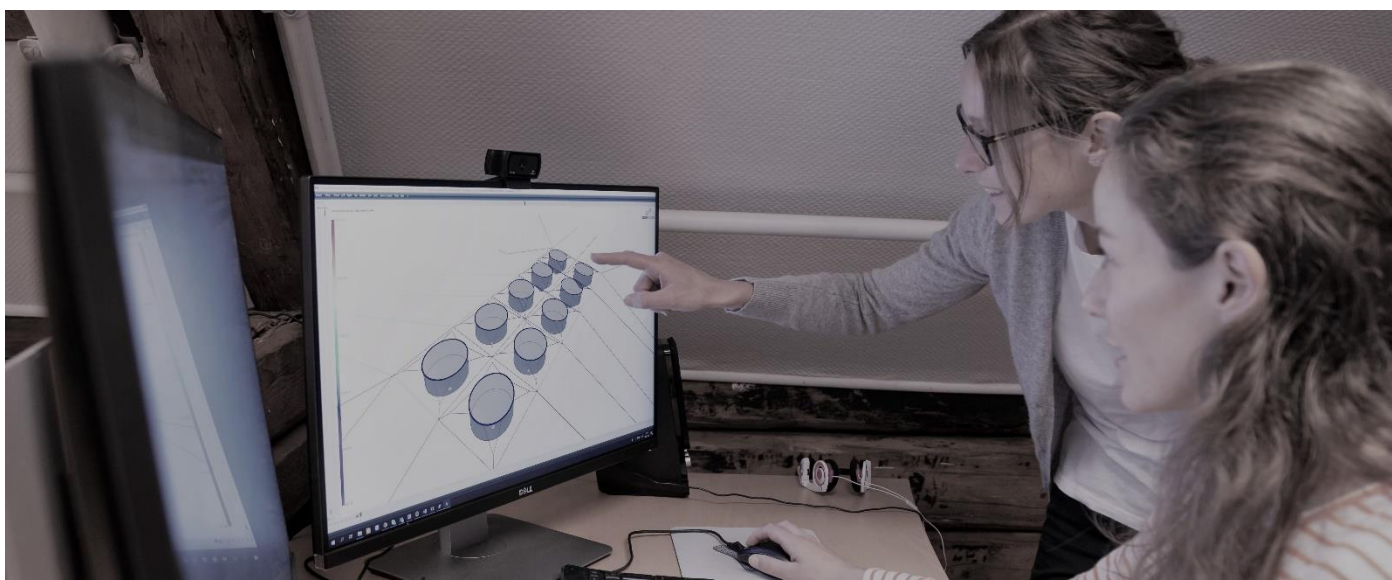


AquaSim training courses

- Spring: Displaced



Revision: 1.0

AquaSim version: 2.18

Aquastructures AS
Kjøpmannsgata 21, 7013 Trondheim
Norway

Content

1	Prerequisites.....	3
2	Case study – Spring: Displaced	3
2.1	Learning objectives	3
2.2	Introduction	3
2.3	Principles of Spring: Displaced.....	4
2.4	Pre processing	4
2.5	Calculations static analysis.....	8
2.6	Post processing static analysis	9
2.7	Changing Displaced amount	10
2.8	Calculations dynamic analysis	11
2.9	Post processing dynamic analysis	11
2.10	Summary.....	12
3	Revision comments.....	13

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 Case study – Spring: Displaced

2.1 Learning objectives

Upon completion of this case study, you will be able to:

- Understand the fundamentals of the spring type Displaced
- Define appropriate input parameters for the Displaced spring
- Run static and dynamic analysis with Displaced spring

2.2 Introduction

Consider a system consisting of a float. And a rope anchored to the seabed. The float is exposed to current and waves, which induces motion in the system. When the float moves up and down the rope will displace accordingly. The rope should be allowed to move freely upwards. When it is lowered, it will touch the seabed. The Displaced spring is hence applied with the purpose of rendering the contact between the rope and seabed.

In this case study you will determine the appropriate input values for spring type Displaced. First, you are to conduct static analyses to check your model and validate initial displacement with the spring. Then you should run a dynamic analysis with waves and current.

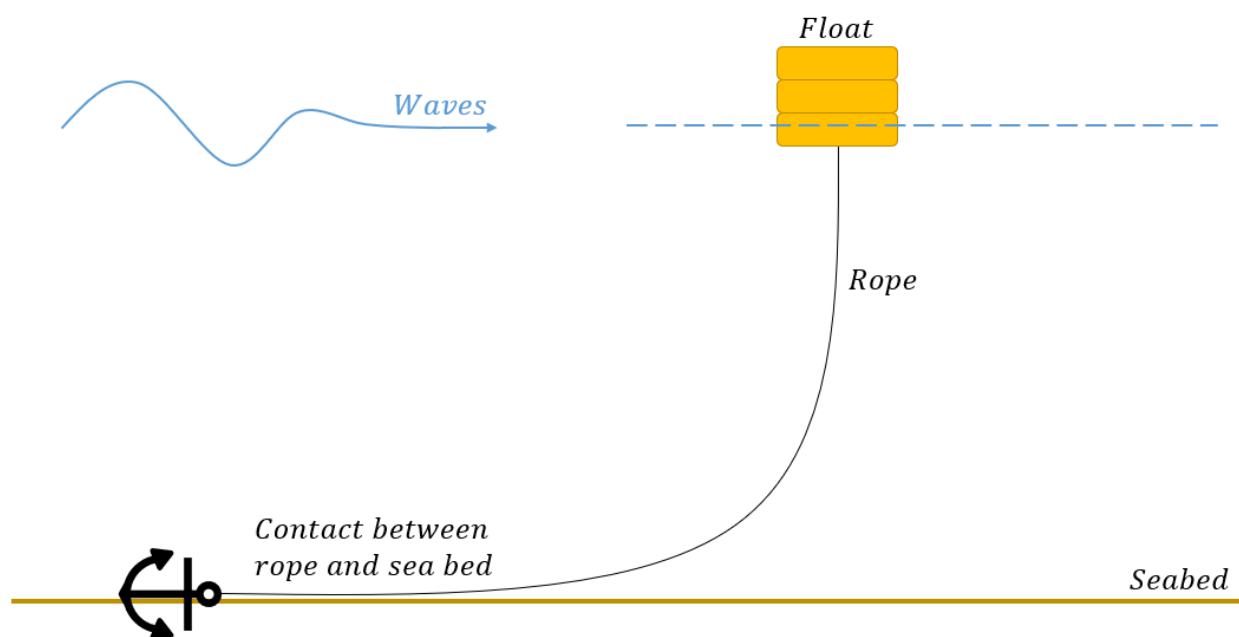


Figure 1

2.3 Principles of Spring: Displaced

In this section, basic principles of Spring: Displaced are explained. The main difference between Spring: Displaced and Bottom contact (found in the Export menu), is the possibility to model a seabed that is not flat. Displaced springs are assigned to nodes individually.

The *Displaced amount* of the node in z-direction is the basis for how the spring works. The user defines a maximum allowed displacement in z-direction (upwards or downwards) and a spring stiffness. The total displacement r of the node has the following relation:

$$r_{TOT} = r_0 + r_{Input} = \sqrt[5]{\frac{F_z}{k_z}} + \text{Displaced amount}$$

where r_0 is the initial displacement due to gravitational force from the rope (or pipe etc.) and r_{Input} is the maximum allowed displacement of the node defined in AquaEdit in the field *Displaced amount*. Furthermore, k_z is the stiffness of the spring in z-direction. Spring stiffness can be applied for all degrees of freedom; translatory and rotational. Rotational stiffness will act as dynamic damping.

2.4 Pre processing

2.4.1 Establish the model in AquaEdit

The model is based on the sketch given in the introduction of this case study. Create one component group of type TRUSS, which should resemble the rope. The main dimensions are described in the figure below.

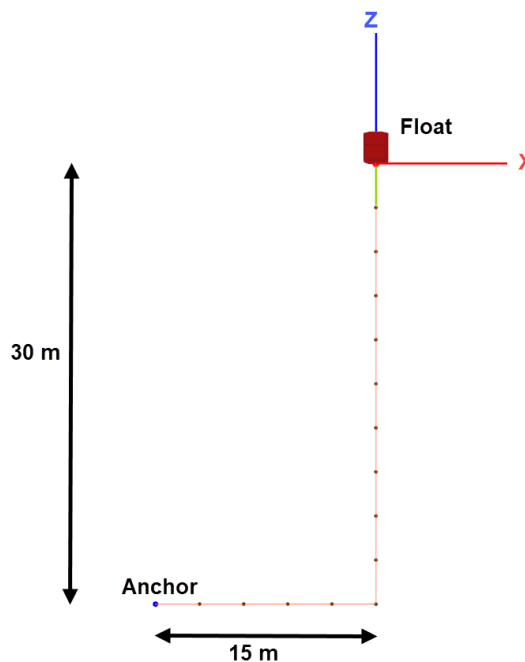


Figure 2

Model according to the figure above. Discretize the horizontal part in 5 elements, and the vertical in 10 elements.

2.4.2 Component properties

Suggested cross sectional properties for the rope is provided in the table below.

1 Rope	
Type	TRUSS
E-modulus	2.1E+09 N/m ²
Area	1.963E-03 m ²
Diameter	50 mm
Weight in air	2.5 kg/m
Weight in water	0.4874 kg/m
Diameter Y / Z	0.05/ 0.05
Drag coefficient Y / Z	1.2 / 1.2
Added mass coefficient Y / Z	1.0 / 1.0

2.4.3 Properties of anchor

To resemble the anchor, this node is for simplicity restrained in all degrees of freedom.

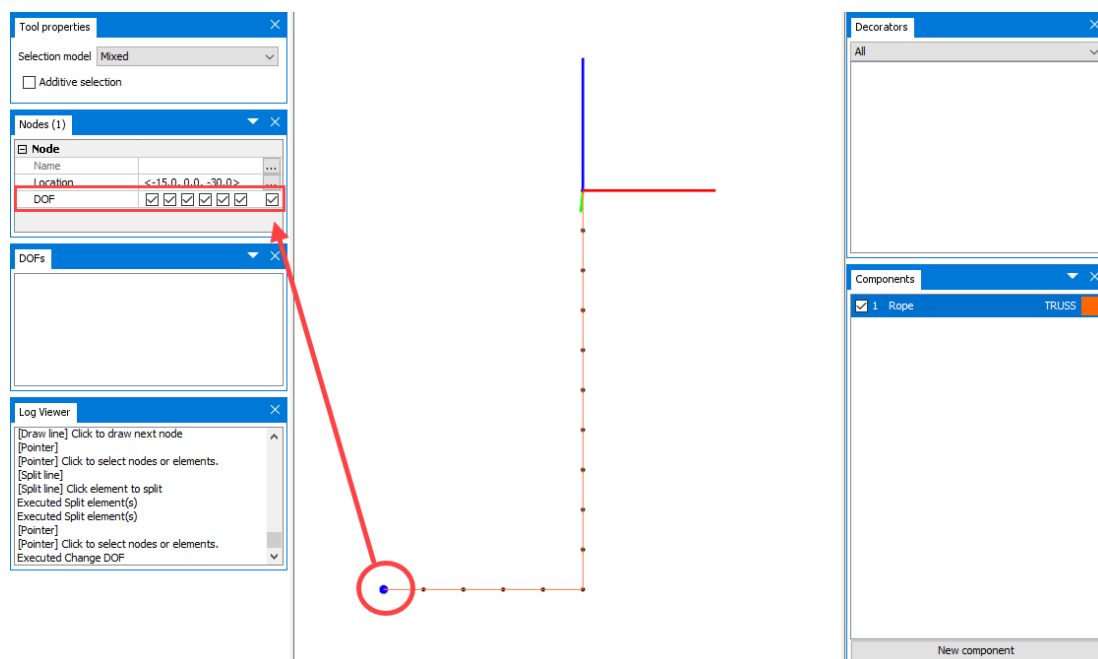
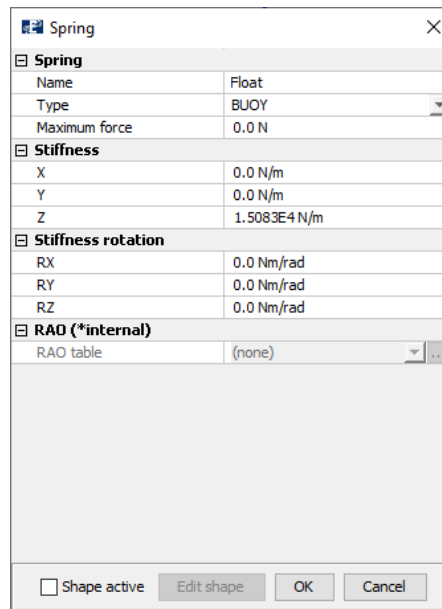


Figure 3

2.4.4 Properties of Float

The float has a rectangular waterplane area of 1.5 m^2 , which correspond to a vertical waterplane stiffness of $A_w \cdot \rho \cdot g = 1.5 \cdot 1025 \cdot 9.81 = 15083 \text{ N/m}$. Create a node decorator spring type Buoy with Stiffness Z equal to 15083 N/m. **Right click** the upper node of the rope > **Node** > **Spring** > **Create new** > **Type: Buoy**.



Spring

Spring	
Name	Float
Type	BUOY
Maximum force	0.0 N
Stiffness	
X	0.0 N/m
Y	0.0 N/m
Z	1.5083E4 N/m
Stiffness rotation	
RX	0.0 Nm/rad
RY	0.0 Nm/rad
RZ	0.0 Nm/rad
RAO (*internal)	
RAO table	(none)

☐ Shape active Edit shape OK Cancel

Figure 4

Select **OK**.

2.4.5 Properties of spring Displaced

In AquaEdit, select the four nodes on the horizontal part of the rope. **Right click** > **Nodes** > **Spring** > **Create new**. By holding **control** while you **click on the nodes**, multiselect is activated.

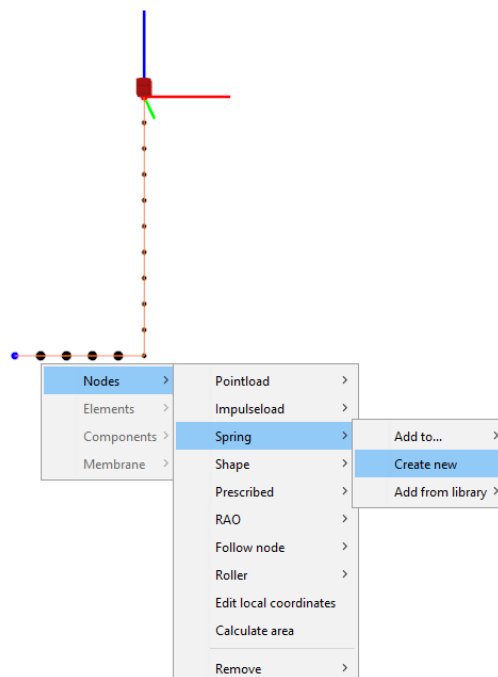
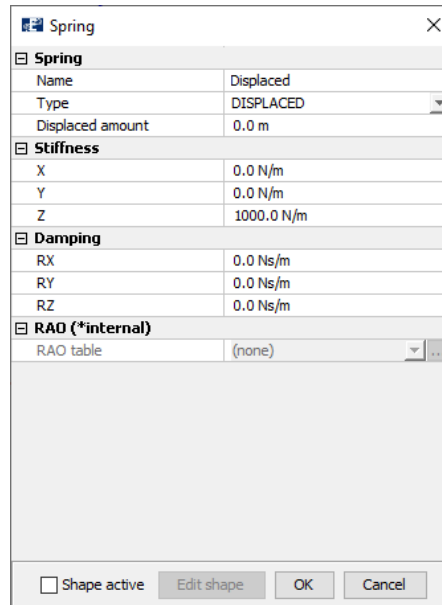


Figure 5

In the Edit spring-window, select **Displaced** as the spring Type. Leave **Displaced amount** to 0.0 m and translatory spring stiffness in z-direction should be 1000 N/m. When the Displaced amount is defined to 0.0, the node will only displace due to gravitational forces of the rope.



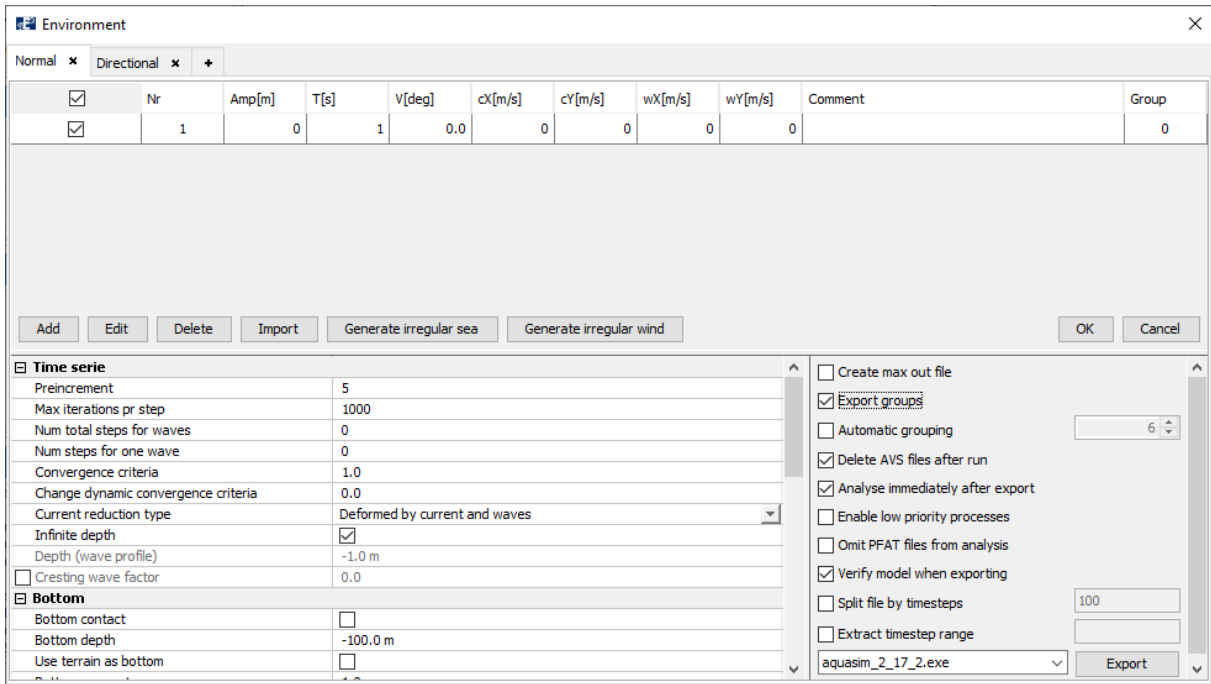
Spring	
Spring	
Name	Displaced
Type	DISPLACED
Displaced amount	0.0 m
Stiffness	
X	0.0 N/m
Y	0.0 N/m
Z	1000.0 N/m
Damping	
RX	0.0 Ns/m
RY	0.0 Ns/m
RZ	0.0 Ns/m
RAO (*internal)	
RAO table	(none)
☐ Shape active Edit shape OK Cancel	

Figure 6

Select **OK**. Remember to save the model from time to time, this will reduce risk of loosing data. The model is now ready for static analysis.

2.5 Calculations static analysis

To check that the modelled system is without substantial errors and that static equilibrium is calculated correctly according to expectations, you should run a static analysis. This analysis is also done to check the initial vertical displacement of the elements close to the seabed. Go to the **Export** menu, navigate to the Normal tab and set **Preincrement** = 5, and **Num steps for waves** = **Num steps for one wave** = 0. Further, activate **Delete AVS files after run** to reduce the amount of data generated from the analysis. Leave the other parameter according to default values.



The screenshot shows the 'Environment' dialog box with the 'Normal' tab selected. The 'Directional' tab is also visible. The 'Time serie' section is expanded, showing the following settings:

Parameter	Value
Preincrement	5
Max iterations pr step	1000
Num total steps for waves	0
Num steps for one wave	0
Convergence criteria	1.0
Change dynamic convergence criteria	0.0
Current reduction type	Deformed by current and waves
Infinite depth	☒
Depth (wave profile)	-1.0 m
Cresting wave factor	0.0

The 'Bottom' section is also expanded, showing the following settings:

Parameter	Value
Bottom contact	☐
Bottom depth	-100.0 m
Use terrain as bottom	☐

The 'Export' section on the right side of the dialog box is also expanded, showing the following settings:

Parameter	Value
Create max out file	☐
Export groups	☒
Automatic grouping	6
Delete AVS files after run	☒
Analyse immediately after export	☒
Enable low priority processes	☐
Omit PFAT files from analysis	☐
Verify model when exporting	☒
Split file by timesteps	100
Extract timestep range	
Export	aquasim_2_17_2.exe

Figure 7

Select **Export** and save the analysis a suitable place on your computer. In this case study we have named the analysis *static_*. **Start** the analysis. The analysis should be finished within seconds.

2.6 Prost processing static analysis

Open the .avz-file in AquaView. Select **Results > Displacement > Displacement Z [m]**. Select the element as indicated in the figure below (on our model this correspond to element no. 4). Start the playback and observe how the model respond; the horizontal part of the rope close to the seabed will slightly displace upwards in vertical direction, and the float will remain at its initial position.

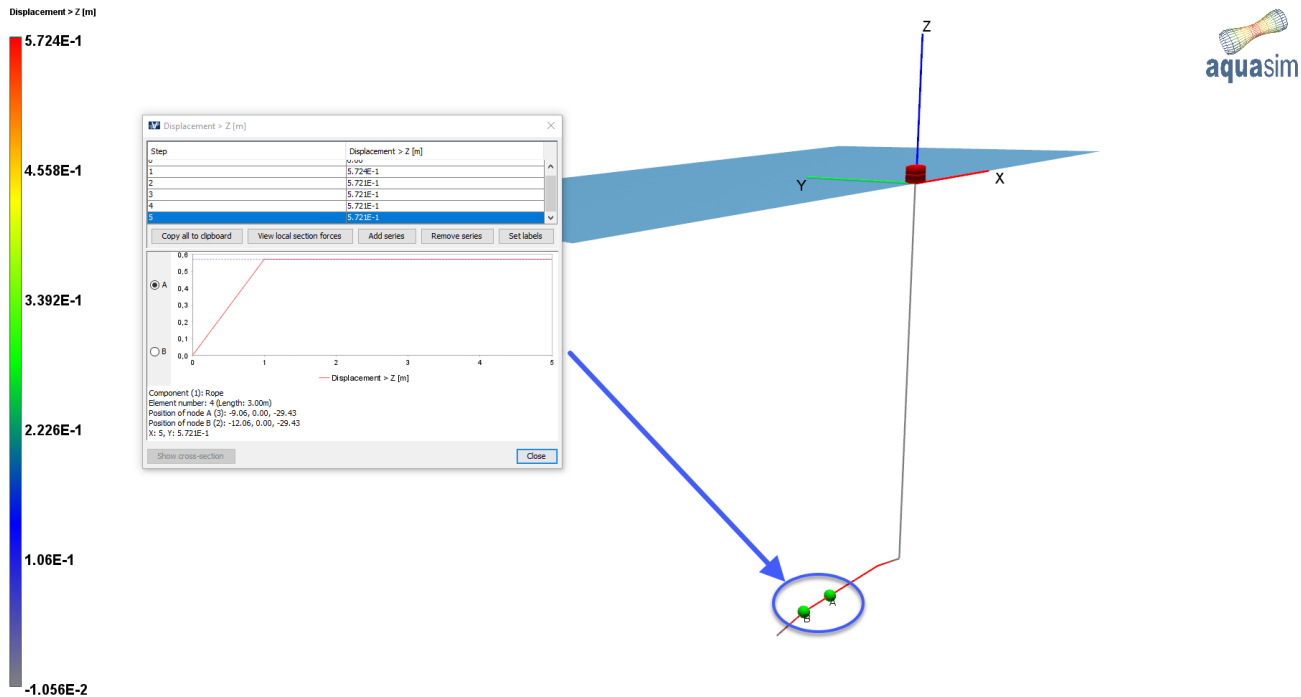


Figure 8

The vertical displacement of the selected node is due to gravitational forces. In our analysis, the displacement is **0.57m** in positive z-direction. Theoretically, the displacement is calculated approximately to be:

$$F_z = m \cdot g = 2.5 \frac{kg}{m} \cdot 3m \cdot 9.81 \frac{m}{s^2} = 73.6N$$

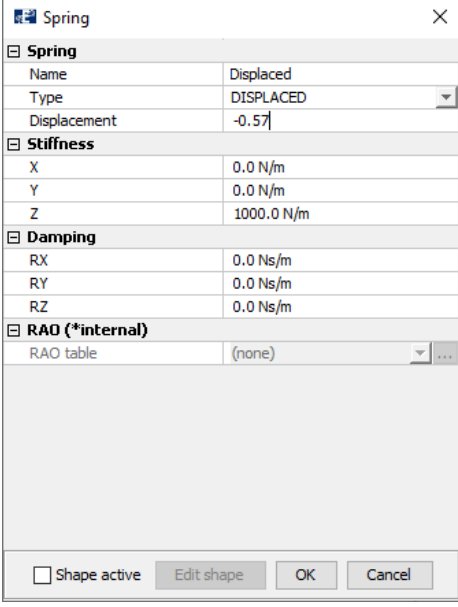
$$r_0 = \sqrt[5]{\frac{F_z}{k_z}} = 0.59m$$

Where m is the weight of the rope kilograms per meter times the length of element no. 4, which is 3 meters. r_0 is the displacement of node A connected to element no. 4 in z-direction.

The first static considerations are finished. **Exit** AquaView and return to your model in AquaEdit.

2.7 Changing Displaced amount

In AquaEdit, re-open the Edit spring-window by **double click** on the **Displaced spring** in the Decorators-window. We should now decide that the nodes close to the seabed should not displace below the depth of -30m. In the field **Displacement** type **-0.57 m**.



The image shows a 'Spring' dialog box with the following fields and values:

Spring	
Name	Displaced
Type	DISPLACED
Displacement	-0.57
Stiffness	
X	0.0 N/m
Y	0.0 N/m
Z	1000.0 N/m
Damping	
RX	0.0 Ns/m
RY	0.0 Ns/m
RZ	0.0 Ns/m
RAO (*internal)	
RAO table	(none)

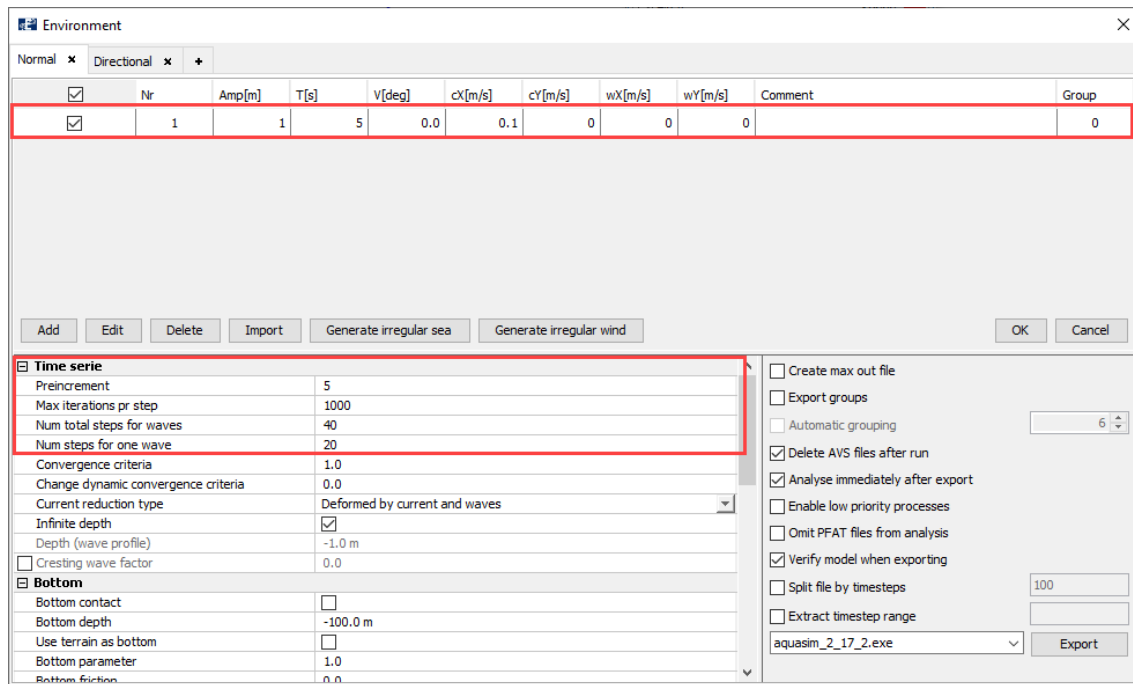
At the bottom, there is a checkbox for 'Shape active' (unchecked), an 'Edit shape' button, and 'OK' and 'Cancel' buttons.

Figure 9

Select **OK**. The model is now ready for dynamic considerations. Select **Export** from the Toolbar menu.

2.8 Calculations dynamic analysis

You should now prepare a dynamic analysis with waves and current. Go to the Normal-tab in the Environment-window. In the field where the wave- and current parameters are defined, type a wave amplitude of 1 m, wave period of 5 seconds and current 0.1 m/s in x-direction. Leave **Preincrement** = 5, and type **Num total steps for waves** = 40 and **Num steps for one wave** = 20. Then press **Export** and save the analysis a suitable place. In this case study we have named the analysis *dynamic_*. **Start** the analysis. The analysis should be finished within seconds.



☒	Nr	Amp[m]	T[s]	V[deg]	cX[m/s]	cY[m/s]	wX[m/s]	wY[m/s]	Comment	Group
☒	1	1	5	0.0	0.1	0	0	0		0

Time serie
 Preincrement: 5
 Max iterations pr step: 1000
 Num total steps for waves: 40
 Num steps for one wave: 20
 Convergence criteria: 1.0
 Change dynamic convergence criteria: 0.0
 Current reduction type: Deformed by current and waves
 Infinite depth: ☒
 Depth (wave profile): -1.0 m
 Cresting wave factor: 0.0

Bottom
 Bottom contact: ☐
 Bottom depth: -100.0 m
 Use terrain as bottom: ☐
 Bottom parameter: 1.0
 Bottom friction: 0.0

☐ Create max out file
☐ Export groups
☐ Automatic grouping
☒ Delete AVS files after run
☒ Analyse immediately after export
☐ Enable low priority processes
☐ Omit PFAT files from analysis
☒ Verify model when exporting
☐ Split file by timesteps
☐ Extract timestep range
 aquasim_2_17_2.exe

Figure 10

Select **Show messages** to check for possible convergence issues.

2.9 Post processing dynamic analysis

Open the .avz-file in AquaView. Select **Results > Location > Position Z [m]**. Select the same element as from the static analysis (in our model this correspond to element no. 4). Start the playback and observe how the bottom nodes displace when exposed to waves and current. The position of the node A and B in element no. 4 will not exceed the depth of -30m substantially.

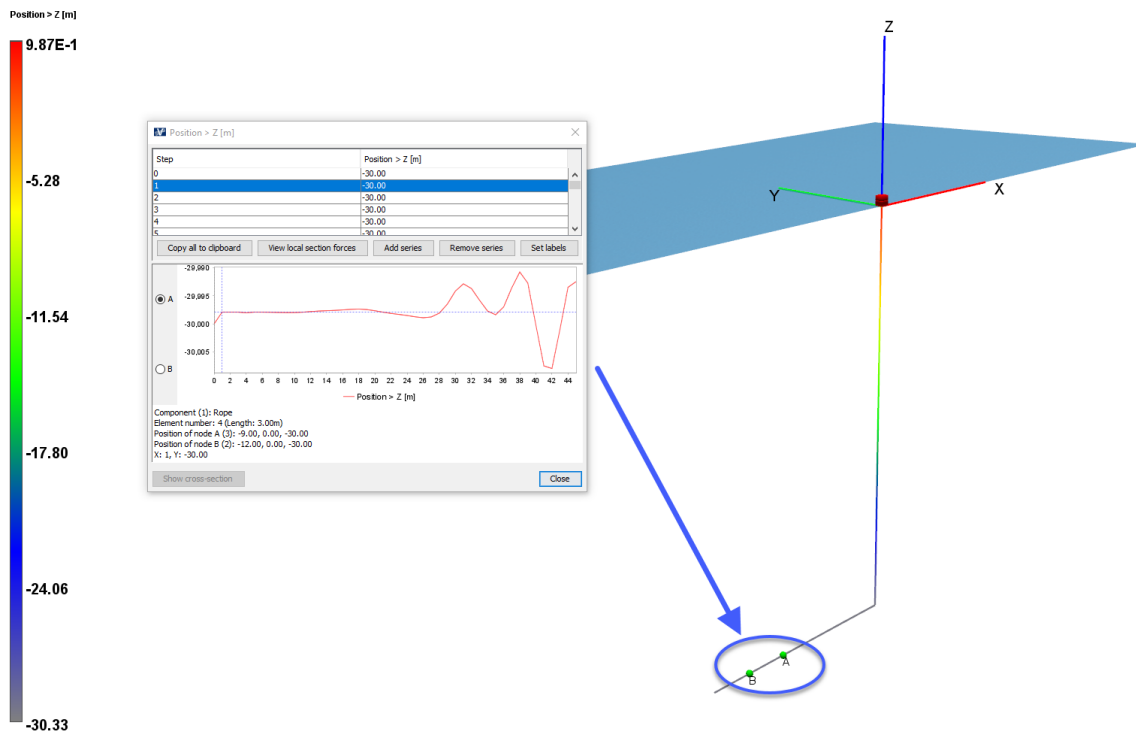


Figure 11

The case study tutorial is finished.

2.10 Summary

In this case study you have established a system consisting of a float and a rope anchored to the seabed. The aim of this case study was to demonstrate how contact between the rope and seabed can be modelled with the use of spring type Displaced. When the rope approaches the defined maximum allowed displacement the spring stiffness increase.

The purpose of this case study was to demonstrate how the **Displaced** spring can be applied to render contact with the seabed. The demonstrated case only considers one spring dedicated to several nodes. The user can add several springs with different type of input-values to simulate seabed with varying depths.

It should be noted that, as for any other dynamic simulations, that applying too high values of stiffness in the spring, in combination with a small value for the allowed amount of displacement, can cause numerical challenges. The model can be too stiff and cause convergence issues. To find a good combination, try with different input-values.

3 Revision comments

Revision no.	Comment
1.0	First publication

--- End of document ---