AquaSim – verification and benchmarking

TR-20000-1755-1

Revision 11

Report no.:	TR-20000-1755-1		
Date of this revision:	07.09.2023		
Number of pages:	58		
Distribution:	Open		
Author:	Frida Klaussen	Keywords:	Verification, benchmarking, AquaSim

Summary:

With support from the Norwegian Research Council, the simulation and analysis software tool AquaSim has been developed over a 20-year period. The report outlines several benchmarking tests to verify the accuracy and capabilities of the software tool.

Revision 2: all verification tests have been re-run to validate the 2.10- and 2.11-release of the AquaSim solver.

Revision 3: all verification testes have been re-run to validate the 2.12-beta version and 2.12-release version of the AquaSim solver.

Revision 4: all verification tests have been re-run to validate the 2.14- beta and -release version of the AquaSim solver.

Revision 5: all verification tests have been re-run to validate the 2.14_191213-beta version (dated 16.12.2019) solver, and then the 2.14_200110-beta version (dated 10.08.2020). The latter is the 2.15-beta candidate for release of 2.15.

Revision 6: all verification tests have been re-run to validate the 2.16-beta candidate for release of the AquaSim solver. The results are all identical to the 2.15-release version of the solver. Minor changes to the report are done.

Revision 7: all verification tests have been re-run to validate the solver 2.16.2-beta (dated 27.08.2021). Visual changes in the report, and text modifications are done. Density of air is changed from 1.21kg/m^3 to 1.27kg/m^3, this affects results in chapter 8. New analytic results are calculated and AquaSim results compared. This report is also regarded as valid for solver 2.16.2 (dated 13.09.2021). Justification is found in chapter 2.

Revision 8: all verification tests are re-run to validate release-candidate 2.17.0 (dated 06.01.2022) of the AquaSim solver.

Revision 9: all verification tests are re-run to validate release-candidate 2.17.2 (dated 27.05.2022) of the AquaSim solver.

Revision 10: all verification tests are re-run to validate release solver 2.18.0 if the AquaSim solver.

Revision 11: all verification tests are re-run to validate release solver 2.18.1 if the AquaSim solver.

9	16.06.2022	ISH	Solver 2.17.2
10	01.03.2023	ISH	Solver 2.18.0
11	17.08.2023	FK	Solver 2.18.1
Revision no.	Date	Author	Description

Content

1 Executive summary

AquaSim has been developed over a 20-year period with support from the Norwegian research council. The program can carry out static as well as dynamic time domain simulation of structures exposed to time varying loads, and structure response such as current and wave loads as well as operational conditions. The program accounts for the hydro-elastic relation between fluid and structure.

Several element types are included in the program. This report validates many of the different elements by doing analysis for cases where analytic solutions can be derived. The loads from waves and current are described and validated. Node properties are validated.

2 Scope of this report

This report is issued in order to verify the software tool AquaSim, developed by Aquastructures. AquaSim is applied for calculating structural response of systems and structures in a marine environment.

Analyses and calculations applying AquaSim are compared to handbook formulas and analytical solutions for the elements used, loads and boundary conditions.

This report was fist issued in 2006, see (Aquastructures, 2006). Recalculation of all the test cases was done in 2012 by Line Heimstad and Are Berstad, this version also includes new validation cases. New recalculations were done by Marie Salthaug Fævelen and Fredrik Mürer in 2014 and 2015 applying the AquaSim solver version 2.10 and 2.11 respectively. This revision also includes new validation cases.

Revision 6: All test cases have been reanalysed with the AquaSim solver version 2.15 in 2020 by Katarina Staalesen. Several cases that were used in earlier revisions are reintroduced in this version. In 2021, all test cases were reanalysed applying the 2.16 (dated 14.01.2021) version of the AquaSim solver, work was done by Katarina Staalesen.

Revision 7: Recalculation of all test cases were done in 2021, applying the AquaSim solver version 2.16.2 (dated 27.08.2021). Work was done by Ida Hystad. The density of air is increased from $1.21 kg/m^3$ to $1.27 kg/m^3$. New analytic results are calculated and compared to AquaSim results. A new solver version of 2.16.2 was published 13.09.2021. The difference between these solvers are changes in how the fluid velocity is treated in the drag-term of Impermeable nets. Impermeable nets are not included in this validation program. Validation of the mentioned changes are treated elsewhere. On basis of this, it is concluded that revision 7 of the validation report (applying solver dated 27.08.2021) is also valid for solver dated 13.09.2021.

Revision 8: All test cases have been reanalyzed with the AquaSim solver version 2.17.0 (dated 06.01.2022). This is the release candidate for the major release in 2022. Work was done by Ida Hystad. Results from applying solver 2.17.0 (06.01.2022) is compared with 2.16.2 (13.09.2021), no significant deviances are found.

Revision 9: All test cases have been reanalyzed with the AquaSim solver version 2.17.2 (dated 27.05.2022). This is the release candidate for minor release in 2022. Work was done by Ida Hystad. Results from applying solver 2.17.2 is compared with 2.17.0 and analytical calculations, no significant deviances are found.

Revision 10: All test cases have been reanalyzed applying the AquaSim release solver 2.18.0. This is the solver included in major release 30.01.2023. Work was done by Ida Hystad. The wind velocity profile for beam elements with wind type 1 has been changed and are now calculated according to TR-FOU-2328-4 Spectral wind loads in AquaSim Ch. 2.1.1. This will cause differences in results when comparing with previous released solver 2.17.2. Ch. 8 Wind loads is updated according to new implementations. It is also noted that calculation of torsion moment due to wind loads are not included in this solver release.

Revision 11: All test cases have been reanalyzed with the AquaSim solver version 2.18.1. This is the solver included in major release 17.08.2023. The work was done by Frida Klaussen. The main difference is that this new version uses a different set value. In addition, the calculation of torsion moment has been reintroduced, as it was not included in the previous version.

3 Benchmark cases static element response

Case studies have been carried out to verify the results calculated by AquaSim. The results are compared to analytical results.

3.1 Truss elements

Truss elements can only take axial loads and are hence applicable for e.g., mooring lines.

3.1.1 Truss element with clamped ends

Based on geometry considerations, the truss element used in the computer program has been tested against analytical results. The geometry of this first case is shown in Figure 1.

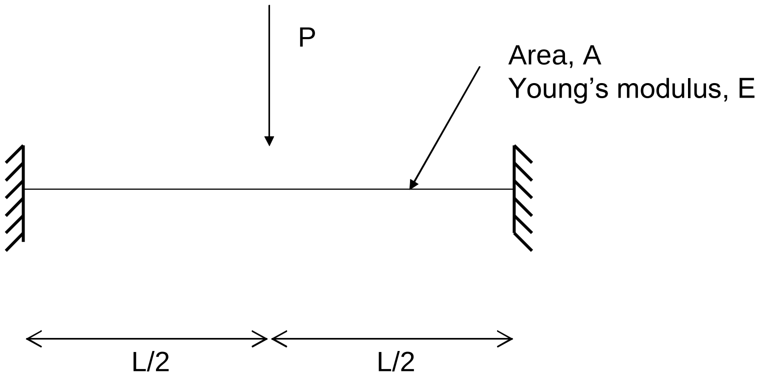

Figure 1 Cable with both ends clamped

Figure 2 presents the deformed geometry of the cable. The material and load data used in the calculation is given in Table 1. AquaSim results and analytical results are compared in Table 2.

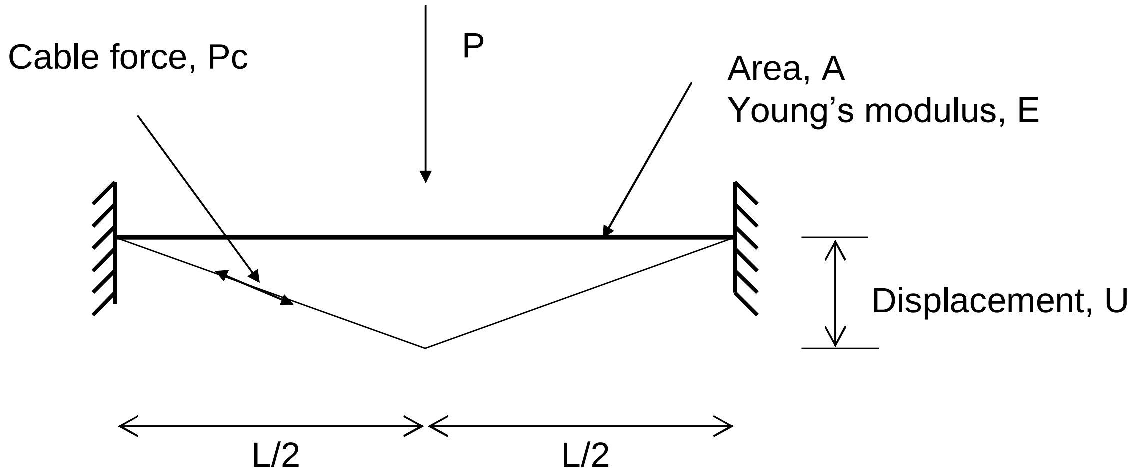

Figure 2 Cable in initial and deformed condition. U is the cable displacement under the point load P, and Pc is the cable force

Table 1 Structural data, cable with both ends clamped

Structural data	Abbreviation	Value
Length of cable from end to end	L	10 m
Cross sectional area of cable	A	100 mm
Modulus of elasticity of cable	E	2.1E9 N/m^2

Table 2 results for cable. U is the displacement in meters and Pc is the cable force in Newton

Applied load	AquaSim results		Analytic results		Difference [%]	
	U [m]	Pc [N]	U [m]	Pc [N]	U	Pc
-100 N	0.391	641.270	0.391	641	0	0.270
-10 000N	1.870	14252.490	1.873	14252	0	0.490
-1 000 000N	16.690	521900	16.695	521943	-0.005	-0.430

As seen from Table 2 the program calculation corresponds well with analytical results. This means that AquaSim calculates geometric nonlinearities of truss elements in a proper manner.

3.1.2 Two crossing cables

A case has been considered where two crossing cables have been modelled as shown in Figure 3. Both cables have the same length, and the load is applied to the centre point.

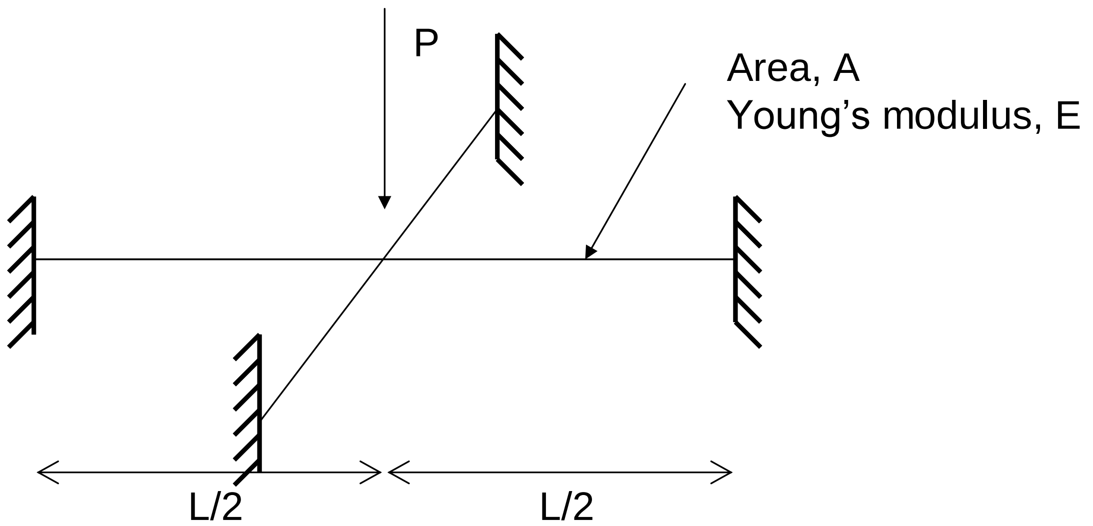

Figure 3 Case with two cables crossing each other with 90deg between the cables

Results for the geometry shown in Figure 3 were calculated with structural data as given in Table 3. The cables have the same properties as the cable in Figure 1.

Table 3 Structural data, case with two cables crossed 90deg

Structural data	Abbreviation	Value
Length of cable from end to end	L	10 m
Cross sectional area of cable	A	100 mm
Modulus of elasticity of cable	E	2.1E9 N/m^2

Results from AquaSim are compared to analytical results in Table 4. Analytical results are derived by considering the load situation in the case of Section 3.1.1. With two times the load applied to the centre point, the displacement U of the crossing cables will be equal to the displacement found for the case of a single cable in Section 3.1.1.

Table 4 Results for crossing cables. U is the displacement in meters and Pc is the cable force in Newton

Applied load	**AquaSim results**		**Analytic results**		**Difference [%]**	
	U [m]	Pc [N]	U [m]	Pc [N]	U	Pc
-200 N	0.391	641.270	0.391	641	0	0.270
-20 000N	1.870	14252.490	1.873	14252	-0.300	0.490
-2 000 000N	16.690	521900	16.695	521943	-0.005	-0.430

As seen from Table 4 multiplying the applied loads with 2 compared to the applied loads in the case of Section 3.1.1 gives equal results for the two cases.

3.1.3 Cable with axial tension load

An axial force is applied to the end of a vertically hanging cable as shown in Figure 4.

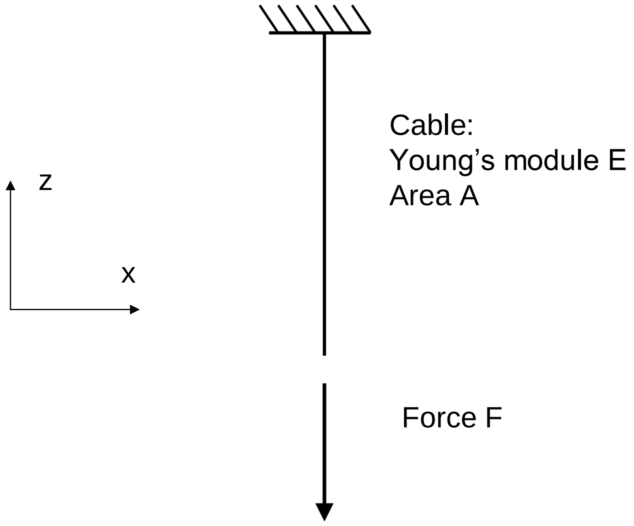

Figure 4 Cable with applied axial force

The structural data for this case is given in Table 5.

Table 5 Structural data for the vertically hanging cable

Structural data	**Abbreviation**	**Value**
Cross sectional area of cable	A	10 mm^2
Length of cable in original configuration	L_0	10 m
Youngs modulus of the cable	E	1.0E8 N/m^2

The results for the cable with applied axial force are given in Table 6. The analysis was done using both 3 and 15 elements for the cable.

Table 6 Results from analytic calculations and AquaSim results for the cable with applied axial force

Applied force in z-direction	**Displacement calculated with AquaSim [m]**	**Analytical displacement [m]**
-100 N	-1.0	-1.0
-1000 N	-10.0	-10.0
-3000 N	-30.0	-30.0

As seen from Table 6 analytical and AquaSim results are identical. The results correspond perfectly for cable with both 3 and 15 elements.

3.1.4 Net built up by truss elements

A net structure as shown in Figure 5 has been established.

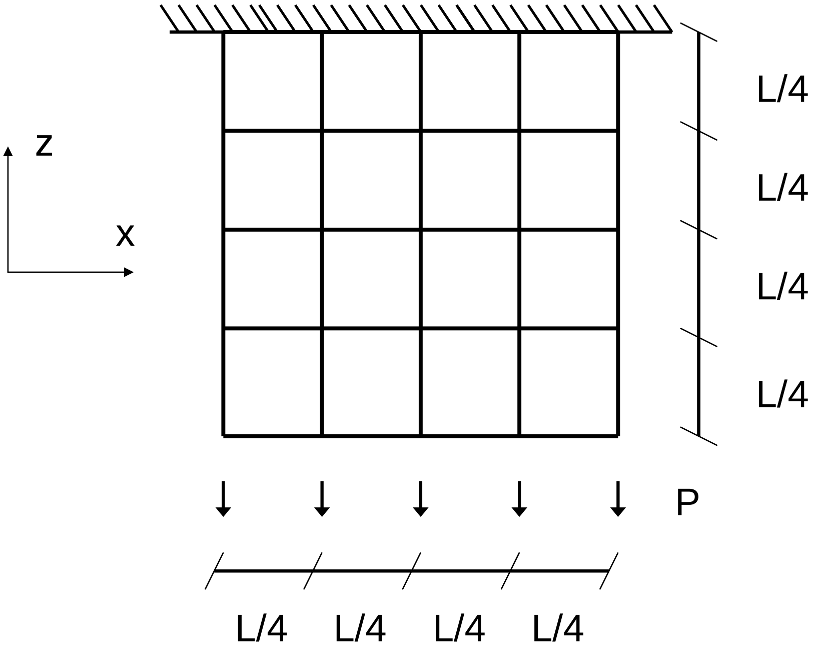

Figure 5 Net structure built up by truss elements

The structural data for the net built up by cable elements are given in Table 7.

Table 7 Values used in net structure built up by cable elements

Structural data	Abbreviation	Value
Cross sectional area of horizontal and vertical cables	A	10 mm^2
Length of cable in original configuration	L_0	10 m
Youngs modulus of the cable	E	1.0E8 N/m^2

Node loads have been distributed on the 5 lowermost nodes as shown in Figure 5. The five upper nodes are all fixed. The results for the net structure built up by cable elements are given in Table 8. AquaSim and analytical results are equal, and only given once in the table.

Table 8 AquaSim and analytic results. Results are equal for the case presented in Section 3.1.3

Node load, each node	Vertical displacement bottom nodes [m]		Horizontal displacement bottom nodes [m]	
	AquaSim	Analytic	AquaSim	Analytic
-100 N	-1.0	-1.0	0.0	0.0
-1000 N	-10.0	-10.0	0.0	0.0
-3000 N	-30.0	-30.0	0.0	0.0

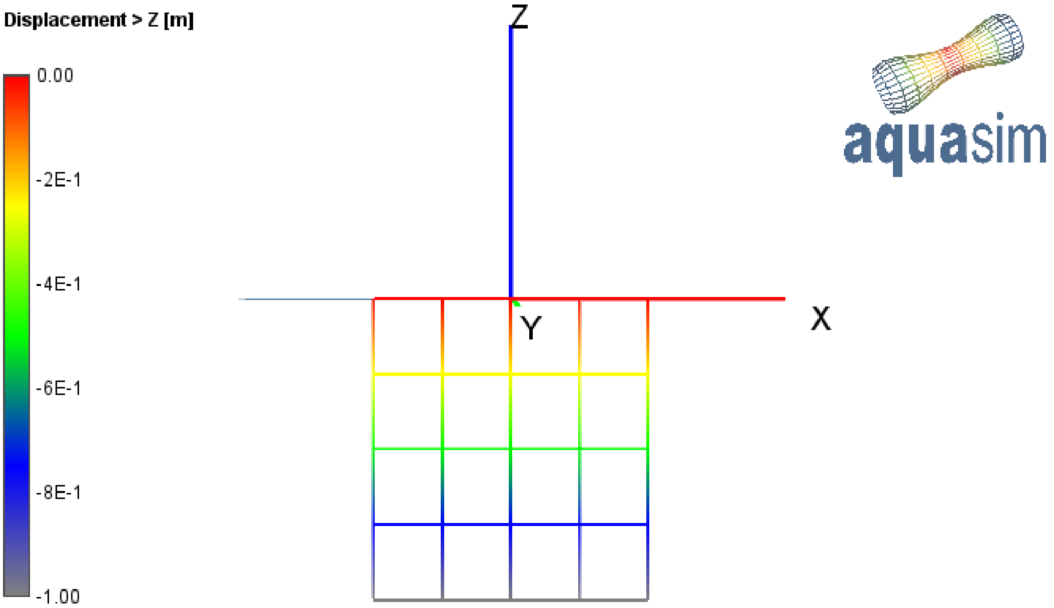

Figure 6 Load of 100 N downwards at each of the five lower most nodes. The legend shows the vertical displacement

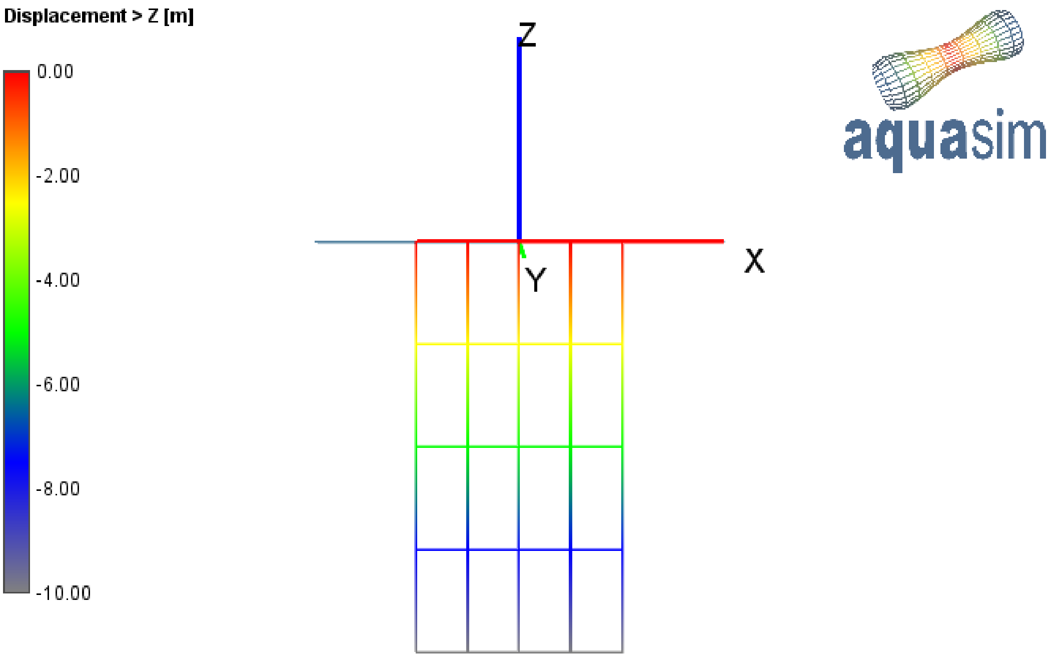

Figure 7 Load of 1000 N downwards at each of the five lower most nodes. The legend shows the vertical displacement

Figure 6 shows the vertical displacement of the net structure for an applied load of 100N. Figure 7 shows the deformation of the net structure when the applied load on each node is 1000N. The maximum elongation in this case is 10 meters, which corresponds well with the analytical results.

A second load case was applied to the net, including a load component in the horizontal y-direction. Load values, analytic solution and AquaSim results are given in Table 9. Figure 8 shows the displacement of the net due to the applied loads.

Table 9 Calculated and analytic results presented

Vertical node loads	**Horizontal node loads**	**Vertical displacement bottom nodes [m]**		**Horizontal displacement bottom nodes [m]**	
		Analytic	AquaSim	Analytic	AquaSim
-800 N	**600 N**	-6.0	-6.0	12.0	12.0

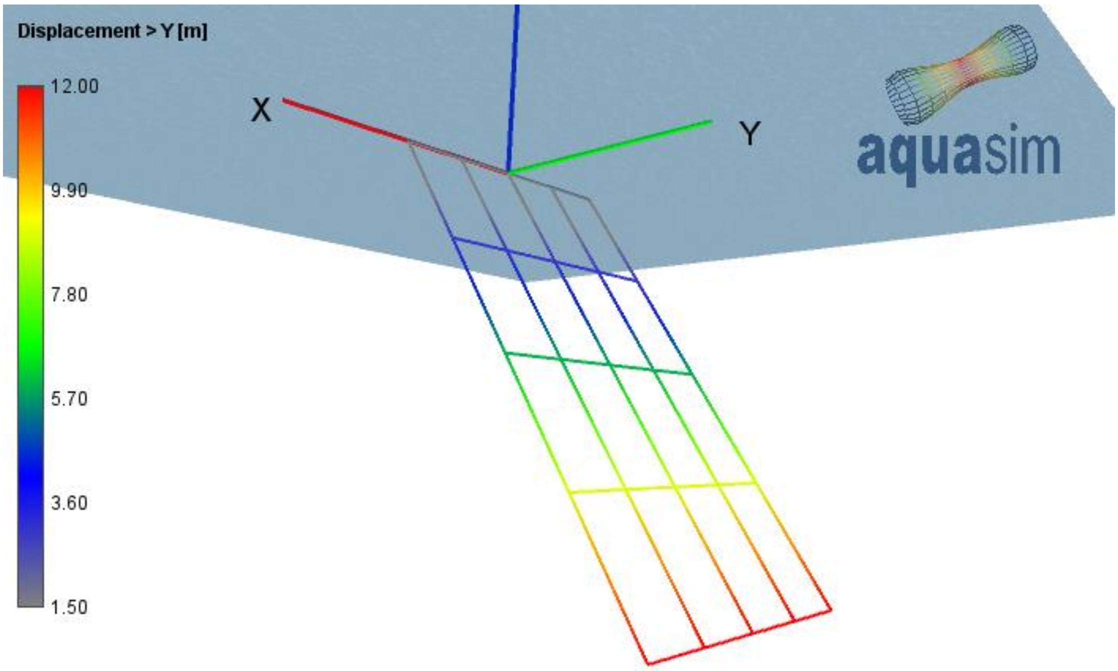

Figure 8 Vertical and horizontal loads at the bottom of the structure. The legend reflects the horizontal displacement of the cables of the net

3.1.5 Net built up with truss elements, 2nd boundary condition

The same structure of cables, as given in Figure 5, is now only fixed at the two endpoints at the top, as shown in Figure 9. The vertical load at each node is 1000N.

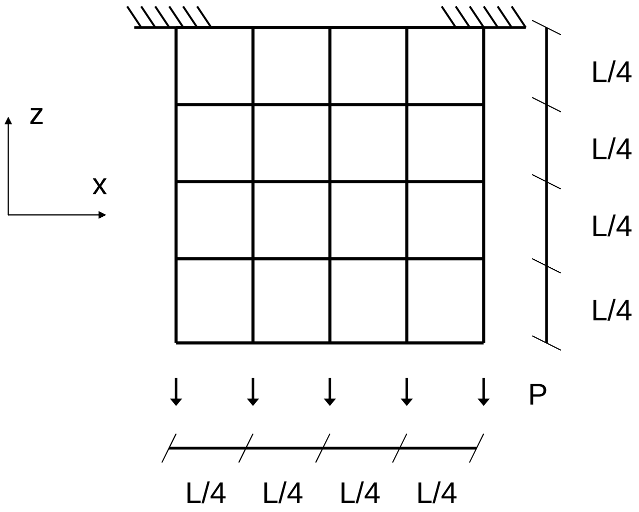

Figure 9 Geometry of net built up of cables, fixed at two nodes at the top

The displacement of the structure of cables with the 2nd boundary condition is shown in Figure 10. The result derived from this 2nd boundary condition is not validated to handbook results. However, it can be shown that the sum of vertical forces at each fixed node is 2500N – half the total force. The sum of horizontal forces at each node is 683N. This is found by extracting the vertical and horizontal force components of the cable elements connected to the node and sum them up. In Figure 10 these are the red and yellow elements connected to the node. The results from this case will be used for validation of the results from an equivalent case with membrane elements in Section 3.3.

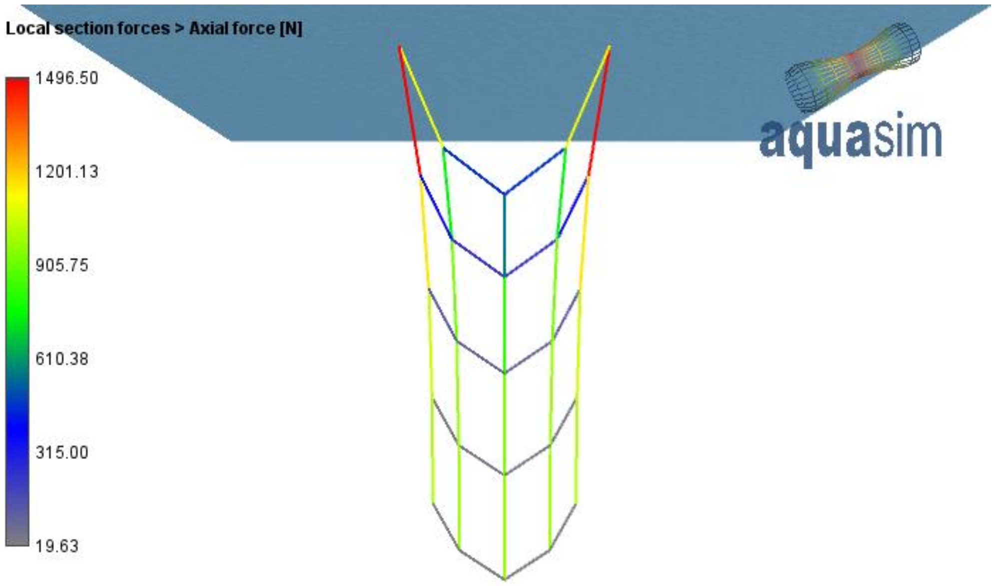

Figure 10 Axial force in the cables for the case with cables only fixed at two nodes, with 1000 N vertical load downwards at each bottom node. The legend shows the values of the axial forces in the cables

3.2 Beam elements

3.2.1 Cantilever beam

A beam element clamped at one end is considered. The case is shown in Figure 11.

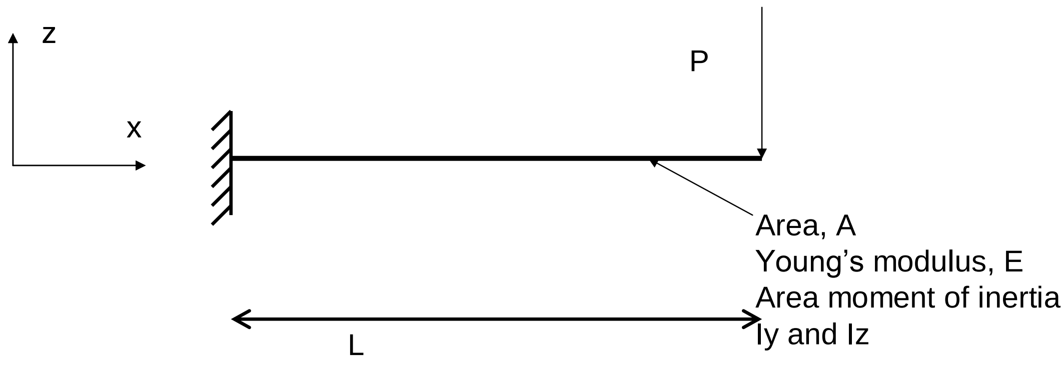

Figure 11 Beam element clamped at one end. A point load is applied at the free end

In this case, analytical linear displacements can be derived as:

$$r_3 = \frac{Pl^3}{3EI}$$

Equation 1

The structural properties for this case are given in Table 10.

Table 10 Structural properties of the clamped beam

Abbreviation	Description	Value
A	Cross sectional area of beam	0.05 m^2
L	Length of beam on original configuration	10 m
I_Y	Area moment of inertia, about local y-axis	0.001 m^4
I_Z	Area moment of inertia, about local z-axis	0.001 m^4
I_T	Torsional moment of inertia	0.002 m^4
G	Torsional modulus	0.8E9 N/m^2
E	Young's modulus	2.1E11 N/m^2

A point load of varying magnitude has been applied as shown in Figure 11. Results from AquaSim and analytical results are given in Table 11.

Table 11 AquaSim- and analytical results for one-element-model

Vertical node load	**Displacement calculated by AquaSim [m]**		**Analytical calculated displacement [m]**	
	z-direction	x-direction	z-direction	x-direction
-100 N	-1.587 E-4	1.508E-9	-1.587 E-4	0.0
-100 000 N	-0.1587	-0.0015	-0.1587	0.0
-1 000 000 N	-1.5500	-0.1446	-1.5870	0.0
-10 000 000 N	-7.0600	-3.7500	-15.8700	0.0

As seen from Table 11 results are almost exactly similar for small loads. This is expected since the beam response is almost linear in this case. As the load increases, the nonlinear effects become important, and the analytical results are no longer correct. AquaSim accounts for the nonlinear effects. The displaced shape of the beam when exposed to the highest load – 10^7 N – is given in Figure 12.

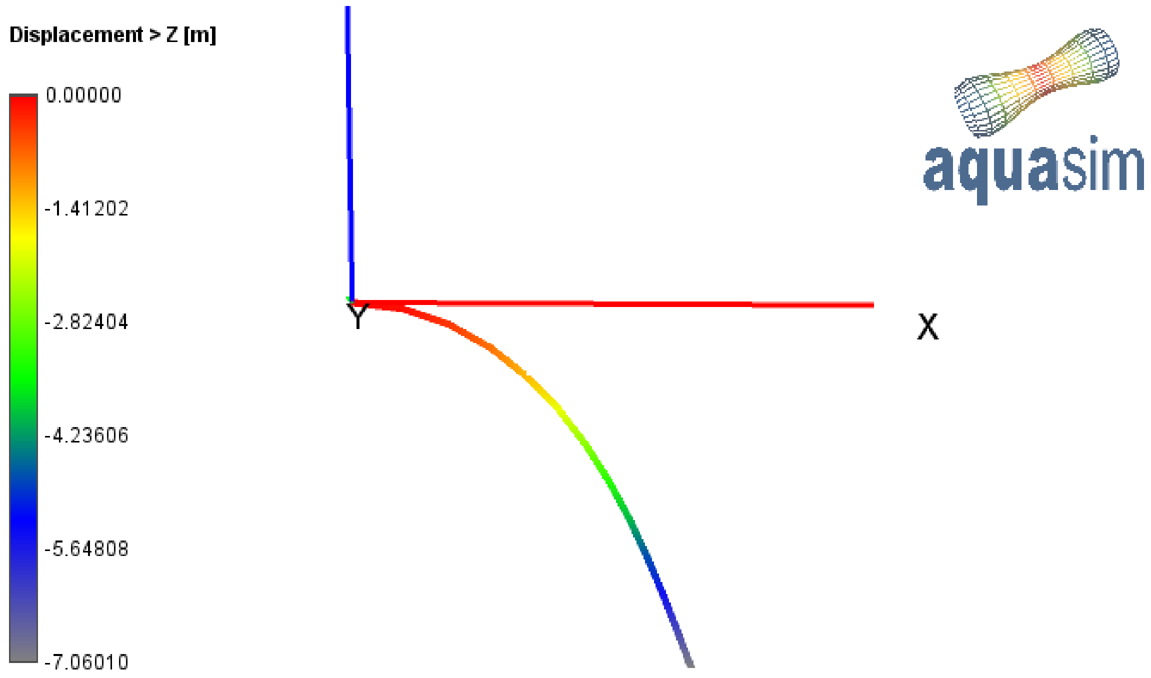

Figure 12 Displacement of cantilever beam with pointload of 1E7 N – nonlinear behavior

3.3 Membrane elements

In the following chapter case studies for membrane elements are presented. Validation of membrane elements are also found in (Aquastructures, 2021c), (Aquastructures, 2020c), (Aquastructures, 2016), (Aquastructures, 2020b) and (Berstad, Walaunet, & Heimstad, 2012).

3.3.1 Membrane elements

One membrane element mesh is shown in Figure 13. In this figure, the membrane element represents a 7 x 7 mesh. Generally, membrane elements represent meshes of m x n, depending on membrane element size and the established mesh size.

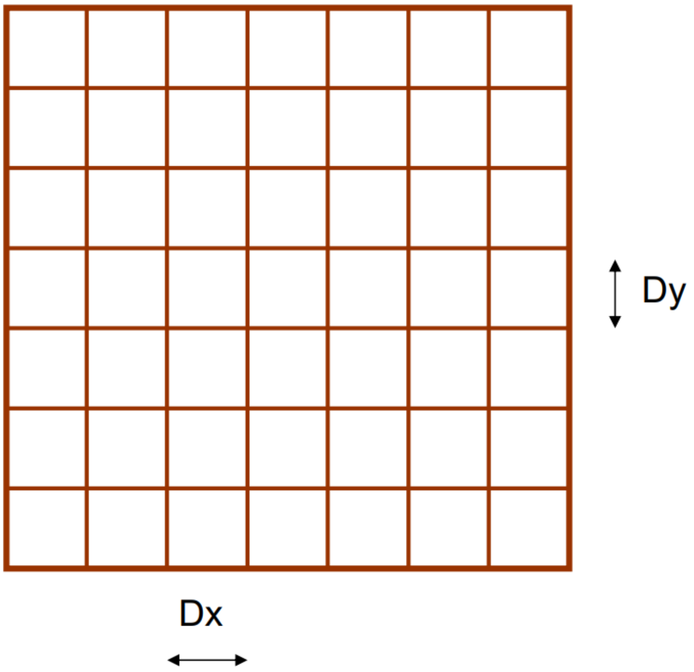

Figure 13 One membrane element representing 7x7 twines of flag shaped net

The mesh structure made up with cable elements shown in Figure 5 and Figure 9 is now "rebuilt" using membrane elements. Results will be compared for the two cases in order to validate the results derived from using membrane elements. Each membrane element is 2.5 x 2.5 meters, and the mesh size is assumed to be the same, meaning the half-mesh size is 2.5 meters. This is not a normal value for aquaculture nets. This is done to obtain a 1 to 1 relation between the net model made of membrane elements and the model built with trusses. This means that each mesh will have a half thread along each side of the mesh. In order to make the model similar to the models shown in Figure 5 and Figure 9 , a cable is arranged around the structure as shown in blue in Figure 14.

The membrane model was first tested with the same boundary conditions as given in Figure 5 i.e., all nodes at the top of the net are fixed. Technical specifications for truss and membrane are given in table below. The load conditions described in Table 8 gave the same results for the membrane case as for the case when the net was modelled by truss elements.

Abbreviation	Description	Value
Young's modulus	E	1.0E8 N/m^2
Thread diameter	d	3.5682E-3 m
Mask width Y	½ Y	2.5 m
Mask width Z	½ Z	2.5 m
Length truss	L	10 m

Analytical solution and AquaSim results are presented in Table 12. This is for the case where all the top nodes in the membrane are fixed.

Table 12 Analytical and AquaSim results. Case study with net structure built up with membrane elements

Node load, each corner	Vertical displacement bottom nodes [m]		Horizontal displacement bottom nodes [m]	
	AquaSim	Analytic	AquaSim	Analytic
-100 N	-9.997E-1	-1.000	0.000	0.000
-1000 N	-10.000	-10.000	1.655E-4	0.000
-3000 N	-30.000	-30.000	01.633E-4	0.000

Now consider the case where the net is only fixed in two upper nodes as shown in Figure 14. This case was run for the same condition as the case shown in Figure 10. The axial force in vertical twines is shown in Figure 15.

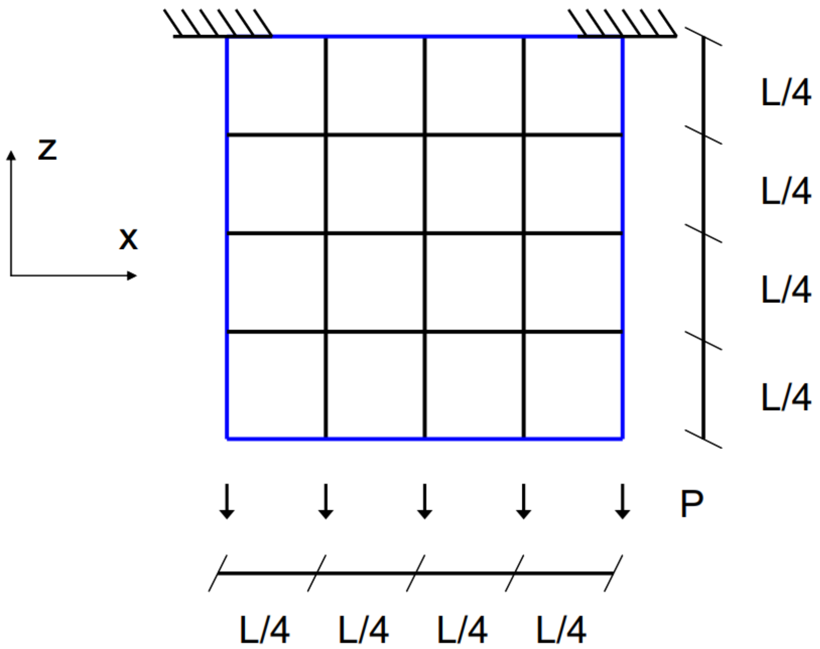

Figure 14 Model for test of the membrane elements. Each square is a membrane element. The blue line corresponds to cable elements with cross sectional area of half the area of one membrane element twine

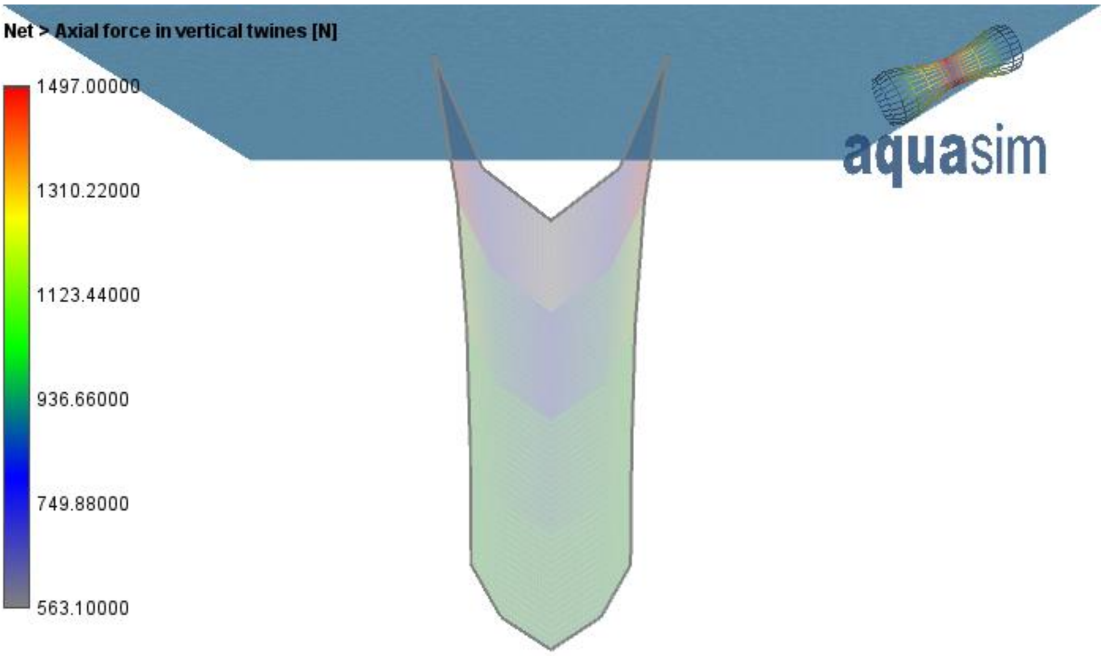

Figure 15 Axial force in vertical twines with the case of two fixed nodes at the top and pointload of 1000N downwards

As seen from Figure 15 the results correspond very well to the results in Figure 10. For the boundary of the membrane element there is only one half twine – and the other half is modelled as a truss element with half the area as the model shown in 3.1.5. The axial forces in the boundary truss frame are shown in Figure 16. This means that cable elements and membrane elements both give the same results for a case study where they represent the same geometry.

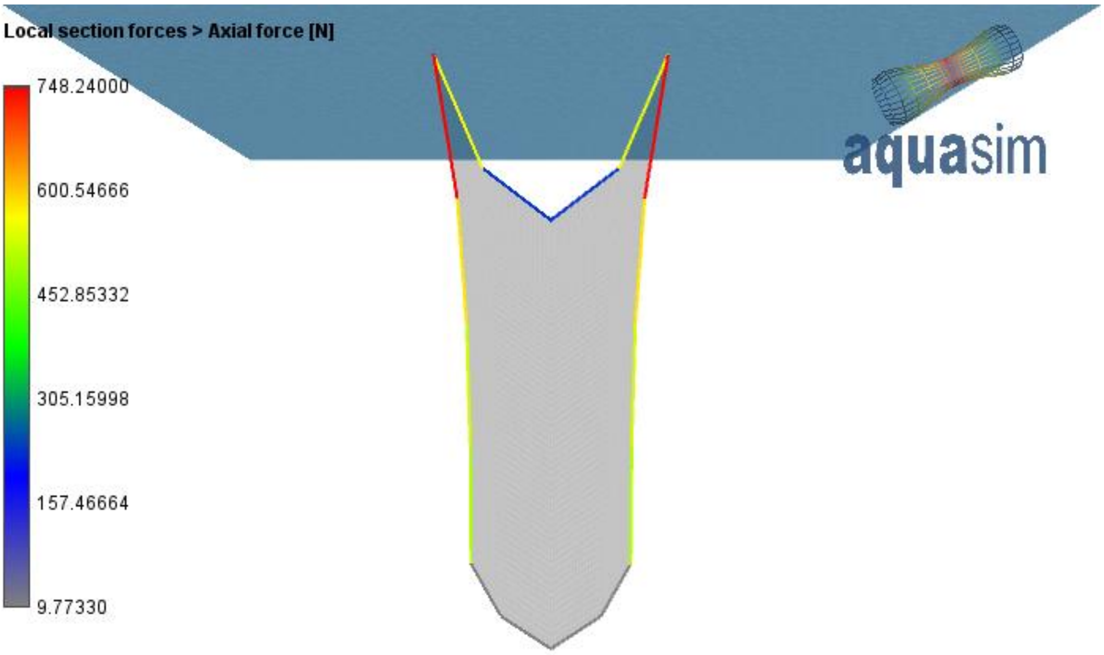

Figure 16 Axial force in boundary truss elements

3.4 Node to node springs

A Node to Node spring (Node2Node) is a component type in AquaSim. This is a spring connected by two nodes. The spring force is proportional to the difference in respective DOFs at the two nodes the spring is connecting. Figure 17 shows a test case where node to node springs have been tested. The beam is the same as in Table 10 of Section 3.2.1.

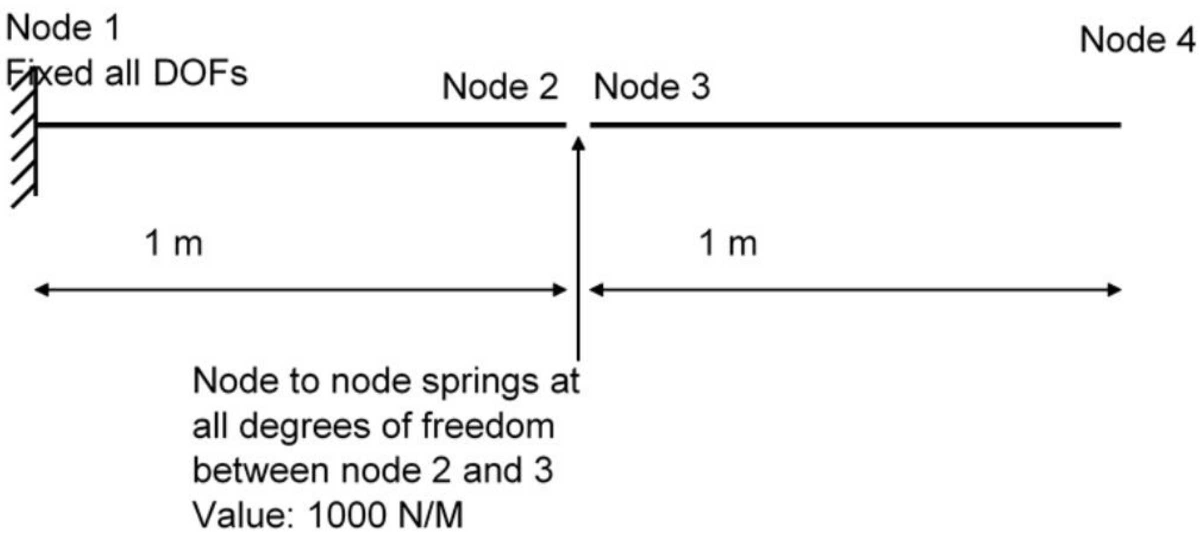

Figure 17 Geometry of beam with Node to Node spring. Beam data is not explicitly expressed in this figure, same beam data as in Section 3.2.1 is applied. The beam is very stiff relative to the stiffness of the spring

Table 13 shows results from the analyses of the beam with Node to Node spring. As the table shows, results using AquaSim corresponds well with analytic results.

Table 13 Results verifying Node to Node spring elements. Node numbers are referring to Figure 17

Applied force/ moment	Force/ moment applied at	Result parameter	Analytic result [m]	AquaSim result [m]
Force X = 1000 N	Node 3 DOF 1	x/y/z-translation at node no. 4	1.0/ 0 / 0	1.0 / 0 / 0
Force Y = 1000 N	Node 3 DOF 2	x/y/z-translation at node no. 4	0 / 1.01 / 0	0 / 1.01 / 0
Force Z = 1000 N	Node 3 DOF 3	x/y/z-translation at node no. 4	0 / 0 / 1.01	0 / 0 / 1.00
Momentum X = 1000 Nm	Node 3 DOF 4	x/y/z-translation at node no. 4	0 / 0 / 0	0 / 0 / 0
Momentum Y= 1000 Nm	Node 3 DOF 5	x/y/z-translation at node no. 4	-4.6 / 0 / -8.42	-4.6 / 0 / -8.41
Momentum Z= 1000 Nm	Node 3 DOF 6	x/y/z-translation at node no. 4	-4.6 / 8.42 / 0	-4.6 / 8.41 / 0

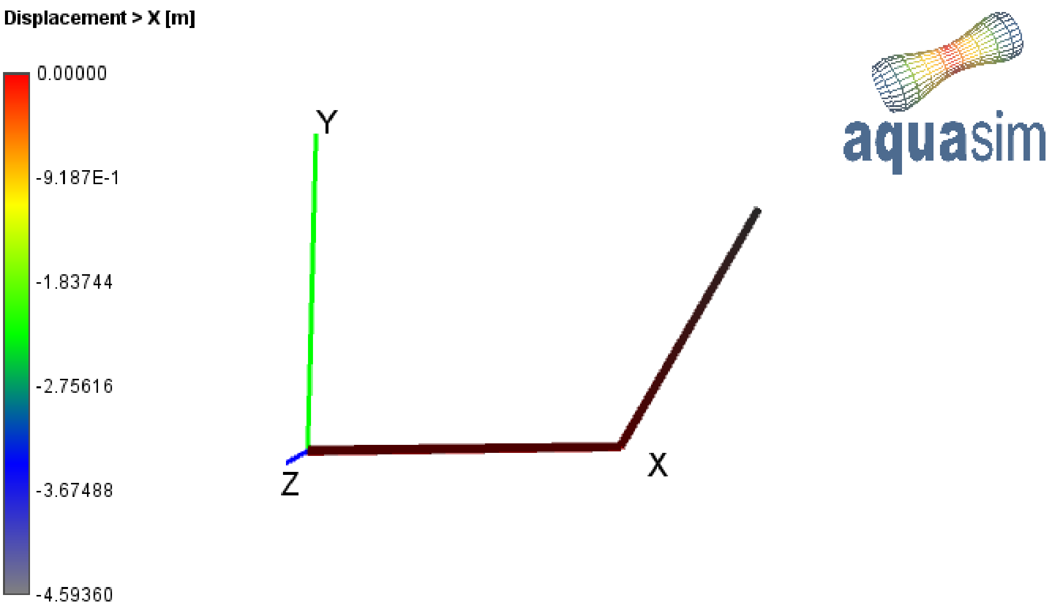

Figure 18 Displacement of beam with elements connected by a Node to Node spring, seen from above. This figure shows the result of a moment applied about z-axis

4 Benchmark cases – element dynamics

4.1 Swinging spring case

The objective of this case is to verify that the load, mass, and stiffness calculation give correct results for a simple case where the eigenfrequency and amplitude can be found analytically. Consider a weight with mass M hanging at the end of a truss element as shown in Figure 19. The end node of the truss element, where the weight is located, is free to move in z-direction only i.e., one degree of freedom.

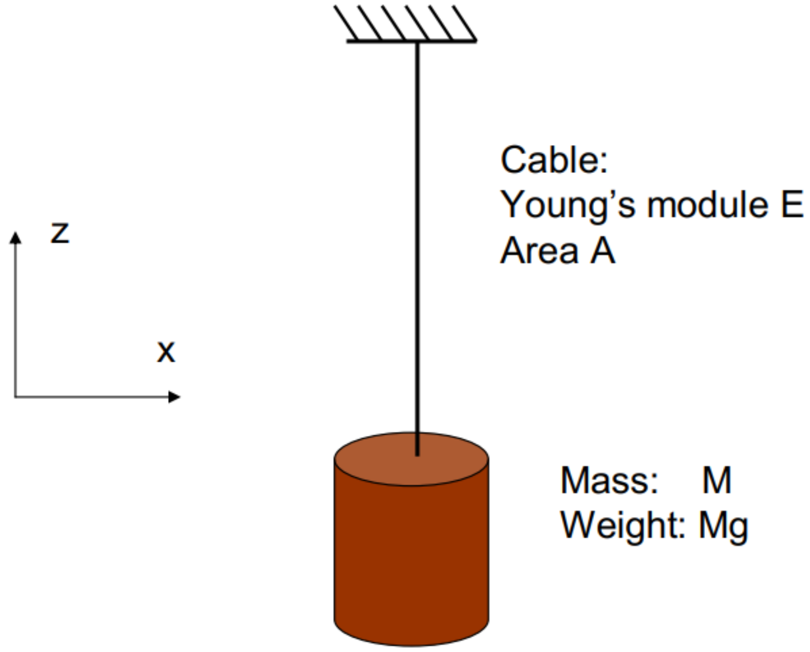

Figure 19 Test of dynamic loading – weight in an axial spring, modelled as a truss element

The following values have been applied to the configuration shown in Figure 19.

Table 14 Structural data in case study with a weight swinging freely

Abbreviation	Description	Value
A	Cross sectional area of the truss	10 mm^2 = 1.0E-05 m^2
L_0	Length of cable in original configuration	10 m
E	Young's module of the truss	1.0E08 N/m^2
M	Mass of weight	305.81 kg
F	The force of the weight = Mg (g = 9.81 m/s2)	-3000 N

4.1.1 Static displacement

Analytic static solution is as follows:

$F = k \cdot \Delta L$, (F is the force in Newton and ΔL is the elongation of L_0)

$k = EA/L_0 = 100$ N/m , (k is the cross-sectional stiffness of the cable)

$\Delta L = L - L_0 = F/k = -3000/100 = -30$m.

This means that the vertical displacement of the cable when the weight is applied statically to the cable is 30 meters. This corresponds well to the values calculated by the program, shown as a blue line in Figure 20.

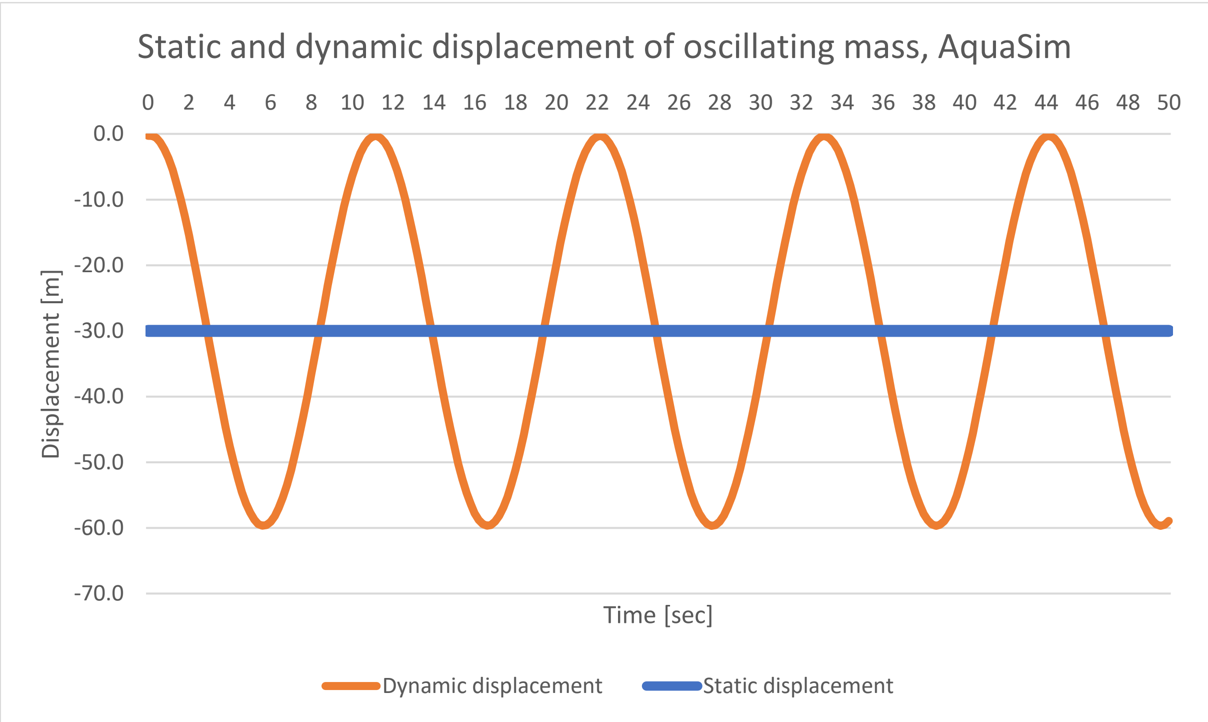

Figure 20 Static and dynamic displacement for the cable with weight. The blue line shows static displacement and the orange line shows dynamic displacement as a function of time

4.1.2 Dynamic displacement

In general, the motion of a system without damping can be describes by

$$r(t) = \frac{\dot{r}_0}{\omega} sin(\omega t) + (r_0 - r_s) cos(\omega t)$$

Equation 2

Applying loads in the present case, $\dot{r}_0$ will be 0 since there is no velocity of the weight at t = 0. $r_0 - r_s$ is the deviance from the position of static equilibrium at t = 0. In the present case this deviance is 30 meters. This means that according to Equation 2 the amplitude of the harmonic motion will be 30, and the motion can be described as:

$r(t) = (r_0 - r_s) \cos(\omega t) = 30 \cos(\omega t)$

Where ω is the eigenfrequency of the motion being found as:

$$\omega = \sqrt{\frac{k}{M}}$$

The analytically calculated period T will in this case be:

$$T = \frac{2\pi}{\omega} = 2\pi\sqrt{\frac{M}{k}} = 2\pi\sqrt{\frac{305.81kg}{100\ N/m}} = 10.99s$$

The analytical value was compared with results from the analysis in AquaSim, shown as an orange line in Figure 20, and matched almost perfectly. A summary of the results is given in Table 15.

Table 15 Analytic and AquaSim results for the case study with weight swinging freely in the z-direction

Abbreviation	Description	AquaSim results	Analytic results
k	Cable stiffness [N/m^2]	100	100
r_s	Static displacement [m]	-30	-30
r_d	Dynamic amplitude [m]	30	30
T	Eigen period [s]	10.99	10.99

4.2 Swinging pendulum

In this case a cable was modelled along the x-axis as shown in Figure 21. A weight was put on one end of the cable, and the other end was fixed. This means that the structure will act as a swinging pendulum.

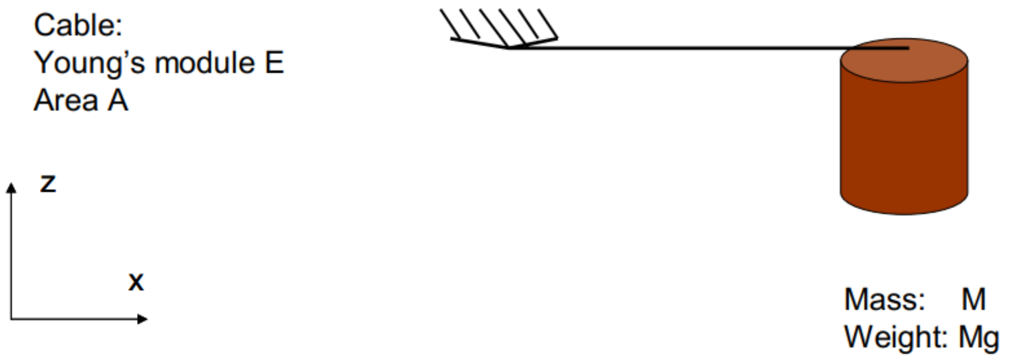

Figure 21 Horizontal cable with weight at the end

The data used for this case is given in Table 16.

Table 16 Structural data for the case study with cable swinging sideways

Abbreviation	Description	Value
A	Cross sectional area of cable	1000 mm^2 = 1.0E-3 m^2
L_0	Length of cable in original configuration	10 m
E	Young's module of the cable	1.0E12 N/m^2
M	Mass of weight	305.81 kg
F	The force of the weight =Mg	-3000 N

The analytic results in this case will give a weight swinging from one side to the other with minimum z- displacement being -10 meters. This corresponds to the AquaSim results shown in Figure 22. So does the horizontal displacement.

The period for the swinging pendulum can be found as:

$$T = 4\sqrt{\frac{L}{g}}\int_0^{\pi/2}\frac{d\varphi}{\sqrt{1-k^2\,sin^2\,\varphi}}$$

Equation 3

where T is the swinging period, L is the cable length and g is the acceleration of gravitation. $k = \sin\frac{\alpha}{2}$, where α is the angle of the pendulum in the original configuration relative to the vertical axis. In the present study α = 90 degrees, giving k = ½. Introducing this into Equation 3, the period T is found to be 7.49 s. Comparing with results found from AquaSim shown in Figure 22, it is seen that the period calculated by AquaSim is approximately the same. Both are abbreviated to 7.49 seconds.

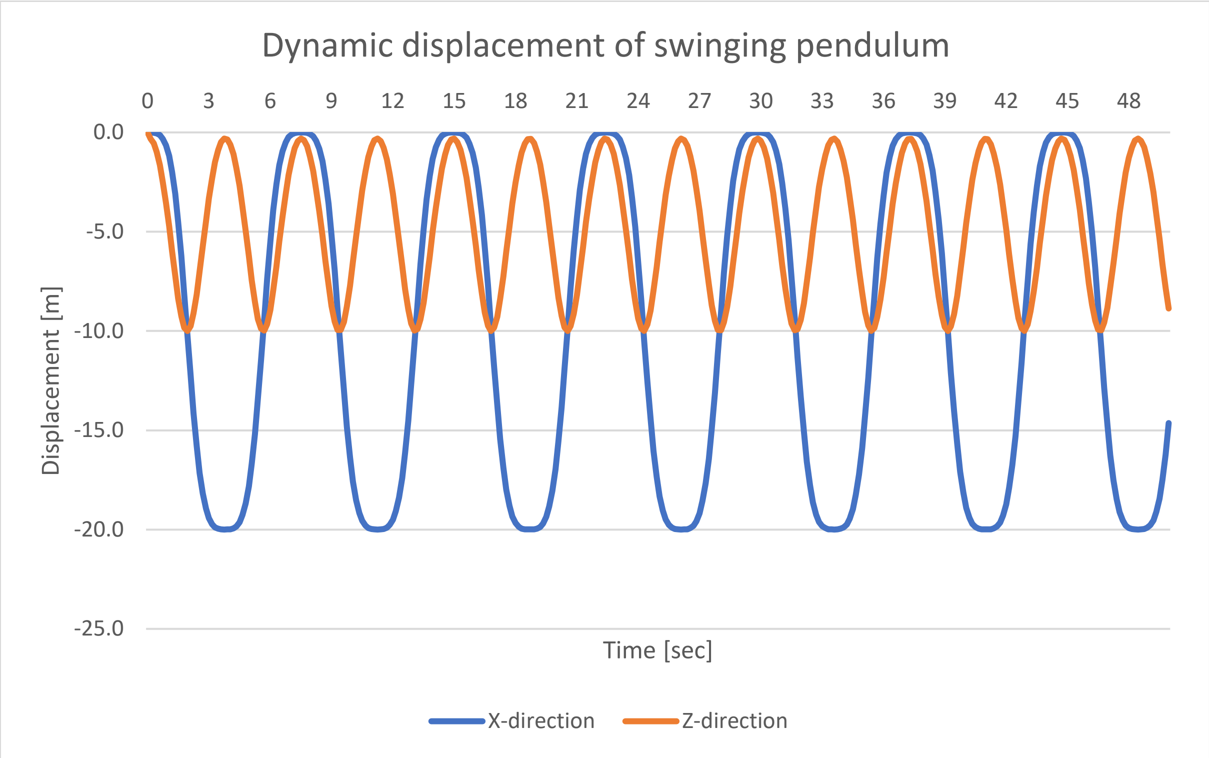

Figure 22 Weight attached to cable end. Cable is swinging sideways. The blue line shows vertical displacement, and the orange line shows horizontal displacement

5 Benchmark cases Morison load application

On beams and cables, the cross-flow principle is used to derive the Morison loads. Referring to a local coordinate system where the beam or cable is located along the local *x*-axis, forces in the local y-direction can be found as given in Equation 4.

$$F_2 = \frac{\rho_w C_{dy} D_N L_0}{2}(u_2 - \dot{v}_2)\sqrt{(u_2 - \dot{v}_2)^2 + (u_3 - \dot{v}_3)^2}$$
$$+\rho_w A L_0 C_{my} a_2 - \rho_w A L_0 (C_{my} - 1)\ddot{v}_2$$

Equation 4

Here C_{dy} is the drag coefficient in the local y- direction. D_N is the diameter of the cross section. $\sqrt{(u_2 - \dot{v}_{2m})^2 + (u_3 - \dot{v}_{3m})^2}$ is the relative velocity vector between the beam and fluid in the cross-sectional plane. $u_2 = u_{2wave} + u_{2current}$ where u_{2wave} is the fluid velocity due to waves and $u_{2current}$ is the current velocity in the local y-direction. a_2 is the fluid acceleration in the local y-direction. $C_{my} = C_a + 1$ is the mass coefficient, where C_a is the added mass coefficient. A is the cross-sectional area of the element. The expression will be similar in the local z-direction.

5.1 Beam exposed to current

Consider a case with a vertical beam exposed to uniform current as shown in Figure 23. The beam has a circular cross section.

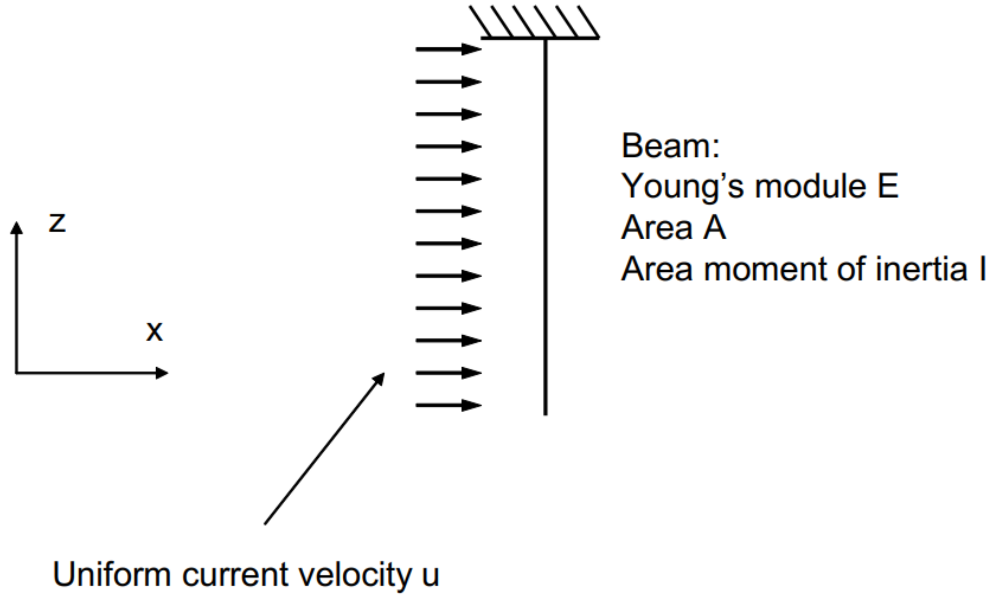

Figure 23 Beam exposed to uniform current

Structural data for the case study is provided in Table 17.

Table 17 Structural data applied for beam exposed to current load

Abbreviation	Description	Value
A	Cross sectional area	0.1 m^2
L	Length of beam in original configuration	10 m
I_y	Area moment of inertia about local y- axis	0.001 m^4
I_z	Area moment of inertia about local z- axis	0.001 m^4
I_T	Torsional area moment of inertia	0.002 m^4
G	Torsional module	4E10 N/m^2
E	Young's modulus of the beam	1.0E11 N/m^2

The distributed load over the cross section can be found by using the Morison equation (Equation 4). Applied to this static case, the equation reads:

$$q = \frac{\rho}{2} C_D D|u|u$$

Equation 5

where q is the uniformly distributed load and D is the diameter of the cross section. The displacement r of the lower end of the beam, and the shear force V_z and moment M_y at the upper end can then be found as:

$$r_{zEnd} = \frac{qL^4}{8EI}, \qquad M_{yUpperend} = \frac{qL^2}{2}, \qquad V_{zUpperend} = qL$$

Equation 6

Some additional data for the beam is given in Table 18.

Table 18 Data for test of beam exposed to current load

Abbreviation	Description	Value
C_D	Drag coefficient	1
ρ	Water density	1025 kg/m^3
D	Diameter of cross section	0.35 m
u	Current velocity	1 m/s
C_m	Mass coefficient	2

Three element configurations for the beam are analysed; one with 10 elements, one with 100 elements, and one with 2 elements where the top element is 0.1 meters and the bottom element is 9.9 meters. Analytical and computed results are compared in Table 19.

Table 19 Analytic and AquaSim results for beam exposed to current load

Response parameter	**Analytic results**	**AquaSim results**		
		With 10 elements	With 100 elements	With 2 elements (0.1m at top, 9.9m lower part)
r lower end [mm]	2.24 mm	1.585	2.242	2.242
M_y upper end	8969 Nm	8954	8969	8790
V_z upper end	1794 N	1704	1785	1785

As seen from Table 19 the results from AquaSim corresponds well with the analytical results. The shear force is computed as constant over each element meaning that the response at the clamped beam end will be “under predicted” proportional to element size. As can be expected, the results are more precise the more elements used in the analysis. When dividing the beam into two elements of different sizes, the moment at the fixed end is found very precisely using *one* short element at the fixed node. It also gives a precise value for the displacement at the bottom of the beam. This configuration works well for this specific case but will not give precise values for every point of the beam. The configuration with 10 and 100 elements will give more precise results over the whole length of the beam.

For this circular cross section, the test is repeated by exposing the beam to a current of 45 degrees. This means that $u_x = u_y = 0.707 m/s$. All other data are the same.

Table 20 Analytic- and AquaSim results for beam exposed to current 45 degrees relative to the x- and y-axis

Response parameter	**Analytic results**	**AquaSim results**	
		With 10 elements	With 100 elements
$r_x = r_y$ lower end	1.59 mm	1.585	1.585
|r| lower end	2.24 mm	2.242	2.242
$M_y = M_z$ upper end	-6342 Nm	-6329	-6340
|M| upper end	8969 Nm	8951	8966
$V_y = V_z$ upper end	1268 N	1205	1262
|V| upper end	1794 N	1704	1784

Applying a current of increasing velocity, from 0 m/s to 3.0 m/s shows the relationship between current velocity and displacement of the beam. Figure 24 shows the displacement of the tip of the beam as a function of the current velocity.

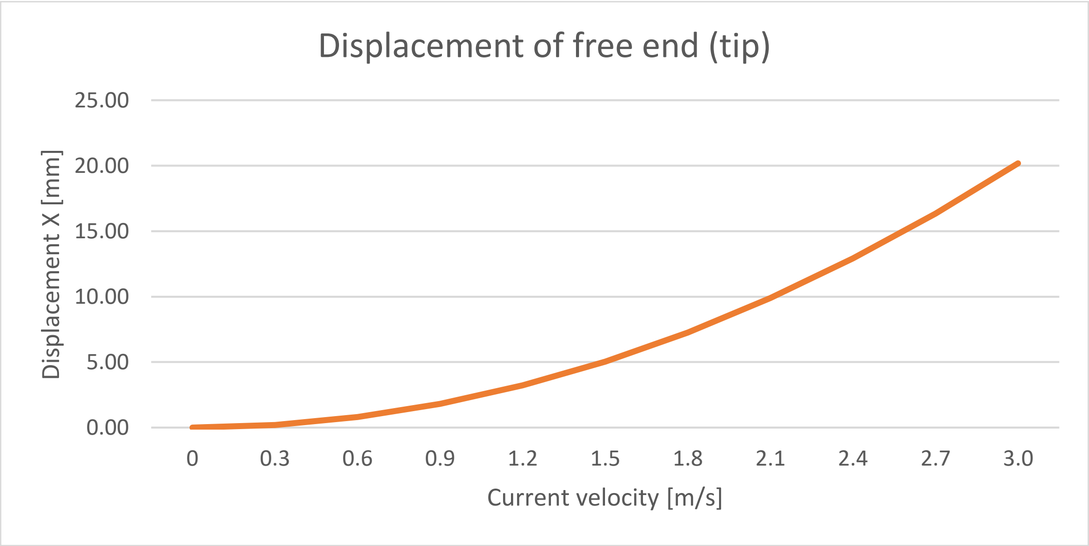

Figure 24 Displacement of beam end as a function of increasing current velocity

5.2 Beam exposed to wind loads

Wave loads are considered, using the Morison formula (see Equation 4). The same beam as described in Section 5.1 is considered (see Figure 18 and Equation 6). In the present case the beam is positioned horizontally, along the y-axis, 5 meters below wave surface. The beam response is assumed static. Infinite wave depth is assumed.

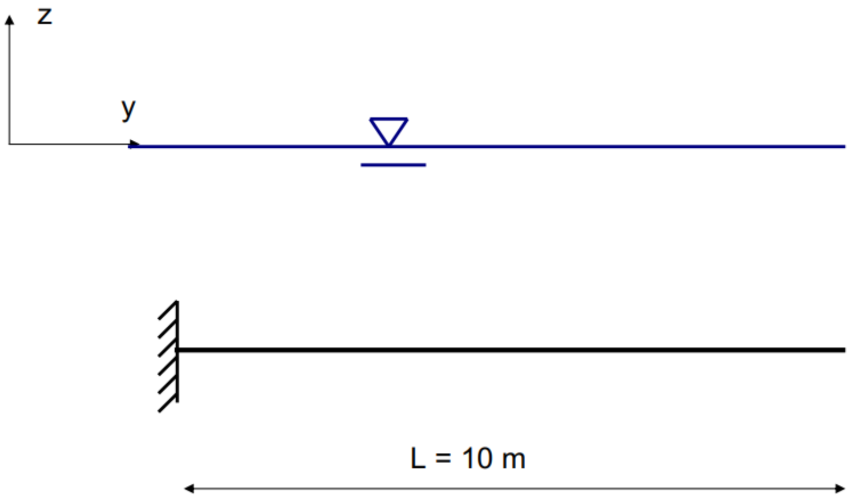

Figure 25 Beam exposed to waves

The beam is exposed to waves in the x-direction with 5 meters wave height. The wave frequency, ω is 1.0 rad/sec. As seen from Figure 26 the program calculated results correspond very well with analytical results calculated according to Equation 4.

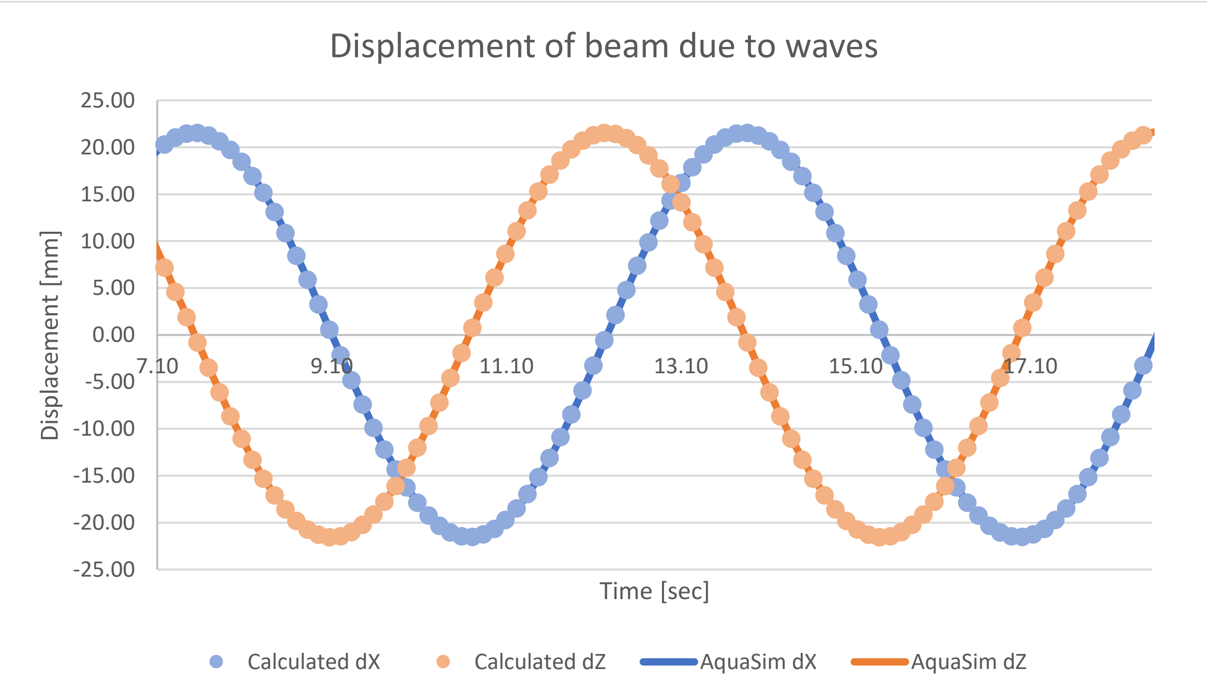

Figure 26 Horizontal beam exposed to waves. Wave elevation is 5 meters, wave frequency ω= 1rad/sec. Displacement at the beam's free end is shown. The graph show the displacement after transient phase in AquaSim

5.3 Beam exposed to wave and current

A similar case is considered applying both current and waves. In this case the wave frequency is 1.0 rad/sec and a current velocity of 1.0 m/s is applied in addition to the wave. It is assumed that the waves are "riding" on top of the current field. As seen from Figure 27, results from AquaSim corresponds well with the analytically calculated results.

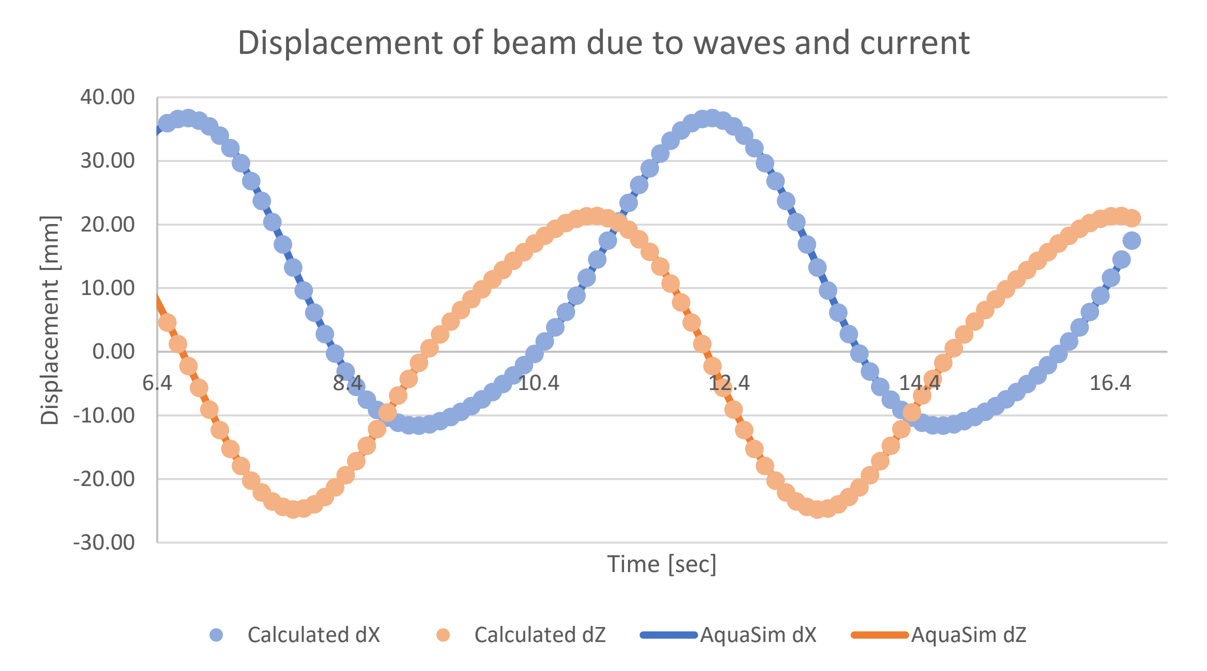

Figure 27 Horizontal beam exposed to waves and current. The wave elevation is 5 meters and the wave period is 6.28 sec. Current velocity is 1 m/s in the x-direction. Displacement at the beams free end is shown. The graph shows the displacement after transient phase in AquaSim

5.4 Horizontal hanging cable exposed to current loads

This case considers a vertical cable with length L = 10 m with a point load of 5000 N applied to the lower end, as shown in Figure 28. The diameter D is 0.35 m and the water density ρ = 1025 kg/m^2. Displacements are calculated from a simplified formula based on moment equilibrium:

$$y = \frac{F_D}{2P}(z^2 + 2Lz)$$

Equation 7

In Equation 7, F_D is the drag force per length unit $F_D = \frac{1}{2}\rho C_D D u^2$. The vertical coordinate z is 0 at the top, negative downwards.

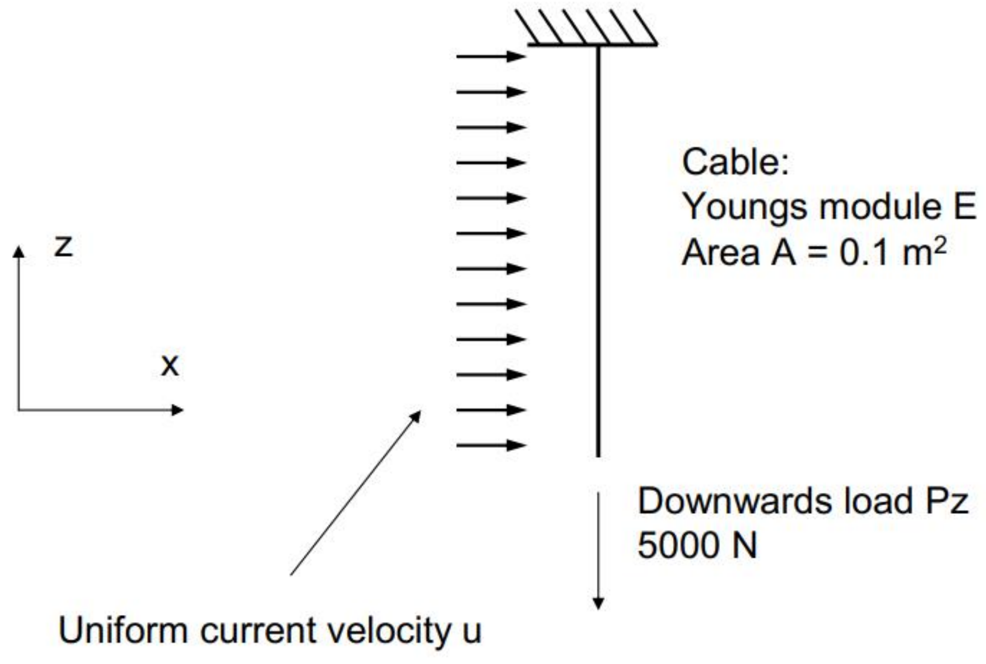

Figure 28 Cable, with a hanging weight, is exposed to current flow

In contrast to the AquaSim simulation program, the simplified formula does not account for the updated geometry of the cable when loads from current is derived. This means that the simplified formula will differ from the nonlinear results calculated by AquaSim as the displacements increase. This is clearly seen in Figure 29. The results predicted by AquaSim and the simplified formula corresponds well when the horizontal displacement is less than 0.2 meters. For this case with current velocity of 0.3 m/s the nonlinear geometry effect is not very important. Also seen in Figure 29 is the case where the current velocity increases from 0.3 m/s to 1 m/s. This gives a displacement of the cable of almost two meters in the horizontal plane. As seen from this figure it is a deviation between results predicted by AquaSim and the simplified formula. This was expected since the simplified formula does not account for the effect of the geometric nonlinearities. Figure 30 shows the displacement of the cable in the 1 m/s current.

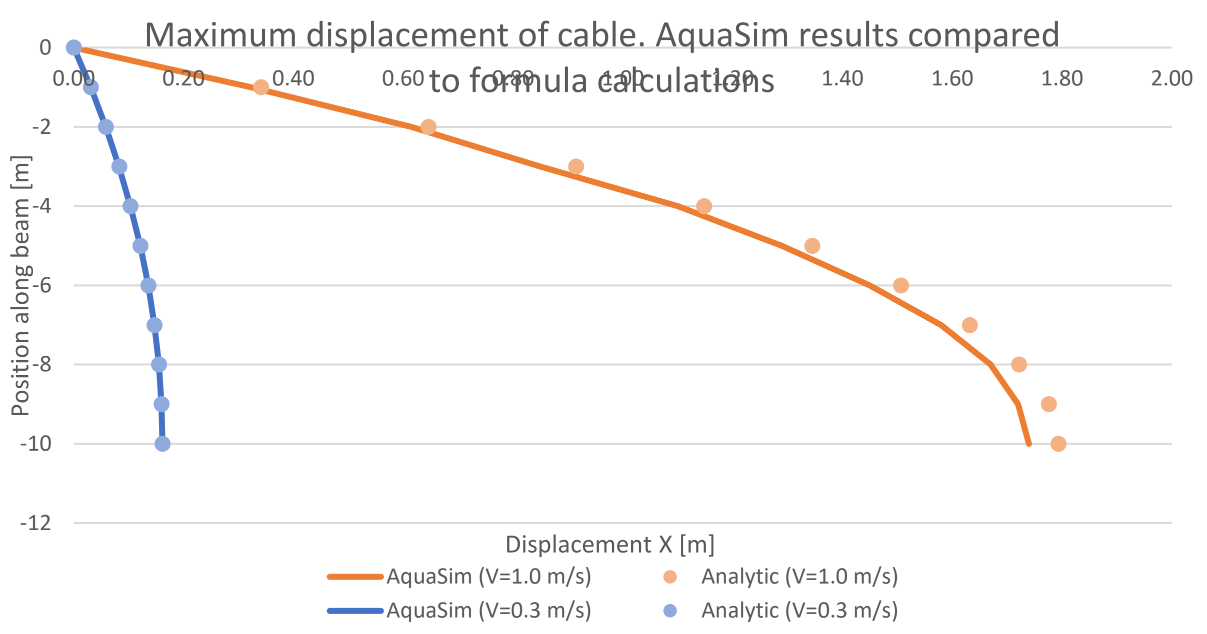

Figure 29 Cable displacement predicted by the simplified formula and AquaSim results. Results are shown for current velocities of 0.3 m/s and 1.0 m/s. The continuous lines are AquaSim results, while dots are the results predicted with the simplified formula

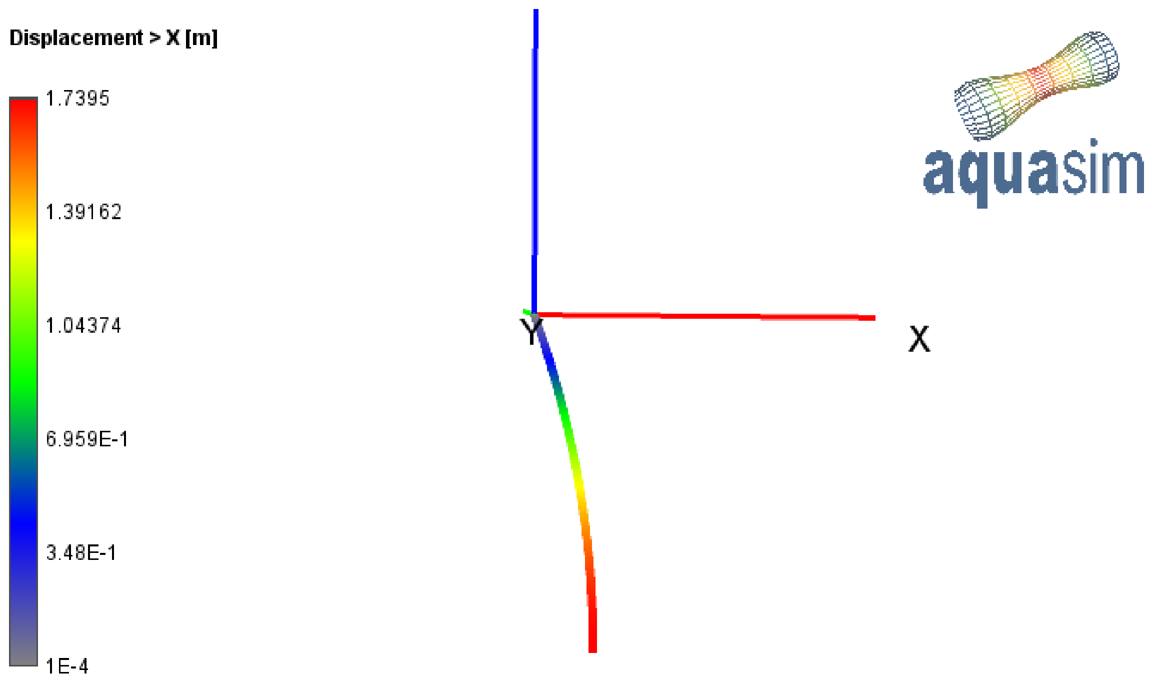

Figure 30 Cable with weight at the bottom, current velocity is 1.0 m/s. The legend gives the horizontal displacement in the cable

6 Benchmark cases hydrodynamic load application

This section shows results for calculations based on hydrodynamic loads. Case studies are used to investigate how the program corresponds with results for cases where asymptotic solutions or textbook solutions exist.

In AquaSim added mass and hydrodynamic damping is established for elements based on linear coefficients established at a user specified mean water line. Diffraction properties are established in the same manner. During time domain simulation, the Froude-Krylov and diffraction of the pressure is applied at the actual horizontal location of the elements.

6.1 AquaSim results compared with the small body asymptote

When a submerged body is small relative to the wavelength, the forces acting on the body can be approximated as (see e.g. (Faltinsen, 1990) pp. 60-62):

$$F_i = -\iint_S p n_i ds + A_{i1}a_1 + A_{i2}a_2 + A_{i3}a_3$$

Equation 8

where p is the dynamic pressure in the undisturbed wave field, $\mathrm{n} = [n_1, n_2, n_3]$ is the unit vector normal to the body with positive direction into the fluid. a_1, a_2, a_3 are the acceleration components along the *x*-, *y*- and *z*-axis of the undisturbed wave field, which are to be evaluated at the geometrical mass centre of the body. For a totally submerged body:

$$-\iint_S p n_i ds = \rho V a_i$$

Equation 9

where V is the volume of the body. For a body not totally submerged, the above equation is valid only in the horizontal direction. A case as shown in Figure 31 is used as a case study. The beam is located with its origin in the free surface (z = 0).

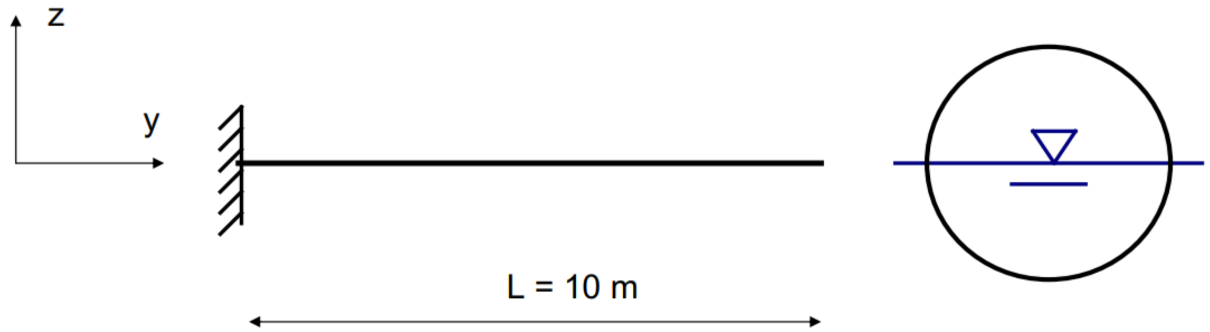

Figure 31 Geometry of the beam in the case study. The diameter of the cylinder is 0.4 meters

A wave with an amplitude of 1 meter and a period of 8 seconds is applied. Based on the long wave approximation, the evenly distributed force amplitude due to Froude-Krylov and added mass can be derived from Equation 8 and Equation 9, and is found to be equal to 79.45 N/m. The force will be harmonic with this as the force amplitude. This force is introduced into the equation for free end displacement for a clamped beam:

$$r_{zend} = \frac{qL^4}{8EI}$$

Equation 10

where q is the sum of Froude-Krylov and added mass force, i.e. $q = 79.45N/m$. Structural and environmental data for this case is given in Table 21.

Table 21 Values used in beam exposed to wave loads, applying numerical strip theory and wave diffraction theory

Abbreviation	Description	Value
A	Cross sectional area	0.12566371 m^2
	Submerged part of circular cross-sectional area	50 %
L	Length of beam in original configuration	10 m
I_y	Area moment of inertia about local y- axis	0.001 ($1/m^4$)
I_z	Area moment of inertia about local z- axis	0.001 ($1/m^4$)
I_T	Torsional area moment of inertia	0.002 ($1/m^4$)
G	Shear module	4E10 N/m^2
E	Elasticity module	1E11 N/m^2
Cd	Drag coefficient	0
β	Wave direction	Beam seas (90^o)
H	Wave amplitude	1.0 m
T	Wave period	8.0 s

Using the beam properties for this case gives a sinusoidal response shown in Figure 32. In this figure, analytic results using the small body approximation is compared to the results predicted by AquaSim which use a strip theory panel method. As seen from the figure, predicted response corresponds well. Note that beam response is calculated statically. A comparison plot of the results from small body approximation and results obtained with AquaSim analysis is shown in Figure 32.

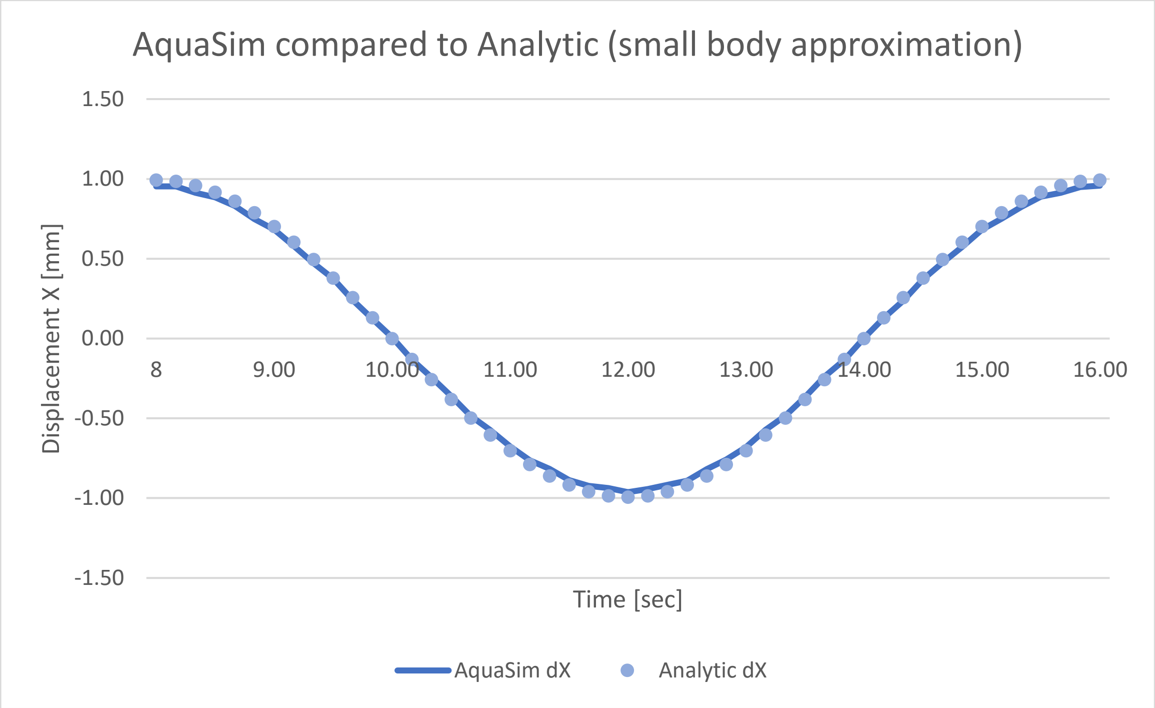

Figure 32 Comparison of AquaSim results with results calculated analytically applying the long wave approximation. The results show horizontal displacement at the beam tip for the 10 meter long beam with hydrodynamic and structural properties given in Figure 31 and Table 21, and with response as in Equation 10. Wave amplitude is 1.0m, and wave period 8 seconds

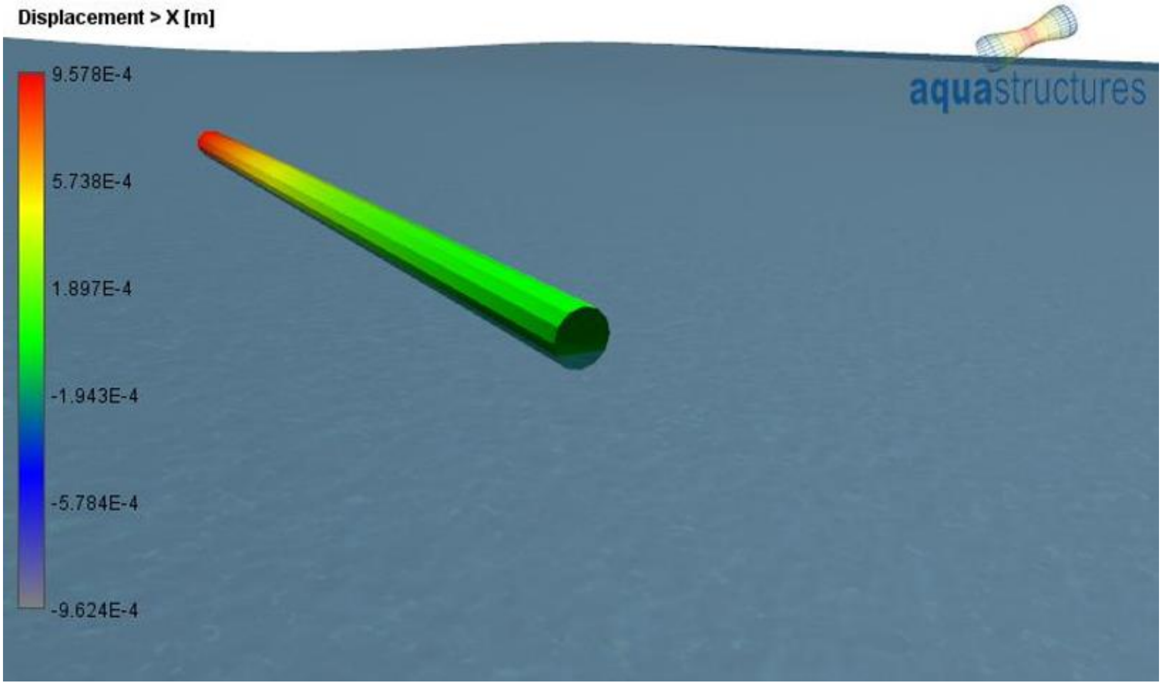

Figure 33 Calculated displacement from AquaSim, with 1 meter wave amplitude

Figure 33 shows visualization of the calculated displacement close to the maximum value. Note that the results depend on the numerical calculation of the hull. AquaSim generates straight lines between the given input points, which indicate that with fewer points a true circle is not predicted correctly. In this analysis case there is no viscous drag load added to the hydrodynamic loads. Figure 34 shows the same as Figure 33 but for a 10-meter wave amplitude. As seen, the results compare well as this load component is linear with respect to wave height.

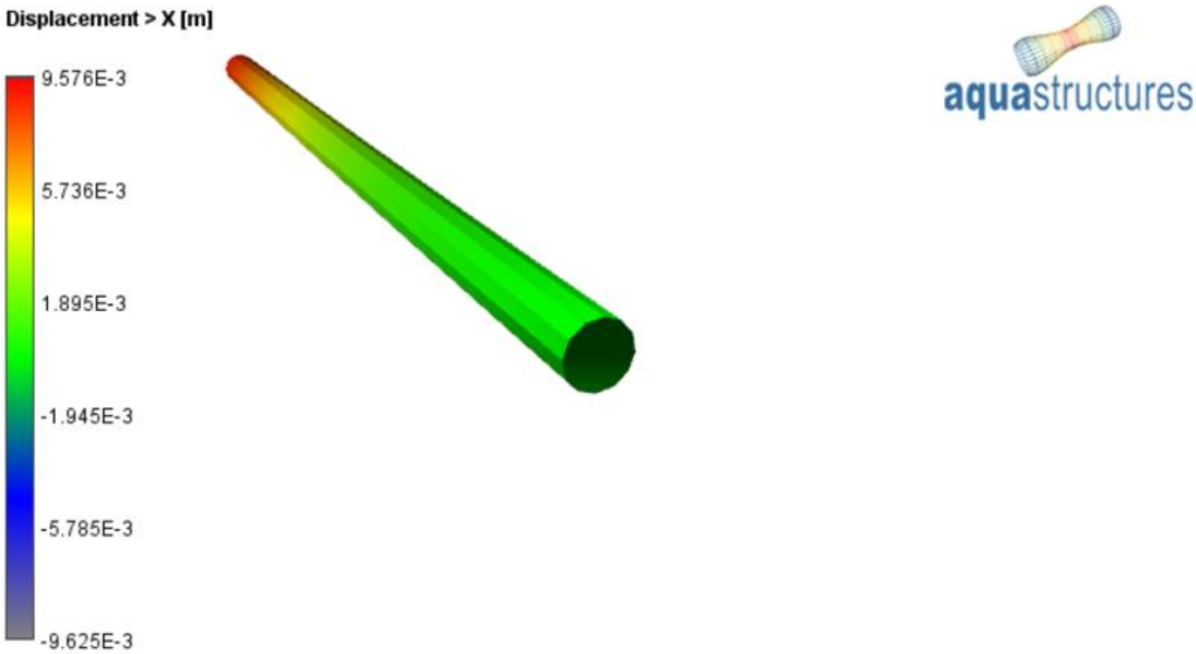

Figure 34 Calculated displacement from AquaSim, with 10 meter amplitude

6.2 AquaSim results compared with the wave reflection asymptote

In this section a cross section is established in order to compare calculated beam displacement using the reflected wave asymptote with results using the AquaSim program. The reflected wave asymptote is valid for waves approaching a wall. The wave will then be reflected, causing an oppositely directed wave of equal magnitude.

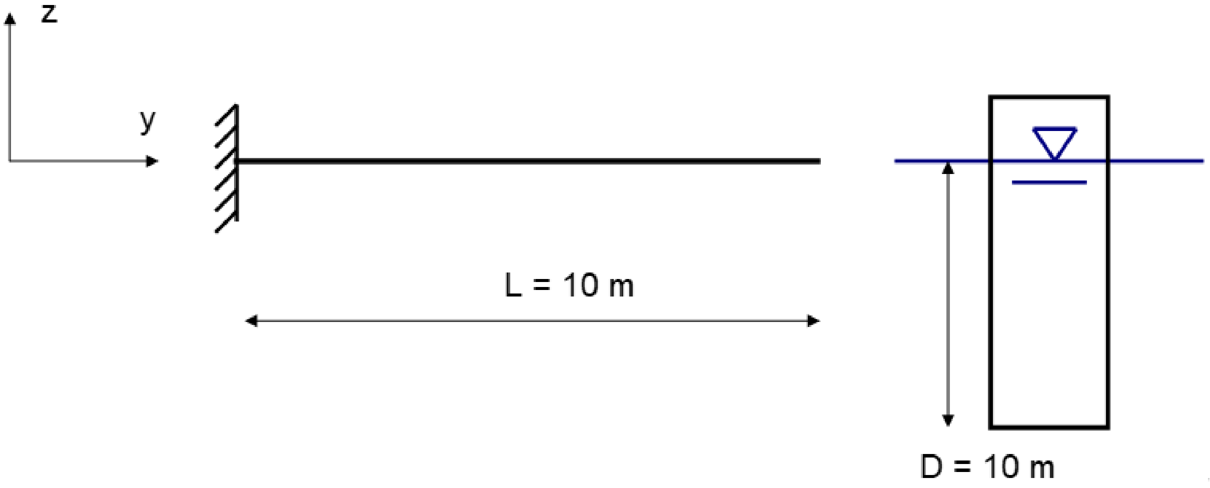

Figure 35 Information for the case study where results from AquaSim are compared to results from using the reflected wave asymptote. The depth of the beam below the water surface is 10 meters and the width of the beam is 1 meter

Structural data for this case is given in Table 22.

Table 22 Beam properties in case study using a beam to test closeness to the wave reflection asymptote

Abbreviation	Description	Value
A	Cross sectional area	0.1 m^2
L	Length of beam in original configuration	10 m
I_y	Area moment of inertia about local y- axis	4.0 ($1/m^4$)
I_z	Area moment of inertia about local z- axis	0.4 ($1/m^4$)
I_T	Torsional area moment of inertia	2.0 ($1/m^4$)
G	Torsional module	4E10 N/m^2
E	Young's modulus of the beam	1.0E11 N/m^2

The applied wave data is given in Table 23.

Table 23 Wave data

Description	Abbreviation	Value
Wave amplitude	A	5 m
Water density	ρ	1025 kg/m^3
Direction	β	Beam seas (90^o)
Current velocity	U	0 m/s

The analytical results for this case have been derived by integrating the Froude-Krylov pressure over the weather side of the beam. This leads to the following expression for the horizontally distributed load q in this case:

$$q_{fc} = \frac{\rho g A}{k}(1 - e^{-kD})$$

Equation 11

The total distributed force using the above expression is then:

$$q = 2\frac{\rho g A}{k}(1 - e^{-kD})$$

Equation 12

These analytic values have been introduced to the above expressions and compared with results from AquaSim. Figure 36 shows this comparison for a wave period of 4 seconds. In this case most of the wave will be reflected since the particle velocities are reduced downwards proportional to e^{kz}. k is the wave number, expressed as $k = \frac{2\pi}{\lambda} = \frac{4\pi^2}{T^2 g}$. In this case, this means that at z = -10 the wave velocity will only be approximately 8 % of the velocity at the surface. Due to continuity, most of the wave will have to be reflected. As seen from Figure 36 this clearly happens.

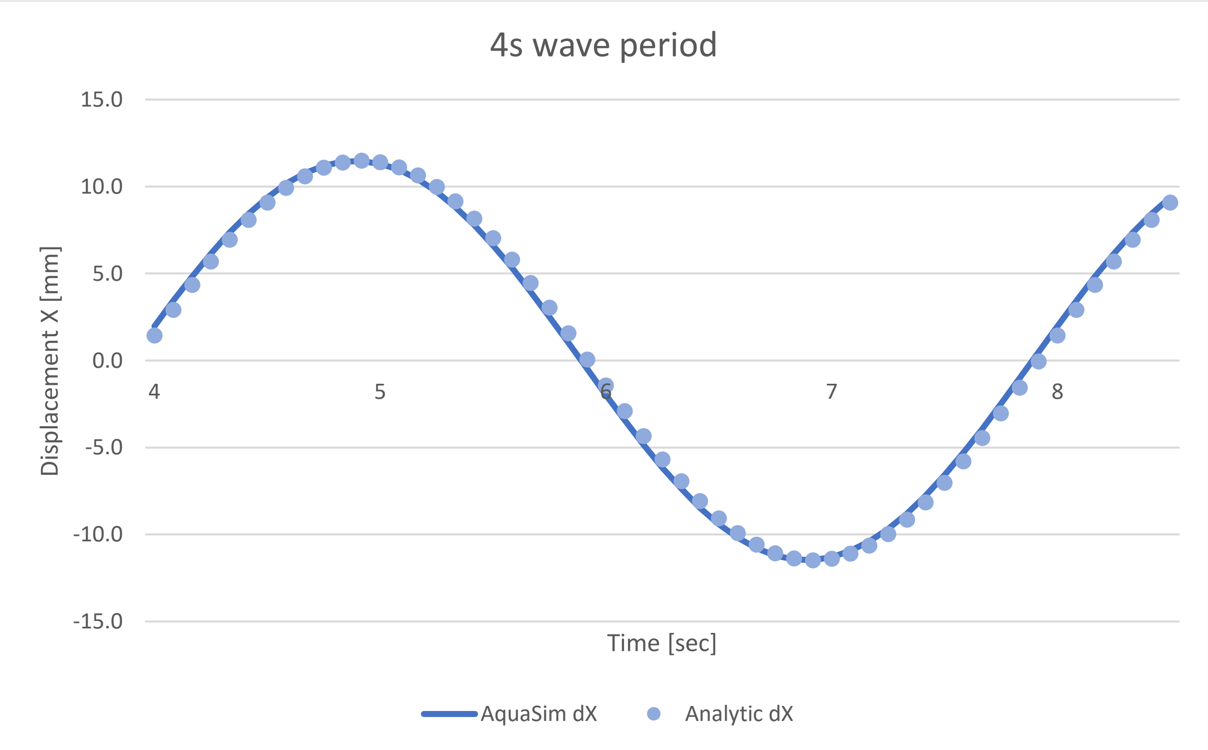

Figure 36 Horizontal displacement at the end of the beam. Comparison between analytical results and AquaSim results

As seen from Figure 36 results calculated by AquaSim correspond very well with the analytic expression based on the small body approximation. The results are deviating because small body approximation is not fully valid for this case. AquaSim reflects the actual beam in a proper manner whereas the small body approximation is a slight simplification. Figure 37 shows the same as Figure 36 but in this case the wave period is 10 seconds. This means that the wave velocity at z = -10 meter is approximately 65 % of the wave velocity at the surface. This means that much of the water is transported below the beam, and a smaller part is reflected. Hence it is not expected to find results close to the reflected wave asymptote for this case. As seen from Figure 37 this is also the case.

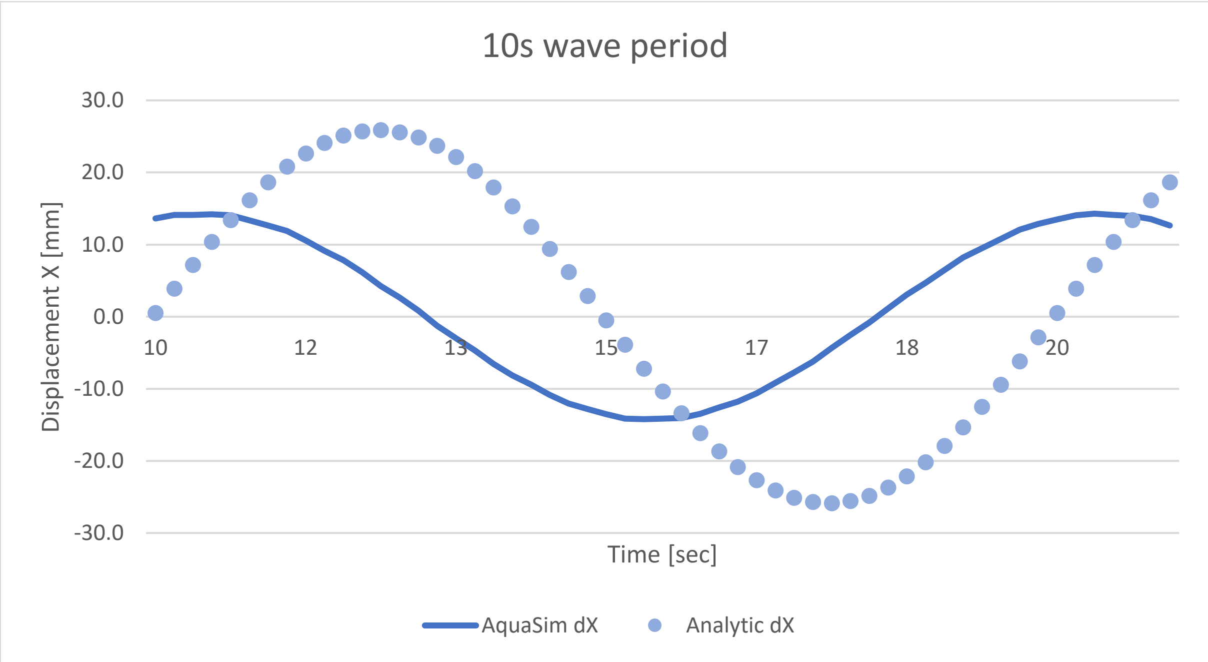

Figure 37 Horizontal displacement at the end of the beam. Comparison between analytical and AquaSim results

As seen from Figure 37 the response calculated by AquaSim is smaller than the response predicted by the small body asymptote. This is in good correspondence with the physics since AquaSim accounts for the fact that much of the wave is not reflected in this case. The wave reflection asymptote assumes that the whole wave is reflected.

7 Properties on nodes

Several types of properties may be introduced to nodes in AquaSim. Basically, nodes can be free or have prescribed properties. Otherwise, node loads or springs may be attached to nodes.

7.1 Fixed nodes

Several papers have been issued and several analyses have been carried out where nodes have been fixed. Reference is made to (Aquastructures, 2006) which validates this properly.

7.2 Linear “node to ground” spring

Node to ground springs may be attached to nodes in 6 degrees of freedom. Translation along the x-, y- and z-axis, as well as rotation about the x-, y- and z-axis are possible inputs. Using DOF as abbreviation for degree of freedom means that x-translation is DOF 1, y-translation is DOF 2 and z-translation is DOF 3. Respectively, rotation about the x-axis is DOF 4, y-axis is DOF 5 and z-axis is DOF 6.

In the present case a very stiff beam is considered. The beam consists of ten elements of 1 meter length, modelled along the y-axis, with the first element as shown in Figure 38. The spring resistances is in this case 1000 N/m and 1000 Nm/rad. The springs are applied to node 1, the other nodes are free. Loads are also applied to node 1. The displacement of the beam is calculated by AquaSim and compared to analytical results. Conservative node loads are applied to various DOFs, and the DOFs are given in Table 24. As seen from this table the results from AquaSim and analytical calculations correspond very well. Note that node to ground springs are conservative, meaning that a spring attached to a node will not rotate proportional to other elements attached to the nodes. By using local coordinates such effects can be introduced.

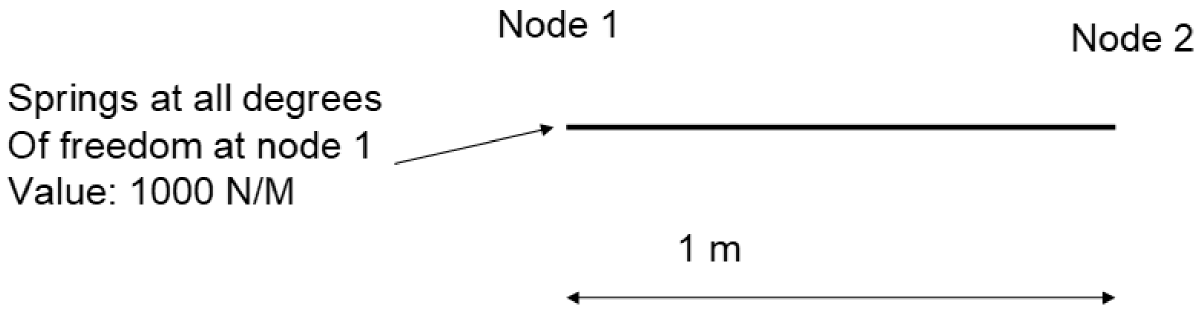

Figure 38 Geometry of test case node to ground springs. One element is 1 meter, and the total length is 10 meters

For beam data not expressed explicitly in Figure 38, they are the same as in Section 6.1, Table 21. The beam is very stiff relative to the stiffness of springs.

Table 24 Results verifying node to ground spring element. Node numbers are referring to Figure 38

Applied force/ moment	Force/ moment applied at	Result parameter	Analytic result [m]	AquaSim result [m]
Force X = 1000 N	Node 1 DOF 1	x/y/z-translation node no. 2	1.0/ 0 / 0	1.0/ 0 / 0
Force Y = 1000 N	Node 1 DOF 2	x/y/z-translation node no. 2	0/ 1.0 / 0	0/ 1.0 / 0
Force Z = 1000 N	Node 1 DOF 3	x/y/z-translation node no. 2	0/ 0 / 1.0	0/ 0 / 1.0
Moment X = 1000 Nm	Node 1 DOF 4	x/y/z-translation node no. 2	0/ -0.46 / 0.84	0/ -0.46 / 0.84
Moment Y = 1000 Nm	Node 1 DOF 5	x/y/z-translation node no. 2	0/ 0 / 0	0/ 0 / 0
Moment Z = 1000 Nm	Node 1 DOF 6	x/y/z-translation node no. 2	-0.84/ -0.46/ 0	-0.84/ -0.46 / 0

Table 24 presents the results predicted by AquaSim and lists the analytical results. The results correspond well.

7.3 Local coordinates

AquaSim allows for the introduction of a local coordinate system at any node. The advantage of this is that one may introduce for example hinges in any direction by specifying which nodes and DOFs that are coupled and which that are not in any direction. The local coordinate system at a node may "follow" node rotations such that the location of the local coordinate system always follows any global rotations. In such a case, the elements the local coordinate system shall rotate in proportion to must be specified.

In this case a local coordinate system was introduced to the case study in Section 7.2. In the local coordinate system, a totally fixed hinge was introduced in node 2. The same load cases as in Section 7.2 was tested in this case. As the hinge is fixed in all directions, the results are expected to be the same as in the previous section.

The analysis shows that the beam behaves as if there were no hinge present, as was expected. The displacements in x-, y- and z-direction are exactly the same as in the previous case, shown in Table 24, and a new separate table is therefore not presented.

7.4 Buoys

Consider a buoy located at the free surface. The buoy will then act as a spring relative to the sea surface in the z-direction with a spring force of

$\rho g A_w$,

where A_w is the cross sectional area of the buoy in the horizontal plane at the water surface. Consider a case with a beam located at the free surface. Assume the beam have no water plane area or weight itself, but that there is one buoy connected to the beam at each side as shown in Figure 39.

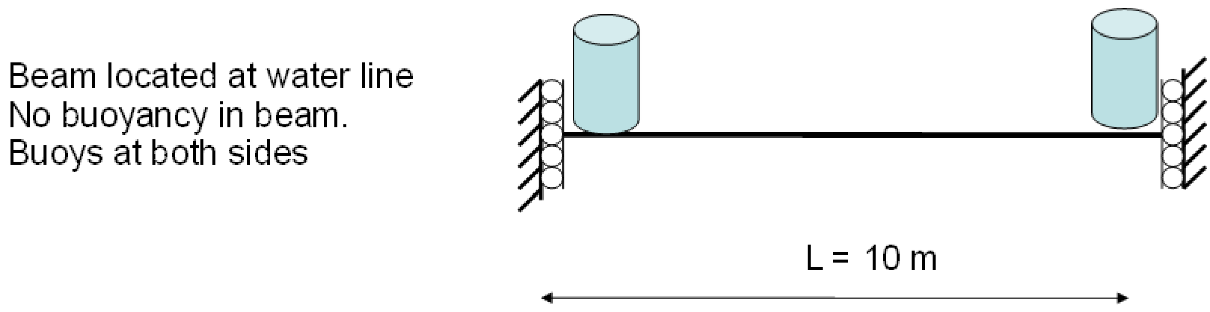

Figure 39 Beam located at the water line with a buoy at each end

Figure 39 shows the AquaSim analysis model. The wave amplitude is 5 meters and the wave period is 8 seconds. The beam data not expressed explicitly in Figure 39 are the same as in Table 22. The buoy force is 10000 N/m in positive z-direction.

In Figure 40 the results from AquaSim are compared to the results from the analytic expression. It is seen that the results correspond very well. Figure 41 shows the connection between the elevation of the buoys and the waves. It shows that the beam and buoys follow the wave elevation perfectly.

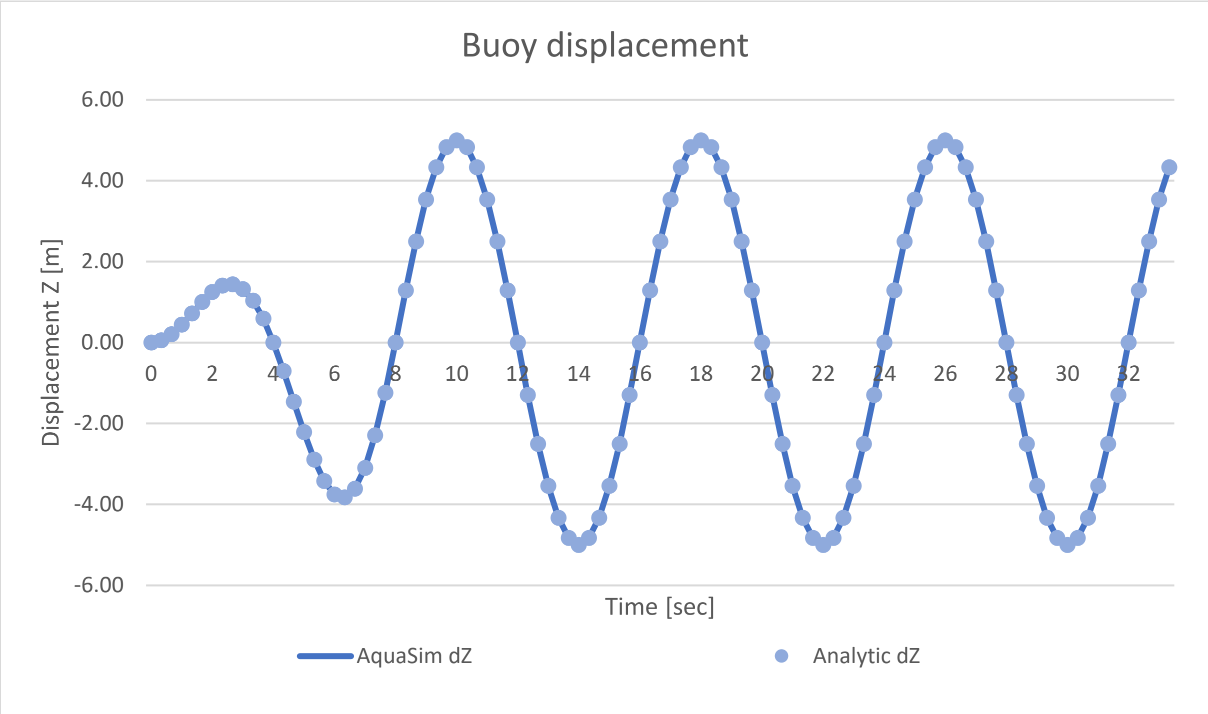

Figure 40 Comparison of buoy displacement calculated by AquaSim and analytic formula

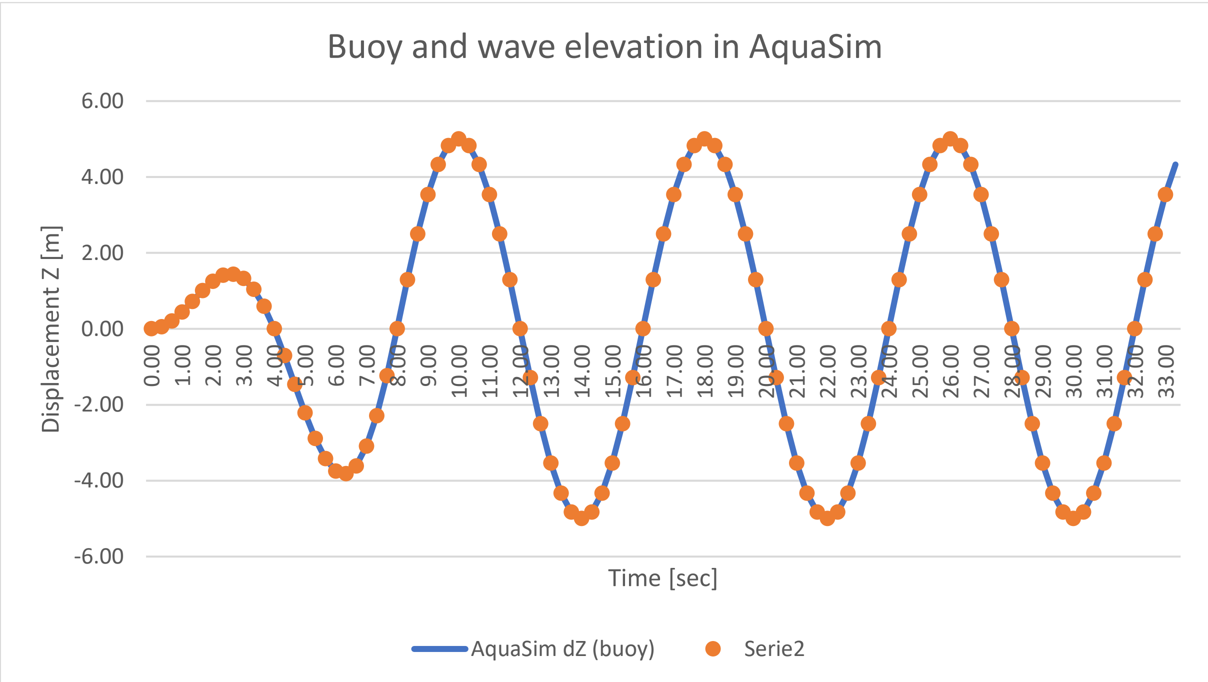

Figure 41 Buoy elevation compared to wave elevation

7.5 Prescribed displacement

Prescribed displacements may be used in AquaSim. A test case study has been established. The structural data for this case is given in Table 25.

Table 25 Values used in beam case study testing prescribed displacements

Abbreviation	Description	Value
A	Cross sectional area	0.1 m^2
L	Length of beam in original configuration	10 m
I_y	Area moment of inertia about local y- axis	0.001 m^4
I_z	Area moment of inertia about local z- axis	0.001 m^4
I_T	Torsional area moment of inertia	0.002 m^4
G	Torsional module	4E10 N/m^2
E	Young's modulus of the beam	1.0E11 N/m^2

The beam is exposed to a set of different boundary conditions at Node 2, whereas at Node 1 the beam is fixed as shown in Figure 42.

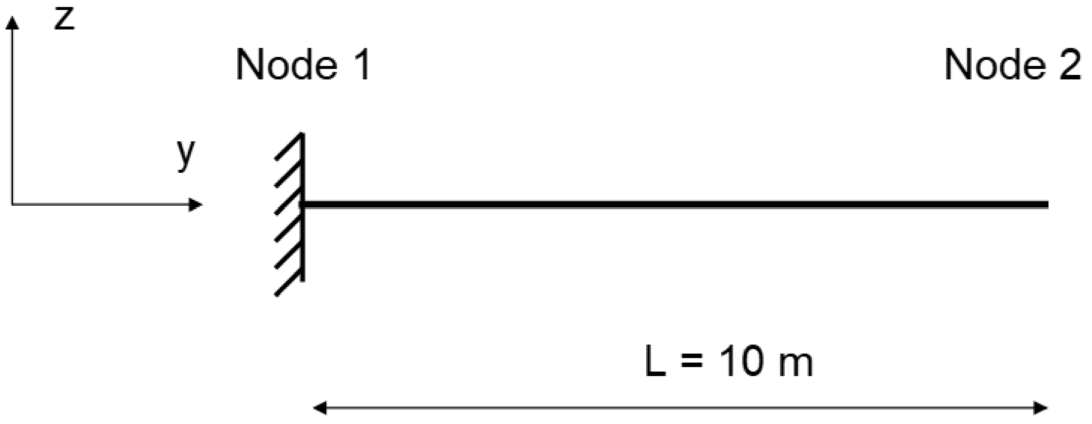

Figure 42 Beam for testing of prescribed displacement

Different prescribed displacements have been applied for Node 2. The status of the DOFs and results are given in Table 26. As seen from this table AquaSim and analytical results corresponds very well. AquaSim results for load case 1, 2, 3 and 4 are shown in Figure 43, Figure 44, Figure 45 and Figure 46, respectively.

Table 26 Results from the 4 prescribed displacement load cases

Case no.	Prescribed displacement at node 2 [m]	Result parameter	Bending moment at node 1 [kNm]	
			Analytical	AquaSim
1	DOF 1 = 0.1	M_x	0	0
		M_y	0	0
		M_z	-300	-300
2	DOF 3 = 0.1	M_x	0	0
		M_y	-300	-300
		M_z	0	0
3	DOF 3 = 0.0 DOF 4 = 0.1	M_x	0	0
		M_y	2000	2000
		M_z	0	0
4	DOF 1 = 0.1 DOF 6 = 0.0	M_x	0	0
		M_y	0	0
		M_z	-600	-600

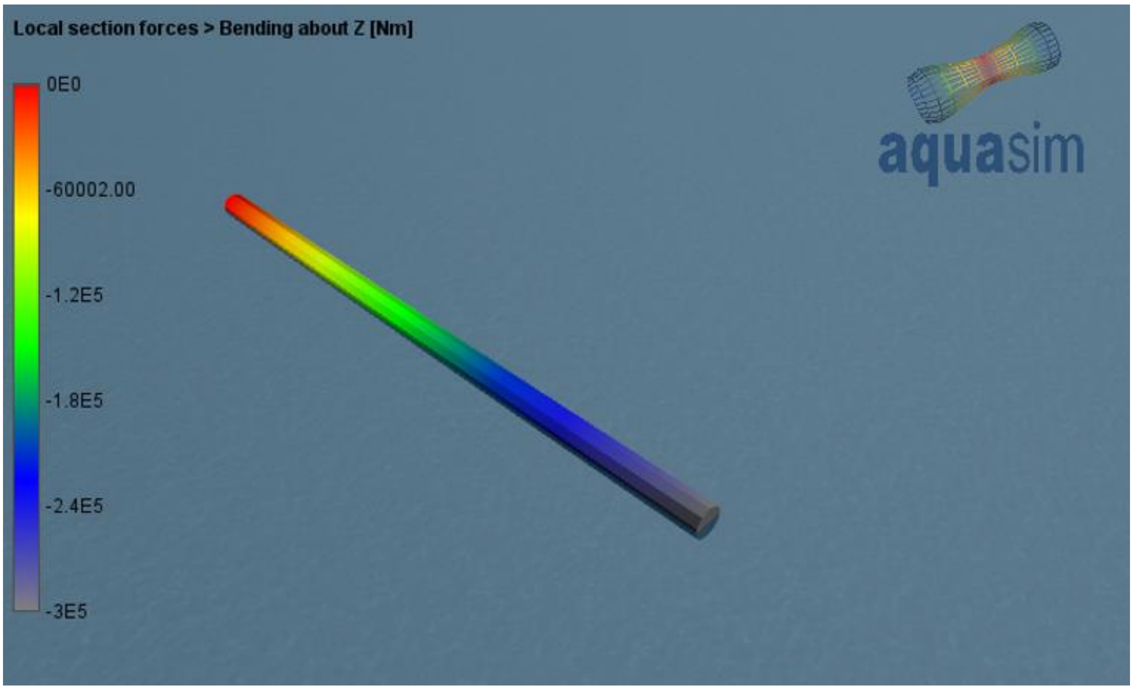

Figure 43 Load case no. 1. Bending moment about z-axis (vertical axis)

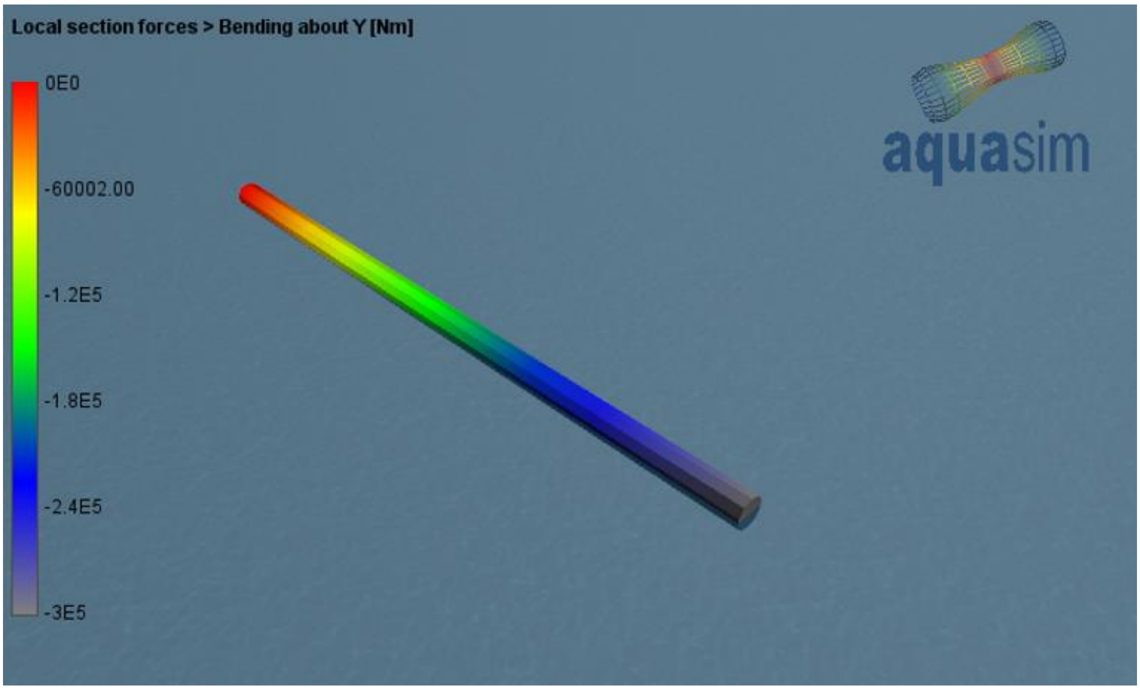

Figure 44 Load case no. 2. Bending moment about y-axis (neutral axis of the beam)

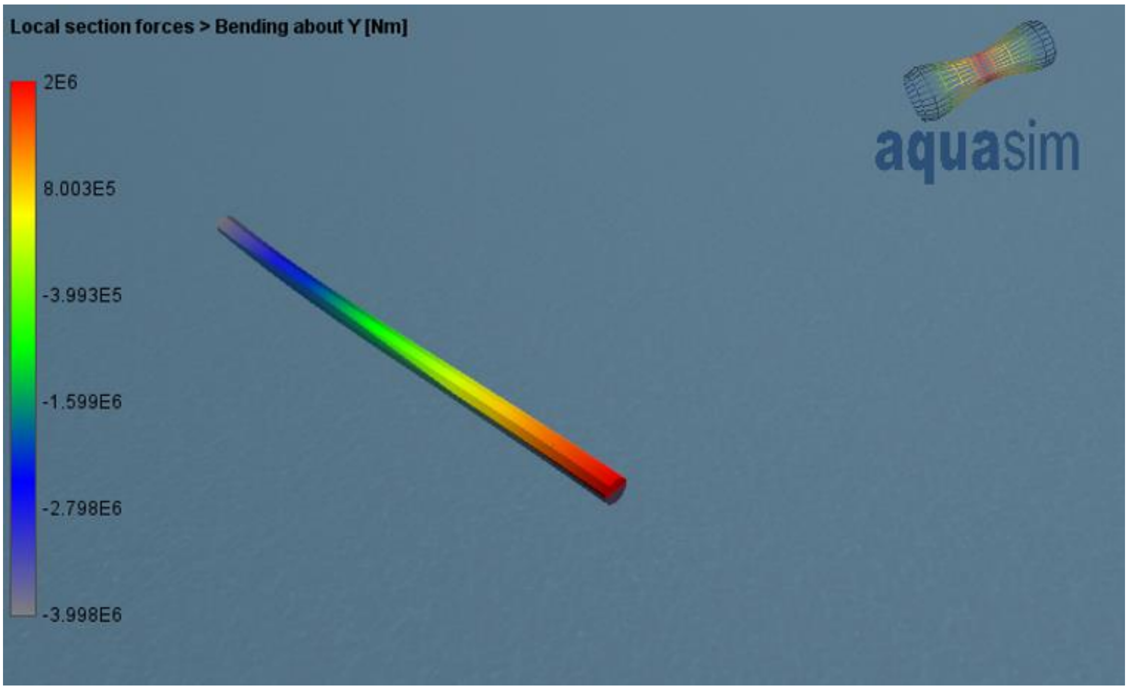

Figure 45 Load case no. 3. Bending moment about y-axis (neutral axis of the beam)

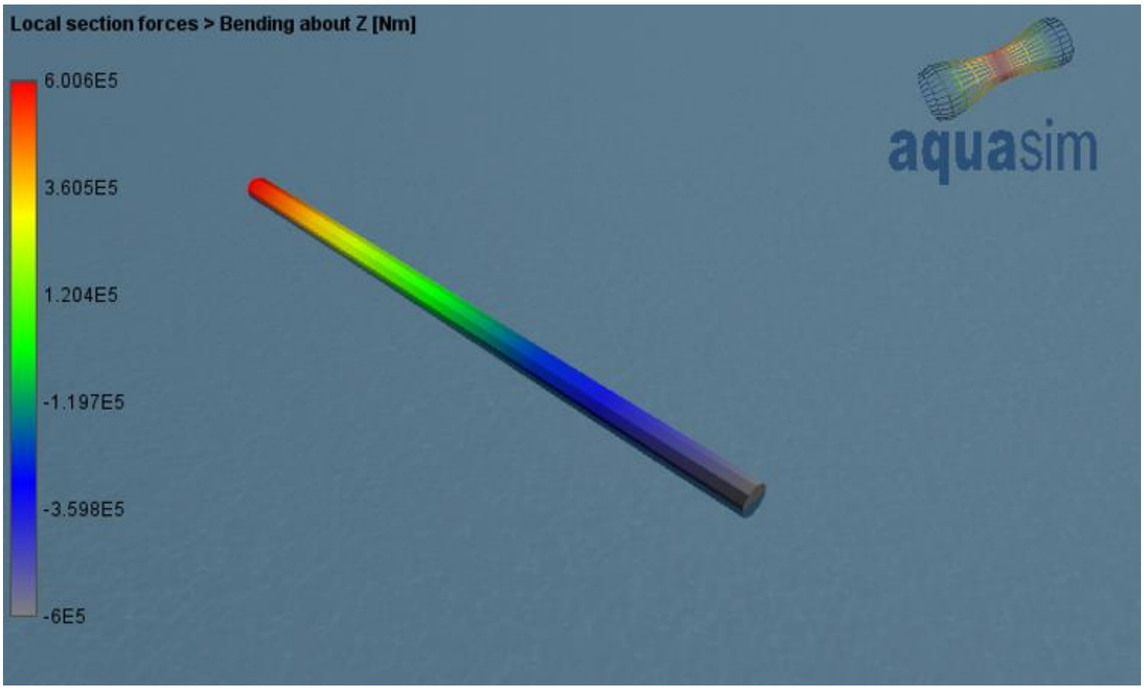

Figure 46 Load case no. 4. Bending moment about z-axis (vertical axis)

7.6 RAO on nodes

AquaSim has a command for applying RAO (response amplitude operator). There is a large variety in possibilities on how to arrange the input data. The following two cases compares the results from AquaSim with the results that are expected based on the input data in the model. A truss element of 10 meters with two nodes are used to test the two RAO cases. The cross-sectional area is 0.001 m^2 and the E-modulus is 1.0E9 N/m^2. Node 1 is retained in x-, y- and z-direction, and Node 2 is retained in y-direction.

7.6.1 RAO displacement case

In this case a forced RAO displacement of 1.0 meter in positive z-direction is applied to Node 2. The results will show how the response of the truss element evolves when a wave passes. Two load conditions are tested, Hs = 1 m and 2 m. The RAO is set to evolve as a function of wave height, with an amplitude response of 1 meter. This means that for Hs = 1 m, H_{max} is approximately 1.9 meters, and the difference between the lowest and highest point the end point of the truss reaches should be the same ($1.0\,m\,Hs \cdot 1.9 \frac{m\,H_{max}}{m\,Hs} \cdot 1.0\,m\,RAO = 1.9\,m$). For Hs = 2 m, this value is then 3.8 meters. Tp for the wave is 4 seconds.

Figure 47, Figure 48 and Figure 49 shows the results from the analysis. During the initial steps (before time 0 in Figure 47) Node 2 is moved to z = 1 meter, and for the two waves the motion of the truss varies around 1 meter. The difference between the highest and lowest point on the graph is 1.89 meters for Hs = 1 m, and 3.78 meters for Hs = 2 m, which is as expected.

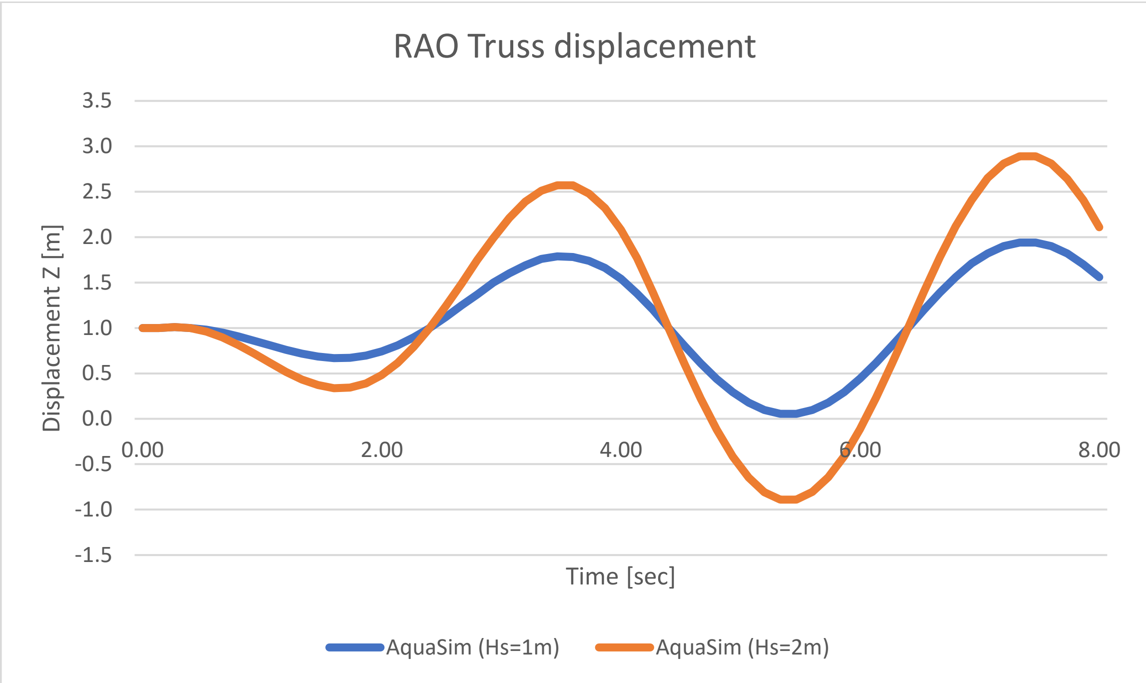

Figure 47 Displacement of Node 2 of truss with RAO displacement. Load condition Hs = 1m and 2m

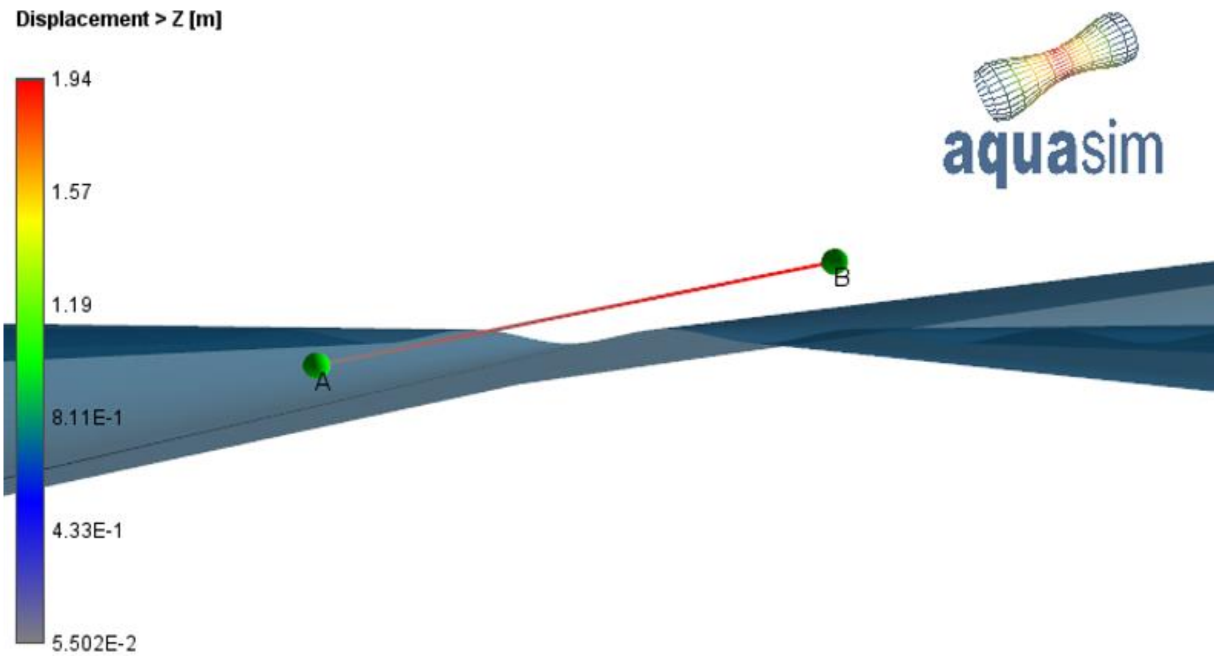

Figure 48 AquaView, dispalcement in z-direction. Hs=1m. The legend shows the displacement

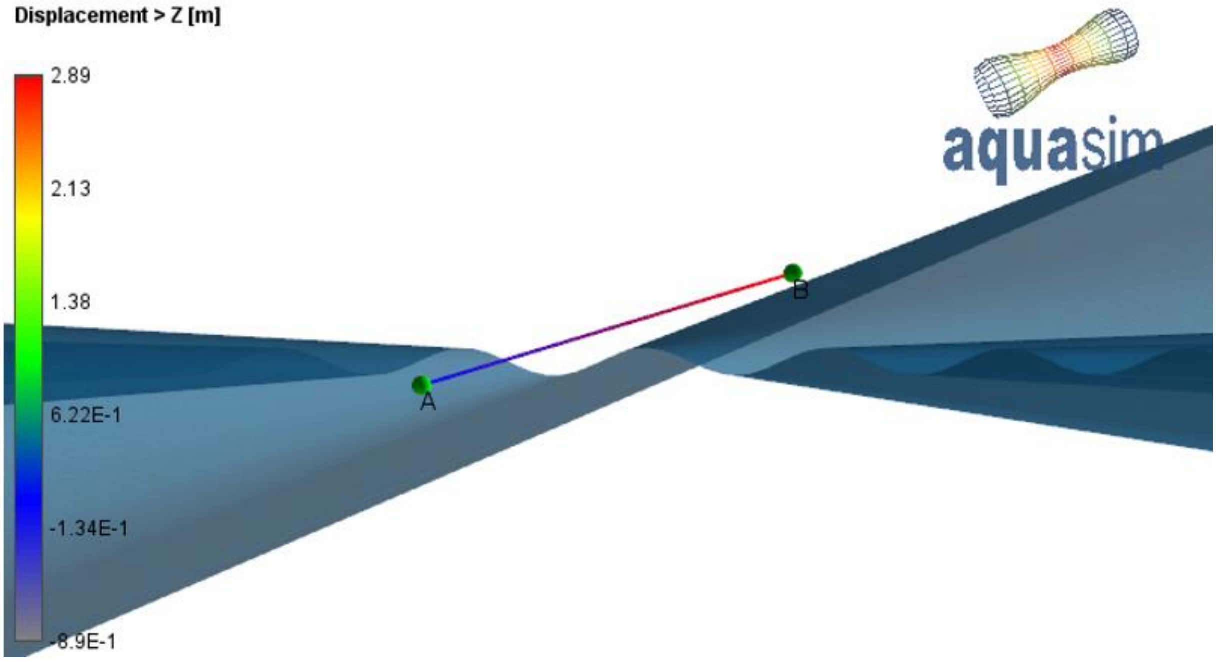

Figure 49 AquaView, displacement in z-direction. Hs = 2m. The legend shows the displacement

7.6.2 RAO load case

In this case an RAO unit load of 1.0 N in positive x-direction is applied to Node 2. The results will show that the axial load in a truss-element is equal to the unit load, when a wave passes. The amplitude response in x-direction is set to 1, so that the axial load in the truss element is at its maximum when the crest passes, and the response amplitude is a function of the unit load. The same two load conditions are tested, Hs = 1 m and 2 m, and the RAO is set to evolve as a function of wave height, with an amplitude response of 1 meter, this time in x-direction. The expected results are similar to the previous case, except that they now are expressed as the axial force in the truss. This means that the maximum axial force is expected to be close to 0.95 N (1*1.9*1*0.5) with Hs = 1 m, and 1.9 for Hs = 2 m.

Figure 50 shows the results from the analysis. The maximum axial force in Node 2 occurs when the 2nd crest passes the node. This value is 0.945 for Hs = 1 m, and 1.89 for Hs = 2 m. Figure 51 and Figure 52 shows the axial force in the truss when the 2nd crest is passing Node 2.

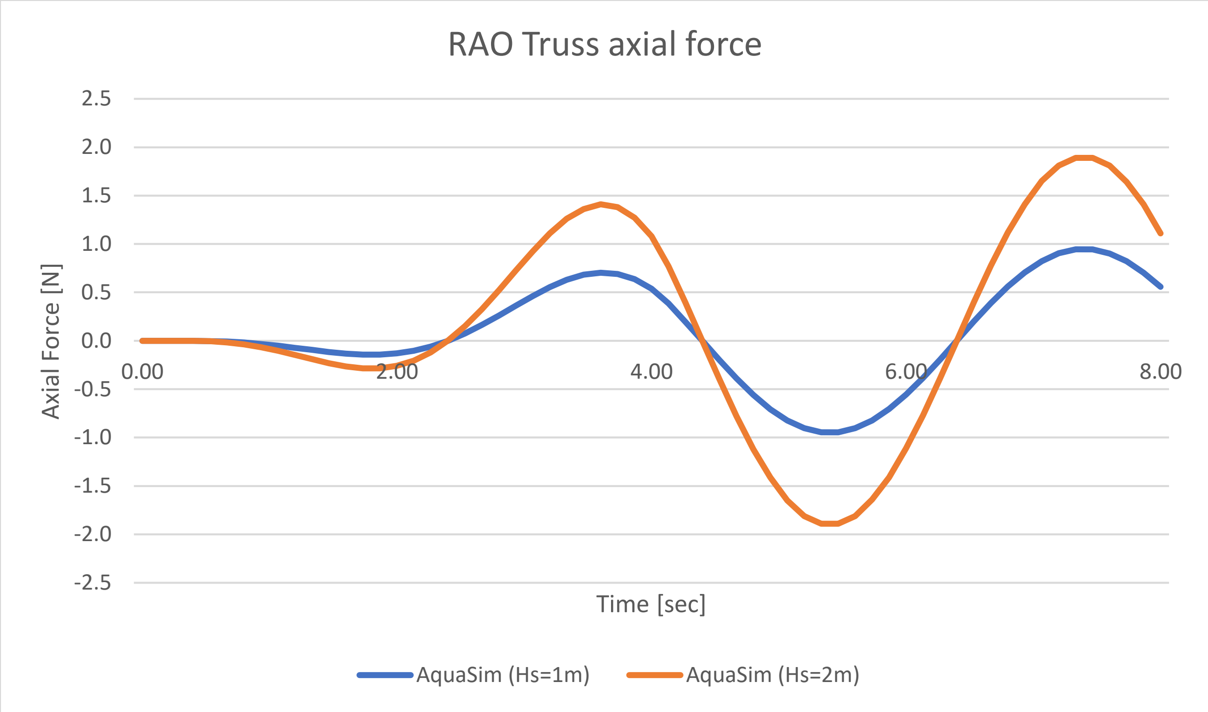

Figure 50 Axial force in Node 2 of the truss with RAO unit load. Load conditions Hs = 1m and Hs = 2m

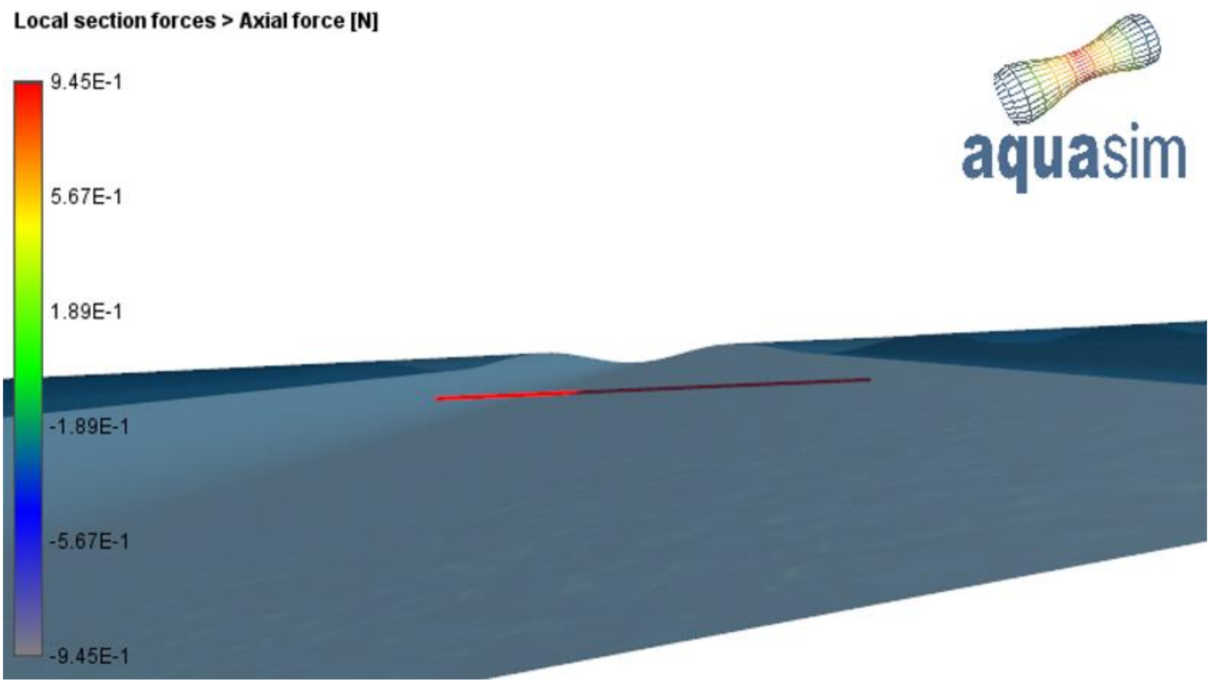

Figure 51 AquaView, axial force in the RAO node. Load condition Hs = 1m

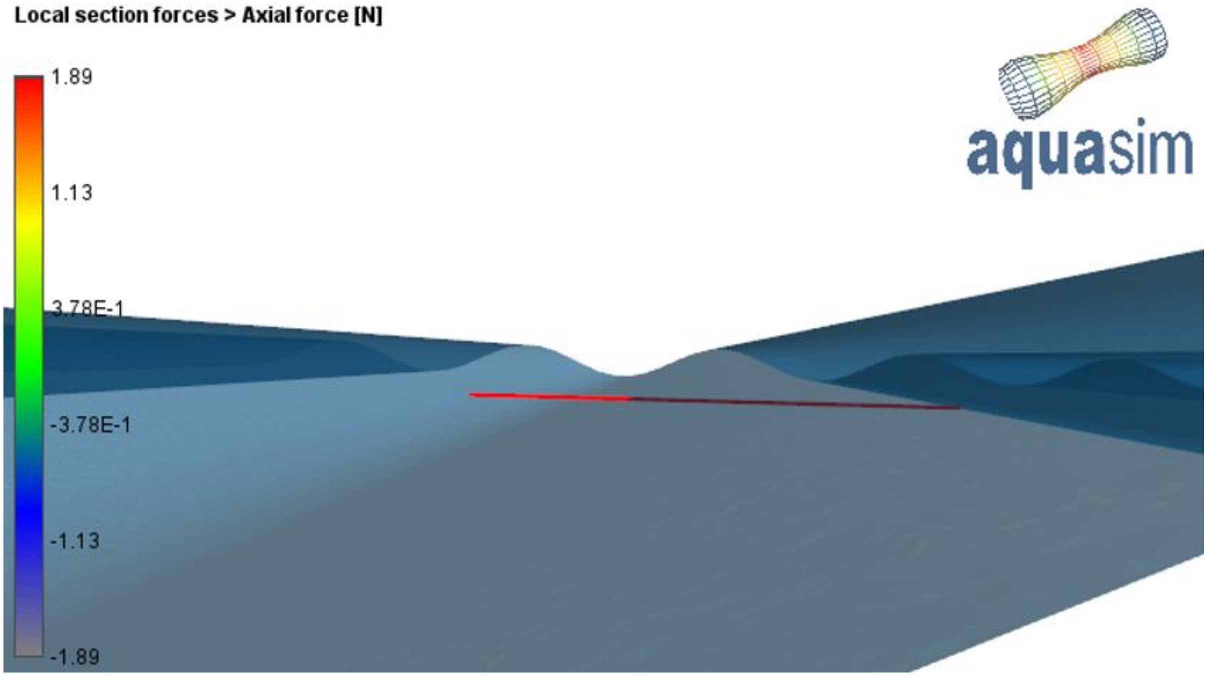

Figure 52 AquaView, axial force in the RAO node. Load condition Hs = 2m

The possibility for prescribing a time series for node motions and rotations in AquaSim works as they should. Hence, the AquaSim calculations are validated.

7.7 Node loads

AquaSim holds several possibilities for node load application. Depending on type of node load, the node loads are treated differently. There are 11 different load types. Load type 0 and 1 are tested in this case.

Type 0 and 100: Conservative node loads are conserving the same magnitude and direction throughout the full analysis. This can be used with (0) or without (100) automatic introduction of mass corresponding to negative vertical force.

Type 1, 2 and 3: Input values are multiplied with $V_{xr}*|V_{xr}|$, where V_{xr} is the relative velocity between the fluid and the node. This means that for a flow along the x-axis, this can be used to introduced drag, lift or moment in any direction. There are corresponding effects in y- and z-direction.

Consider a test case with the data given in Table 27.

Table 27 Data for case study

Description	Abbreviation	Value	Unit
Node load at tip	P	10000	N
Length	L	10	m
Cross sectional area	A	0.1	m^2
Youngs module	E	1.00E+11	N/m^2
Area moment of inertia	I	0.001	m^4
Displacement beam (analytic solution)	r	33.33	mm

The displacement calculated by AquaSim are the same as the results calculated by the analytical formula. Figure 53 compares the results, both for load case 0 and for load case 1.

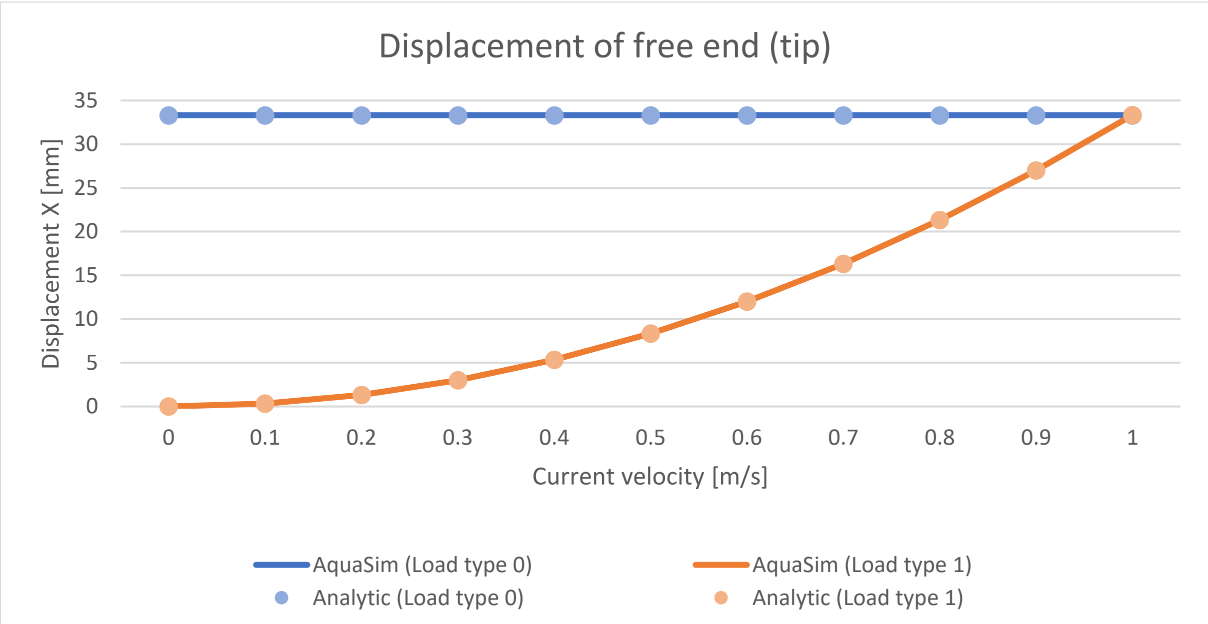

Figure 53 Comparison of results from analytical formula and AquaSim results

8 Wind loads

AquaSim may account for wind loads. Wind loads are calculated in the same manner as drag loads on an element. It is specified by the wind area of the element. The wind velocity profile is calculated according to the mean wind component of NORSOK spectra, read more about this in (Aquastructures, 2021b). This wind velocity profile is implemented as from solver release version 2.18.1.

The mean wind velocity profile is expressed as:

$$U(z) = U_{10} \cdot \left(1 + C \cdot ln\left(\frac{z}{10}\right)\right)$$

Equation 13

where U_{10} is the wind velocity 10 meters above the surface, and z is the distance upwards from the surface. C is a constant:

$$C = \frac{5.73}{100}(1 + 0.15 \cdot U_0)^{0.5}$$

Equation 14

U_0 is the 1-hour mean wind velocity at 10 meters above the surface. The relation between U_0 and U_{10} is:

$$U_0 = \frac{U_{10}}{\left[1 - 0.41 \cdot I_u(z) \cdot ln\left(\frac{t}{t_0}\right)\right]}$$

Equation 15

The turbulence intensity factor $I_u(z)$ is described as:

$$I_u(z) = 0.06 \cdot (1 + 0.043 \cdot U_{10})\left(\frac{z}{10}\right)^{-0.22}$$

Equation 16

The force on a surface caused by the wind is then calculated by the following expression

$$F_D = \frac{\rho_{air} C_D}{2} A U^2(t)$$

Equation 17

ρ_{air} is the density of the air = 1.27 kg/m^3, C_D is the drag coefficient of the surface and A is the area of the surface of the element. The vertical position z is calculated as the mean position of the element plus half the height of the wind exposed area. A case study is investigated where wind is applied on a beam which as shown in Figure 54. Beam data not expressed explicitly in this figure are the same as in Table 22.

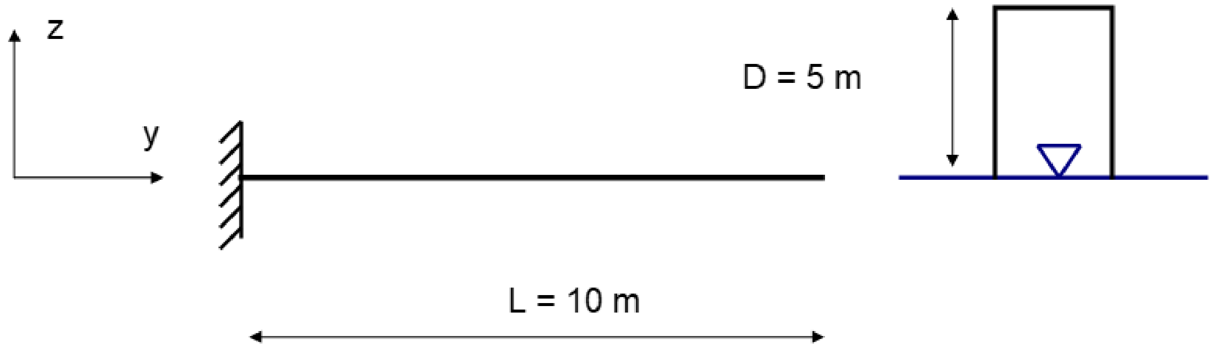

Figure 54 Case study for testing wind loads. Beam data is the same as in Table 22

Input data to the analysis is specified in Table 28.

Table 28 Input data to the wind load case

Description	Abbreviation	Value
Drag coefficient, C_D	1	-
Length of beam	10	m
Length of beam, origin to mid-point last element	9.5	m
Height of wind area	5	m
Lower part of wind catch area	0	m
Transverse drag area	47.5	-
Average wind at 10 meters height	10	m/s

Results from the analysis is presented in Table 29. The values are taken at the node positioned in the origin. Figure 55 shows the distribution of the bending moment about z-axis.

Table 29 Results from analytical calculations and AquaSim

	Analytic results	AquaSim results
Shear force [N]	-2311.2	-2317.87
Torsion moment [Nm]	5777.9	5794.68
Moment about z-axis [Nm]	-12164.0	-12179

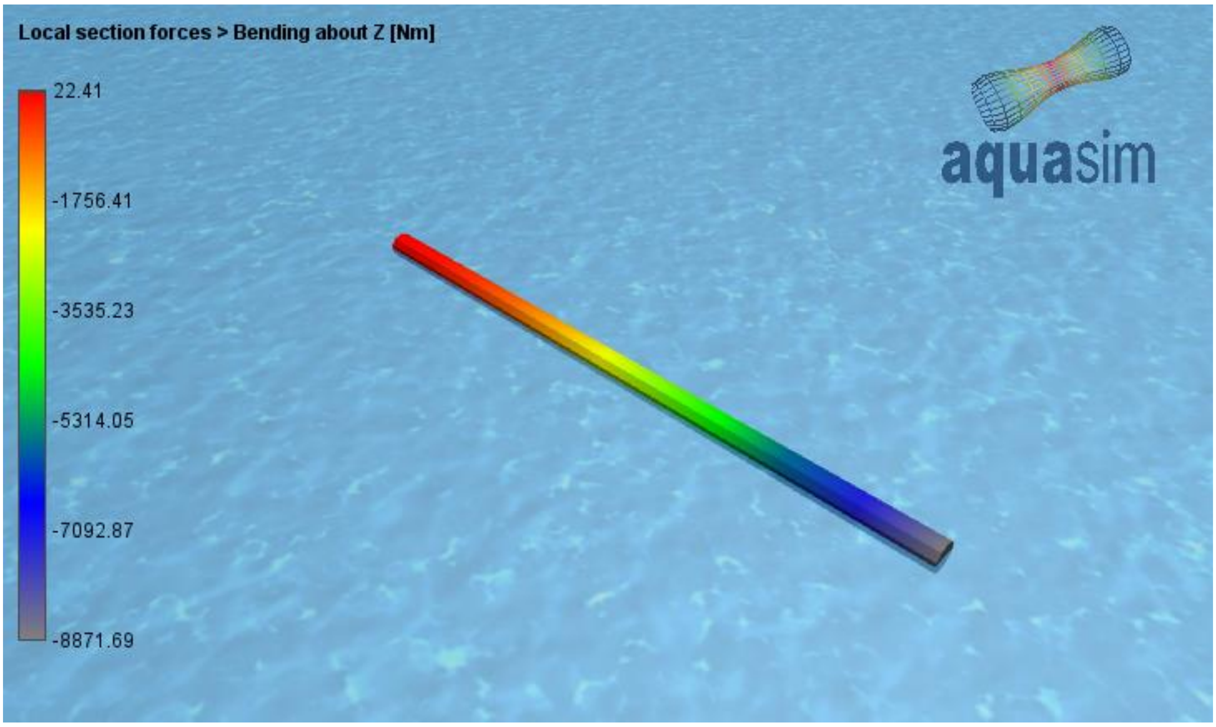

Figure 55 Bending moment about z-axis

9 References

Aquastructures. (2006). *AquaSim - Theoretical Formulation of Structure and Load Modeling.* Tech. rep. 2006-Fo06.

Aquastructures. (2016). *Nets with bending resistance in AquaSim.* Tech. rep. TR-FOU-2222-5.

Aquastructures. (2020b). *Shell elements in AquaSim, case studies for validation.* Tech. rep. TR-FOU-2328-2.

Aquastructures. (2020c). *Morison free plate load formulation to shell and membrane elements.* Tech. rep. TR-FOU-2328-8.

Aquastructures. (2021b). *Spectral Windloads in AquaSim.* Tech. rep. TR-FOU-2328-4.

Aquastructures. (2021c). *Loads on impermeable nets and large volume objects in AquaSim.* Tech. rep. TR-FOU-2328-5.

Berstad, A., Walaunet, J., & Heimstad, L. (2012). Loads From Currents and Waves on Net Structures. *31th International Conference on Offshore and Arctic Engineering. OMAE2012-83757. Vol. 7*, pp. 95-104.

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