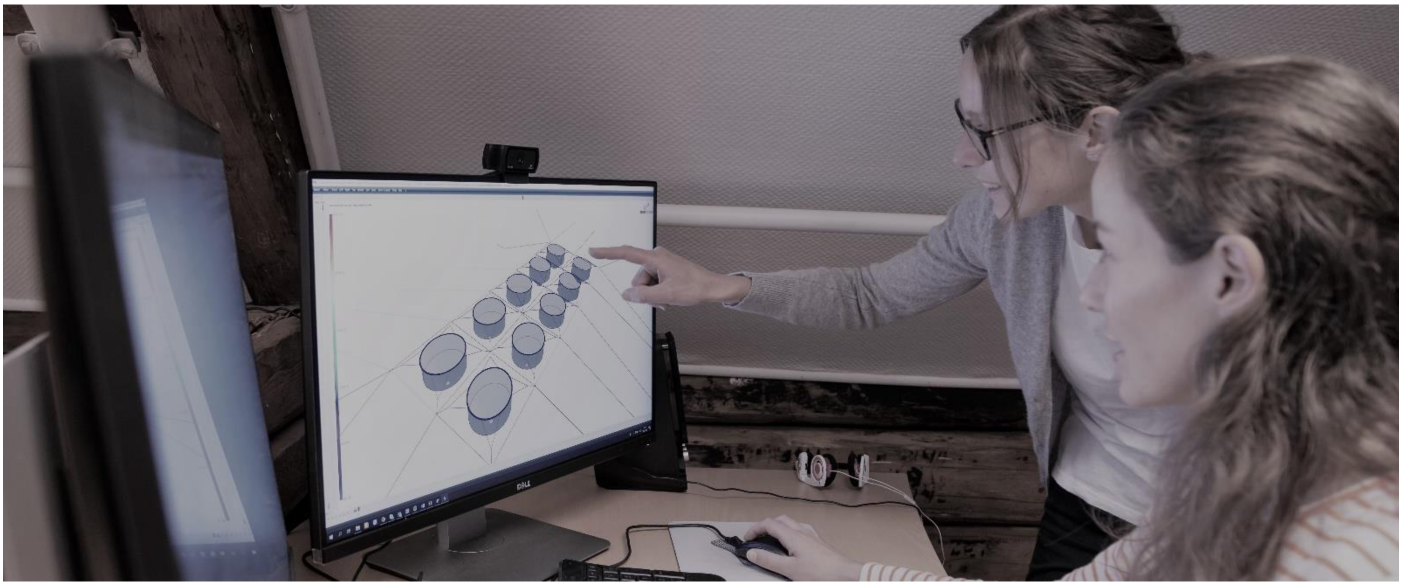
AquaSim training courses

- **Component contact: Falling box**



Revision: 1.0

AquaSim version: 2.18

Aquastructures AS
Kjøpmannsgata 21, 7013 Trondheim
Norway

Content

1 Prerequisites 3
2 Case study – Component contact 3
2.1 Learning objectives 3
2.2 Introduction 3
2.3 Principles of Component contact 4
2.4 Pre-processing 5
2.5 Analysis 10
2.6 Post processing analysis 11
2.7 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 – Component contact

2.1 Learning objectives

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

- Model a system with a falling object on a net
- Establish a Component contact-table between two components
- Contact contact-table:
    - First part
    - Second part
    - Distance/Radius
    - Stiffness
    - Damping stiffness
    - Dynamic friction damping
- Evaluate the forces that arise due to contact in AquaView

2.2 Introduction

Using component contact enable to perform analyses where contact and collision between elements are of significance. This can be contact between ship and quay, mooring line contact, falling equipment and so on. AquaSim holds the opportunity to define that there should be contact between selected component groups.

In this case study you will define contact between a falling object and a net. The object could be some sort of box or equipment, and the net could be for safety reasons. When the box hits the net, kinetic energy will be transferred from the box to the net and be distributed to the suspension ropes holding the net. You are to establish a table which defines contact between the box and the net and investigate force and displacement in AquaView.

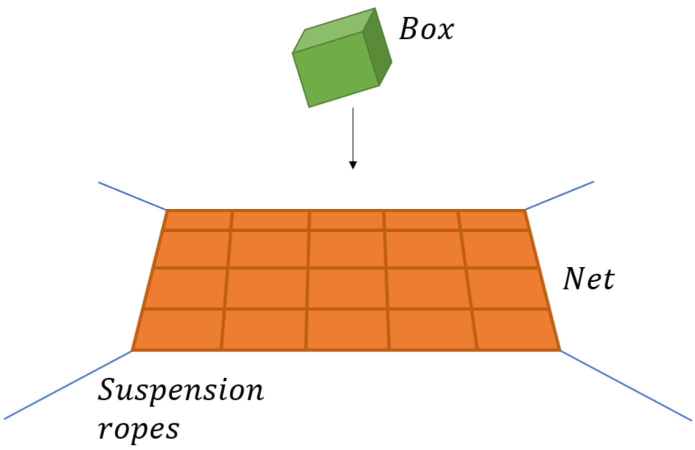


Figure 1

2.3 Principles of Component contact

The basic principle of establishing contact between elements in AquaSim is to first define contact between component groups in AquaEdit. Two-and-two component groups can be assigned contact. From this a Component contact-table is created which defines how the contact should take place.

Contact between elements in the component groups is based on a spring, as illustrated in the figure below. We have the following relation between force, spring stiffness and displacement:

$$F = k \cdot r^5$$

where F is the contact force, k is the spring stiffness and r is the relative distance between the elements. In addition, you can also assign friction, damping and distance when contact should start.

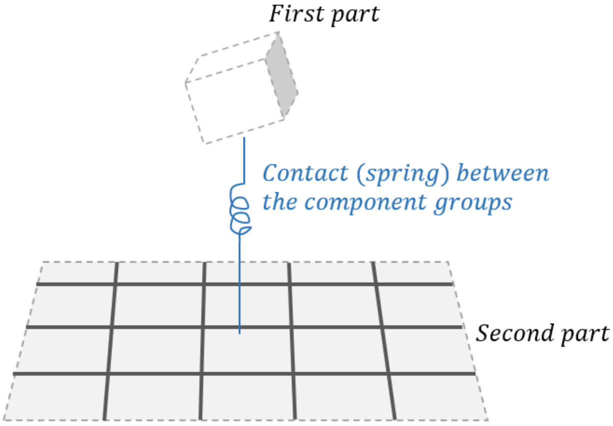


Figure 2

To establish contact, you need to select a “First part” and a “Second part”. The choice of order is for most practical purposes indifferent. AquaSim will keep track of the position and distance between the “First part” and “Second part” in all timesteps in the analysis, this is illustrated in the figure below. If elements in “First part” gets closer to “Second part” than the specified values for where contact is assumed a force will push the elements apart.

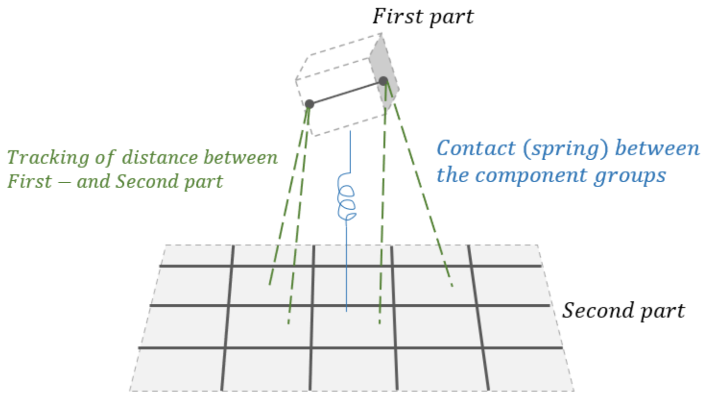


Figure 3

2.4 Pre-processing

2.4.1 Modelling in AquaEdit

Open the .amodel that is associated with this case study in AquaEdit. The model consist of three component groups; Falling box, Net and Suspension rope.

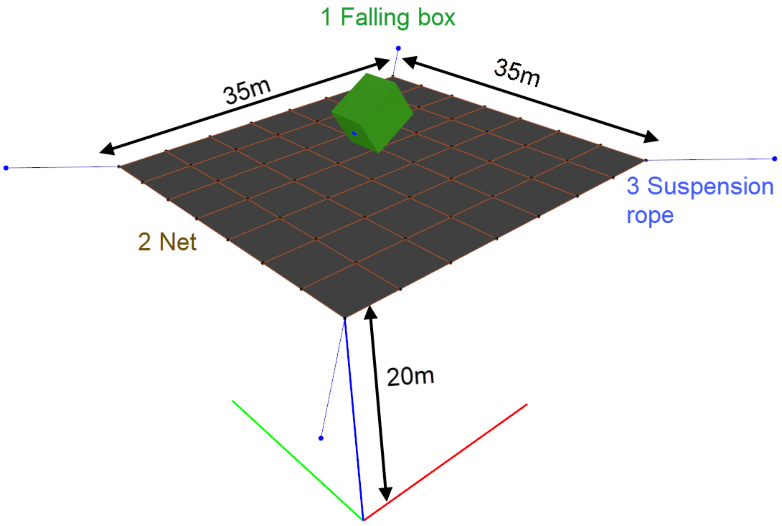


Figure 4

2.4.2 Component properties

The box is modelled as a component Beam with an arbitrary angle in the xz-plane, to illustrate a random drop. The chosen load formulation for the Beam is Morison submerged. In combination with the Water volume correction: With slamming, AquaSim will automatically adjust the properties that is valid in air. Properties of the beam is shown in the table below:

1 Falling box	
Type	BEAM
Height x width	4 m x 4 m
E-modulus	2.1E+11 N/m2
Area	16 m2
Volume	16 m3/m
Weight in air	20 kg/m
Weight in water	-1.638E4 kg/m
Drag coefficient Y / Z	1.2 / 1.2
Adde mass coefficient Y / Z	2.0 / 2.0

The Net is modelled as a conventional net of type Normal in combination with load formulation Normal.

2 Net	
Type	Normal
Load formulation	Normal
Height x width	35 m x 35 m
E-modulus	1.0E+09 N/m2
Thread diameter	2.0E-3 m
Growth coefficient	1.0 -
Maskwidth Y / Maskwidth Z	0.025 m / 0.025 m
Net in air	Checked

The Suspension rope is modelled as trusses. The purpose of the suspension trusses is to restrain the net from falling. The truss is made fairly soft, with a low E-modulus. That is because they are meant to act as shock absorbers.

3 Suspension rope	
E-modulus	2.1E07 N/m2
Area	2.46E-03 m2
Weight in air	2.62 kg/m
Weight in water	0.0985 kg/m
Diameter Y / Z	0.056 m / 0.056 m
Drag coefficient Y / Z	1.2 / 1.2
Added mass coefficient Y /Z	1.0 / 1.0

2.4.3 Add Spring type OFFLOADED

For the current model-setup the box is free to translate and rotate in any direction. This will cause large initial translations, and potential unstable analysis which can lead to divergent results. To dampen this, one can apply a spring of type OFFLOADED. This spring is constant in the initial steps. In the dynamic part of the analysis, the effect of this spring is removed.

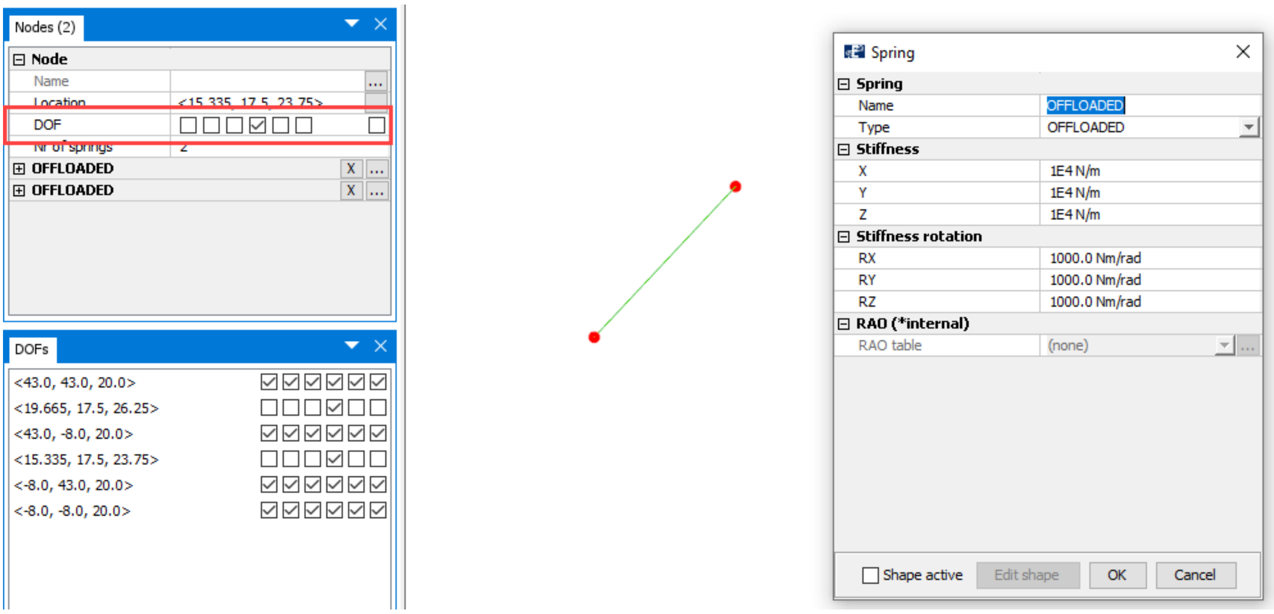


Figure 5

The end-nodes are also restrained from rotation about x-axis.

2.4.4 Add component contact

Contact between Falling box and Net should now be established. Adding contact between two component groups can be done in two ways in AquaSim:

1. Select the two component groups from the Components-window > **Right click** > **Operations** > **Add contact**.

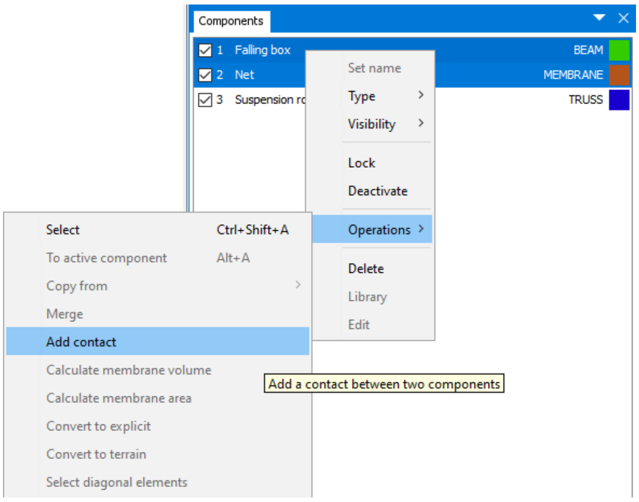


Figure 6

2. Alternatively, select **Tools** > **Tables** > **Show component contacts** and the select **Add**.

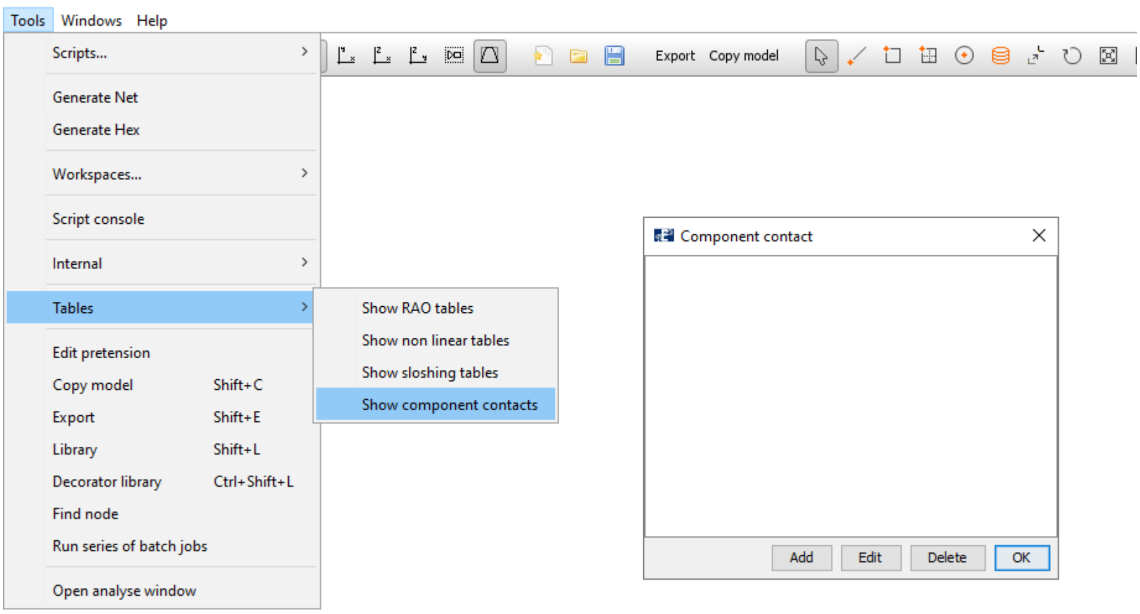


Figure 7

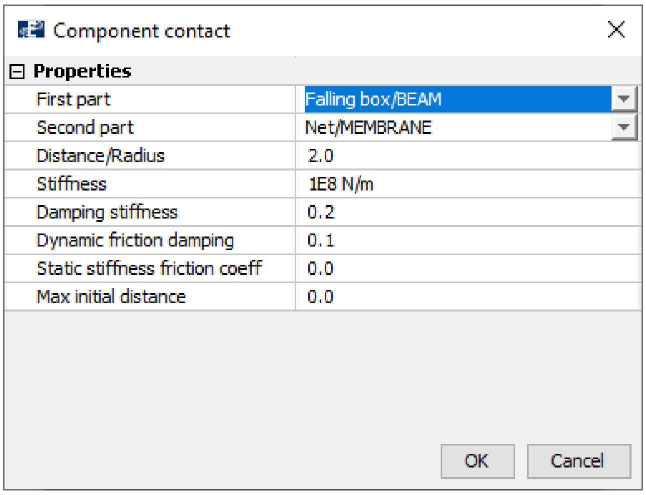


Figure 8

Component Falling box is selected as **First element** and Net as the **Second element**:

- **Distance/ Radius:** defines when contact should occur. Aquasim treats this parameter a bit different depending on the type of component group that is included here. If contact is defined between a Beam (or Truss) and Beam (or Truss) this parameter can be seen as a radius to Second element in which contact should start. In this case, we have contact between Beam and Membrane. Then this parameter defines at what distance between the Beam and the surface of the Membrane. When this equals 2.0, then contact will start when one of the nodes on the beam are 2.0 meters from the Membrane-surface. If it is 0.0, then contact start when the distance is 0.0 m. In our case, we have a rectangular shaped box of 4 x 4 m, and since we want contact to start when the outer edge of the cross section hits the Net, we set **Distance / Radius = 2.0**. Unit is meters.
- **Stiffness:** is the spring stiffness of the contact. The higher the stiffness, the tighter the contact.
- **Damping stiffness:** provides damping in the direction normal to the contact. This factor is proportional to Stiffness. If an impact is elastic, the box will bounce back and if the impact is plastic the energy of the impact will be drained, and the box will not bounce back. We assume some damping and set **Damping stiffness** = **0.2**.
- **Dynamic friction damping:** this is a factor that is proportional to the stiffness in the tangential direction to the element. This can be interpreted as a friction that prevents the box from sliding back to the impact point. The factor is dependent on the relative velocity between the colliding elements, and hence only works when we have a dynamic situation. We assume that when the box hits the net, it can to some extent slide and set **Dynamic friction damping** = **0.1**.
- **Static stiffness friction coefficient:** is a friction force coefficient that is proportional to the relative displacement between the colliding elements. This coefficient is constant and pulls the object back to the impact point. We do not assume such effect in this case and keep **Static friction coefficient** = **0**.
- **Max initial distance:** when establishing contact between two component groups, AquaSim, will keep tract of the position and relative distance between all elements in all timesteps. If you have a large model with many elements, this can contribute to slow down the analysis time. Max initial distance adjusts when AquaSim should start to include contact between elements in the two component groups. If this is equal to 0, contact will be established from the first time-step. If this

is greater than 0, contact will only be established when the distance between the elements is equal or less than this value. Since this is a small model, we keep **Max initial distance** = **0**.

We accept this table and select **OK**.

2.5 Analysis

Go to the **Export** menu to open the **Normal** tab in the Environment widow. We should now conduct a dynamic analysis. Analysis is run without current, waves or wind. This is seen in the highlighted load line section.

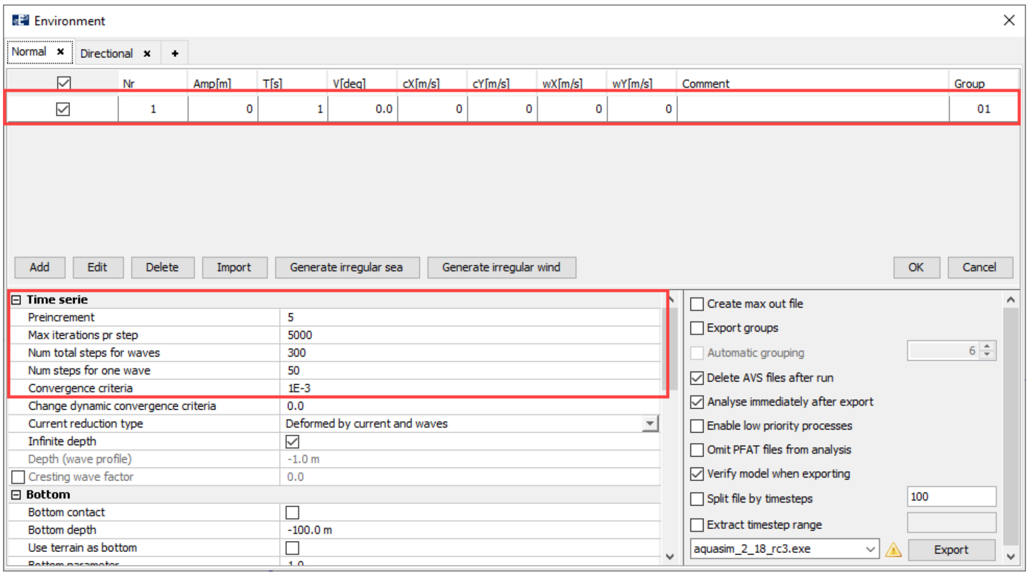


Figure 9

We set **Preincrement** = **5**. As the vertical displacement of the box will be large over a short period of time, one must consider increasing the number of steps in the dynamic part and reduce the convergence criteria. Setting **Num total steps for waves** = **300** and **Num steps for one wave** = **50** will result in an analysis of 6 seconds when wave period T = 1. This should be enough for the box to hit the net.

Select **Export** in the lower right corner and save the analysis a suitable place on your computer. Select **Export** and **Start** the analysis.

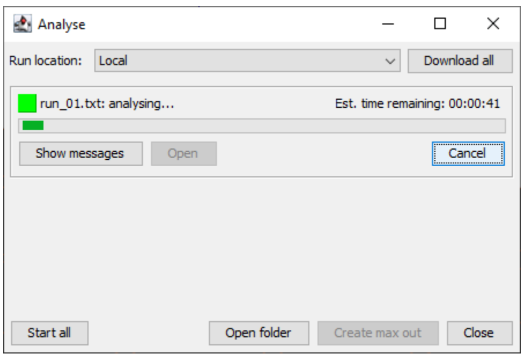


Figure 10

The analysis should take a couple of minutes to finish.

2.6 Post processing analysis

We open the .avz-file in AquaView and select **Result** > **Local section forces** > **Axial force [N]** to see how the axial force in the Suspension rope develop when the box hits the net. **Start the playback** and watch the animation.

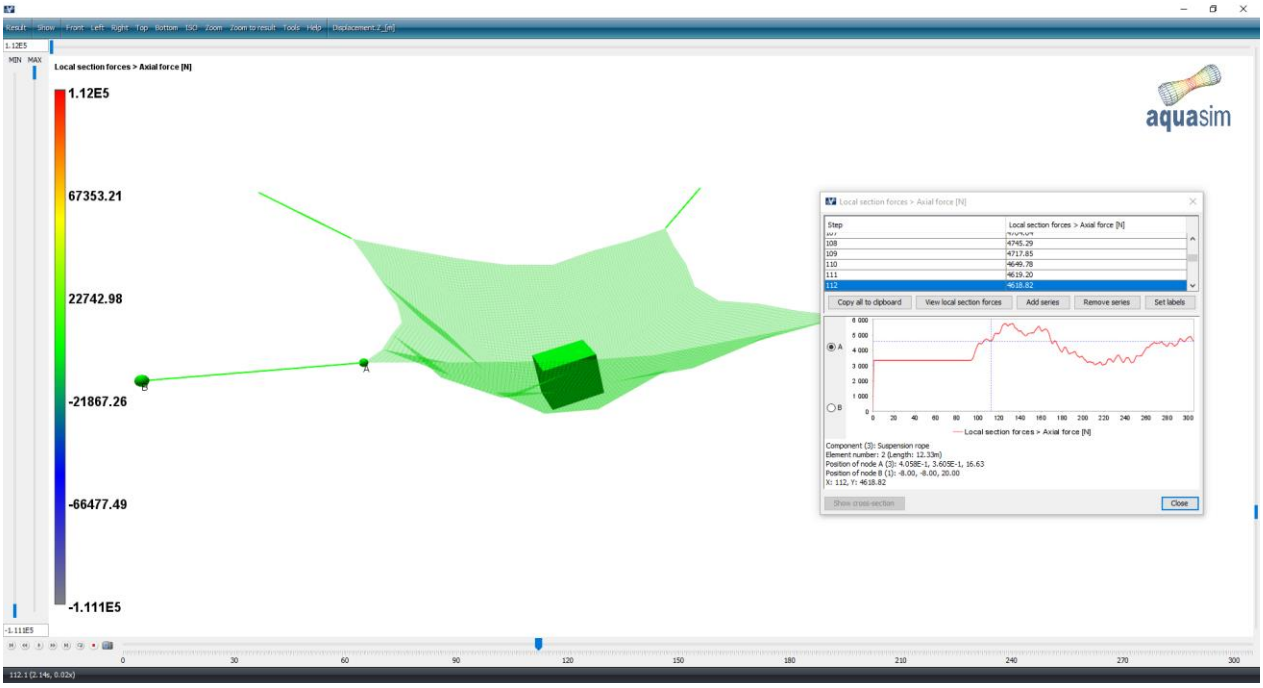


Figure 11

In the figure above, the axial force in one of the Suspension ropes are plotted. One can see that the forces increase and fluctuates when contact between the box and net is established. There are some fluctuations in the force, and the time-stepping settings may need more assessing to find the best fit. Fluctuations can also be due to the Suspension ropes being soft and that resonance effects may play a role. The user is encouraged to first establish simplified models of the problem investigated and conduct sensitivity studies to find a suitable combination for time-stepping and impact-parameters for the contact.

Other results in AquaView that is of interest is the ones found in **Result** > **Contact**. Here you can plot relative distance between elements, forces that arise from impact and damping forces. In the below figure, the relative distance between the box and the net is shown.

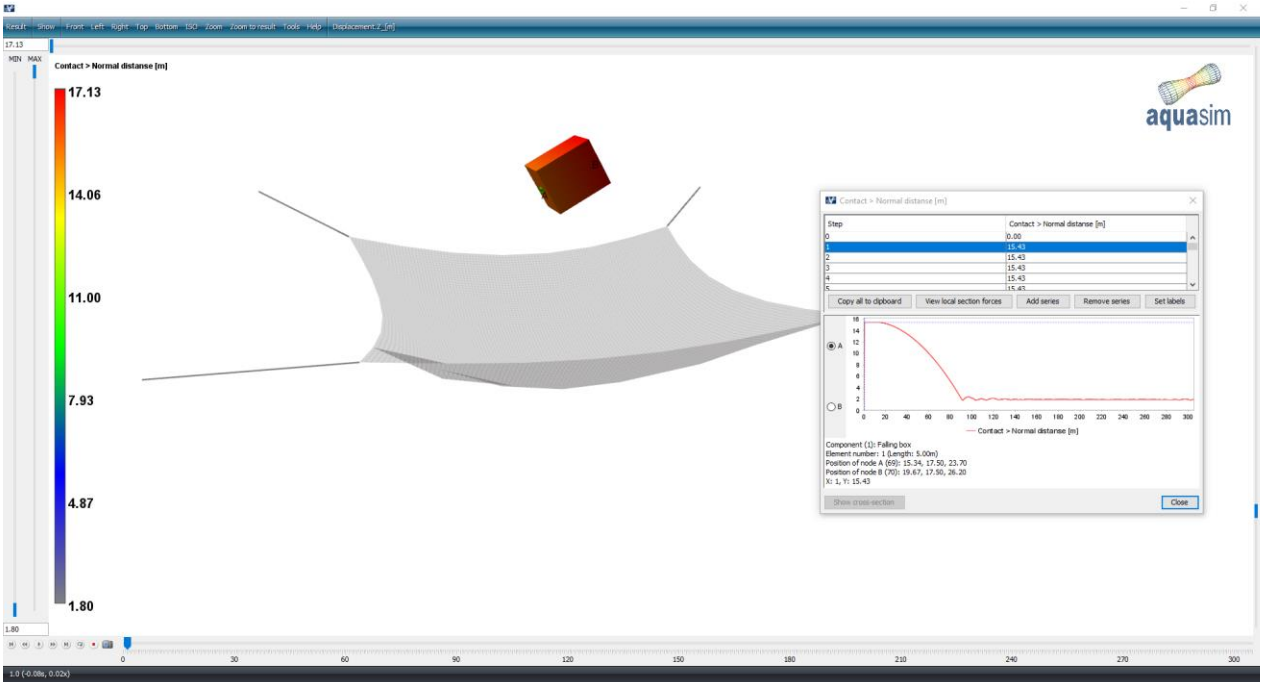


Figure 12

2.7 Summary

This case study introduces how contact can be established between elements in AquaSim. This is done by analyzing a box that is free to fall into a net. The basic principles of establishing a Component contact-table with its parameters are explained. When deciding for values of these parameters, the user should assess the nature of the problem; what type of structures are to interact and material properties. Can the impact be considered elastic or plastic, or something in between? Is damping substantial? In addition, the time-stepping settings in the Export menu should be subject to investigation; should the number of steps be increased, and the convergence criteria lowered to make sure that essential response is captured? This can be evaluated through a sensitivity study for example.

At last, an analysis is run to demonstrate how impact and collisions can be resolved and which result options that is available for contact simulations.

3 Revision comments

Revision no.	Comment
1.0	First publication

--- End of document ---