AquaEdit
» **AquaCross**
AquaView
AquaTool
Other

User manual

Document ID: TR-20000-583-2
Date: 01\03\2024

AquaSim version: 2.19

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 preprocessing tool AquaCross. AquaCross is an addon to AquaEdit for drawing and computation of section parameters and stiffness of cross sections. It handles both predefined and arbitrary geometry.

The cross sections are analyzed either as thinwalled- or massive sections. Thinwalled sections are built up by series of straight-line segments, connected by nodes, and assigned a thickness. Massive sections are meshed, using straight-lined triangles. AquaCross is compatible with the component type beam in AquaEdit.

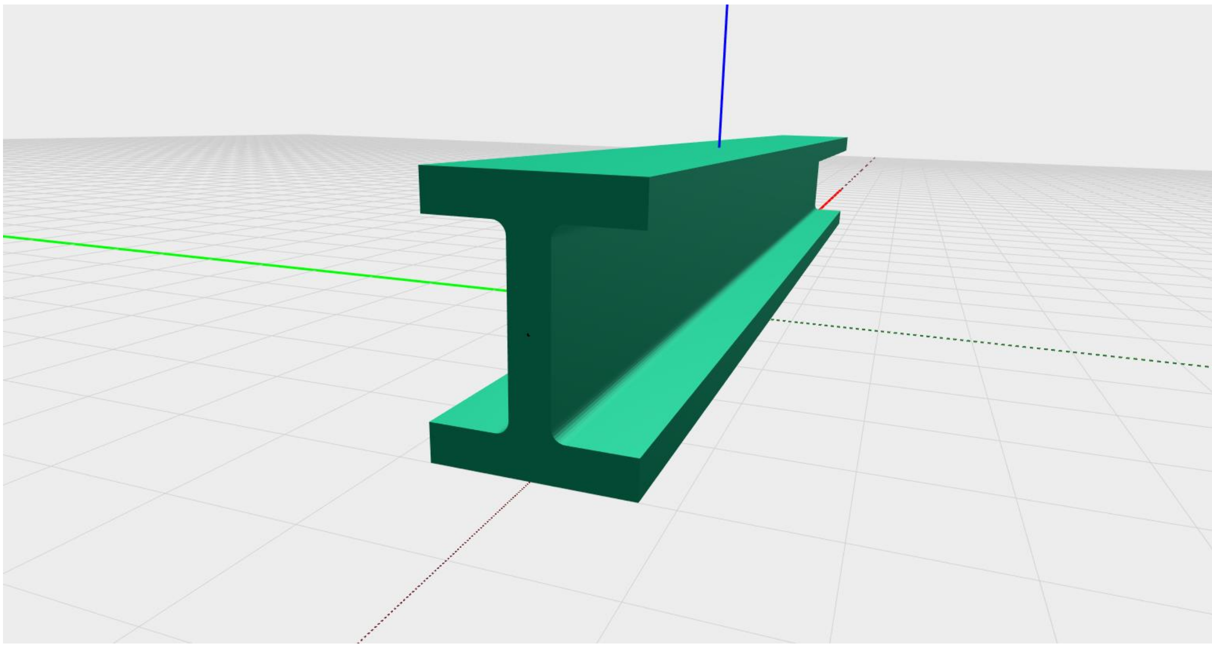

Figure 1 Massive cross section in AquaEdit, using AquaCross

1.1 Thinwalled- and massive section

AquaCross differentiates between thinwalled and massive cross-sections. Upon opening AquaCross, the user must choose between one of these two types.

Thinwalled sections consists of an assemblage of straight-line segments with a defined thickness. The section may consist of several materials, open and/or closed segments. Closed segments are referred to as cells. Figure 4 illustrate a section of two materials (green and blue), with open and closed (cell) segments. For calculation purposes, AquaCross assumes that each line segment has constant properties.

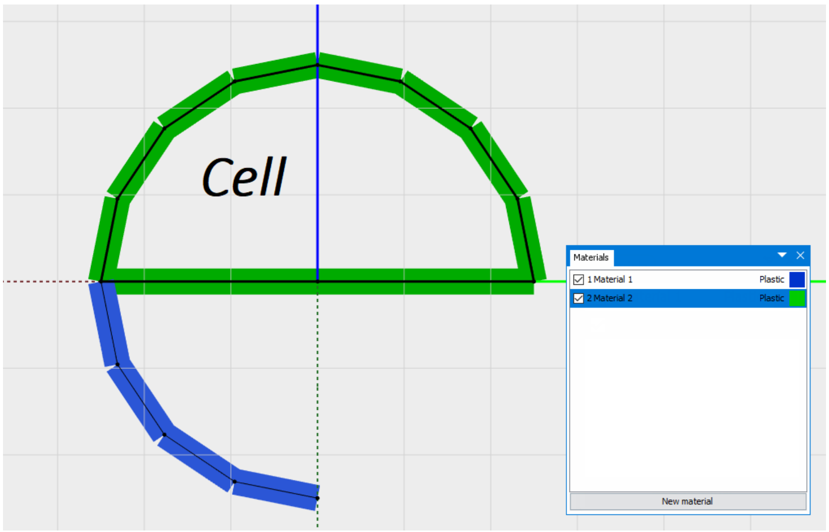

Figure 2 Thinwalled section with open and closed segments (cell)

Note! For cross sections covered by the Wizard (e.i. Data source: Tube, Hollow rectangular, Massive rectangular, I/H-beam) it is advised to apply the Wizard. This is because the Cross sectional properties, distance to neutral axis and shear area is more accurately estimated by this method. The Wizard is based on defined formulas, rather than FEM.

Massive sections are areas closed by elements and nodes; each assigned a particular material. The area is divided into subsections of straight-line triangles, i.e. a mesh. This is exemplified in Figure 3.

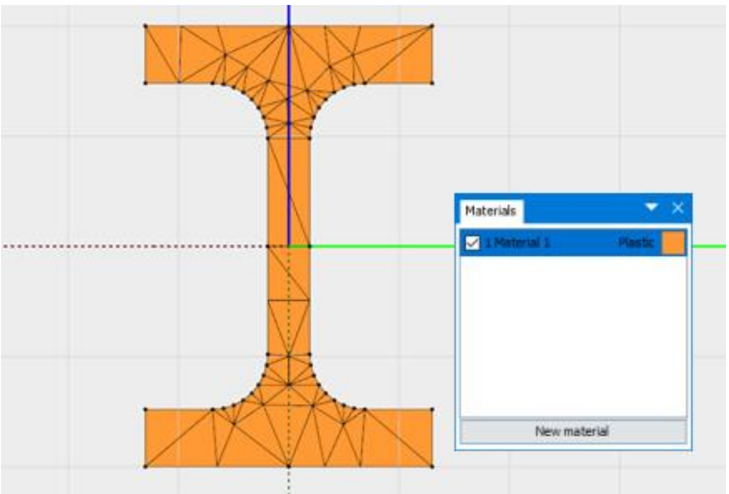

Figure 3 Massive section with triangle mesh

1.2 Coordinate system

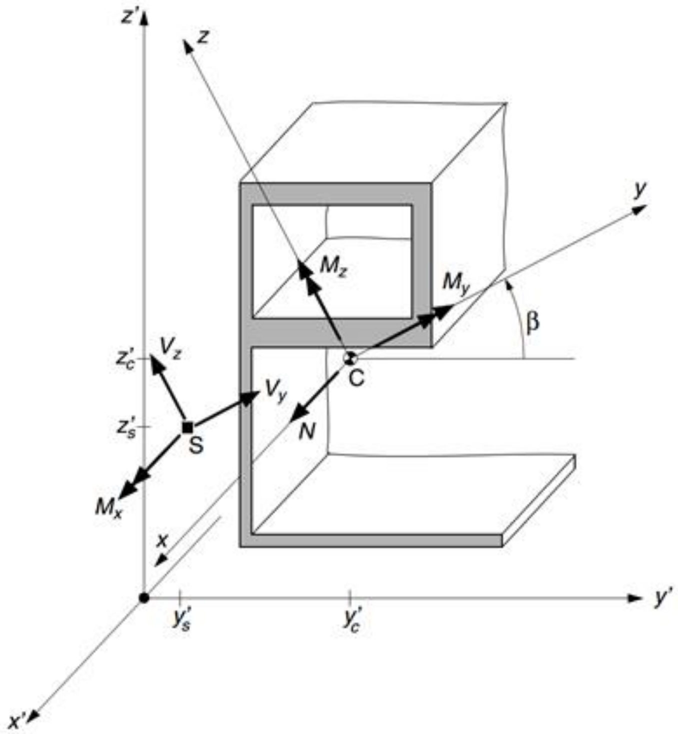

Figure 4 Coordinate system, and positive section forces

Figure 4 shows a typical cross-section. The cross-section is referred to a right-handed “global” coordinate system x’, y’ and z’, where x’ is parallel with the beam axis and y’ and z’ are the cross-section axes. These axes are referred to in coming chapters as reference axes. Figure 6 shows the z’ axis as blue and the y’ axis as green, the same coloring scheme known from AquaEdit (AquaEdit User Manual) main window. **Note:** All length units in AquaCross is in mm.

Point C is the center of the cross-section area (center of gravity, c.o.g.). If the cross-section is composed of more than one material, the location of C is weighted with respect to the stiffness parameter E (Young’s modulus). For instance,

$$z'_c = \frac{\sum A_i E_i z'_{ci}}{AE} \qquad \text{where } AE = \sum A_i E_i$$

A_i and E_i apply to a subsection of a material.

Axes y and z through C are the principal axes of the (stiffness weighted) cross-section. The corresponding x-axis is parallel with x’. **Note:** The y-axis is the major axis and z-axis the minor axis.

Point S is the shear center of the (stiffness weighted) cross-section. AquaCross always determines the location and orientation of the principal axes, and the location of S. **Note:** The forces calculated in the AquaSim solver hits c.o.g., and the moment of area around the principal axes are the ones used in the calculations. If the principal axes and reference axes are not aligned (the cross-section is non-symmetric) then the geometrical point3 in the exported analysis files are automatic rotated from the visual point3 to ensure that the response of the beam is correct. This is rotated back when opened in AquaView (AquaView User Manual), so that the visual representation of the beam in AquaView is the same as the one drawn in AquaEdit.

1.3 Parameters and stiffnesses

For a cross-section, the following "stiffness weighted" properties are found,

- Location of the area center C (center of gravity, c.o.g.)
- Location of shear center S
- The true section area A
- Orientation of the principal axes y and z (defined by the angle β between axis y' and y)
- The weight per unit length of beam
- The mass per unit length of beam

The following parameters are computed with respect to the principal axes:

- Second moment of area (or moment of inertia) I_y
- Second moment of area (or moment of inertia) I_z
- St. Venant torsion constant I_t (often designated by J)
- Shear deformation factor (in y-direction) κ_y
- Shear deformation factor (in z-direction) κ_z

The moments of inertia about reference axes through C, $I_{y'}$ and $I_{z'}$, are also computed. As is the product of inertia $I_{y'z'}$.

The parameters and stiffnesses are calculated and compared against results from formulas in Appendix A – Accuracy.

1.4 Material properties

Any cross-section, thinwalled or massive, may be composed of one or several different materials. Each area of a massive section is assigned a particular material. For thinwalled sections, each line segment is associated with one material. This is similar as for AquaEdit, where line segments are associated with a component. Each material is assumed to be homogenous and isotropic, and linearly elastic. In other words, it behaves according to Hooke's law and is characterized by the following elastic constants:

- E: modulus of elasticity (Young's modulus)
- G: modulus of elasticity in shear, or just shear modulus
- v: Poisson's ratio

These three constants are coupled by the well-known equation

$$G = \frac{E}{2(1+\nu)}$$

AquaCross permits the user to violate this equation since he or she may specify values for all three constants independently for a material. In addition, the mass density is also specified for each material. This enables AquaCross to determine both mass and weight per unit length of beam.

A legal cross-section cannot have any disconnected sections. However, two disconnected sections may connect via a section of VOID material to form a legal section. For such a situation the VOID material will automatically be assigned a very small shear stiffness (= G_{min}/200000, where G_{min} is the smallest shear stiffness of the non-VOID materials in the model)

2 Interface

2.1 Loading AquaCross

AquaCross is accessed through beam elements in AquaEdit. In the Edit beam window, select from the dropdown menu **Data source** > **AquaCross**, see Figure 5.

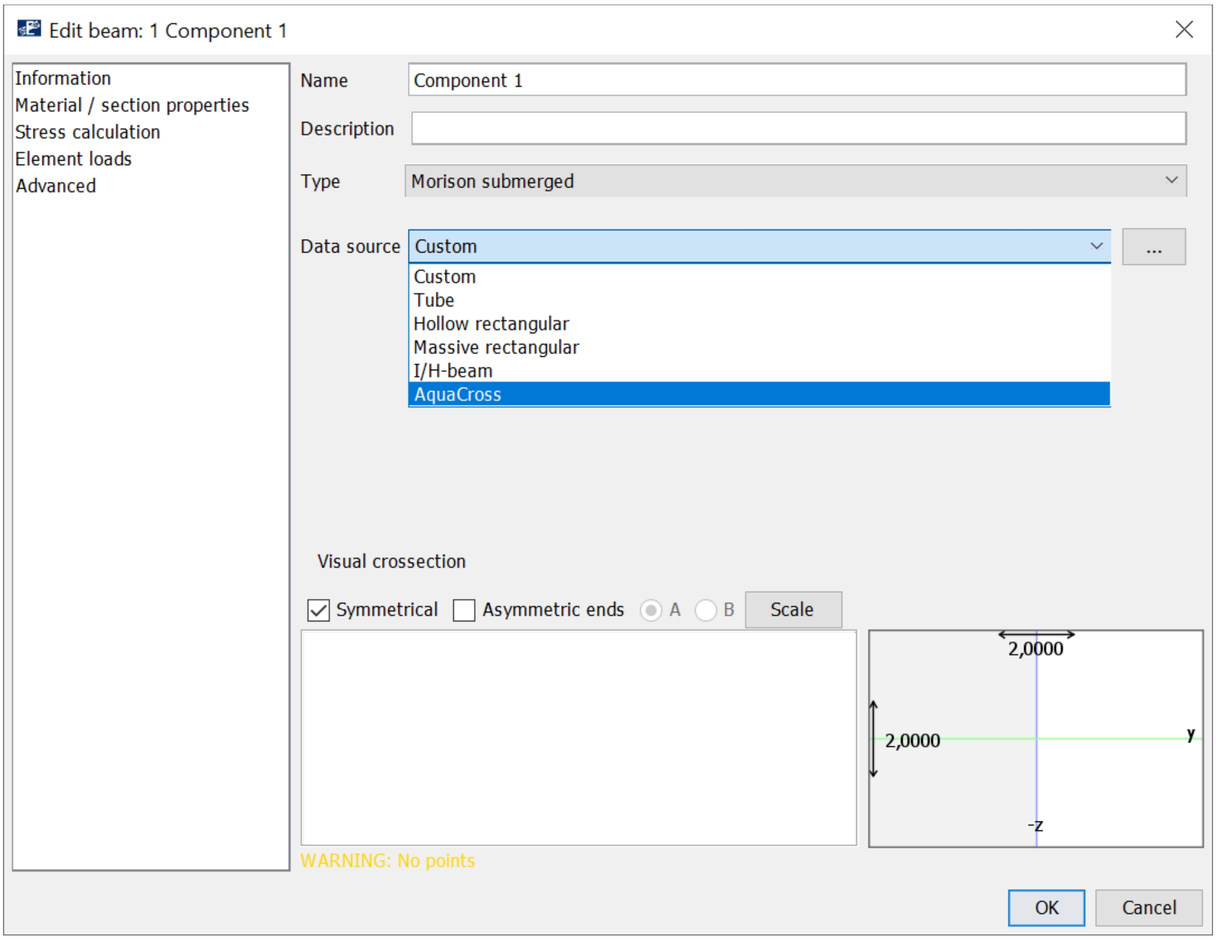

Figure 5 Loading AquaCross

The user must then choose between applying a thinwalled cross-section, or massive.

2.2 Main View

The main view of AquaCross is shown in Figure 6. It is almost identical to the main window in AquaEdit, and most of the functions and interactions is also identical. Only that AquaCross is drawn in 2D, the x'-axis is pointed towards the user. An empty cross-section will begin with one material with type Plastic and thickness of 10 mm.

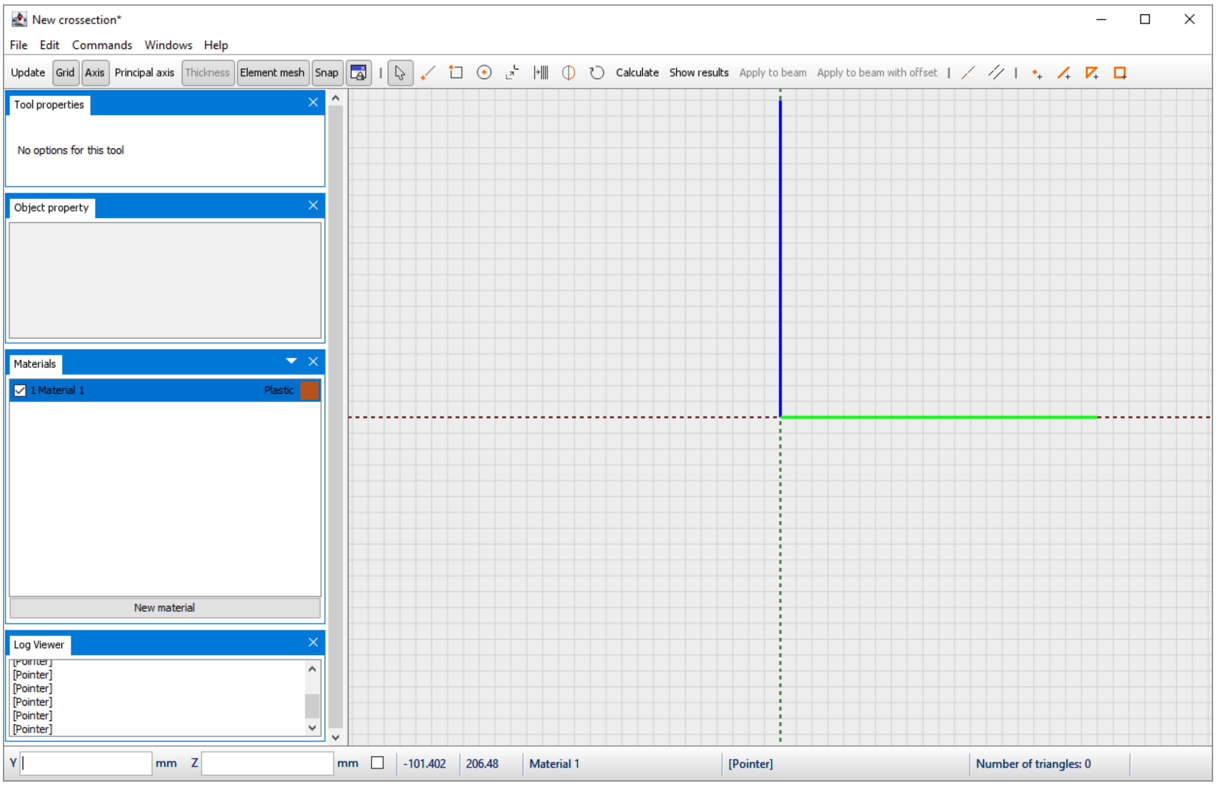

Figure 6 The Main View

2.3 Toolbar

The Toolbar is found on the upper part of the Main view. It resembles the one in AquaEdit, with some differences, see Figure 7.

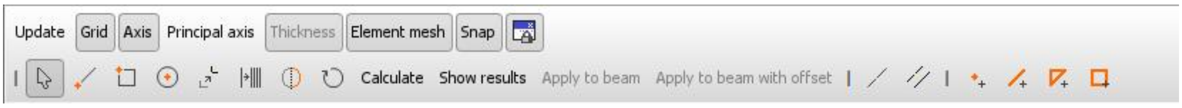

Figure 7 The Toolbar in AquaCross

The options on the toolbar not found in AquaEdit is described in Table 1.

Table 1 Options found on the Toolbar

Option	Description
Principal axis	Toggles on/off the principal axis, only available after calculations are completed successfully.
Thickness	Toggles on/off the visibility of the thickness of the elements.
Calculate	Calculates the currently drawn cross-section and finds parameters and stiffnesses.
Show result	Shows the calculated parameters and stiffnesses after a successful calculation.
Apply to beam	Applies the currently drawn cross-section to the beam.
Apply to beam with offset	Applies the cross-section with the drawn offset to the principal axis. The COG from AquaCross is applied as the center of the geometry, and the whole geometry is moved from (x,y)=(0,0) to where COG is.

2.4 Material

To determine parameter such as mass density and E-modulus, the section has to be assigned a material type. The material window shows a list of all the materials in the current cross-section, see Figure 8. AquaCross allow the user to define one or more materials in one section. For more information about the Materials window, reference is made to the AquaEdit User Manual.

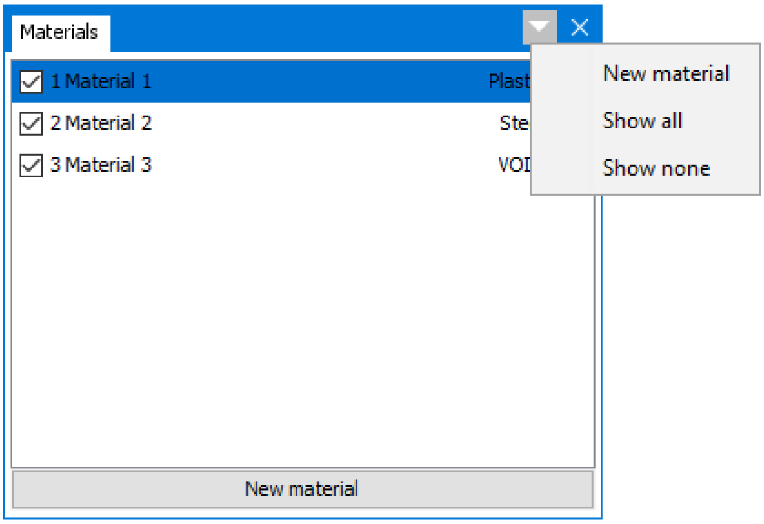

Figure 8 Left: materials window. Right: popup window for additional options

2.4.1 Material options

By right clicking on a material, a list of options for the selected material comes up, see Figure 9. The options are described in Table 2.

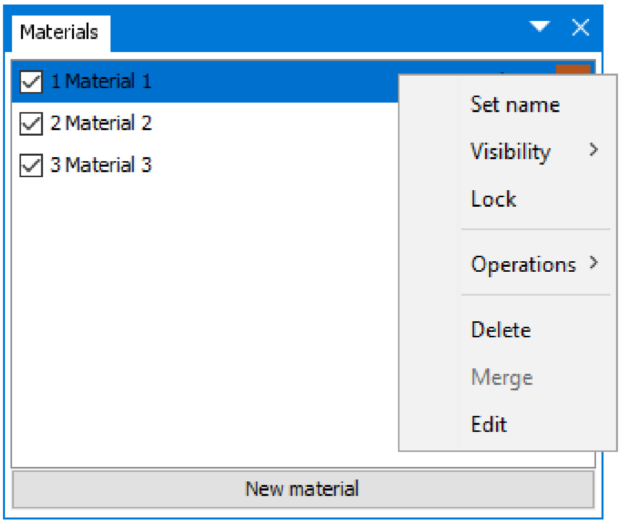

Figure 9 Material options

Table 2 Material options

Option	Description
Set name	Sets the name of the material, keyboard shortcut F2.
Visibility	Toggle visibility of the material on/ off. Using option "Show only this" will turn the visibility of the material on, while turning visibility of other materials off.
Lock	Disables any editing of the material.
Operations	Select: selects the elements of the material. To active material: move selected materials to this material.
Delete	Deletes the selected material(s).
Merge	If multiple materials are selected, they will be merged into one. The first selected material will be the material the rest is merged into.
Edit	Brings up the property window for the selected material, see chapter 2.4.2. The property window can also be activated by double-clicking on a material, or by pressing **ENTER**.

2.4.2 Material properties window

The Material properties window enable the user to specify properties of the selected material, see Figure 10. The different properties are described in Table 3.

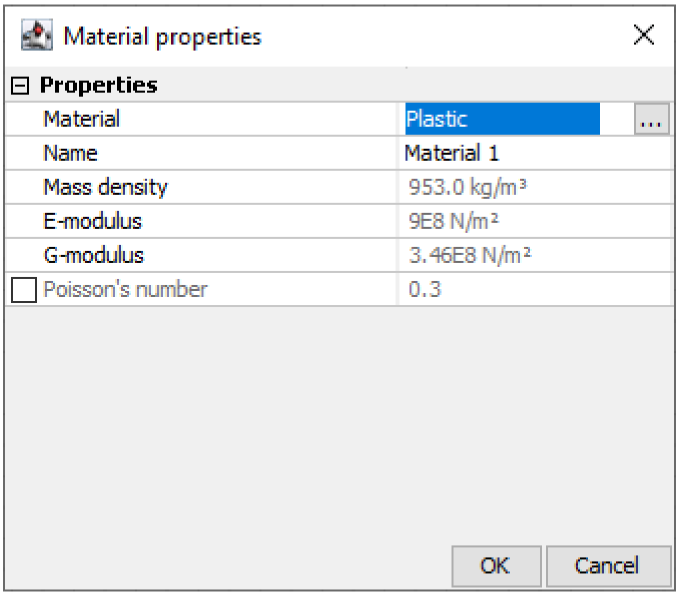

Figure 10 Material properties window

Table 3 Material properties

Property	Description
Material	Provides a dropdown list of available materials. Defaults are Plastic, Steel and VOID. Selecting **<Custom>** allows for editing of the material properties, enabling the user defining a custom material. [**…**]-button allows the user to save a custom material into the folder specified in **Settings** > **Materials folder**. Upon selecting a saved material, the saved properties are brought up. The material properties can be edited and saved, and the presets may also be deleted by pressing the [**…**]-button.
Name	Enable the user to type a name of the material.
Thickness	Thickness of the material. This will be applied to new elements drawn in the Main view window.
Mass density	Mass density of material (i.e. mass per unit volume).
E-modulus	Young’s modulus.
G-modulus	Shear modulus.
Poisson’s number	Poisson’s ratio.

The elements will be drawn with the properties of the activated material.

2.5 Nodes

By clicking a node, it will be highlighted and options for nodes are enabled. Figure 11 view the options provided.

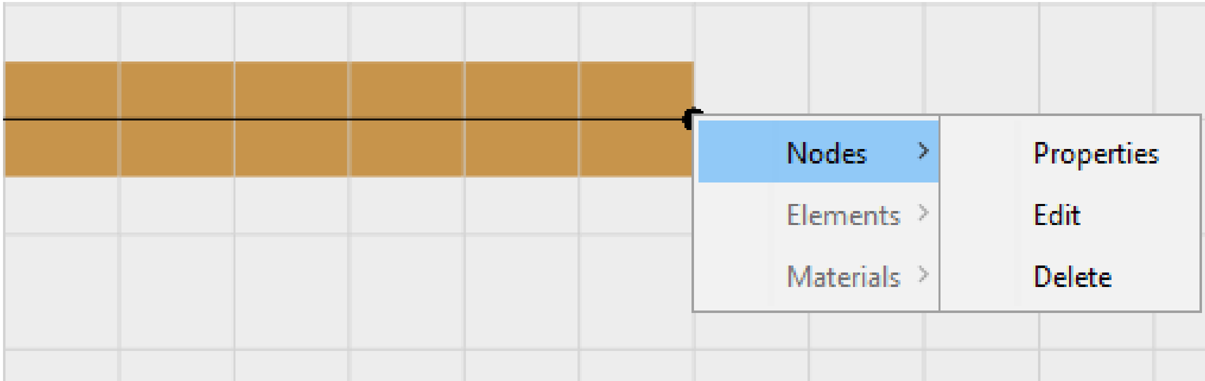

Figure 11 Options for nodes

Table 4 Node options

Option	Description
Properties	Activates the Node property panel.
Edit	Opens the Edit node window, see Figure 12. The user can input new coordinates for the node here. This window will also be opened by double click the node. If more than one node is selected, the user may enter in relative coordinates. The selected nodes will then move accordingly to the input value.
Delete	Deletes all selected nodes.

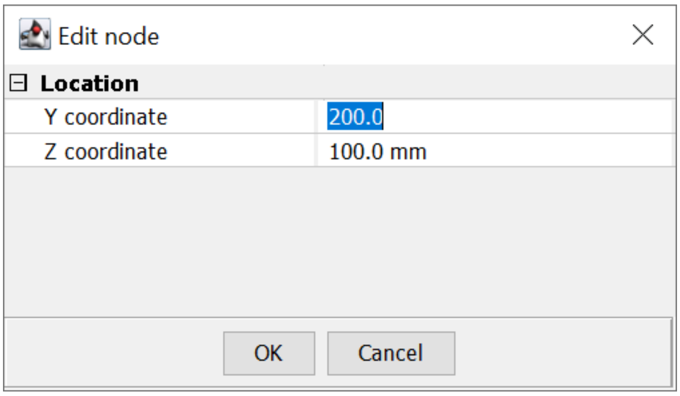

Figure 12 Edit node window

2.6 Elements

By clicking one or more elements they will be highlighted, and options are enabled. The available options are found in Figure 13 and described in Table 5.

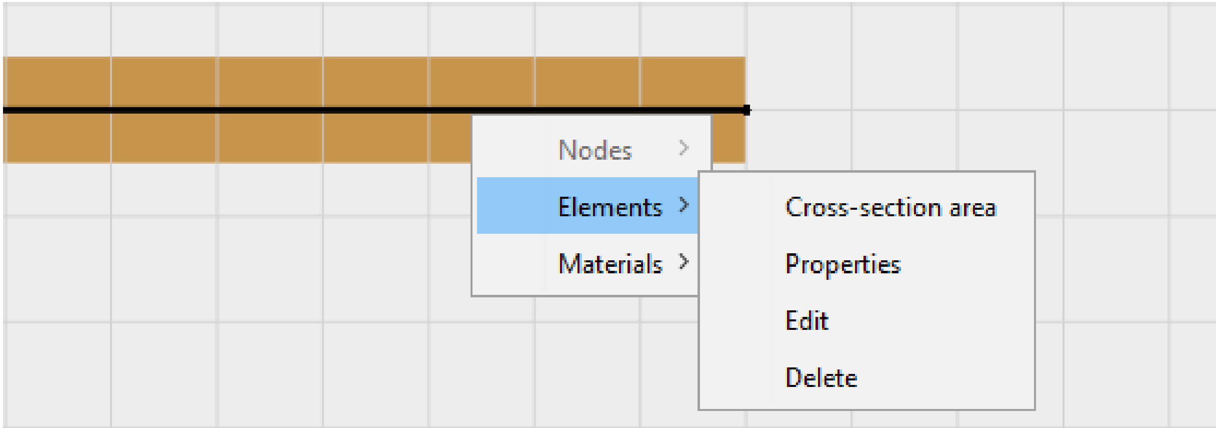

Figure 13 Options for elements

Table 5 Element options

Option	Description
Cross-section area	Shows the cross-section area of the selected element(s). Presented in mm2.
Properties	Activates the Element property panel.
Edit	Opens the Edit element window, see Figure 14. The user may move the selected element(s) to another material.
Delete	Deletes the selected element(s).

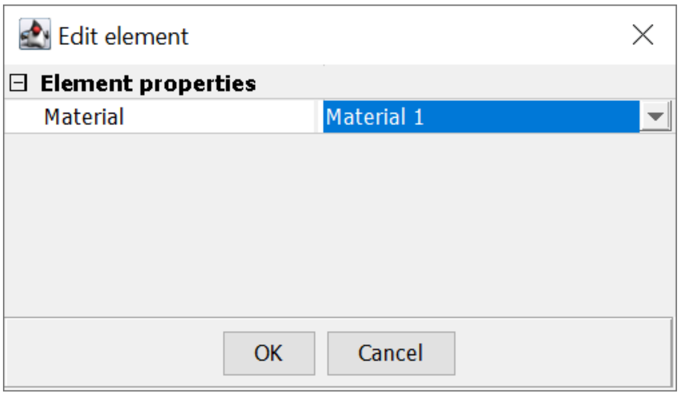

Figure 14 Edit element window

2.7 Insert section F3

AquaCross holds a library of standardized sections. This is accessed through **Commands** > **Insert section F3**. The **Standard sections**-window is then opened. By pressing the dropdown-menu on the top left corner, the user may choose between IPE, HEA, HEB, SHS, L etc. AquaCross also allows for adjusting the standardized sections to the right in the **Standard sections**-window.

3 Calculation and results

Once a cross-section is established, the section parameters and stiffness must be calculated. The **Calculate**-button is found on the Toolbar, or by select **Commands** > **Calculate**. The user will be notified if the calculations are successful or not. The results are stored and may be viewed using the **Show results**-button in the Toolbar, alternatively **Commands** > **Show results**.

3.1 Mesh

Massive cross-sections need to be meshed in order to calculate cross sectional properties. Meshing is a part of the Finite Element Method (FEM), where the model is divided into subsections of straight-line triangles. Calculations for the subsections are then assembled into the larger system that constitutes the section area.

The cross section is meshed automatically when pressing the **Calculate**-button. A default mesh is established based on preset parameters. Options for adjust and refine the mesh are found under **Commands** > **Mesh**, see Figure 15.

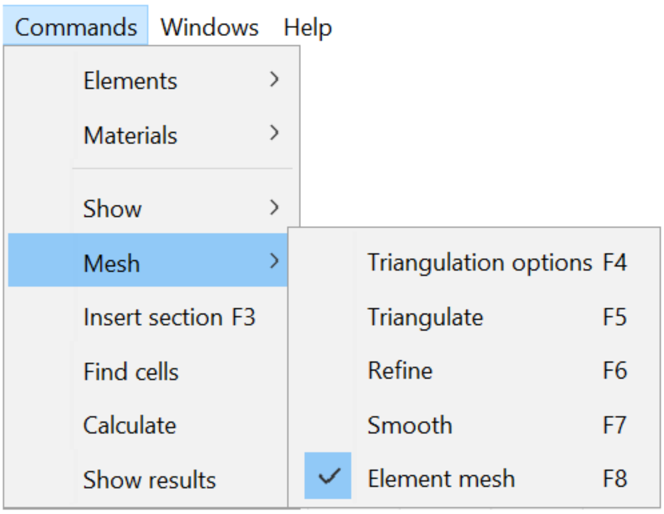

Figure 15 Mesh options

Triangulation options brings up a window for adjusting the mesh parameters, see Figure 16. The options are described in Table 6.

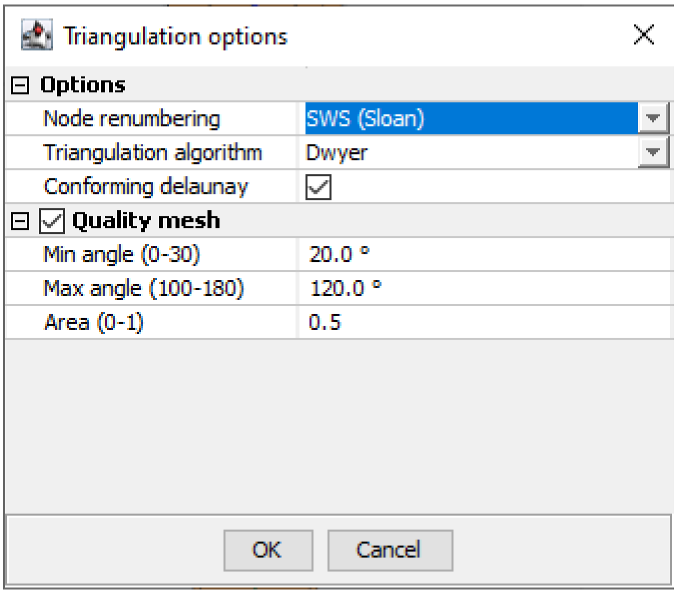

Figure 16 Mesh Triangulation options

Table 6 Triangulation options

Option	Description
Node renumbering	Nodes in a massive section are automatically numbered. This option allows the user to change the numbering system. Changing the numbering system will influence the computational efficiency. Renumbering reduces the bandwidth of the calculation matrices, so that the calculations go faster. Choose between: SWS (Sloan), PFM (Profile Front Minimization) And ELW (Wilson).
Triangulation algorithm	Allow the user to change the algorithm conducting the triangulation. Dwyer: a divide-and-conquer algorithm for construction of Delaunay triangulations. By Dwyer. SweepLine: an algorithm that 'sweeps' across a plane for constructing of Delaunay triangulations. Incremental: algorithm that repeatedly add one vertex at a time for construction of Delaunay triangulations.
Conforming delaunay	Delaunay triangle is such that if one draws a circle around each triangle that passes through the three triangle points, no other points lies inside that circle. When toggle Conforming Delaunay ensures that extra points are inserted so that all the triangles created are a Delaunay triangle.
Min angle (0-30)	Restrictions for minimum allowed angle in the mesh triangles. The minimum angle may be set between 0° and 30°. By default, this is set to 20.0°.
Max angle (100-180)	Restrictions for maximum allowed angle in the mesh triangles. The maximum angle may be set between 100° and 180°.
Area (0-1)	Restrictions for allowed area of the triangle mesh. This is a coefficient, and the value is set between 0-1. If one chooses to use the Refine option after triangulation, the area of the smallest triangle is taken in the mesh and multiplied with this. Then the new area becomes a target for all the triangles that are made when pressing Refine. If the value in Area is set low, the number of triangles increase quickly upon pressing Refine multiple times.

3.2 Show results

The **Show results**-button provides the most important cross-sectional parameters. Figure 17 shows an example of a calculated C section (thinwalled), with length of vertical element 200mm and horizontal 100mm. The thickness is 10mm, and applied material is steel.

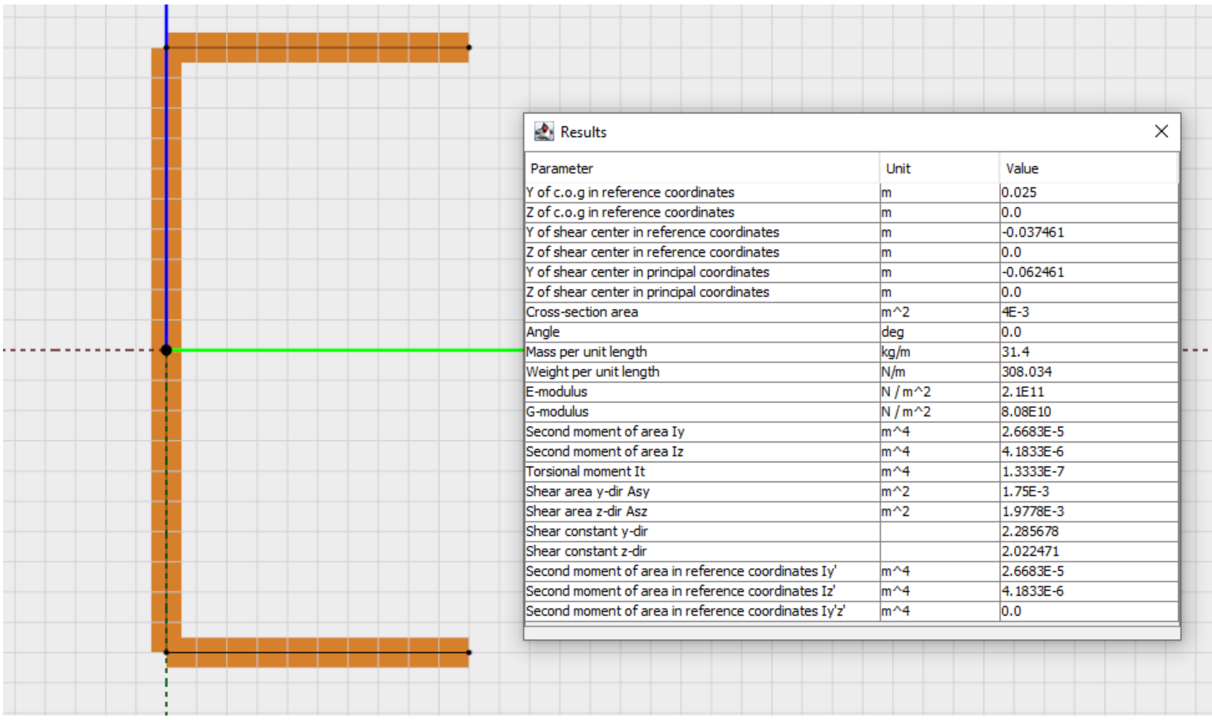

Figure 17 Results for a C section

Note: cross-sections with open segments, just as the one illustrated in Figure 17, will get positive and negative Tau from St. Venant torsion. Therefore, AquaSim may calculate the wrong Von Mises for the elements. To correct for this, use the Show crossection" in AquaView and view the Von Mises there. In these cases, the user will be notified in AquaCross.

3.3 Apply to beam

Having drawn and calculated the section parameters and stiffness, the AquaCross-model must be implemented and applied to the AquaEdit model. This is done by pressing the **Apply to beam**-button in the Toolbar menu. AquaCross will then automatically close and return to the AquaEdit model.

4 View results in AquaView

Figure 18 shows a cantilever beam drawn in AquaEdit, with the cross section from Figure 17 applied to it. One node is held in all DOFs and the other node has a moment of 1000N/m applied to it. This is then analyzed, and we can then view the results in AquaView.

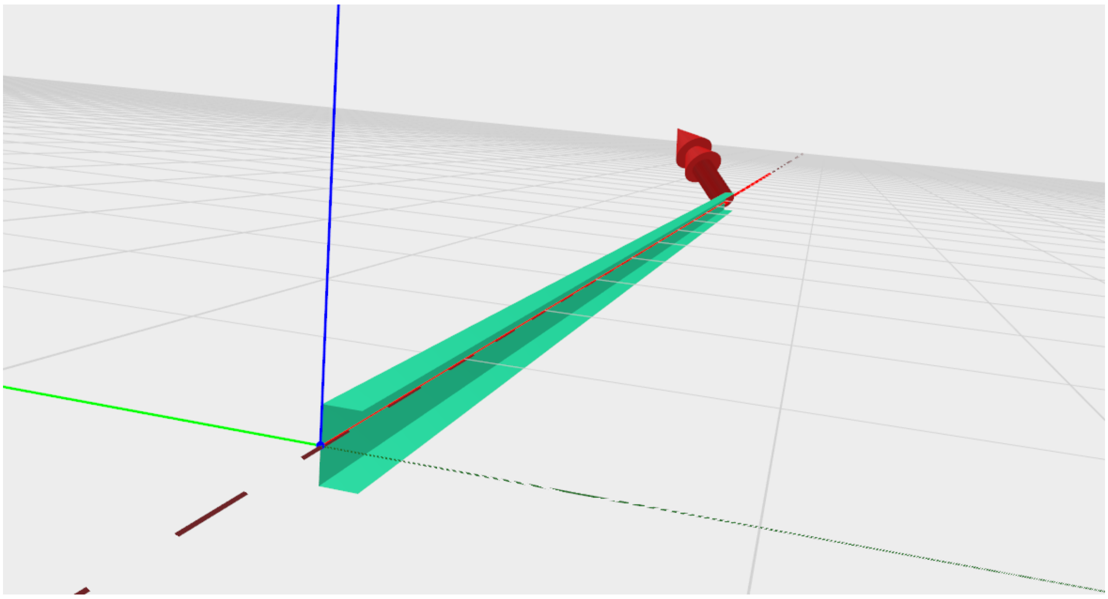

Figure 18 Cantilever beam

4.1 AquaView Von Mises

AquaSim calculates Von Mises stress which may be viewed as a color plot in AquaView, as shown in Figure 19. **Note:** This only applies for symmetrical cross-sections. For asymmetrical cross-sections, as the one illustrated in Figure 17, the Von Mises will be incorrect. Correct Von Mises for asymmetrical cross-sections is found by selecting **Show cross-section** in the Graphing result window for the selected element.

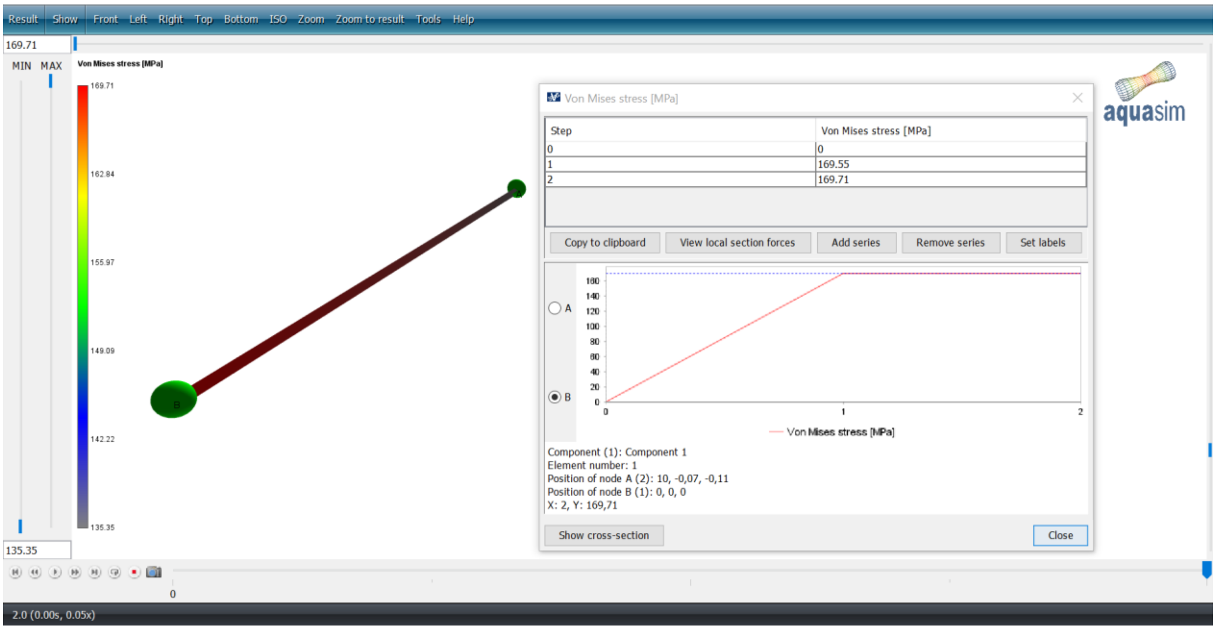

Figure 19 Von Mises plot in AquaView

Having pressed **Show cross-section**, a new window opens, see Figure 20. Comparing Von Mises values presented in Figure 19 and Figure 20 the differences are evident. Having constructed a symmetrical cross-section in AquaCross, the differences in Von Mises in AquaView are small or negligible.

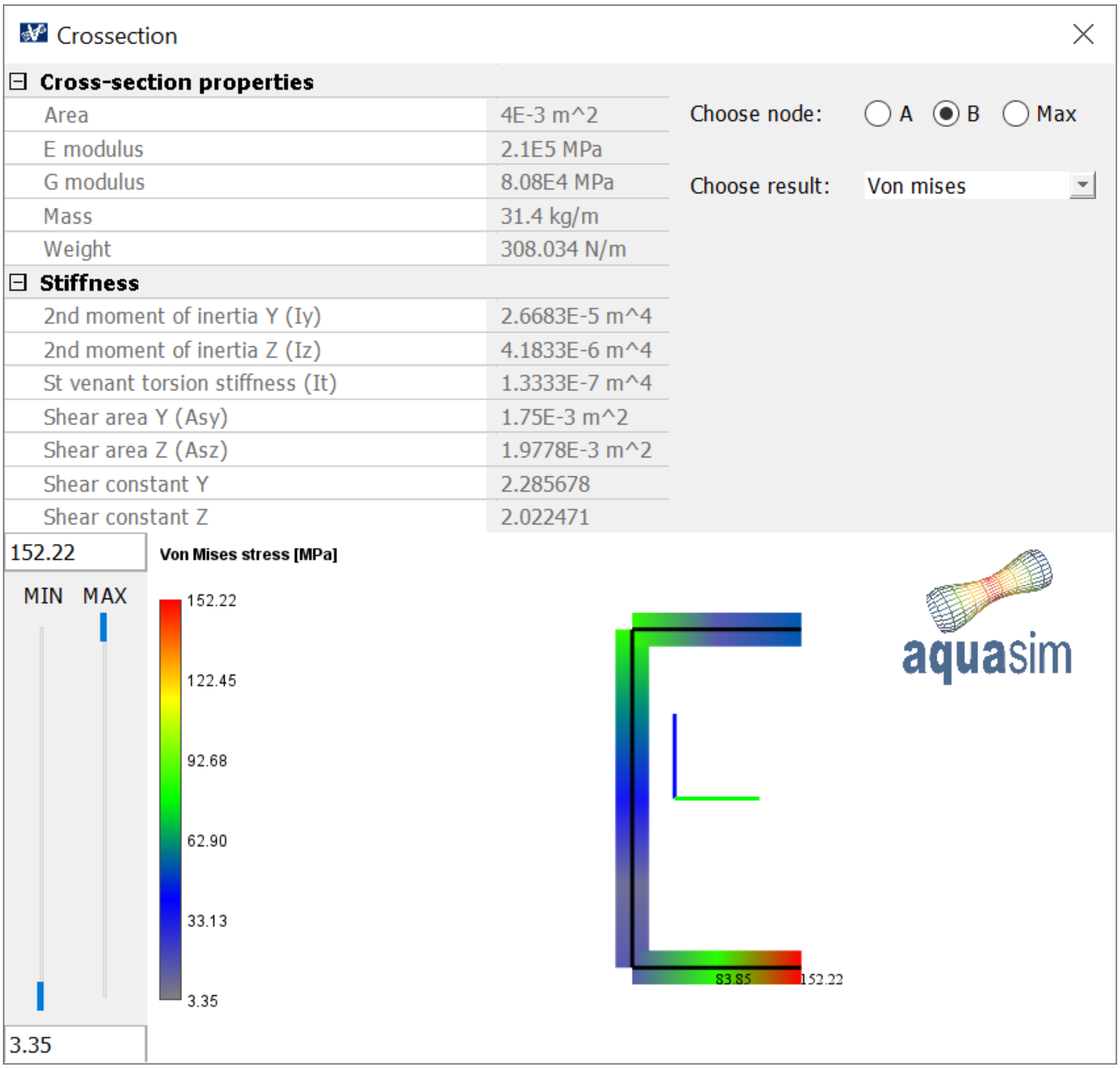

Figure 20 Von Mises plot using AquaCross

The Show cross-section window enable the user to view results as a function of each analyzed timestep. The color plot will hence vary as a function of time when using the playback function. The Show cross-section window har the same functionality as the Main View in AquaView; enabling the user to zoom and pan. By right click on an element in the visual cross-section, results for the selected element are viewed. The local axes (z-axis is blue, y-axis is green) is placed in the cross-section c.o.g.

For more information about viewing results in AquaView, reference is made to the AquaView User Manual.

5 References

Aquastructures. (2022a). *AquaEdit User Manual.* TR-20000-583-1.

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

Bell, K., & et.al. (2000). *CrossX User's Manual.*

6 Appendix A – Accuracy

This chapter presents comparison of section parameters and stiffness using thinwalled and massive sections in AquaCross and formulas. The formulas for known cross-sections are in use today in AquaEdit, in the Data source dropdown menu for beam elements.

6.1 Tube

Given a circular tube of diameter 300mm and thickness 10mm.

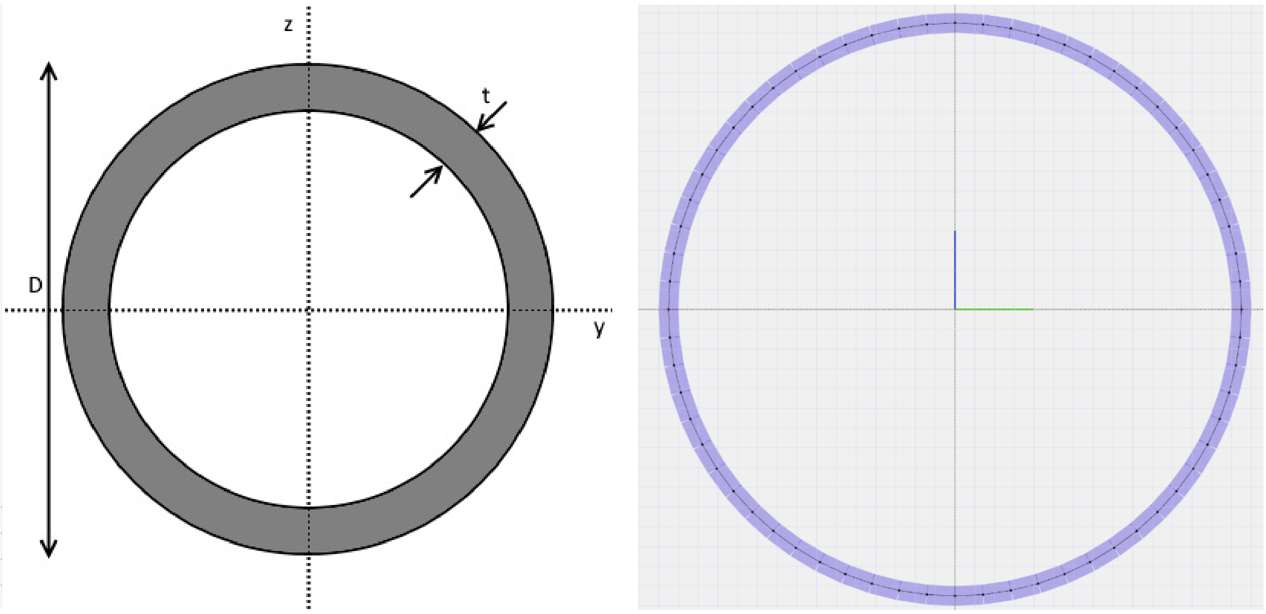

Figure 21 Tube cross-section. Left: from Data source in AquaEdit. Right: from AquaCross

In AquaCross, the number of elements that make up the cross-section are varied. The resulting section parameters from AquaEdit and AquaCross are presented in Table 7.

Table 7 Results from AquaEdit and AquaCross

Parameters	AquaEdit	AquaCross, number of elements						
		32	64	128	256	512	1024	2048
Area	9,111E-03	9,096E-03	9,107E-03	9,110E-03	9,110E-03	9,111E-03	9,111E-03	9,111E-03
Iz	9,589E-05	9,505E-05	9,562E-05	9,577E-05	9,580E-05	9,581E-05	9,581E-05	9,581E-05
Iy	9,589E-05	9,505E-05	9,562E-05	9,577E-05	9,580E-05	9,581E-05	9,581E-05	9,581E-05
It	1,918E-04	1,894E-04	1,910E-04	1,914E-04	1,915E-04	1,915E-04	1,915E-04	1,916E-04
Kappa Y	1,500E+00	1,500E+00	1,499E+00	1,499E+00	1,499E+00	1,499E+00	1,499E+00	1,499E+00
Kappa Z	1,500E+00	1,500E+00	1,499E+00	1,499E+00	1,499E+00	1,499E+00	1,499E+00	1,499E+00

With 32 elements in the AquaCross tube, the largest difference in results are 1.25% for torsional moment. Increasing the elements to 64, reduces the difference to 0.4%.

6.2 Hollow rectangle

Given a hollow rectangle with height 150mm, width 100mm, and thickness 5mm.

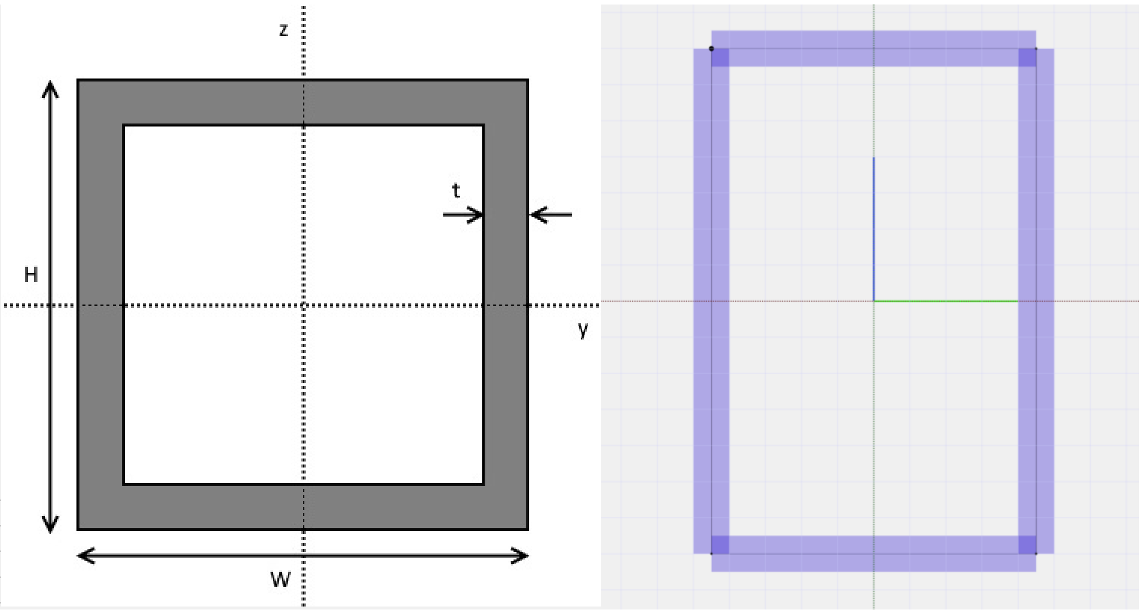

Figure 22 Hollow rectangle. Left: from Data source in AquaEdit. Right: from AquaCross

The number of elements that make up the cross-section is varied in AquaCross. The resulting section parameters from AquaEdit and AquaCross are presented in Table 8.

Table 8 Results from AquaEdit and AquaCross

Parameters	AquaEdit	AquaCross, number of elements	
		5	25
Area	2,400E-03	2,400E-03	2,400E-03
Iz	3,995E-06	3,989E-06	3,989E-06
Iy	7,545E-06	7,536E-06	7,536E-06
It	7,906E-06	7,906E-06	7,906E-06
Kappa Y	2,400E+00	2,539E+00	2,539E+00
Kappa Z	1,600E+00	1,692E+00	1,692E+00

Increasing number of elements in AquaCross does not increase the accuracy. The only difference is the Kappa Y and Kappa Z, which is defined from the shear area in each direction, i.e.:

$$\kappa_y = \frac{A}{A_y}$$

Since AquaCross use line segments in the way it does, the shear area will become a little too small here. It will not count with the small squares at each corner. In this example, the area is 25mm2, and with that added to the shear area it will give a Kappa Y of 2.47, which is closer to the answer from AquaEdit.

6.3 Double tube

Provided a double tube with diameter 300mm, thickness 10mm and length between the two origins are 500mm. This is drawn in AquaCross as two circles, with a VOID element connecting the two circles, see Figure 23.

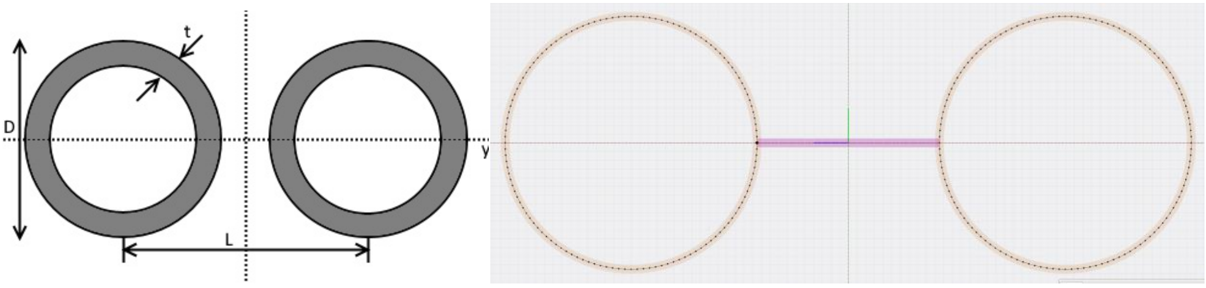

Figure 23 Double tube. Left: from Data source in AquaEdit. Right: from AquaCross

Results are presented in Table 9.

Table 9 Results from AquaEdit and AquaCross

Parameters	AquaEdit	AquaCross
Area	1,822E-02	1,822E-02
Iz	1,918E-04	1,915E-04
Iy	1,918E-04	1,330E-03
It	3,836E-04	3,828E-04
Kappa Y	1,500E+00	1,499E+00
Kappa Z	1,500E+00	2,243E+05

When applying double tube from Data source in AquaEdit the second area of moment and the shear deformation constants will be the same. This because we say that the two tubes are not connected to each other. If this was not the case, then the formula would give I_y of 1.331E-03. This is the same value as from AquaCross calculation.

A value of 2.243E+05 for Kappa Z from AquaCross are clearly incorrect, why this happens is not clear. But the cause is probably due to the shear modulus of the VOID element is so small it causes the calculations to fail. Setting the VOID element to steel will provide a Kappa Z equal to 3.47.

It is recommended to look at the results and evaluate if they make sense. Using a Kappa Z 150000 times larger than expected will cause wrong results in the analysis later.

6.4 I-section

Provided an I-section, using values for an IPE 300 section. This has a flange width of 150mm, flange thickness of 10.7mm, web height of 278.6mm, web thickness of 7.1mm and radius of 15mm.

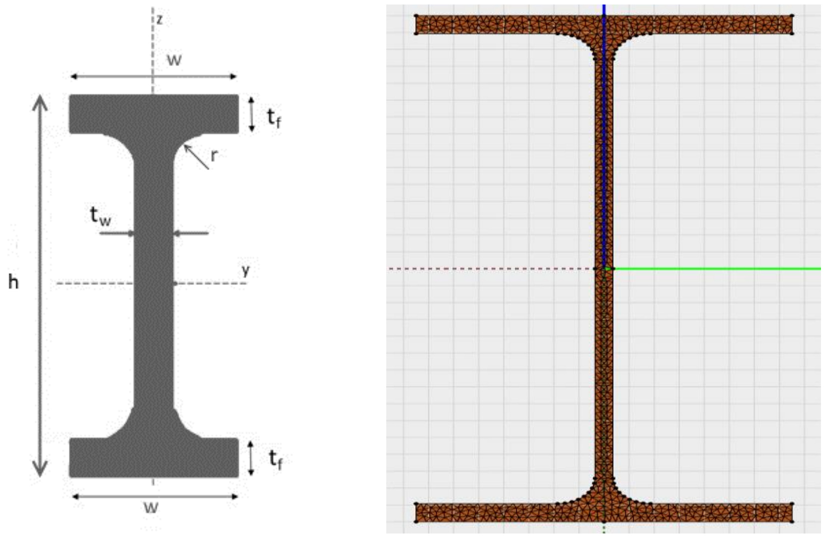

Figure 24 I-section. Left: from Data source in AquaEdit. Right: from AquaCross

The results are presented in Table 10. For IPE-beams official numbers are also available.

Table 10 Results from AquaEdit and AquaCross

Parameters	AquaEdit	AquaCross	Official numbers
Area	5,188E-03	5,386E-03	5,380E-03
Iz	6,027E-06	6,382E-06	6,040E-06
Iy	7,999E-05	8,364E-06	8,365E-05
It	1,951E-07	1,985E-07	2,010E-07
Kappa Y	1,616E+00	1,570E+00	1,571E+00
Kappa Z	2,623E+00	2,569E+00	2,569E+00

The default mesh created in AquaCross is used here. Increasing the number of triangles by creating an own mesh with smaller triangles will not increase the accuracy of the results by much. If a higher *It* is desired to match the official numbers even more, it is possible to decrease the number of fillets used when inserting a standard I-section, this will make the values of *It* to increase a bit.