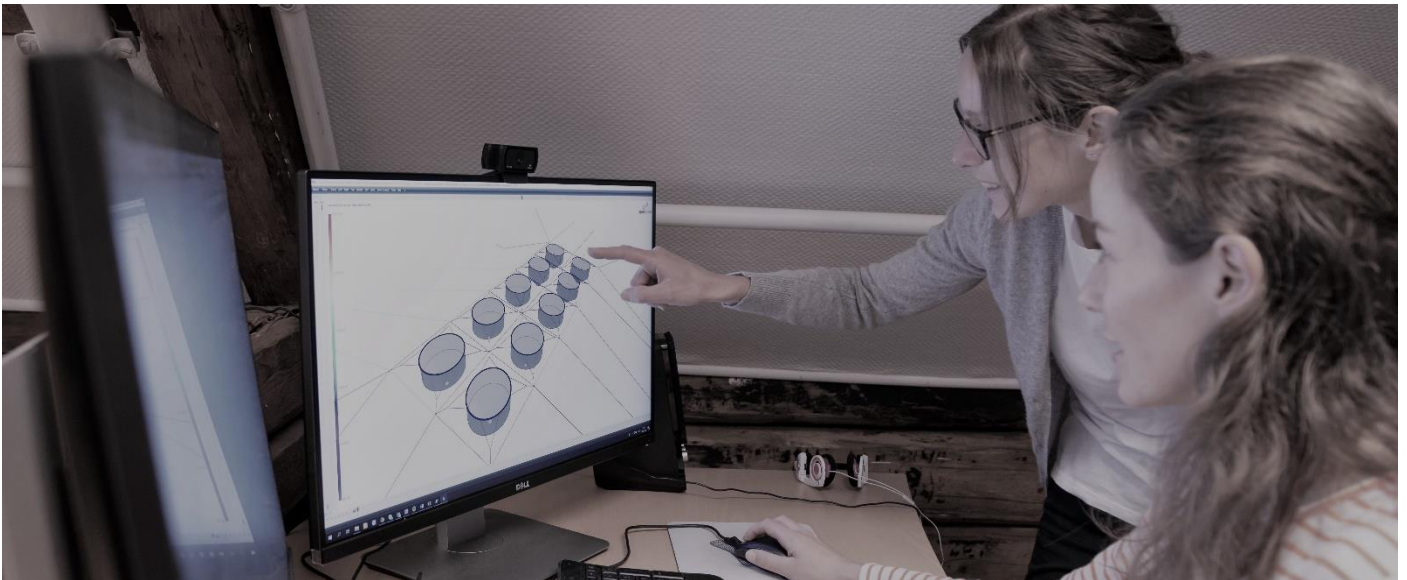


# AquaSim training courses

## - Buckling



Revision: 1.0

AquaSim version: 2.19

Aquastructures AS  
Kjøpmannsgata 21, 7013 Trondheim  
Norway

## Content

1	Prerequisites.....	3
2	Introduction.....	3
3	Principles of buckling analysis in AquaSim .....	4
3.1	Analysis type.....	4
3.2	Amount of eigenvalues written to files .....	5
3.3	Positive eigenvalues closes to 0 to omit.....	5
3.4	Amplitude of eigen vectors .....	5
3.5	Shift value .....	5
3.6	Stress value.....	5
3.7	Analysis valid for .....	5
3.8	Calculations and results.....	6
4	Case study – Euler beam.....	7
4.1	Learning objectives .....	7
4.2	Introduction .....	7
4.3	Buckling analysis of beam modelled with type Morison submerged .....	9
4.4	Buckling analysis of beam modelled with type Shell .....	11
4.5	Buckling analysis of beam modelled with type Normal with bending stiffness .....	14
5	Summary .....	15
6	Revision comments .....	15

## 1 Prerequisites

The tutorial presents a simple case study with the purpose of demonstrating functionality in AquaSim.

It is assumed that the user is familiar with the basic principles of modelling and specifying material parameters in AquaEdit, as well as conducting analyses. If you are looking for an introduction to AquaSim we advise you to start with the Basic program tutorials.

## 2 Introduction

This tutorial presents how buckling can be investigated in AquaSim.

Buckling is a phenomenon that commonly appear in slender structures, or thin plates that is exposed to compressional forces. It can materialize as a rapid change in the shape, such as bulging or wrinkling. Buckling may arise from loads which in themselves are not critical for the structural integrity, but the subsequent deformations may lead to unfavorable load transfer which in turn can cause the load bearing structural part to lose its load capacity.

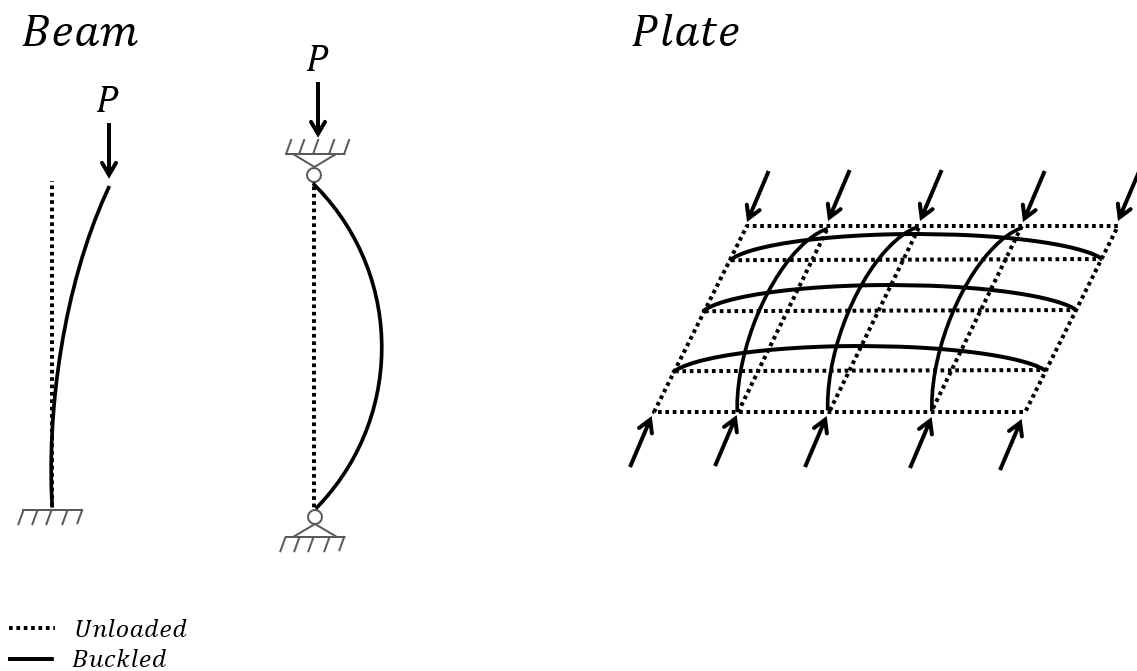


Figure 1

Construction standards often require buckling to be investigated and documented.

### 3 Principles of buckling analysis in AquaSim

As from AquaSim version 2.19, buckling analysis is available for the following component types:

- Beam:
  - Morison submerged
  - Hydrodynamic
- Membrane/ Membrane X:
  - Shell
  - Normal with bending stiffness

In AquaSim, one has the possibility to apply explicit buckling analyses through the option **Buckling /eigen period analysis** found in the **Export** menu. This is illustrated in the figure below.

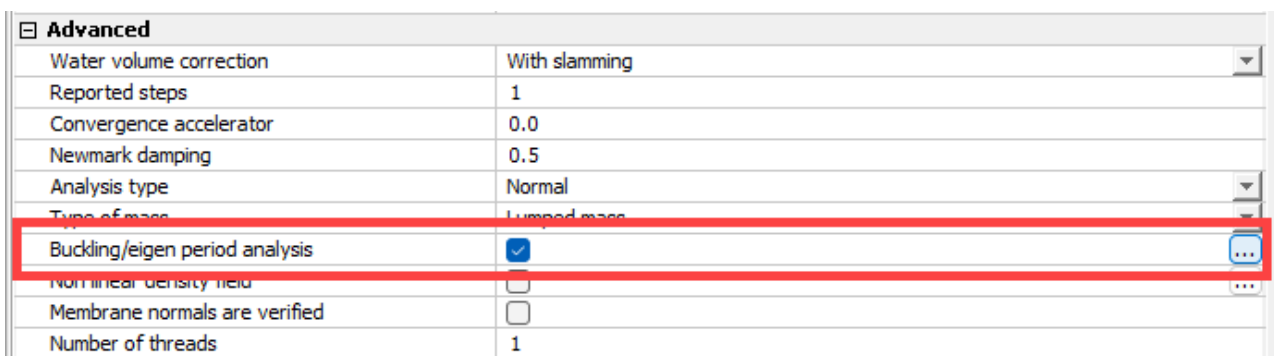


Figure 2

By selecting the three dots [...] to the right, one enters the control window for buckling settings.

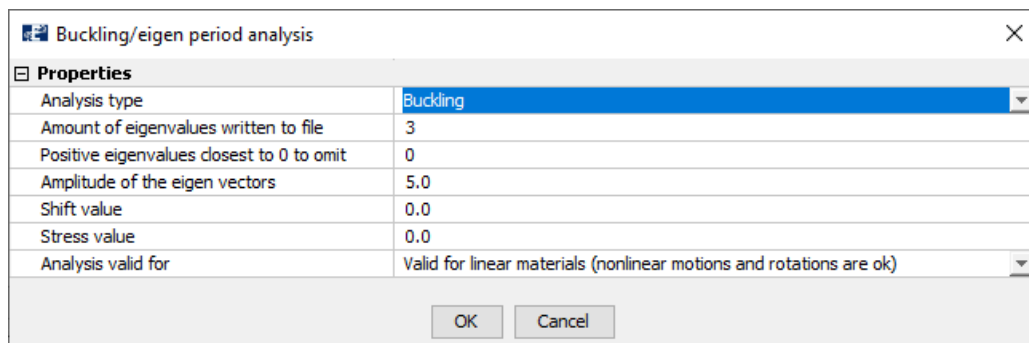


Figure 3

Description of the different options are given in the succeeding sections.

#### 3.1 Analysis type

Buckling analysis solves the equation:

$$K_{elem} - \delta K_{geom} = 0$$

where  $K_{elem}$  is the material stiffness for the structural configuration at the given timestep in the analysis.  $K_{geom}$  is the geometrical stiffness for the structural configuration at the given timestep in the analysis. The result of the analysis is  $\delta$ , which is the factor the load distribution should linearly be increased to obtain linearized buckling. The corresponding eigen vector is the buckling shape for this factor. Buckling shape may also be referred to as buckling mode, in this tutorial we apply the term buckling shape.

**Analysis type** provides four different options:

- Buckling: calculation of buckling factor.
- Eigen periods (excluding mass of truss elements): not the topic for this tutorial.
- Eigen periods (including mass of truss elements): not the topic for this tutorial.
- Eigen periods (excluding mass of truss elements and mass caused by nodal loads): not the topic for this tutorial.

### 3.2 Amount of eigenvalues written to files

AquaSim may calculate up to six different buckling factors. In this case, eigen values will correspond to buckling factor. The amount of buckling factors written to file must be a number that is higher than 0 and lower than 7.

If this parameter is equal to 3, then the three buckling shape closes to the **Shift value** is calculated.

### 3.3 Positive eigenvalues closes to 0 to omit

How many positive buckling factors closest to 0 that shall be omitted in the results written to file.

### 3.4 Amplitude of eigen vectors

The amplitude of buckling shape. The buckling shape indicates the direction of the buckling for the calculated buckling factor. This parameter is unitless and can be interpreted as an amplification factor for the buckling shape. It makes it easier to identify the buckling direction and to evaluate the results.

### 3.5 Shift value

The buckling analysis in AquaSim is conducted by numerical method. This parameter enables a numerical search for buckling factors around this value. I.e. it pinpoints the area in which one wish to find buckling factors.

### 3.6 Stress value

This option is not currently available to use in AquaSim.

### 3.7 Analysis valid for

What type of materials the analysis should be conducted for. Two options are available:

- Valid for linear materials (nonlinear motions and rotations are ok): only material data from Material/ section properties are considered in the analysis. Potential nonlinear relation from NLD-tables is not considered when calculating buckling factors.
- General, but noiser buckling response: materials with both linear and nonlinear relations are included in the buckling analysis. Please note that AquaSim may find numerous of buckling factors when this is selected, and all these can be perceived as “noise”.



### 3.8 Calculations and results

To include buckling study in AquaSim-analysis, the user should select the checkbox for **Buckling /eigen period analysis**. The user should then export the model and run an analysis as normal, see figure below.

Water volume correction	With slamming	<input checked="" type="checkbox"/> Automatic grouping <input checked="" type="checkbox"/> Delete AVS files after run <input checked="" type="checkbox"/> Analyse immediately after export <input type="checkbox"/> Enable low priority processes <input type="checkbox"/> Omit PFAT files from analysis <input checked="" type="checkbox"/> Verify model when exporting <input type="checkbox"/> Split file by timesteps <input type="checkbox"/> Extract timestep range aquasim_2_18_1.exe <b>Export</b>
Reported steps	1	
Convergence accelerator	0.0	
Newmark damping	0.5	
Analysis type	Normal	
Type of mass	Lumped mass	
<b>Buckling/eigen period analysis</b>	<input checked="" type="checkbox"/>	
Non linear density field	<input type="checkbox"/>	
Membrane normals are verified	<input type="checkbox"/>	
Number of threads	1	
<b>Hydrodynamic properties</b>		
Wave headings	90	
Segments on hull	40	

Figure 4

Having carried out a analysis, some additional result files will be generated, they are presented in the figure below.

0_01buckle1.avz	10.01.2024 07:35	Compressed Aquasim result file	91 kB
0_01buckle2.avz	10.01.2024 07:35	Compressed Aquasim result file	88 kB
0_01buckle3.avz	10.01.2024 07:35	Compressed Aquasim result file	90 kB
0_01buckle4.avz	10.01.2024 07:35	Compressed Aquasim result file	88 kB
0_01buckle5.avz	10.01.2024 07:35	Compressed Aquasim result file	91 kB
0_01buckle6.avz	10.01.2024 07:35	Compressed Aquasim result file	89 kB

Figure 5

The <filename>buckle1.avz will include the results for the buckling factor closest to **Shift value**. Please note that each step in the analysis is considered separately, such that each step must be evaluated by the user. The <filename>buckle2.avz contain results for the buckling factor second closest to **Shift value**, and so on.

## 4 Case study – Euler beam

### 4.1 Learning objectives

In this case study you will be presented to buckling analysis of a beam that is exposed to a compressive load. The aim is to find the buckling factor. Upon completion of this case study, you will know:

- basics of beam buckling
- basic concepts of critical buckling load and buckling factor
- conduct a buckling analysis in AquaSim.

### 4.2 Introduction

We should investigate buckling of a Euler beam. A Euler beam is a beam which follow the Euler-Bernoulli beam theory.

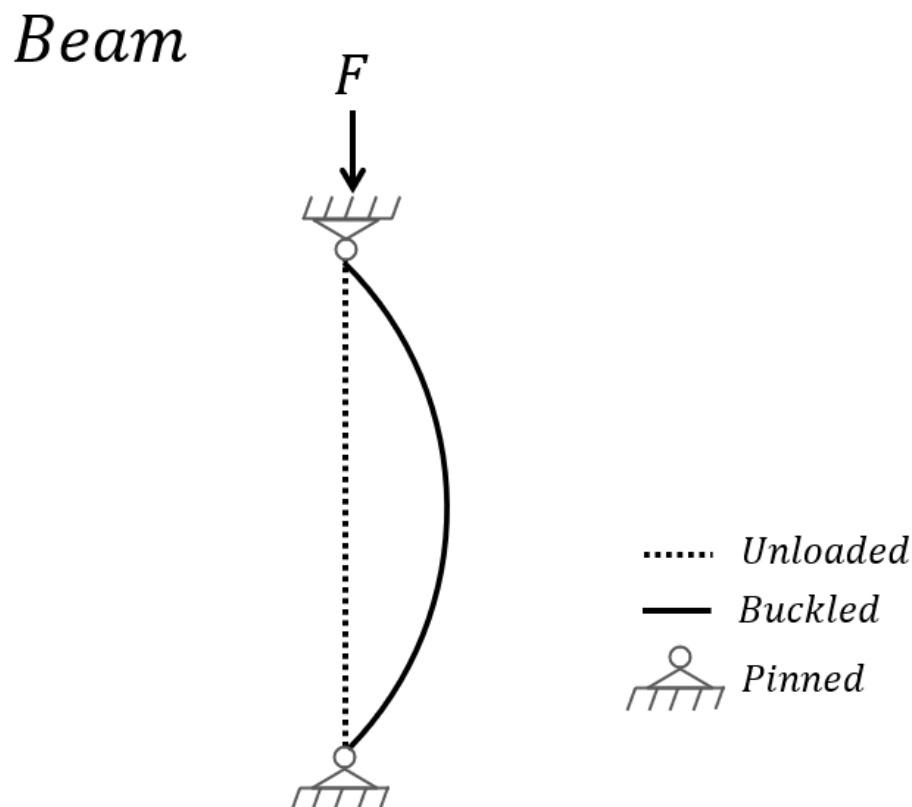


Figure 6

Consider a straight, prismatic beam. The beam is pinned in both ends. When the beam is exposed to compressive load, it deforms until it reaches a point where it becomes unstable, causing the lateral displacement to increase uncontrollably. This instability is known as Euler buckling and occurs when the axial compressive load exceeds a critical value. Note that the theory assumes a linear relation between stress and strain in the material. The critical buckling load  $P_{cr}$  for a beam is described as:

$$P_{cr} = \frac{\pi^2 EI}{(KL)^2}$$

where  $E$  is the modulus of elasticity,  $I$  is the moment of inertia for the cross section,  $K$  is beam effective length factor and  $L$  is the effective length of the beam.

The effective length factor  $K$  depends on the boundary conditions of the beam; whether it is free-fixed, pinned-pinned, fixed-pinned etc. For example, if the beam that is pinned-pinned,  $K$  is equal to 1, while for other boundary conditions  $K$  will be different. Euler's formula provides a theoretical prediction of the critical load at which buckling occurs. When the applied load exceeds the critical load, the beam becomes unstable in the lateral direction and buckling initiates.

The main particular for the beam is presented in the table below. The weight of the beam has been set to zero to be able to compare AquaSim results and calculations easily. The beam is exposed to a compressive load  $F = 10\,000\text{N}$  at the top end.

Parameter	Abbreviation	Value
Modulus of elasticity [N/m <sup>2</sup> ]	E	2.1E+11
Euler effective length factor [-]	K	1.0
Width [m]	W	1.0
Thickness [m]	T	0.1
Area moment of inertia [m <sup>4</sup> ]	I	8.33E-05
Weight in air [kg/m]	m	0.0
Length [m]	L	10.0
Compressive load [N]	F	1.0E+04

Applying the formula presented in the introduction, the following critical buckling load is calculated:

$$P_{cr} = \frac{\pi^2 EI}{(KL)^2} = 1727 \text{ kN}$$



### 4.3 Buckling analysis of beam modelled with type Morison submerged

For this case study the beam is modelled with component BEAM and type Morison submerged. You may apply the AquaSim model *Euler-beam.amodel* that is associated with this tutorial.

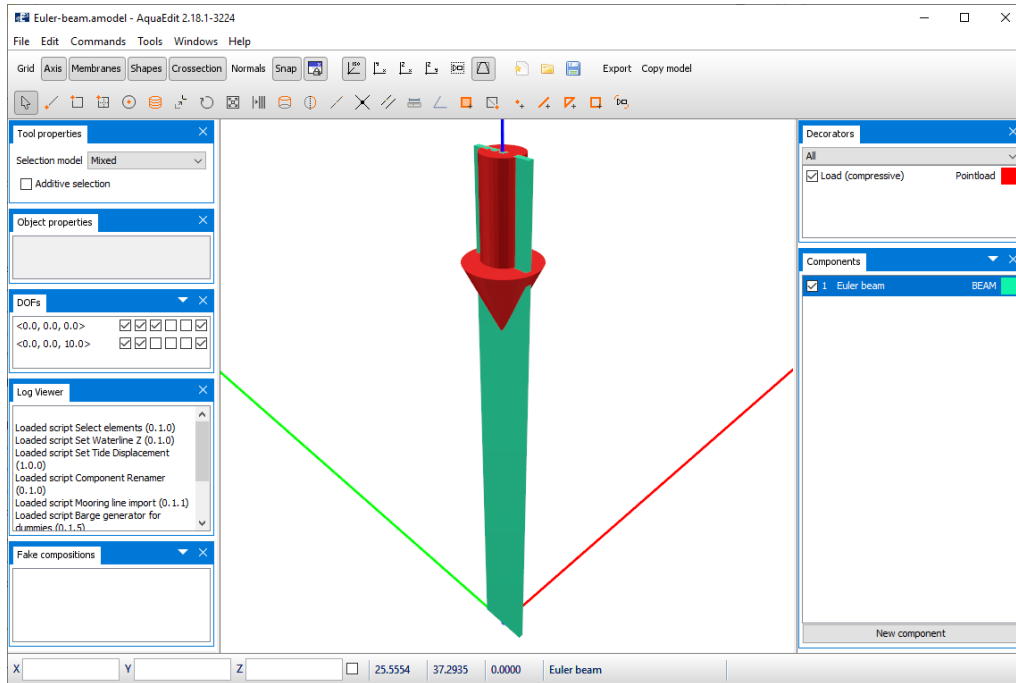


Figure 7

Beam buckling should now be calculated explicitly from the buckling analysis tool in AquaSim. The results are then compared with the formula from the previous section.

In the AquaSim model, go to the **Export** menu, and from the **Advanced** section, select the checkbox **Buckling /eigen period analysis**.

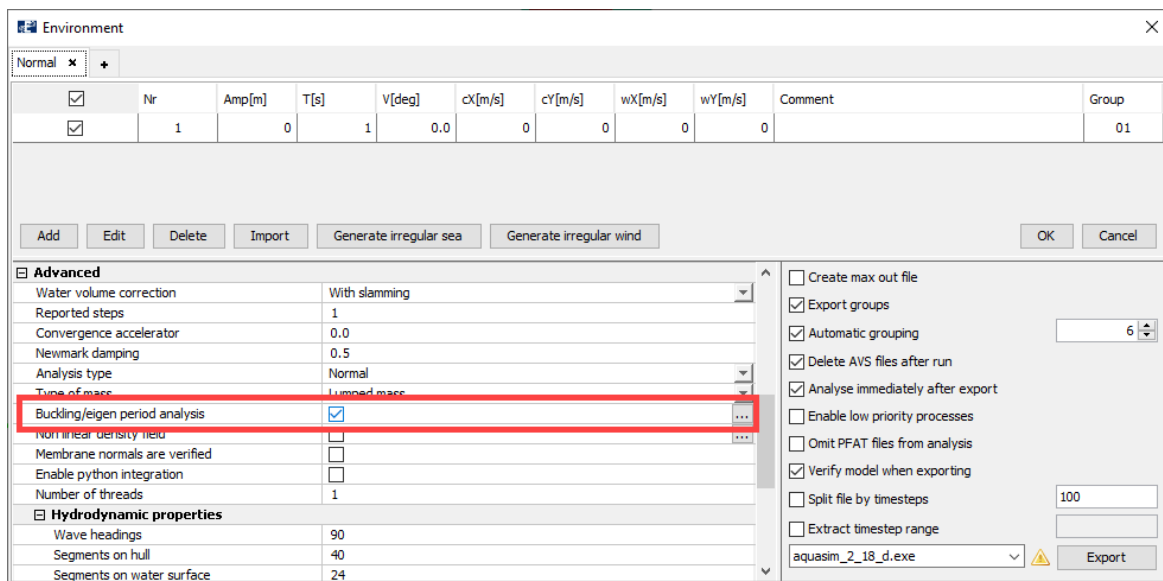
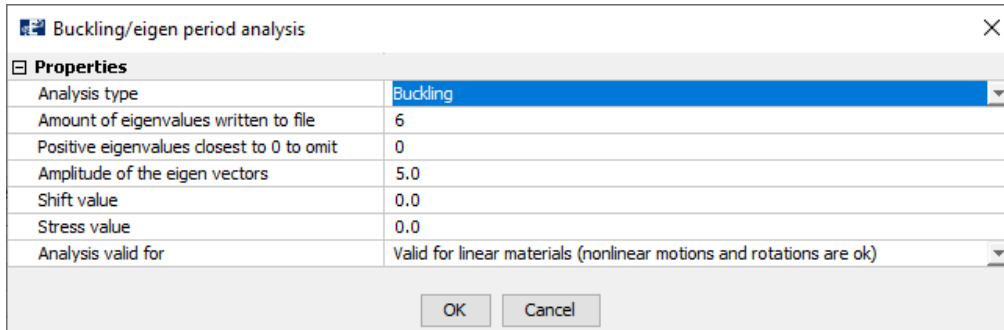


Figure 8

Then select the tree dots [...] to the right to enter the control window for buckling analysis settings. Analysis type should be **Buckling**, **Amount of eigenvalues written to file** should be 6 and the **Amplitude of the eigen vectors** can be left equal to 5.



Buckling/eigen period analysis	
<b>Properties</b>	
Analysis type	Buckling
Amount of eigenvalues written to file	6
Positive eigenvalues closest to 0 to omit	0
Amplitude of the eigen vectors	5.0
Shift value	0.0
Stress value	0.0
Analysis valid for	Valid for linear materials (nonlinear motions and rotations are ok)
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Figure 9

Select **OK**. **Export** the model and **run** the analysis. In this case study we have named this analysis *euler-beam\_*. When the analysis is finished, the files *euler-beam\_01buckle1 -6* should have been generated. These files contain the buckling factors from the analysis.







	euler-beam_01buckle1.avz	10.01.2024 14:41	Compressed Aquasim result file	134 kB
	euler-beam_01buckle2.avz	10.01.2024 14:41	Compressed Aquasim result file	134 kB
	euler-beam_01buckle3.avz	10.01.2024 14:41	Compressed Aquasim result file	134 kB
	euler-beam_01buckle4.avz	10.01.2024 14:41	Compressed Aquasim result file	132 kB
	euler-beam_01buckle5.avz	10.01.2024 14:41	Compressed Aquasim result file	134 kB
	euler-beam_01buckle6.avz	10.01.2024 14:41	Compressed Aquasim result file	138 kB

Figure 10

Load *euler-beam\_01buckle1.avz* in AquaView. Select **Result > Results > Buckling factor**. The buckling factor is found to be approximately 172.72.

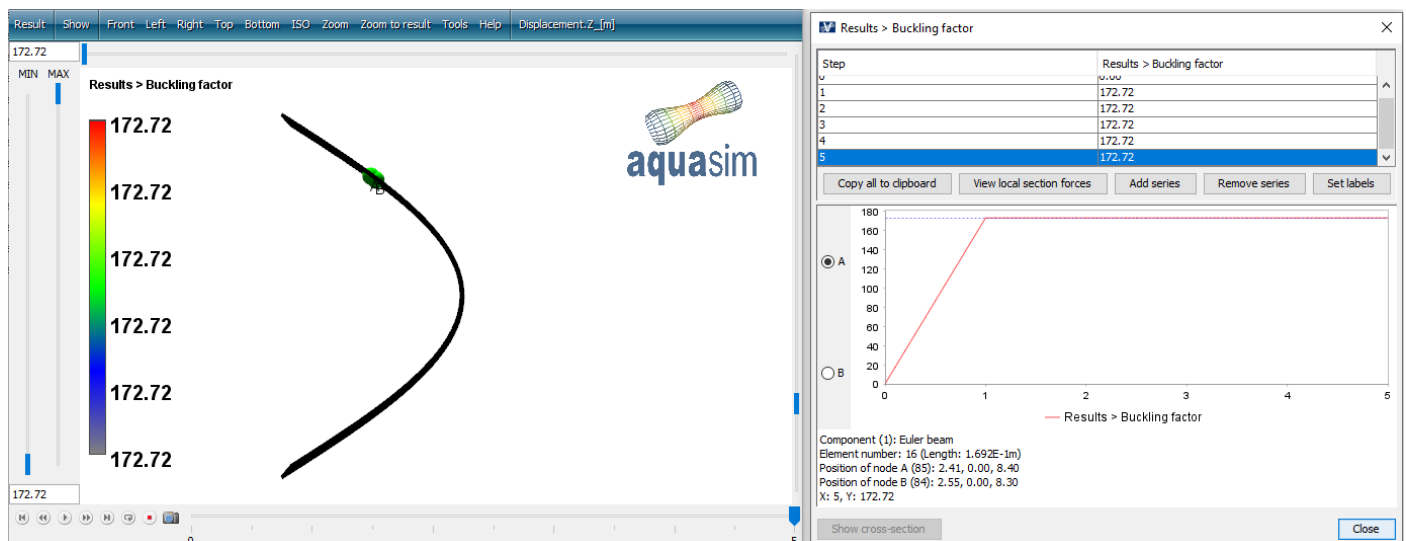


Figure 11

The buckling factor is the critical buckling load  $P_{cr}$  divided by the load in the element:

$$\text{Buckling factor} = \frac{P_{cr}}{F}$$

In this case, the element was subject to a compressive load of 10kN, meaning that the corresponding critical buckling load then will be 1727.2 kN. Comparing this with the calculated  $P_{cr} = 1727 \text{ kN}$  the results compare very well.

#### 4.4 Buckling analysis of beam modelled with type Shell

For this case study the beam is modelled with component MEMBRANE X and type Shell. You may apply the AquaSim model *Euler-shell.amodel* that is associated with this tutorial.

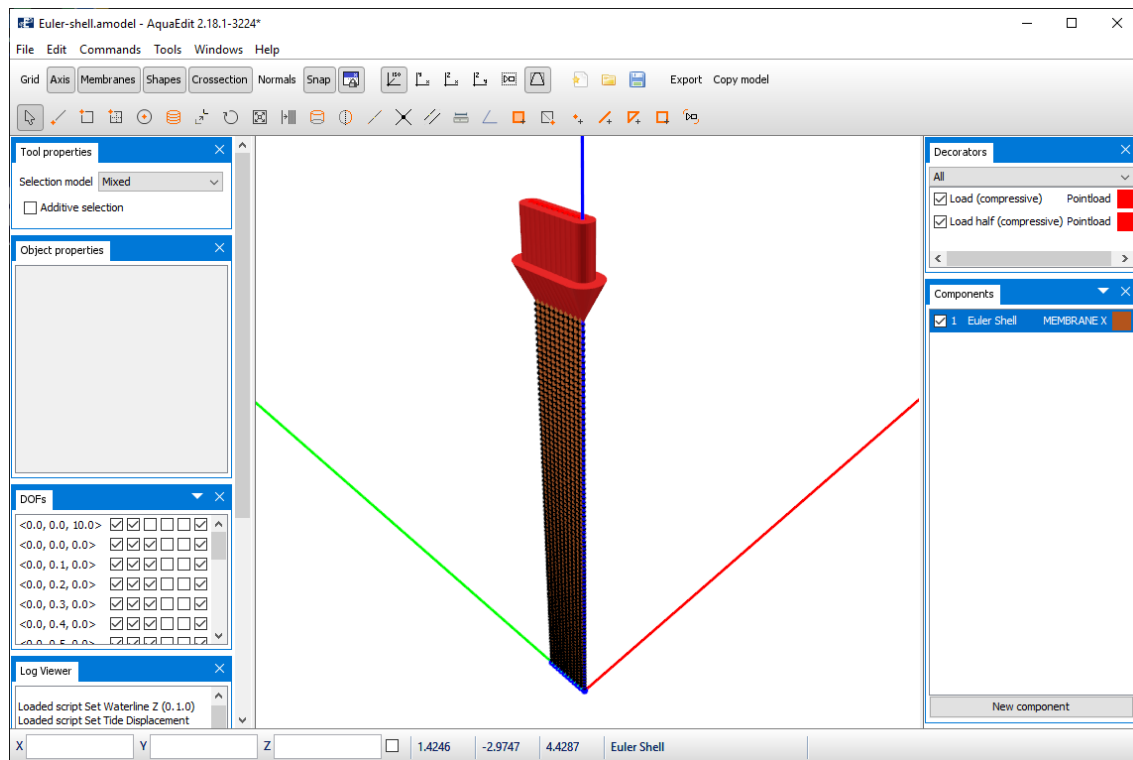


Figure 12

The model is built up with membrane panels of 0.1 x 0.1 m, and the thickness is 0.1 m. This will result in a model with a total of 1000 panels. The boundary conditions at the top and bottom of the model are equivalent as for the beam-model from previous section.

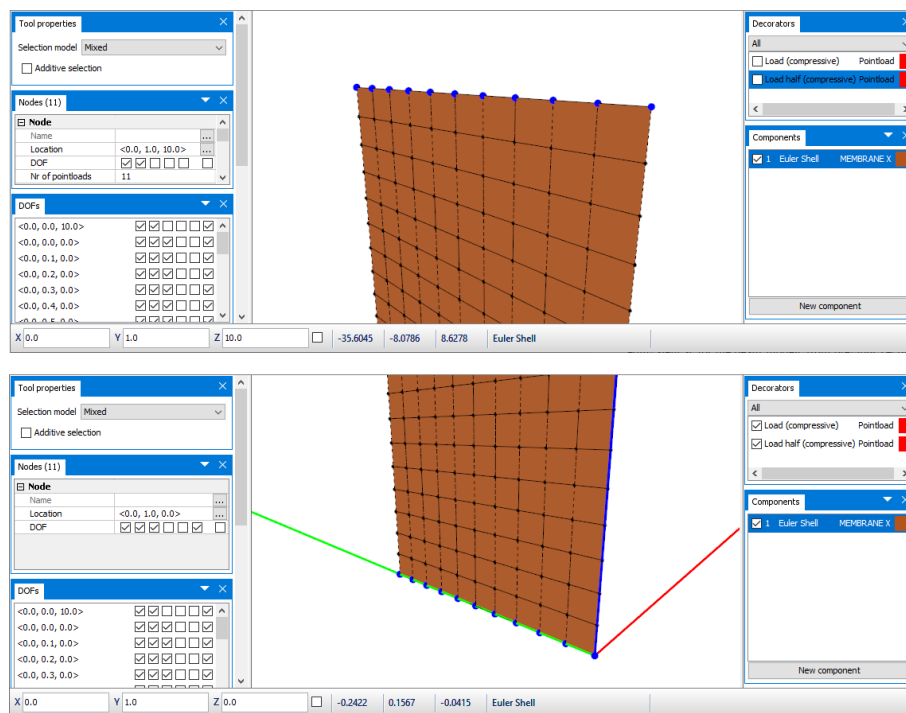


Figure 13

At the top, **Pointloads** are placed on all 11 nodes. The corner nodes have a load of 500 N each pointing downwards, where the middle nodes have 1000 N pointing downwards. **Export** your model and **run** a buckling analysis, the same options as for the beam case is applied for the shell as well. In this case study we have named this analysis *euler-shell\_*.

The applied Pointloads in the model will result in an approximate 10 kN through the cross section of the model, apart from some boundary effects at the top and bottom of the model. This is seen as axial force in local x-direction in the figure below, taken from *euler-shell\_01.avz*.

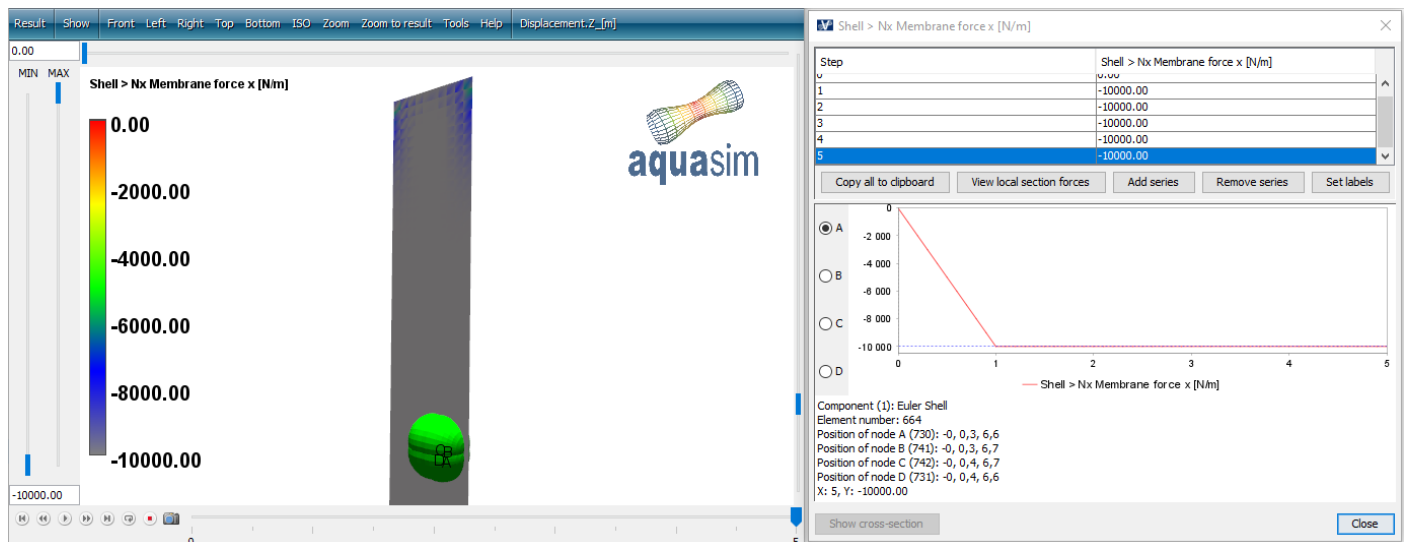


Figure 14

Load the file *euler-shell\_01buckle1.avz* in AquaView. Select **Result > Results > Buckling factor**. The buckling factor is approximately 171.93. This aligns very well with the results for a beam.

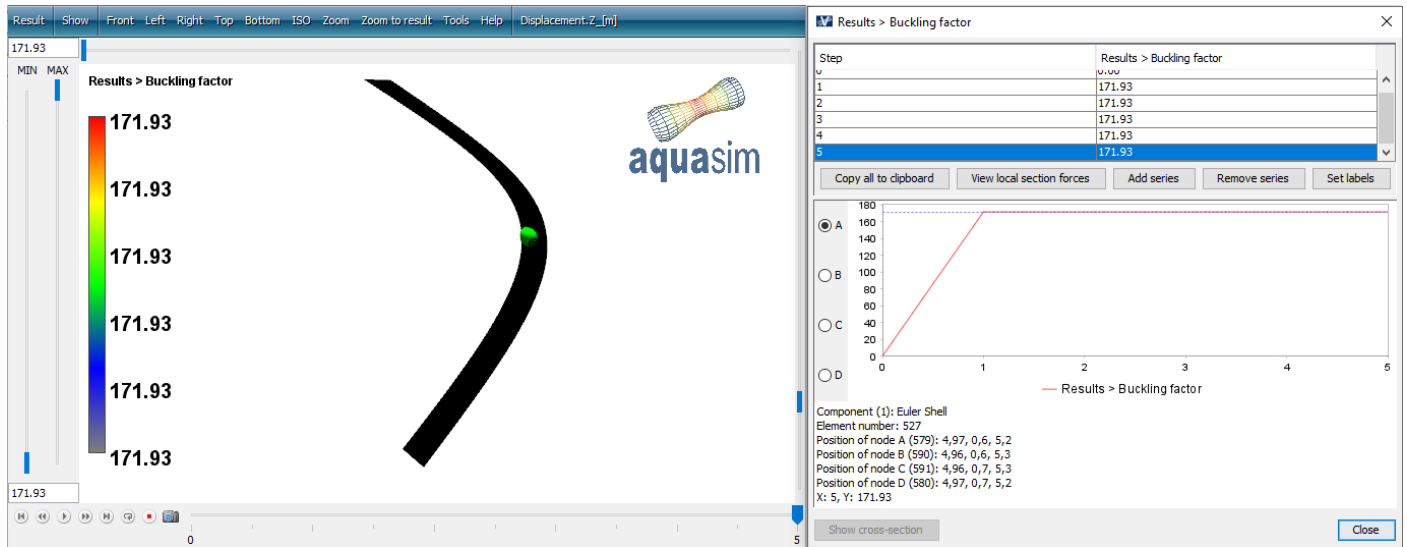


Figure 15

It should be noted that all buckling results in AquaSim is based on the applied element material stiffness matrix and the geometrical stiffness. Euler beams have a stiffness and a geometrical stiffness which is exact relative to the critical buckling load  $P_{cr}$ . Shell elements have a numerical formulation both for element stiffness and geometric stiffness, and results will depend on that.

## 4.5 Buckling analysis of beam modelled with type Normal with bending stiffness

A third version of the beam has been investigated. In this case, the beam is modelled with component MEMBRANE X and type Normal with bending stiffness. You may apply the AquaSim model *Euler-normal.amodel* that is associated with this tutorial.

The thread diameter is set such that the cross-sectional area of the panels corresponds to 0.1 m<sup>2</sup>. Dimensions and material properties are set such that the system is equivalent to the model with beam; **Material properties** and **Bending stiffness** is shown below.

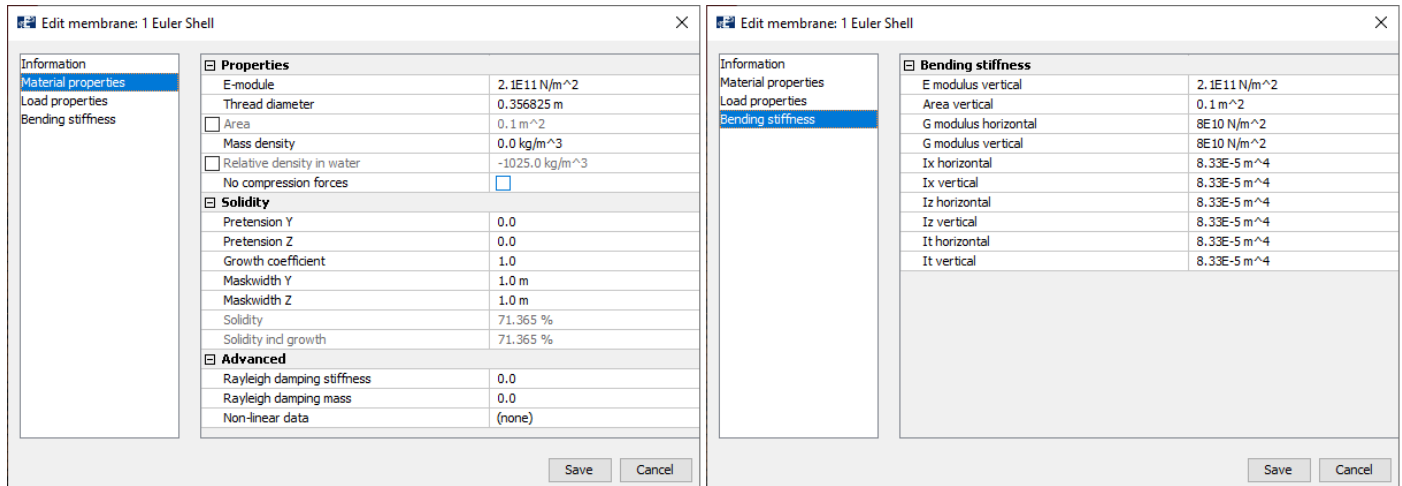


Figure 16

The prepared model can be **Exported** and run as it is. The analysis is named *euler-normal\_*. In the result file *euler-normal\_01.avz*, select **Result > Net > Axial force in horizontal twines [N]** to view the force through the cross section of the model. It corresponds well with the applied **Pointloads** from AquaEdit.

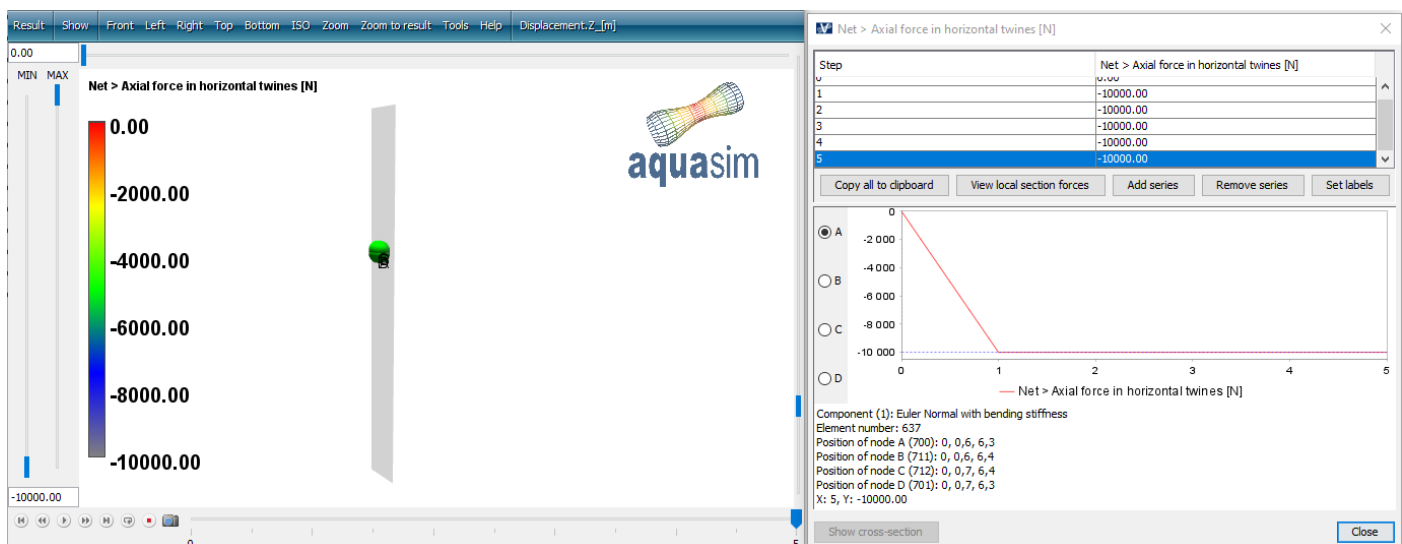


Figure 17

The buckling factor is reported in *euler-normal\_01buckle1.avz*, and is found to be around 172.65. If we multiply this with the compressive load that is applied to the model, we see that also this model correspond well with the calculation of  $P_{cr} = 1727 \text{ kN}$ .

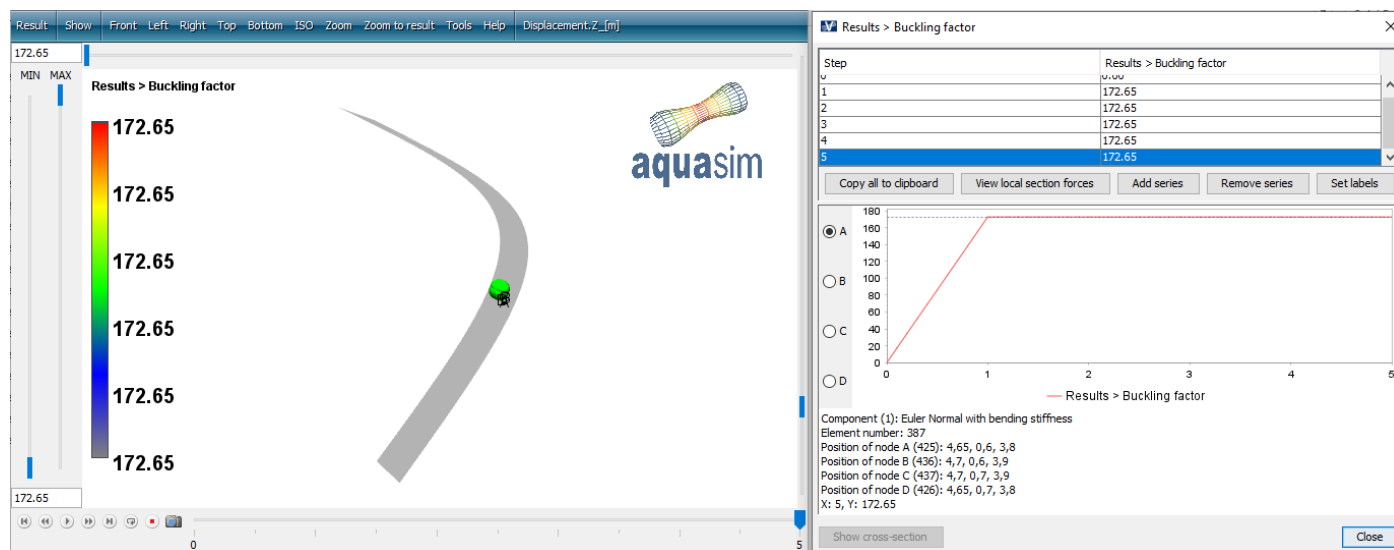


Figure 18

## 5 Summary

In this tutorial you have been introduced to buckling analysis in AquaSim. Buckling analyses can be performed on selected component types. AquaSim account for geometric instabilities.

## 6 Revision comments

Revision no.	Comment
1.0	First publication

--- End of document ---