Type14. Rotational effect (watt) about X-, Y, and Z-axis. More information is found in (Aquastructures, 2024a).
Force X, Y and Z [N]	Spring force in X-, Y and Z-direction respectively. Unit is Newton [N]. Available for both spring type Node2Node and Type14.
Moment X, Y and Z [Nm]	Spring moment in X-, Y- and Z-direction respectively. Unit is Newton-meters [Nm]. Available for both spring type Node2Node and Type14.

3.12 Element Result window

AquaView allows for plotting results for nodes in elements. Right click element to open the Element Result-window, see example in Figure 29. Every element is connected with two nodes: node A and node B. Results are presented for either node A or B.

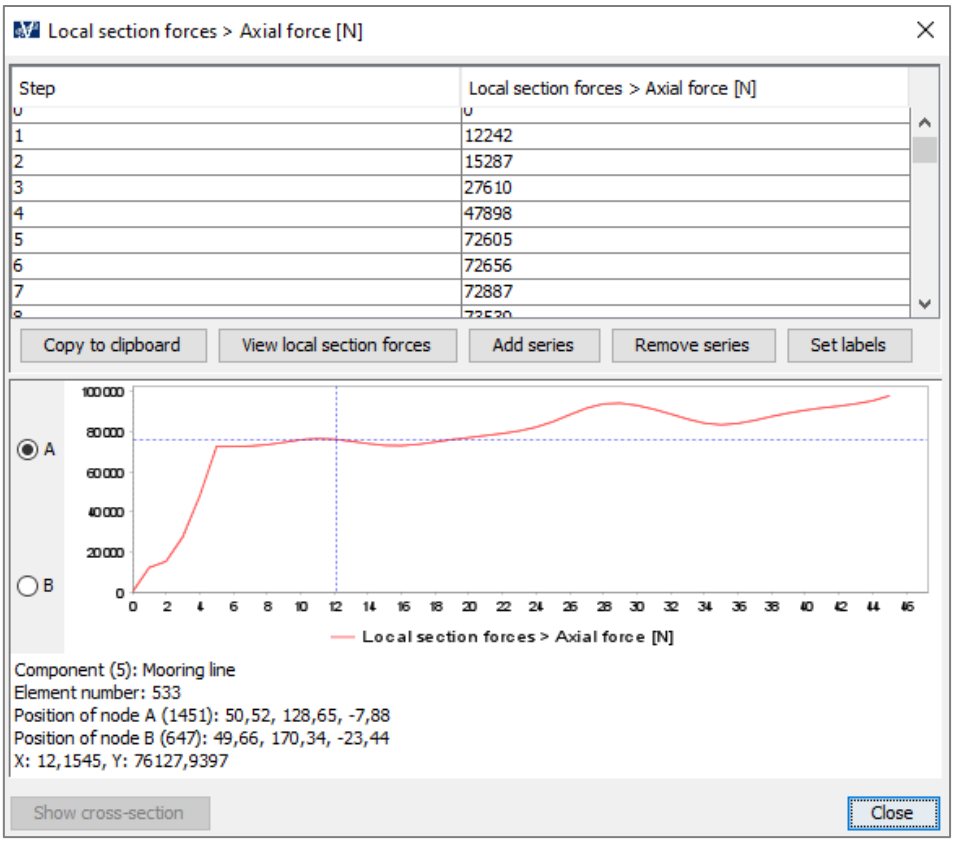

Figure 29 Element Result-window

The window is divided in two main parts: the tabular area and the graph-area. The tabular area presents the results on tabular form for each time step in the analysis. The graph-area provides the same results as a function of time steps. The user may view results for either node A or node B by clicking on **A** og **B**.

Other statistics for the selected element are found below the graph, such as name of the component group, element/ node ID and position.

Table 14 Options in the Element Result-window

Option	Definition
Copy to clipboard	Enables to transfer the data to e.g., MS Excel. More information in chapter 3.12.1.
View local section forces	Allows for view all the result types from the Local section forces option in one table/ graph.
Add series	Display other result types form the Result Menu for the selected element. Such as Convergence norm, Global section force or Von Mises.
Remove series	Option for remove one, or more loaded result types.
Set labels	Having loaded several result types, this option allows for secondary axis in the graph.
Show cross-section	Plots cross sectional results for beam elements. By selecting this option, a new window appears. Provides information about general properties, stiffness, and a visual plot of the modelled cross section. More information in chapter 3.12.2.

3.12.1 Copy all to Clipboard

By selecting **Copy all to Clipboard**, a new window is opened, see Figure 30.

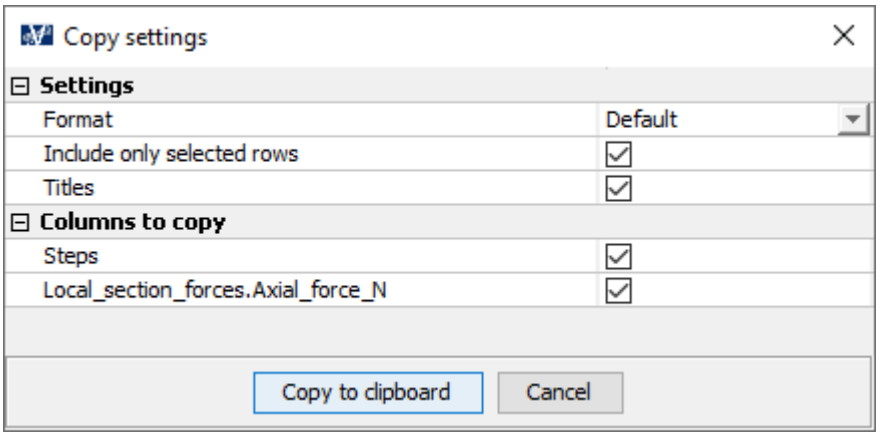

Figure 30 Copy all to Clipboard

Table 15 Options in the Copy all to Clipboard-window

Option	Definition
Format	Selection of number-format. Choose between: - Default: the default format, 0.#### (decimal with four digits) - Locale specific: java-format, %e. Corresponds to the e-notation for the default language installed on your computer.
Include only selected rows	Option to copy only selected rows from the table in Element Result-window. E.g. if only 10 rows are selected for the table, this option only copies this selection.
Titles	Option to include or exclude to result titles to clipboard.
Columns to copy	Selection to include, or omit, columns from the table in Element Result-window. Figure 30 exemplifies this with the option to include both analysis steps and displacement in z-direction.

3.12.2 Show cross-section

By selecting **Show cross-section**, a new window is opened, see Figure 31. Detailed information and results of the cross section is presented. This window is useful when assessing asymmetrical and complex sections. Options and information are described in Table 16. The resulting stresses in this window may be different than those calculated from Ch. 3.8. This is because results in Show cross-section is solely based on the cross-sectional parameters calculated by AquaCross.

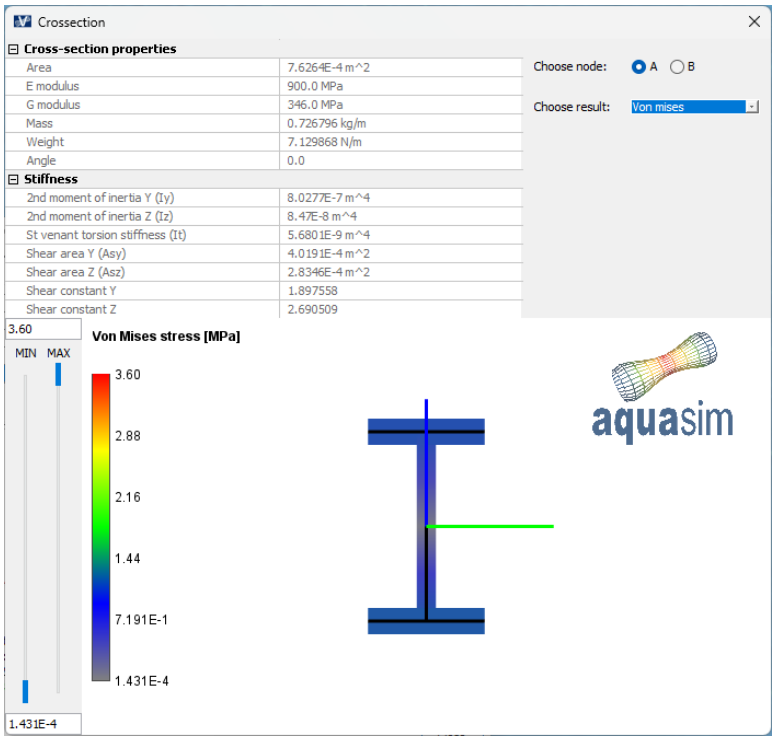

Figure 31 Show cross-section

Table 16 Options in Show cross-section

Option	Definition
Cross-section properties and Stiffness	Shows some of the parameters of the cross-section.
Choose node	One can choose between viewing results from node A or B.
Choose result	Press dropdown menu to view different section results. - σ_x due to N - σ_x due to M_y (bending moment about local y-axis) - σ_x due to M_z (bending moment about local z-axis) - τ due to V_y (shear force in y-direction) - τ due to V_z (shear force in z-direction) - τ due to M_x (torsional moment) - Resulting σ_x - Resulting τ - Von Mises stress $\sigma_{eff} = \sqrt{\sigma_x^2 + 3\tau^2}$, where σ_x and τ are the resulting stresses

3.12.3 Open file (PFAT.avz, PFAT.avs and max_out.avz)

In the Element Result window, for the files mentioned in the header, there is an option to select **Open max file** and **Open file**. This option opens the file the maximum result was found in. See buttons in the lower part of Figure 32 (*Open max file (run_01PFAT.avz)* and *Open file (run_01.avz)*).

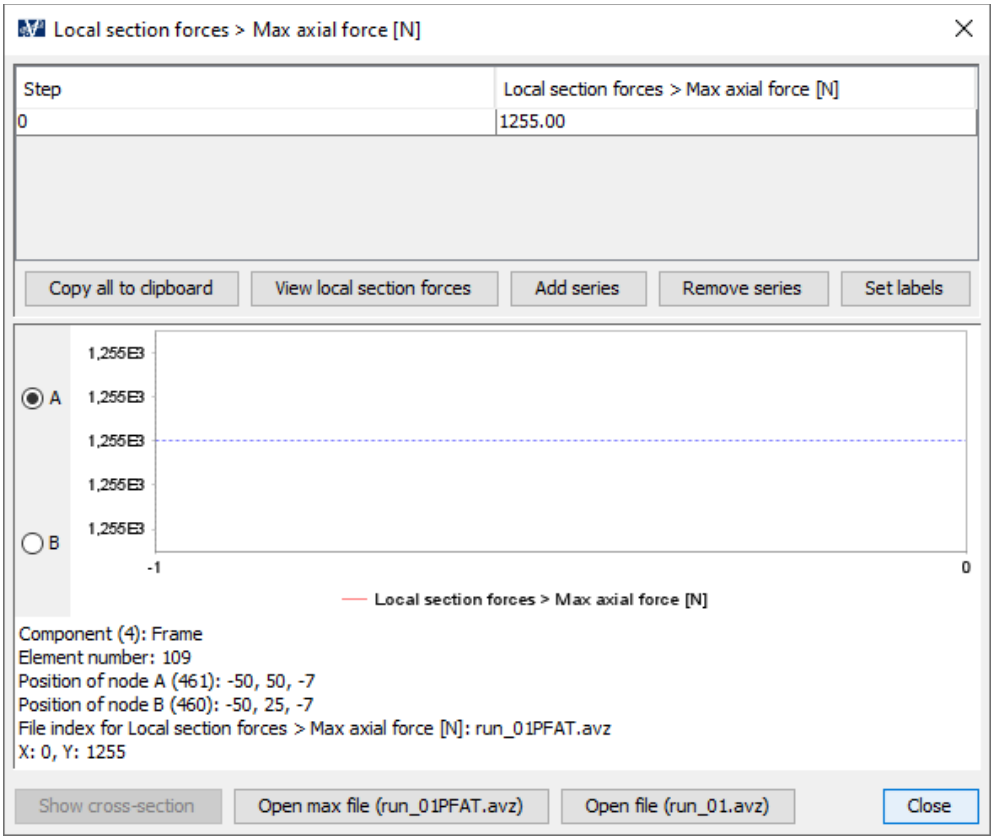

Figure 32 Open file in the Element Result window for PFAT.avz, PFAT.avs and max_out.avz files

The max_out.avz-files is based on the PFAT.avz-files, which in turn is based on the .avz-files. The .avz-files is based on the .avs-files. If the file is not found, the **Open file**-selection is no longer active.

4 Show Menu

The Show Menu provides options for visual effects. The options are available for **Shapes**, **Stresses**, **Components** and **Membranes**. Detailed description is provided in Table 17.

Table 17 Options in the Show Menu

Option	Definition
Shapes	Shapes are defined as objects (.obj-files), visualizing structures at nodes. AquaView have preset objects for point loads, springs, boundary, and environment, see Figure 33. User defined objects is also possible. To view this in AquaView, the user must locate the .obj-file in the Shapes folder and assign the object to a dedicated node in AquaEdit prior to export of analysis. Having included terrain in AquaEdit, the option for toggle this on, is found in this menu.
Stresses	Provides the option of removing results on specified components.
Components	Provides the option of plotting results on only specified component groups.
Membranes	Enables the user to change view of membrane elements.

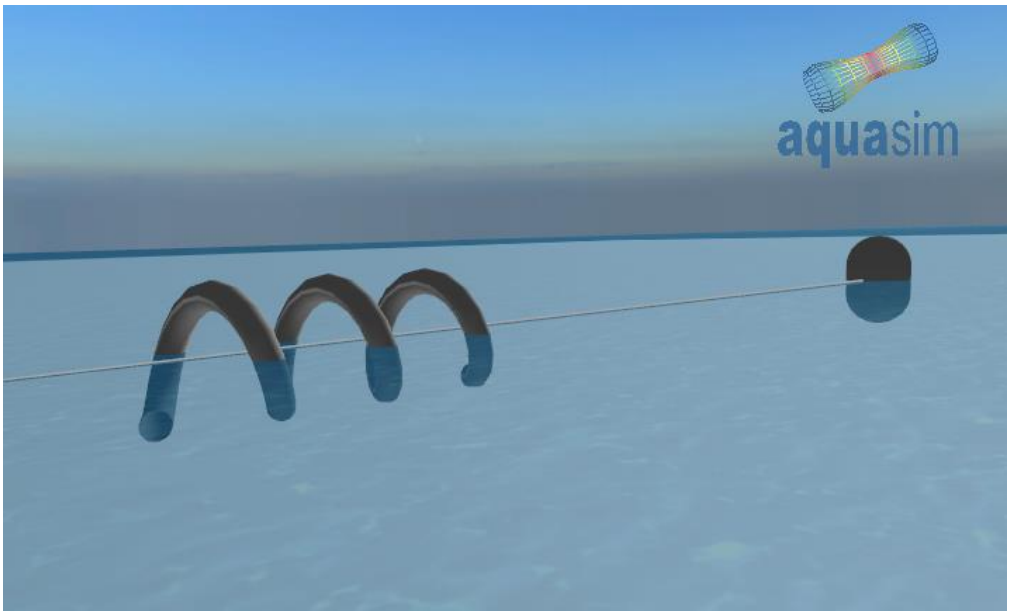

Figure 33 Spring shape (left), point load shape (right) and environment Sunset in AquaView

5 Front/ Left/ Right/ Top/ Bottom and ISO

These menu options are shortcuts for different view angles of the model. Selecting **Front** present the model in XZ-plane, **Left** and **Right** present the model in YZ-plane, **Top** and **Bottom** in XY-plane, and the last, **ISO**, shows the model in isometric view.

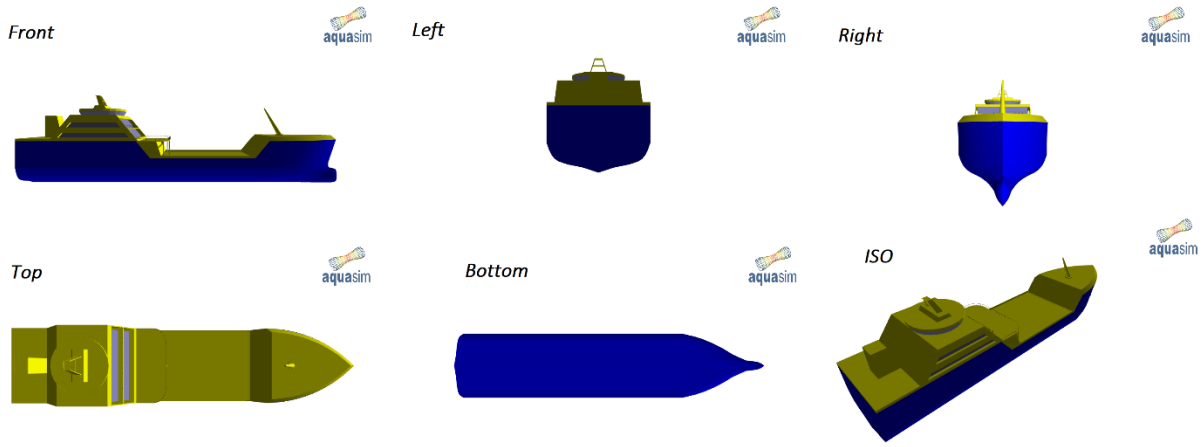

Figure 34 View angles in AquaView

6 Zoom and Zoom to result

These options enable the user zooming to different parts of the model based on either coordinates or result values.

6.1 Zoom

The different options are described in Table 18.

Table 18 Options in the Zoom Menu

Option	Definition
Zoom to smallest X	Centers the view to the node with the lowest X-coordinate.
Zoom to largest X	Centers the view to the node with the largest X-coordinate.
Zoom to smallest Y	Centers the view to the node with the lowest Y-coordinate.
Zoom to largest Y	Centers the view to the node with the largest Y-coordinate.
Zoom to smallest Z	Centers the view to the node with the lowest Z-coordinate.
Zoom to largest Z	Centers the view to the node with the largest Z-coordinate.
Zoom to middle of model	Centers the view to the node in the center of the model.
Zoom to fit	Centers the view so the whole model fits the window.

6.2 Zoom to result

The different options are described in Table 19.

Table 19 Options in the Zoom to result Menu

Option	Definition
Zoom to max in timestep	Enables zoom to the maximum value of a result type for the present timestep.
Zoom to min in timestep	Enables zoom to the minimum value of a result type for the present timestep.
Max component	Having chosen any result type from the Result Menu, this option enables zoom to a maximum value for a specified component group.
Min component	Having chosen any result type from the Result Menu, this option enables zoom to a minimum value for a specified component group.

7 Tools Menu

The Tools Menu provides access to different settings in AquaView. The options are described in Table 20.

Table 20 Options in the Tools Menu

Option	Definition
Set cut plane	Controls how near, or far, the camera is to an object before it vanishes. The user may set custom values, default values for Near plane is 1, and 4000 for Far plane. The values are in meters.
Set background colour	Controls the background color in AquaView. By default, this is white.
Set mouse sensitivity	Properties for mouse sensitivity. Controls acceleration of the mouse cursor. Useful when handling both small and large models.
Set scaling for the axes	Enables scaling of axes.
Views	Same viewing options as on the Top Menu Bar; Front, Right, Top, Bottom and ISO. In addition, **Set camera** provides the user to customize a view by specified coordinates. **Top with rotation** provides a top view with a customized rotation about the Z-axis in the Global Coordinate System.
Water surface	Option for setting size of the water surface (in meters) and extract wave height for each time step in the analysis. Note: the wave amplitude is extracted from origin in the Global Coordinate System.
Calculate volume	Provides a table with information about the volume enclosed by membrane elements. Note: top and bottom of the enclosed volume must be defined in AquaEdit. For more information, reference is made to the AquaEdit User Manual.
Calculate area	Provides information about the surface area of membrane elements.
Load secondary results	Loads additional dataset from an .AVZ-file. Useful for e.g. plotting interference between components from different models. More information in Ch. 7.1.
Show loaded secondary results	Shows a list of loaded datasets. More information in Ch. 7.2.
Show named nodes	Provides a list of nodes with customized identification. Note: only available if node names are specified in AquaEdit prior to export of analysis. Useful in cases when selected nodes are of special interest.
Show named elements	Provides a list of elements with customized identification. Note: only available if specified in AquaEdit prior to export of analysis.
Settings	General settings for AquaView. More information in chapter 7.4.
Save camera view	Enables customized view angle. Use the mouse cursor to a desired view, press Save camera view, and enter a suitable name. Having restarted AquaView the customized camera view is visible in the Top Menu bar.
Extract node positions	Writes all node numbers and associated coordinates to a .txt-file. Upon selecting this option, a window for choosing appropriate directory for saving the .txt-file appears. Type a suitable name for the file and press Save.
Find	Search option for components, elements, and nodes. Type the ID-number in the search field and press enter. AquaView then centers the view to the searched component/ element/ node. The detected component/ element/ node is highlighted with a blue sphere.
Set (for distance calculation)	Settings for calculation of distance between components, elements, and nodes.

Set element for result dialog	Opens the Element Result-window for a specified element. Useful as a search tool, or in the case when user know the element identification number, but not the location in the model.
Max values J	Display the maximum values for the present result type. **J** is the keyboard shortcut for the same option.
Min values K	Display the minimum values for the present result type. **K** is the keyboard shortcut for the same option.
Max/min values I	Display both the maximum and minimum values for the present result type. **I** is the keyboard shortcut for the same option.
Show component lengths	Displays the total length of each component group in the model. Additional option to select specific time step.
On/Off Axis	Toggles on or off the axis for Global Coordinate System.
Load animation	Allows loading nodes from other datasets.
Show model information	Information about applied environmental condition, such as wave condition, wind and current.
Show wave height	Display wave amplitude as a function of analysis step. More information in Ch. 7.3.

7.1 Load secondary results

This option provides the possibility to load additional result files to AquaView. Multiple files can be loaded simultaneously. Keep in mind that large result files require memory on your computer.

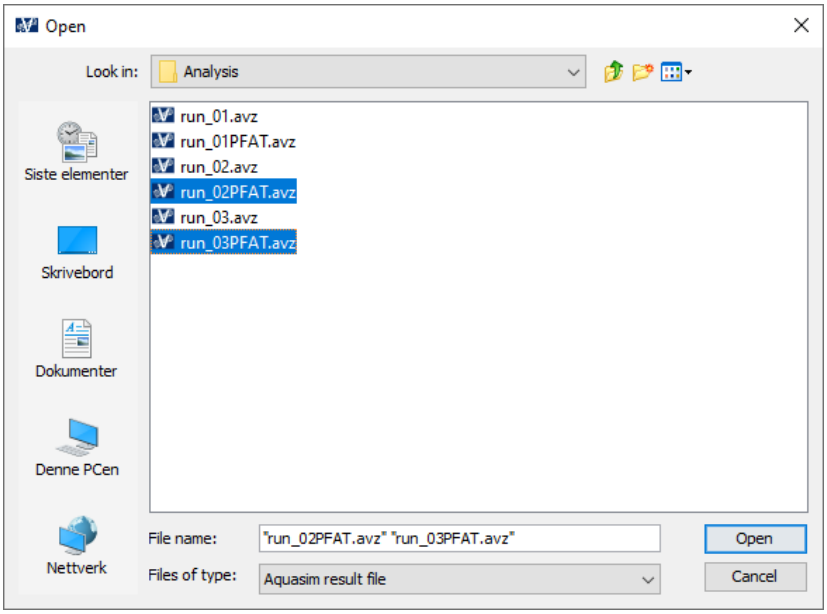

Figure 35 Load secondary results

7.2 Show loaded secondary results

The loaded secondary results can be viewed from the Show loaded secondary results-option. Figure 36 shows a list where two additional result sets have been loaded from the **Load secondary results**-option. The original result set will always be listed first. The order of the result sets can be changed by **left click** + **drag**.

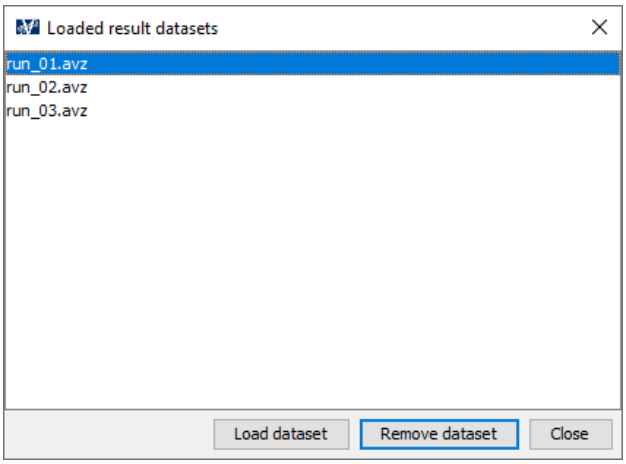

Figure 36 Show loaded secondary results

Additional result sets can be further added or removed from the **Load dataset**- and **Remove dataset**-options.

7.3 Show wave height

The Show wave height-option plots the wave height as a function of analysis step, see Figure 37. **Set point** allows for defining exact coordinates for extracting the wave height.

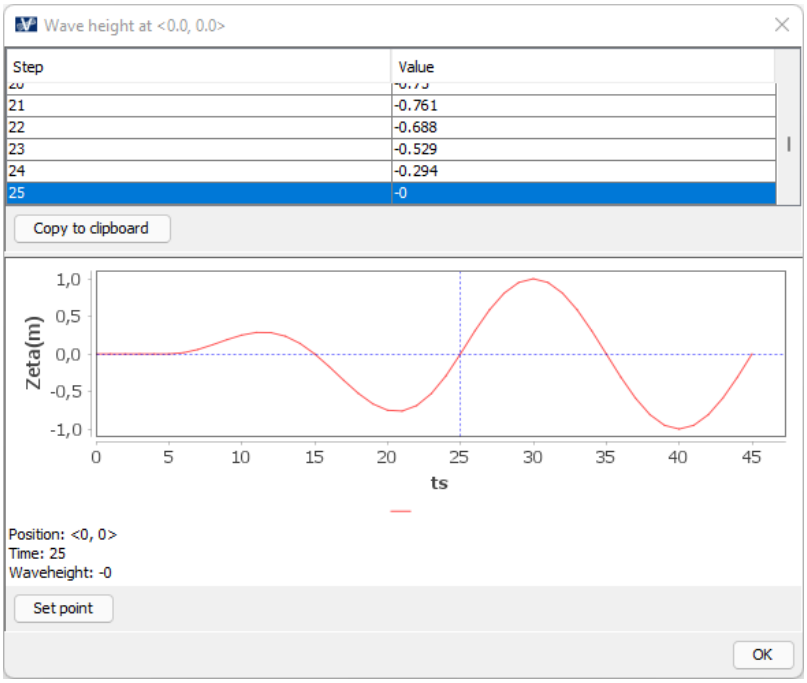

Figure 37 Show wave height

7.4 Settings

The settings window is shown in Figure 38. AquaView must be restarted to activate any changes in Settings.

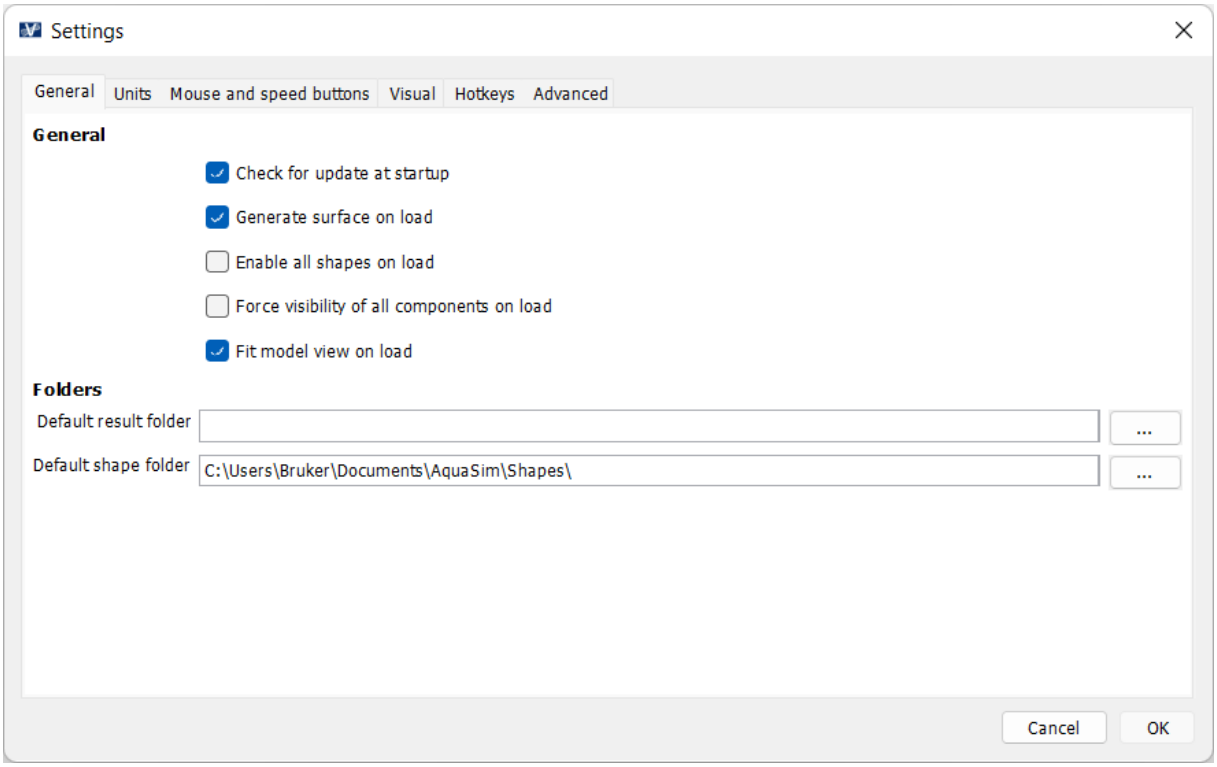

Figure 38 Settings in AquaView

7.4.1 General

Table 21 Options General tab

Option	Definition
Check for update at startup	Toggle for automatic search for a new release of AquaSim.
Generate surface on load	Generates a water surface at startup of AquaView.
Enable all shapes on load	Enables default activation of shapes.
Force visibility of all components on load	Visibility of components can be defined in AquaEdit. Having turn off visibility of components in AquaEdit, this option overrides this setting and display the components anyway.
Fit model view on load	Zooms in or out so you can see the entire model upon start of AquaView
Default result folder	Preset of folder for result files.
Default shape folder	Preset of folder for shape-files, .obj. This directory should be set to the folder where .obj-files typically are stored.

7.4.2 Units

Table 22 Options for Units tab

Option	Definition
Force/ Moment/ Pressure	Preset of units to be displayed in AquaView.
Number format	Number of decimals that is viewed for non-scientific number format.

7.4.3 Mouse and speed buttons

Table 23 Options for Mouse and speed buttons tab

Option	Definition
Mouse settings	Properties for mouse sensitivity. Controls acceleration of the mouse cursor.
Speed buttons	Preset of shortcuts on the Top Menu Bar. For each entry here, a shortcut will be displayed in the Top Menu Bar. The entries must be given on the format as defined in the .AVS-files. Separate entries must be defined for maximum (PFAT)-files and files containing several time steps. Examples of input is provided in Table 24. Add option: Add option allow the user to add additional stress-values for the same speed button. Which stress-value is applied is done through a prioritizing. If the first stress-value is available from the loaded result-file, this value will be shown. If it is not found, then the second stress-value is chosen. This feature is specifically developed for cases with use of PFAT-files. This is so that one may apply the same speed button for PFAT as for other result files.

Table 24 Examples shortcut parameters

Result type	Input for max-files	Input for files containing several time steps
Von Mises stress	Von_Mises_stress_max_MPa	Von_Mises_stress_MPa
Local section force, axial	Local_section_forces.Max_axial_Force_N	Local_section_forces.Axial_Force_N
Global section force, Z	Global_section_Forces.Max_force_Z_N	Global_section_forces.Force_Z_N

7.4.4 Visual

Table 25 Options for Visual tab

Option	Definition
Multisampling	Enable to smooth out edges around lines to they do not irregular.
Projection mode	Changes the projection of the viewport camera. Perspective: the model is distorted to create an impression of perspective view. Orthographic: the model is not distorted; elements stay with the same size regardless of the distance between them.
Fonts	Customization of font type, -size and -style.

Stress color palette	Customization of color rendering for the options in Result Menu. Options for Add/ Remove individual rows of colors or reset to Default. In addition, import or export color maps.
Terrain color palette	Customization of color rendering of terrain. Options for Add/ Remove individual rows of colors or reset to Default. In addition, import or export color maps.
Settings for find component/ element/ node	Customization of how search result is presented in the model view.
Settings for set component/ element/ node (for distance)	Customization of how result is presented in the model view for the Distance option.
Settings for clicked element	Customization of how a selected element is highlighted in the model view.
Cross section options	Toggle on or off the visibility of modelled cross sections from AquaEdit.

7.4.5 Hotkeys

Table 26 Options for Hotkeys tab in Settings

Option	Definition
Hotkey	Presentation of default hotkeys in AquaView. Options for editing and adding custom hotkeys.

7.4.6 Advanced

The Advanced tab includes most of the settings found in the beforementioned tabs in code-like style. Useful for obtaining an overview of the settings in AquaView. Options for adding and remove both default settings and customized. Some examples are shown in Table 27 and Figure 39. Adding new settings, contact Aquastructures AS for more information.

Table 27 Examples of coding settings in the Advanced tab

Key	Value	Description
visual.playback.duration	10.0	Number of seconds a playback of analysis takes from start to finish.
visual_near_plane	1.0	How close the camera is to an object before it vanishes.
Visual_far_plane	4000.0	How far out the camera is to an object before it vanishes.
numberformat.boundryhigh	100000.0	When a result type is displayed, it is displayed either on decimal form or as scientific. If the absolute value of a result is outside the range defined for boundary.low and boundary.high the result value is displayed as scientific. Otherwise on decimal form.
numberformat.boundrylow	0.9999	
numberformat.simple	##0.00	Decimal numbering
numberformat.scientific	0.###E0	Scientific numbering
numberformat.decimalseparator	.	Decimal separator
elementresults.dialog.new	true	Enable a ctrl + right click menu for elements. See Figure 40.

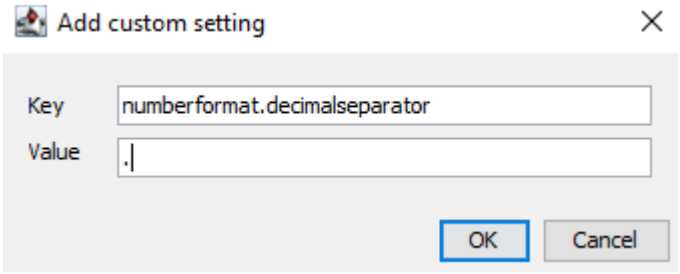

Figure 39 Add custom setting in Advanced tab

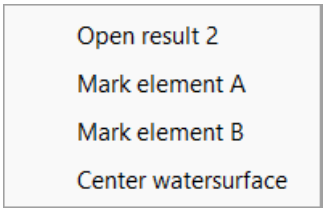

Figure 40 Ctrl + right click element options enabled Advanced tab elementresults.dialog.new

8 Help Menu

Access the AquaView User Manual (the one you are reading now), option for generate an error report in the software.

9 References

Aquastructures. (2024a). *The AquaSim Package Theory User Manual.* Tech. rep. TR-30000-2049-1.

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

Faltinsen, O. (1990). *Sea Loads on Ships and Offshore Structures.* Cambridge University Press. ISBN 0-521-37285-2.

Wikipedia. (2020, 05 20). *Morison equation*. Hentet fra Wikipedia: https://en.wikipedia.org/wiki/Morison_equation