



PennState
College of Engineering

ME 563: Finite Elements in Engineering

Application of the Finite Element
Method to Real World Problems

Exploring Smoothed Particle Hydrodynamics (SPH)

Contents

1. Part Creation and Modification.....	3
2. Material and Section Properties	6
3. Creating the steps.....	10
4. Contacts.....	12
5. Applying Loads and BCs.....	14
6. Field output requests.....	17
7. Meshing.....	17
8. Editing Keywords.....	20
9. Create job and submit it.....	22
10. Post-Processing	22

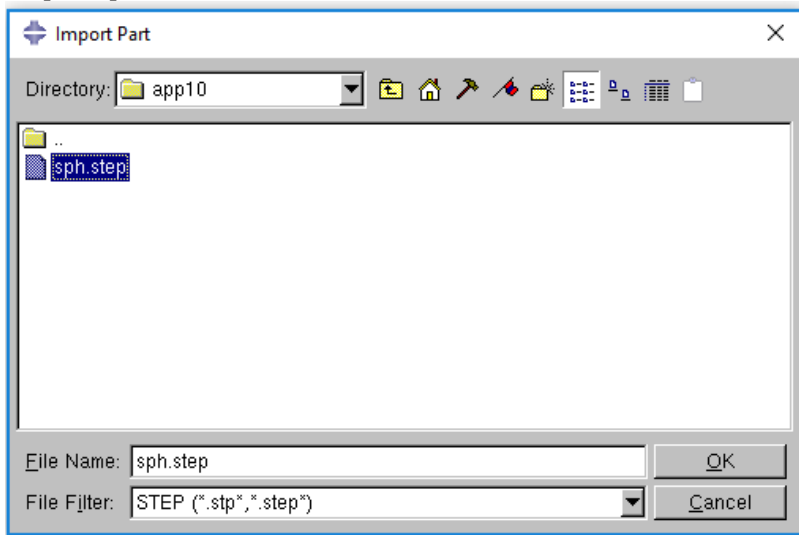
This tutorial gives a basic introduction to SPH modelling in Abaqus CAE. The tutorial will take you through a basic model of g forces acting on a fluid in a typical tank.

1. Part Creation and Modification

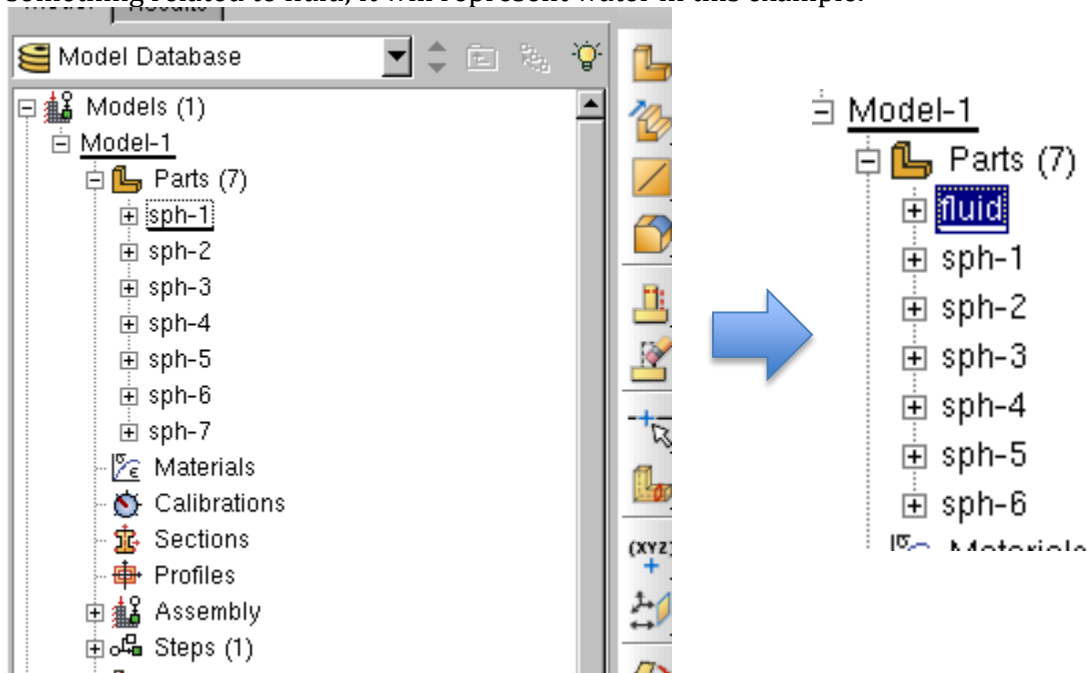
Obtain files from Github:

```
git clone https://github.com/rhk12/sph
```

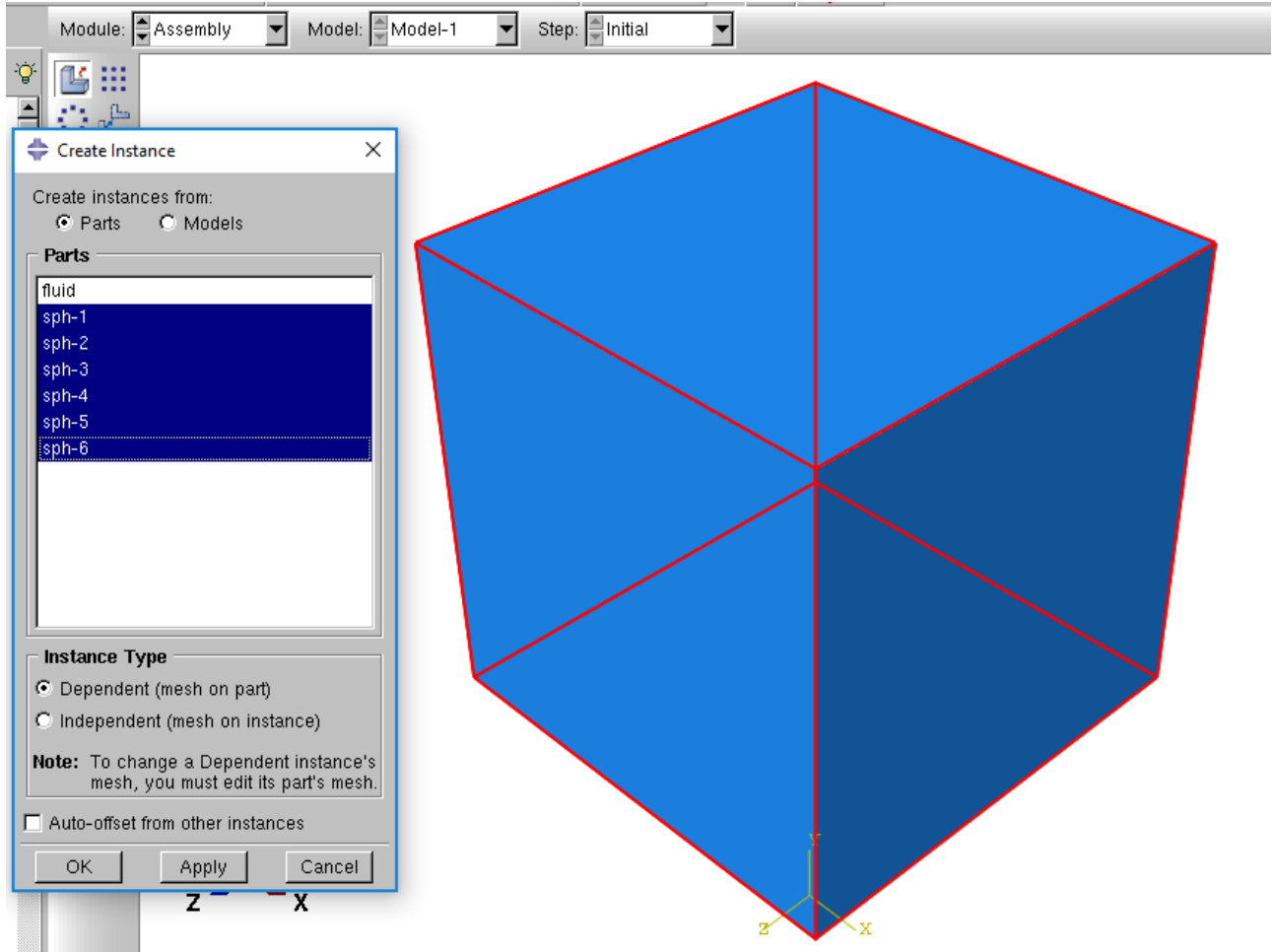
Import part:



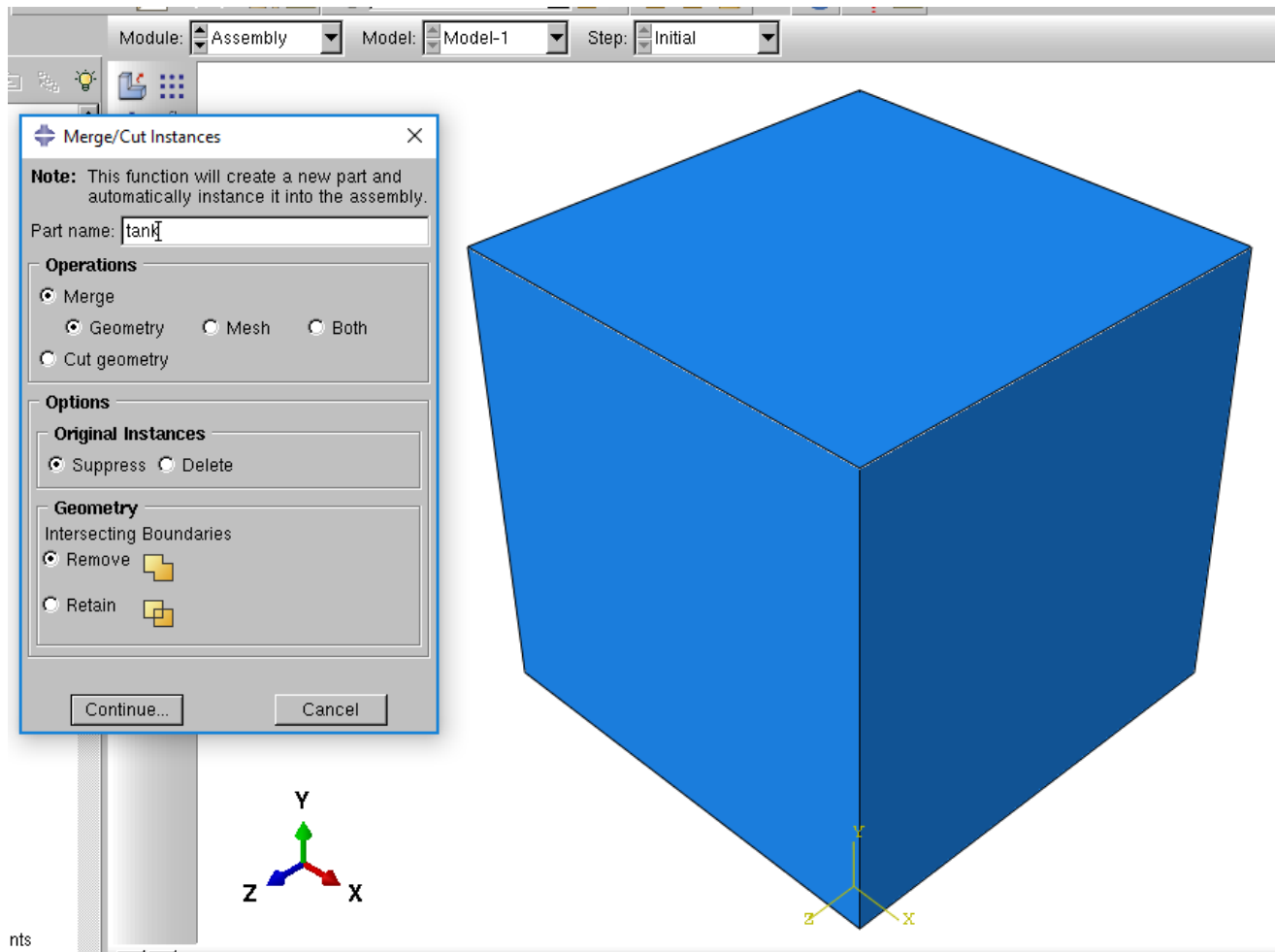
There are 7 different parts. This contains a block and cube made up six faces, rename the block to something related to fluid; it will represent water in this example.



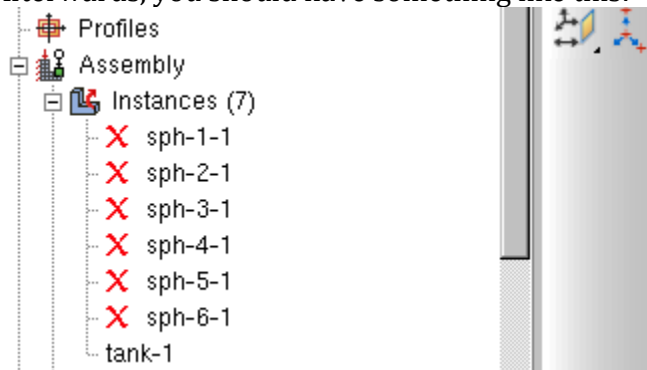
The next step is to combine the faces into a new part to form a box.



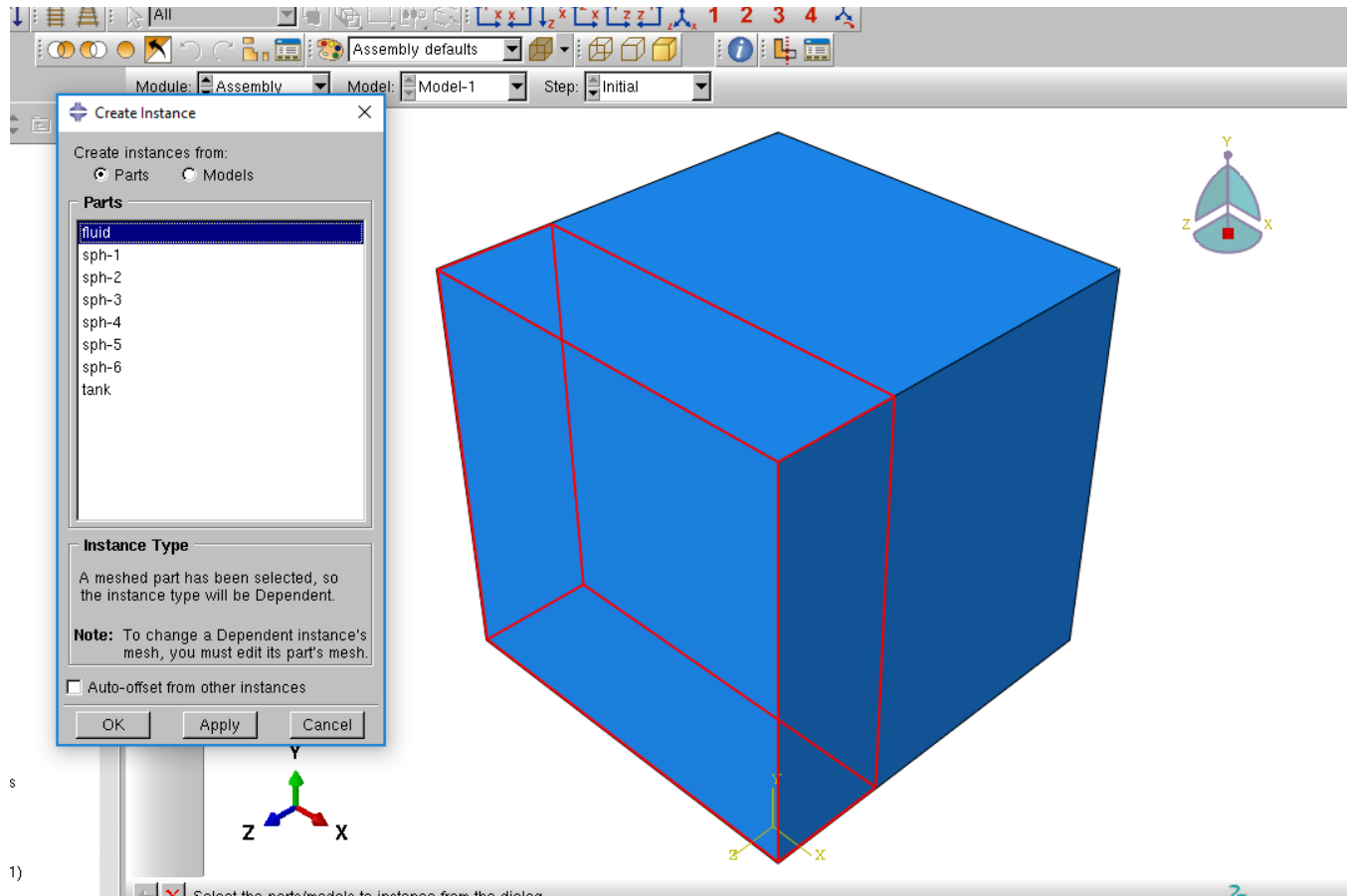
First instance the parts in the **assembly module**, and then once the parts are instanced select the **merge/cut** from the instance drop down menu



Afterwards, you should have something like this:



Now instance the fluid in



2. Material and Section Properties

Water:

Mechanical properties:

- Eos: type = Us – Up, c0 = 1483, s = 0, Gamma0 = 0
- Dynamic viscosity = 0.001

General properties:

- Density = 1000


Aluminum:

Mechanical properties:

- Elasticity-Elastic: Young's Modulus = 70e9, Poisson's ratio = 0.3

General properties:

- Density = 2500

 **Edit Material**

Name: water

Description:

Material Behaviors

Eos

Density

General Mechanical Thermal Electrical/Magnetic Other

Eos


Type: Us - Up

Data

	c0	s	Gamma0
1	1483	0	0

Note: When it comes to assigning sections ignore the 2D squares that merged into the box as they are no longer counted as part of the assembly.

Then create a default solid homogeneous section for water and assign the section to the block.

 **Create Section** X

Name: water

Category

☒ Solid

☐ Shell

☐ Beam

☐ Other

Type

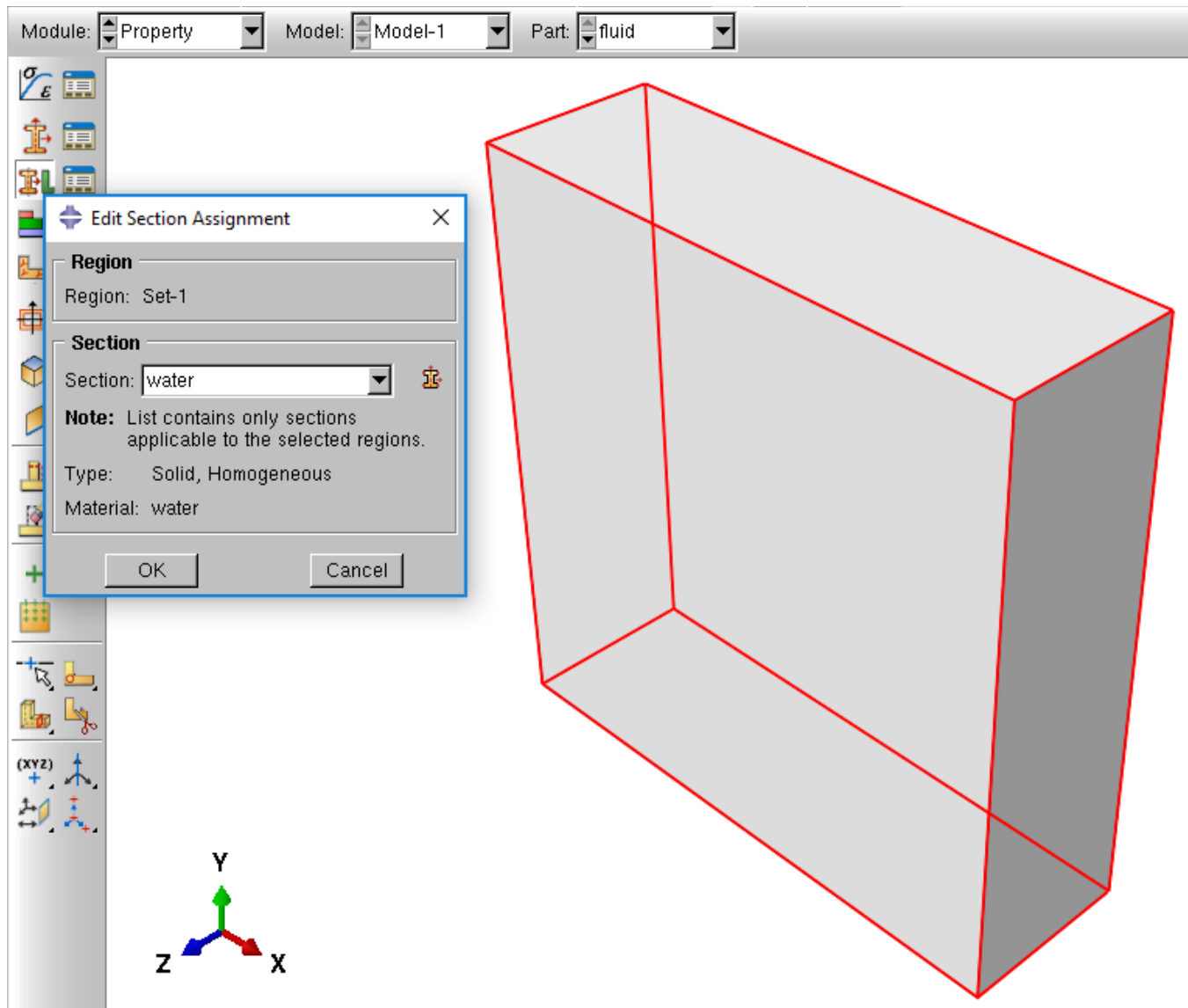
Homogeneous

Generalized plane strain

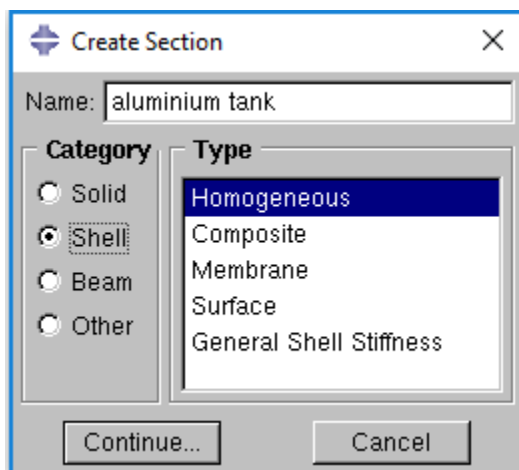
Eulerian


Composite

Continue... Cancel



Then for the aluminum selection, select the '**shell**' option with width of 0.002m and highlight the box



 Edit Section ✕

Name: aluminium tank

Type: Shell / Continuum Shell, Homogeneous

Section integration: ☒ During analysis ☐ Before analysis


Basic **Advanced**


Thickness

Shell thickness: ☒ Value:

☐ Element distribution:


☐ Nodal distribution:

 $f(x)$

Material: 

Thickness integration rule: ☒ Simpson ☐ Gauss

Thickness integration points:

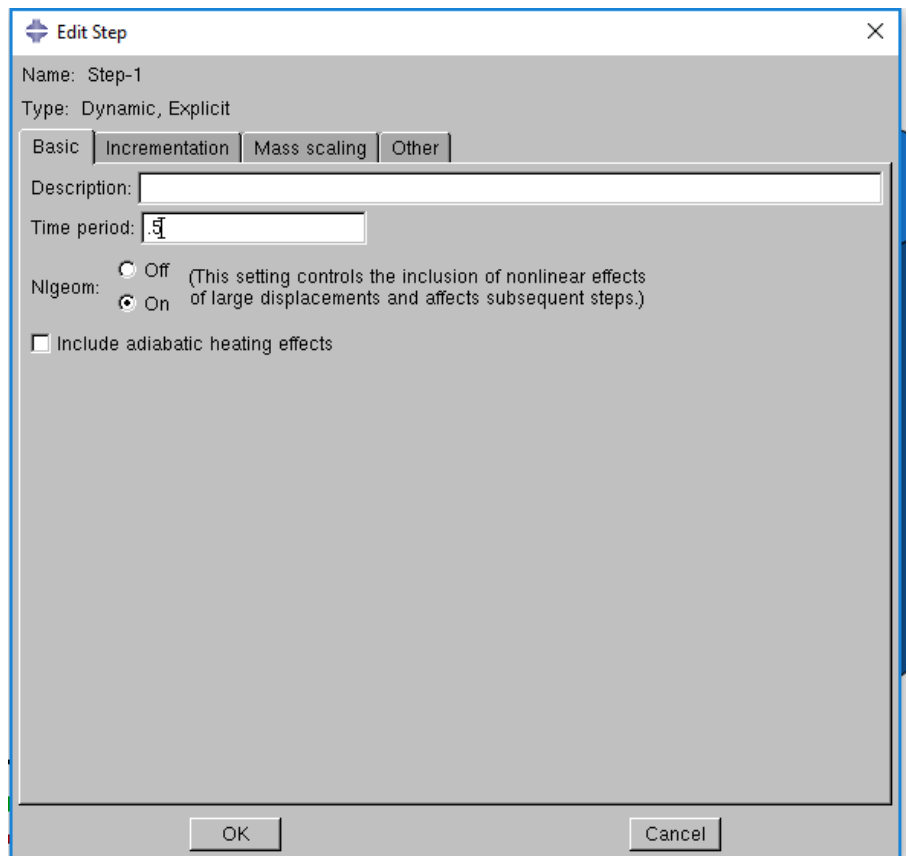
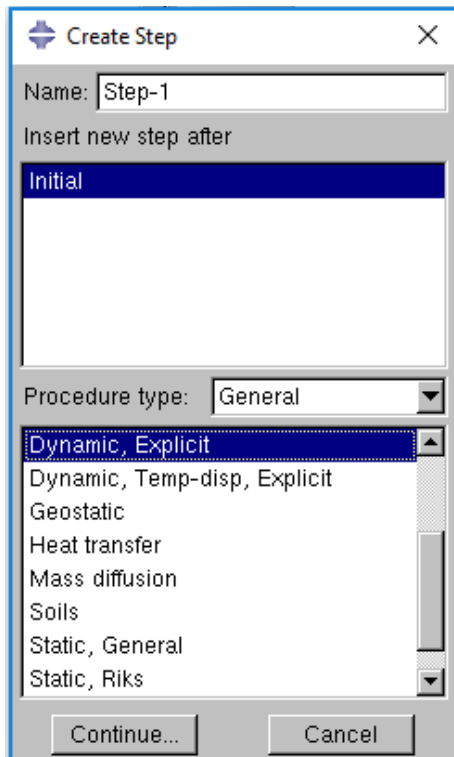
Options: 

OK Cancel

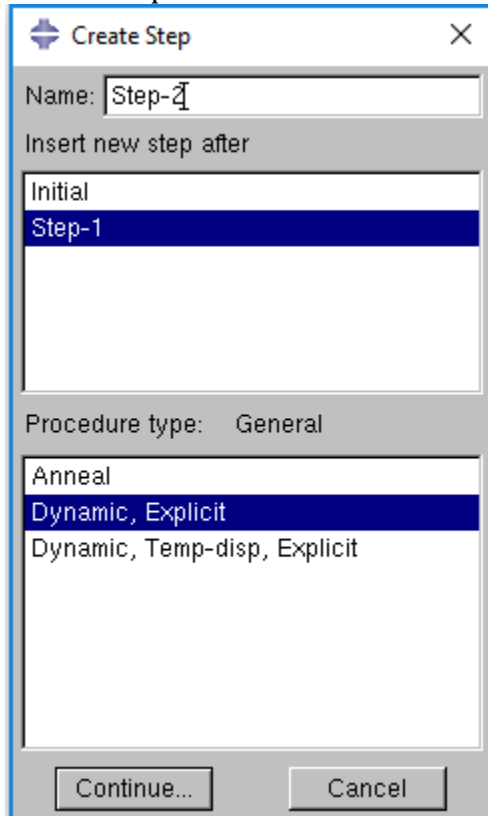
3. Creating the steps

Go to the step module and create two new steps, for all steps in the model use the step option of **'dynamic, explicit'**

For the first step make the time period 0.5 seconds and 1 for the second step



Second step...



Create Step

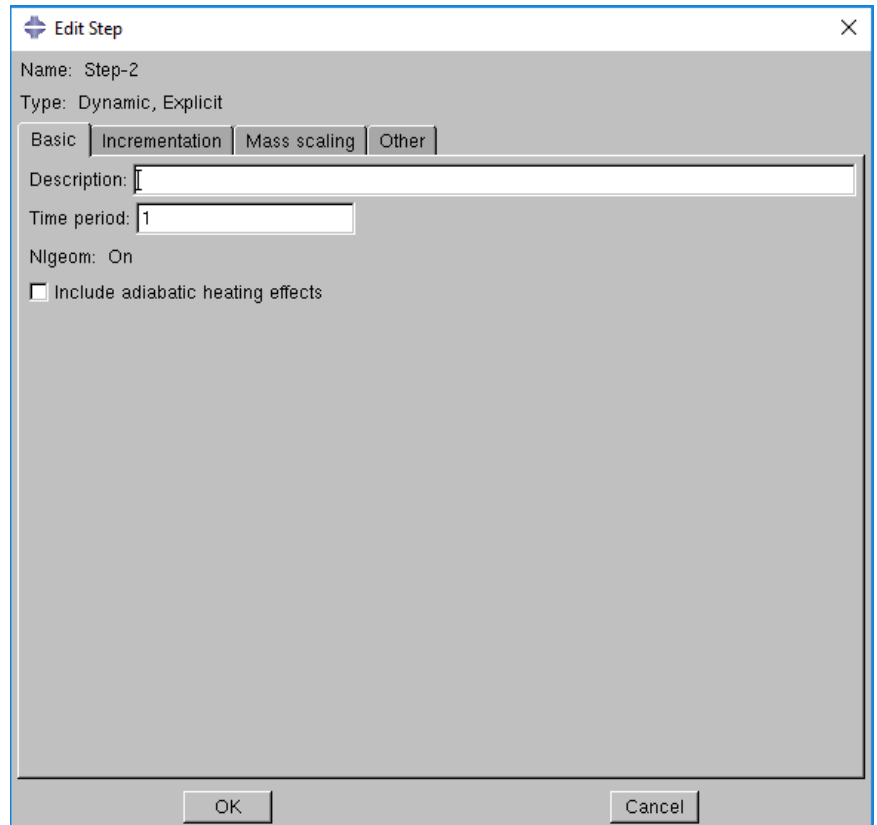
Name:

Insert new step after

- Initial
- Step-1**

Procedure type: General

- Anneal
- Dynamic, Explicit**
- Dynamic, Temp-disp, Explicit



Edit Step

Name: Step-2

Type: Dynamic, Explicit

Basic | Incrementation | Mass scaling | Other

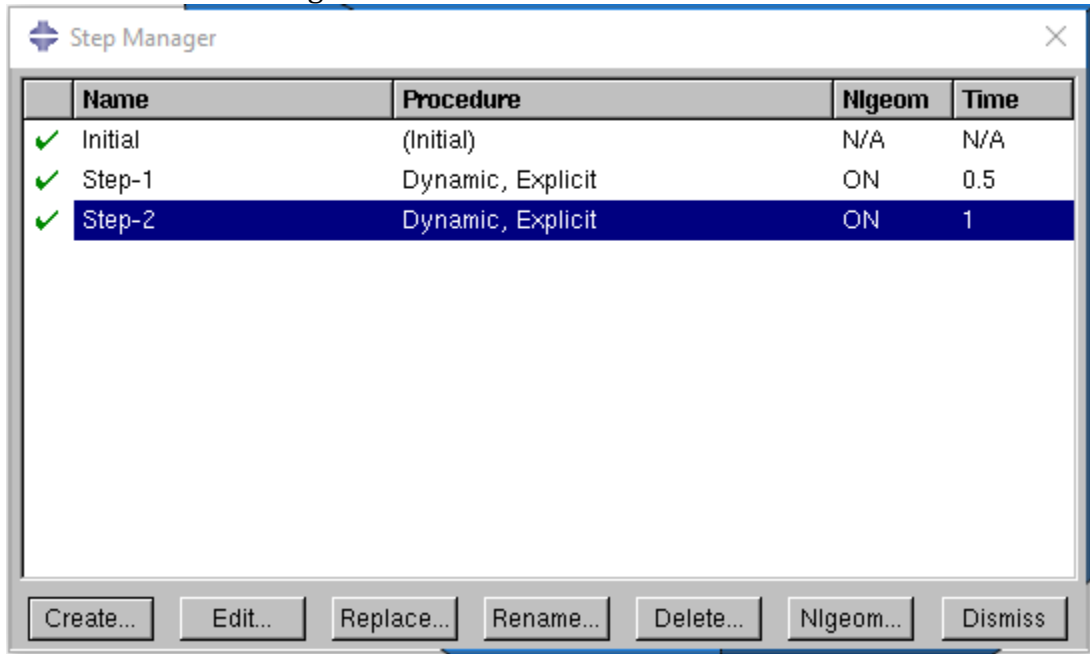
Description:

Time period:

Nlgeom: On

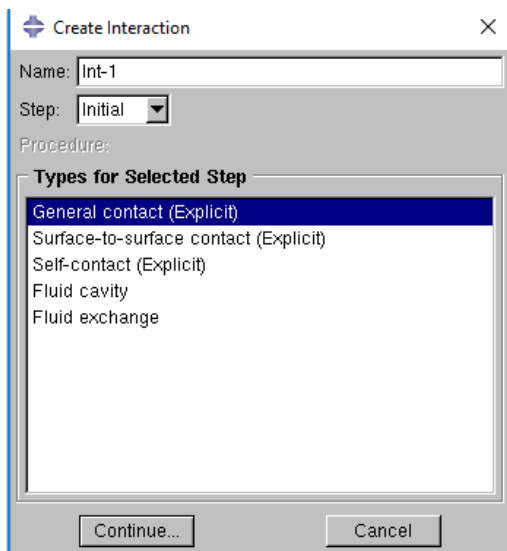
☐ Include adiabatic heating effects

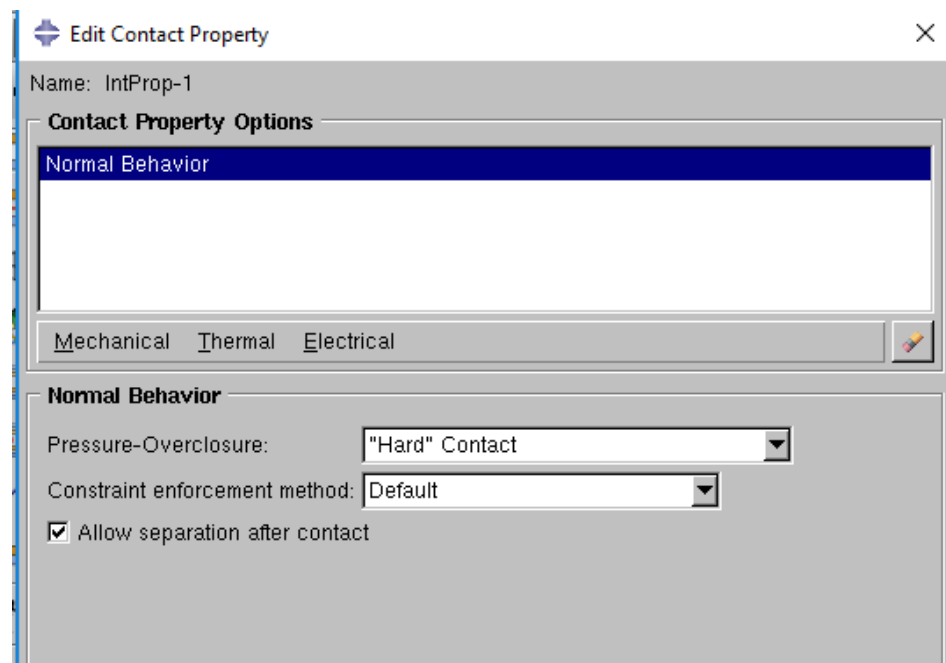
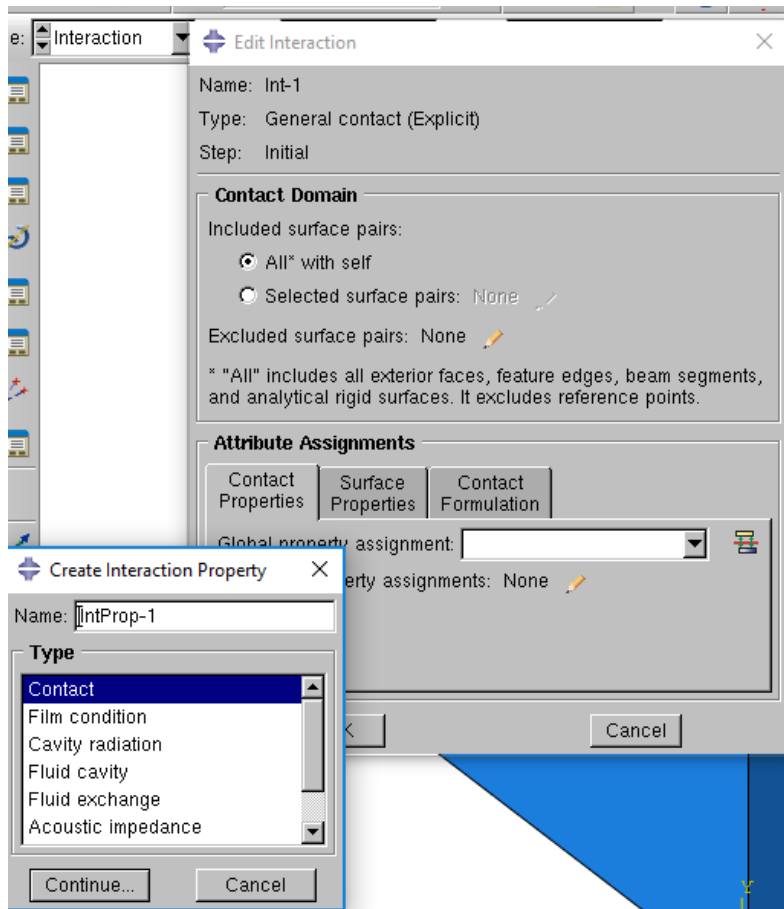
Should have something like this:



4. Contacts

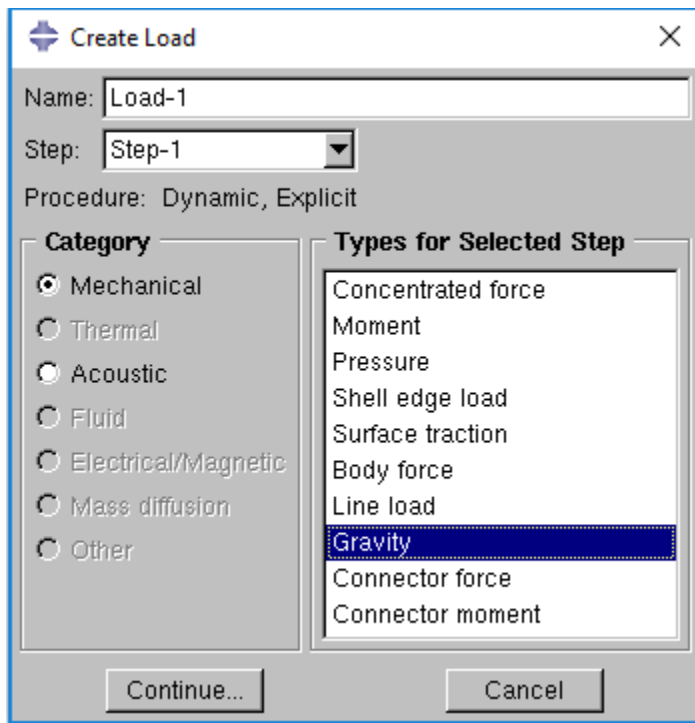
In the interaction module create a default all with self-interaction and default global interaction property.



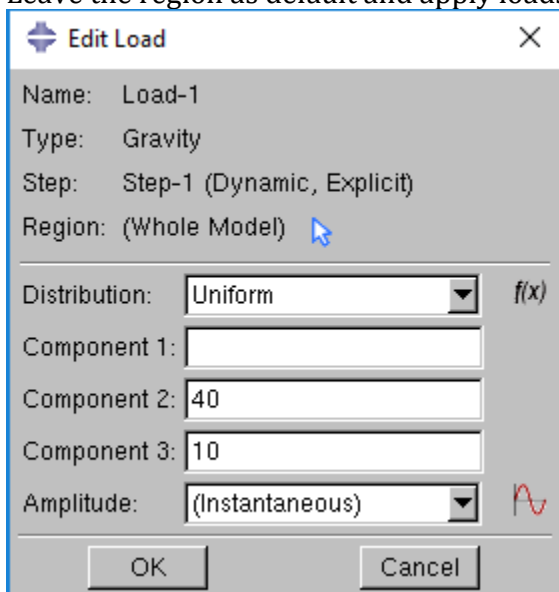


5. Applying Loads and BCs

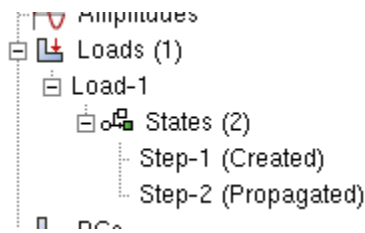
Go to the load module and create a new **Gravity** load



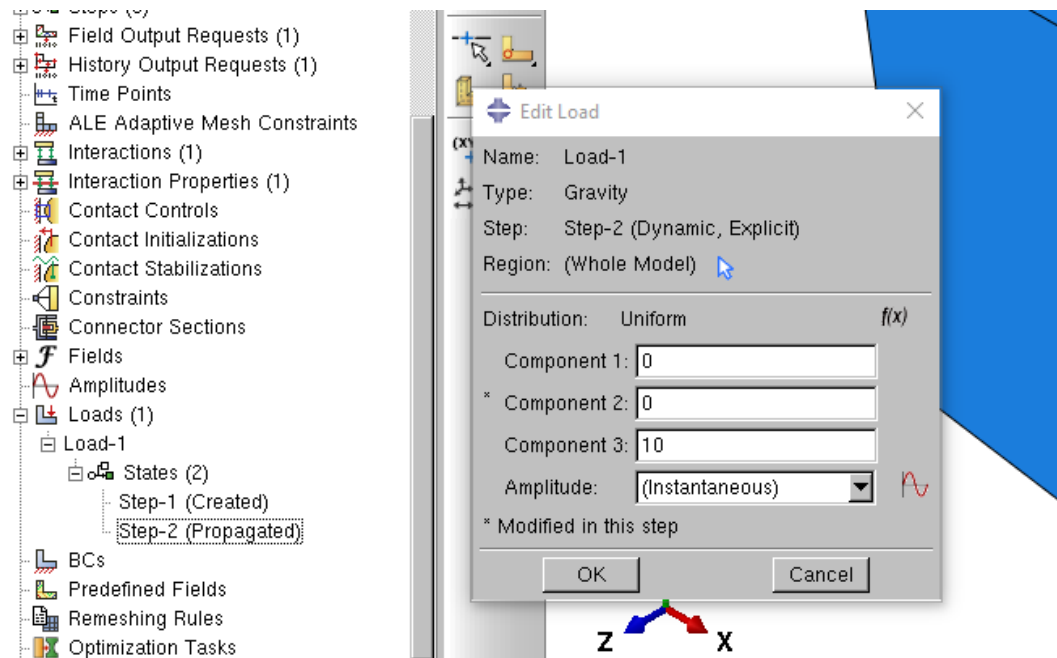
Leave the region as default and apply loads as shown



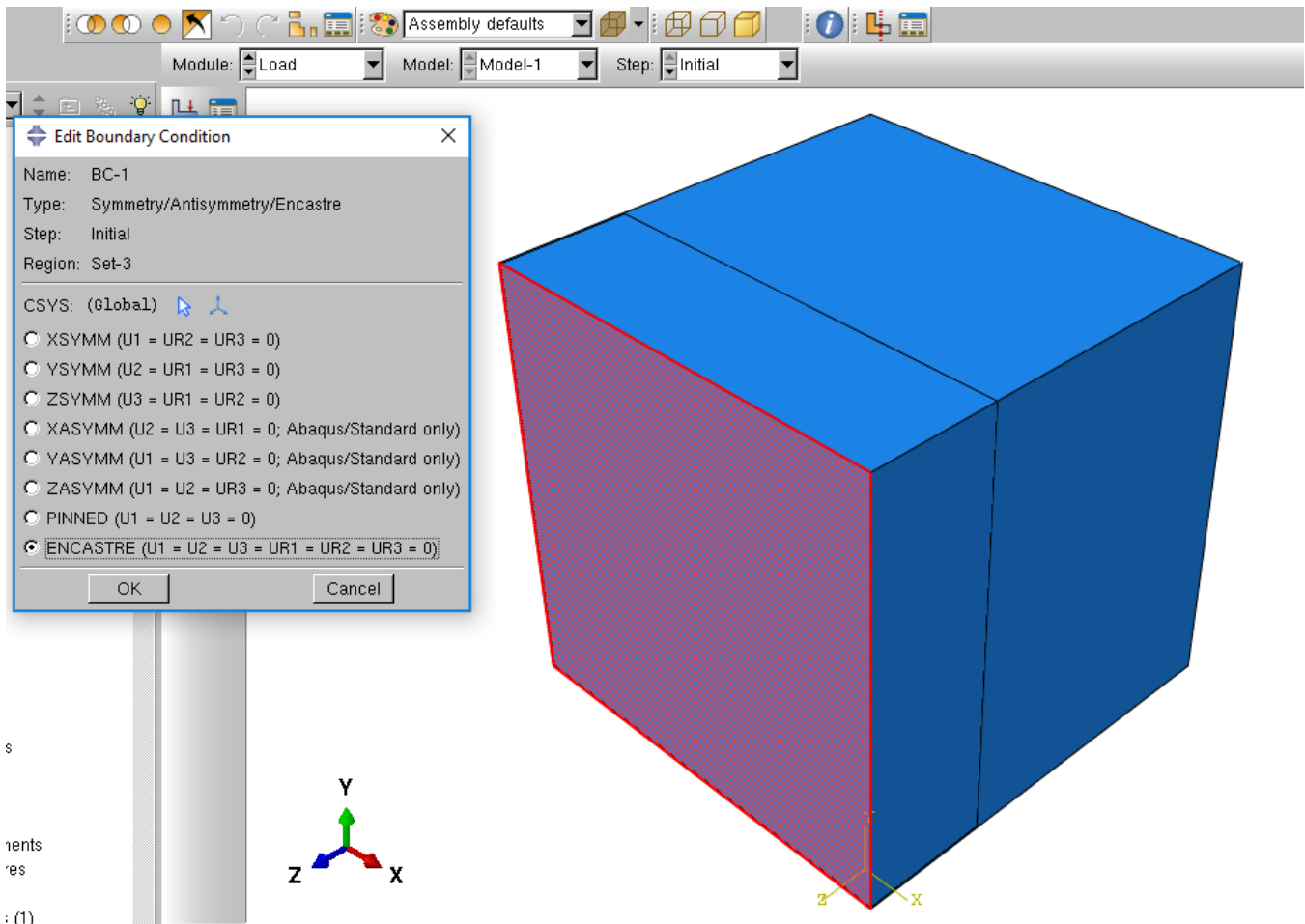
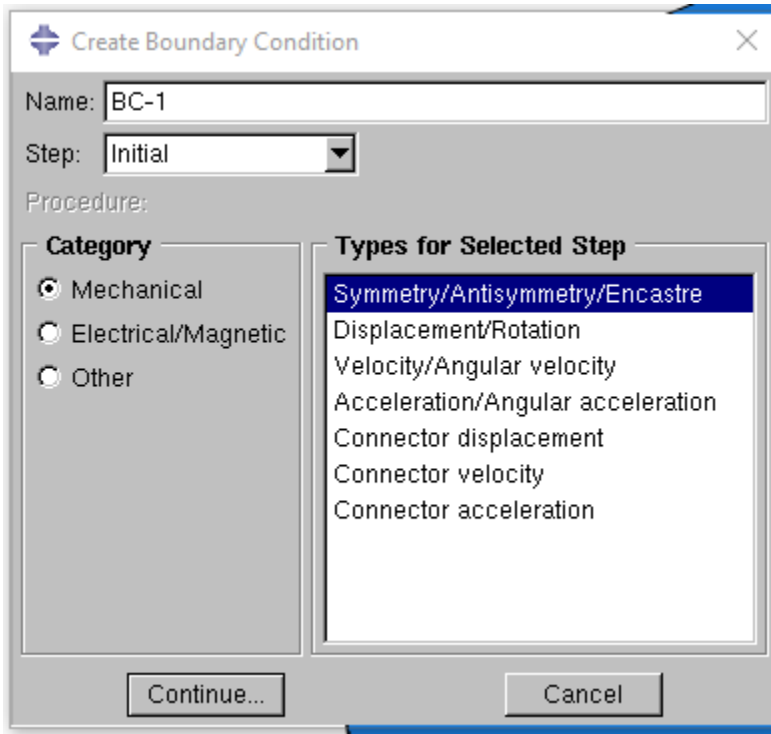
Then in the feature tree window expand the load states and select **step 2 propagated**



Double click on the Step-2 and reduce the 'component two' load to zero:

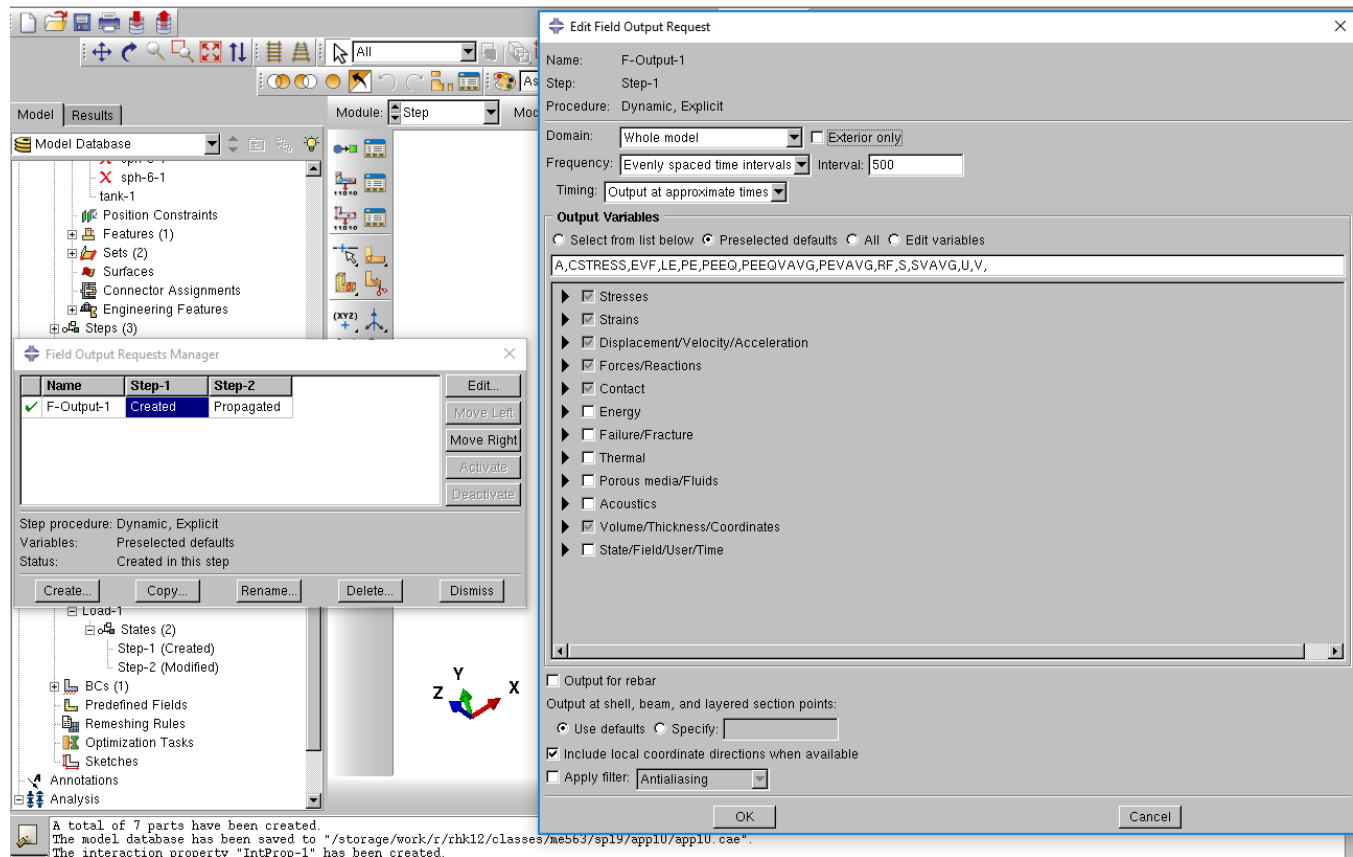


Lastly also apply an entcastre boundary conditions on the bottom four corners



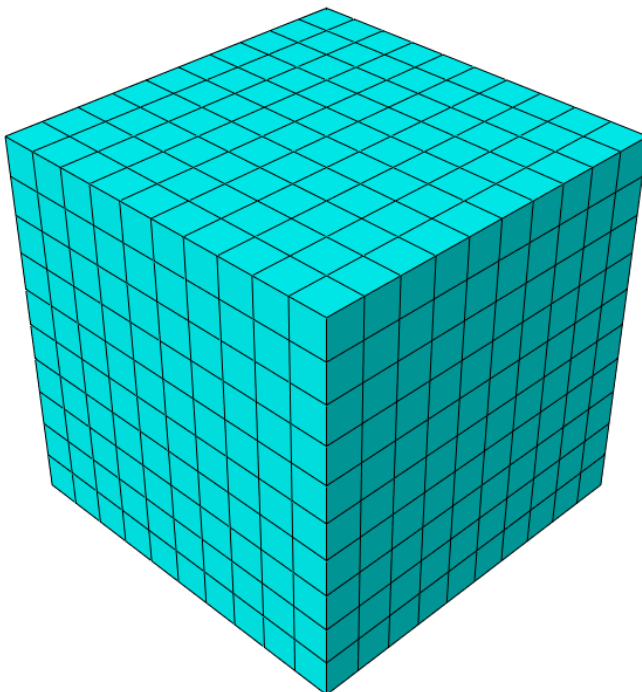
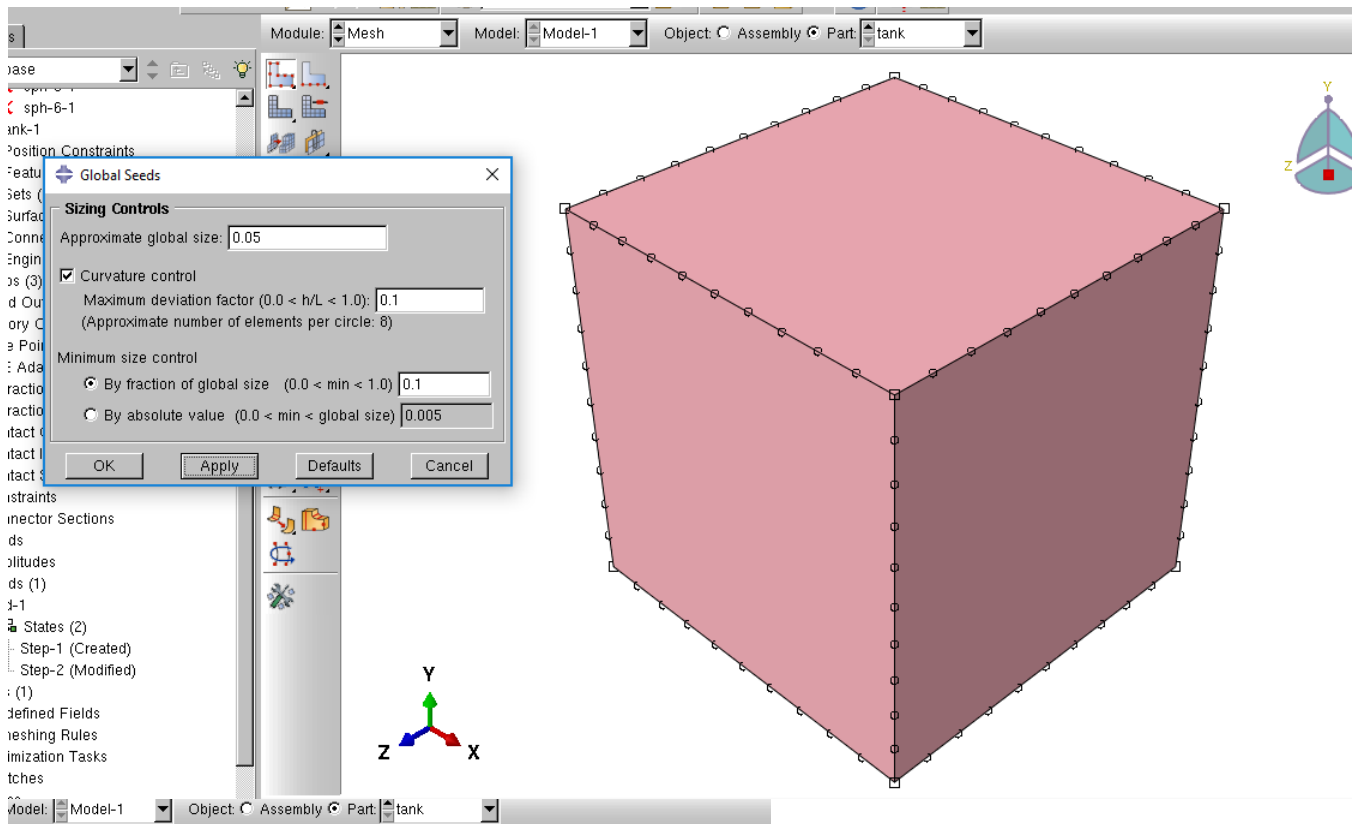
6. Field output requests

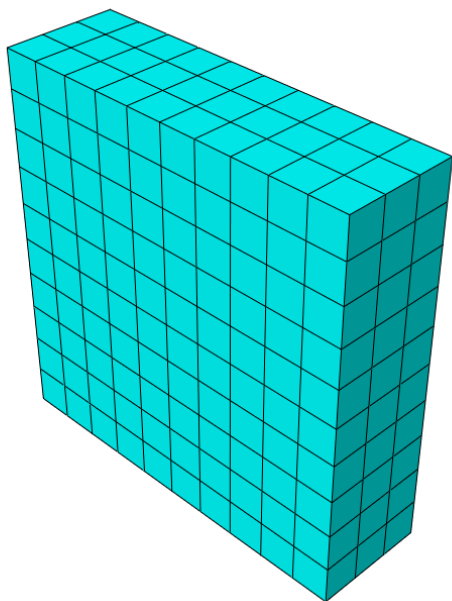
Before meshing select the **field output request** and increase the interval to 500.



7. Meshing

In the meshing module mesh the aluminum tank and water block with spacings of about 0.05. Now assign an element type to the water





Now assign an element type to the water. Change the **element library** to explicit and Conversion to particle to yes, with time threshold = 0

Element Type

Element Library
☐ Standard ☒ Explicit

Geometric Order
☒ Linear ☐ Quadratic

Family
 3D Stress
 Acoustic
 Cohesive
 Continuum Shell

Hex Wedge Tet

☒ Reduced integration ☐ Incompatible modes

Element Controls

Kinematic split: ☒ Average strain ☐ Orthogonal ☐ Centroid
 Second-order accuracy: ☐ Yes ☒ No
 Distortion control: ☒ Use default ☐ Yes ☐ No
 Length ratio: 0.1
 Hourglass control: ☒ Use default ☐ Enhanced ☐ Relax stiffness ☐ Stiffness ☐ Viscous ☐ Combined
 Stiffness-viscous weight factor: 0.5
 Conversion to particles: ☐ Use default ☒ Yes ☐ No
 Criterion: Time Threshold: 0 PPD: 1 Kernel: Cubic
 Element deletion: ☒ Use default ☐ Yes ☐ No
 Max Degradation: ☒ Use default ☐ Specify
 Scaling factors: Displacement hourglass: 1 Linear bulk viscosity: 1 Quadratic bulk viscosity: 1

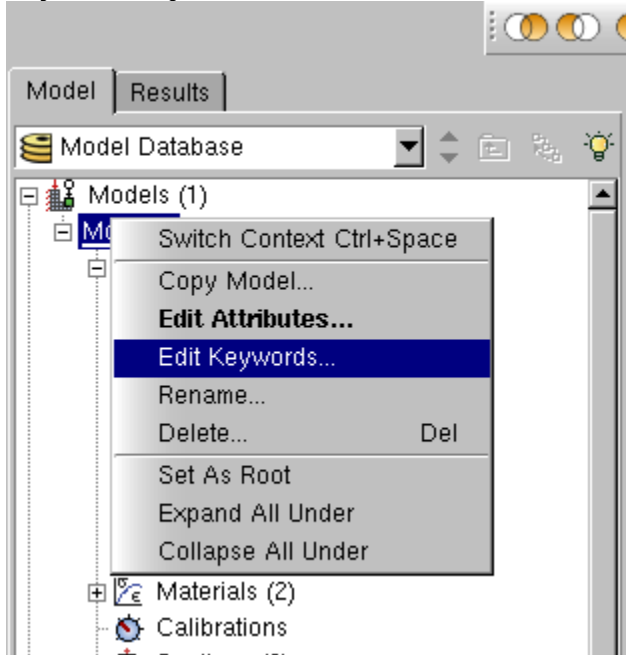
C3D8R: An 8-node linear brick, reduced integration, hourglass control.

Note: To select an element shape for meshing,
 select "Mesh->Controls" from the main menu bar.

OK Defaults Cancel

8. Editing Keywords

Before testing the model the some lines must be added to the model keywords, select the **edit keywords** option on the feature tree window



Insert the line after EoS

***TENSILE FAILURE, element deletion=no, pressure=ductile, shear=ductile
100.0**

Edit keywords, Model: Model-1

```

**
** PARTS
**
*Part, name=tank
*Element, type=S4R
*Nset, nset=Set-1, generate
*Elset, elset=Set-1, generate
** Section: aluminium tank
*Shell Section, elset=Set-1, material=aluminium
0.002, 5
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=tank-1, part=tank
*End Instance
**
*Nset, nset=Set-1, instance=tank-1
*Elset, elset=Set-1, instance=tank-1, generate
*End Assembly
**
** MATERIALS
**
*Material, name=aluminium
*Density
2500.,
*Elastic
7e+10, 0.3
*Material, name=water
*Density
100.,
*Eos, type=USUP
1483.,0.,0.
*TENSILE FAILURE, element deletion=no, pressure=ductile, shear=ductile
100.0
**
** INTERACTION PROPERTIES
**

```

Block:
Add After
Remove
Discard Edits

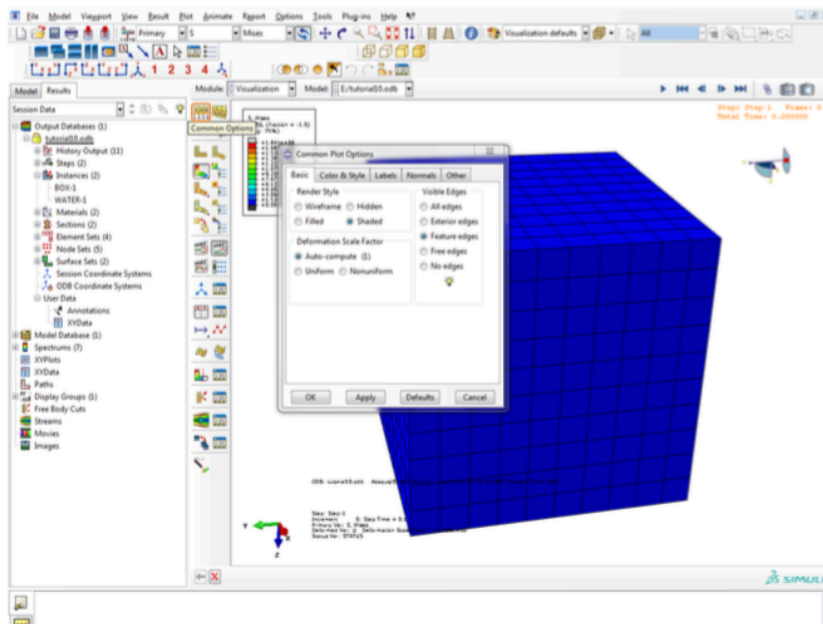
OK
Discard All Edits
Cancel

9. Create job and submit it

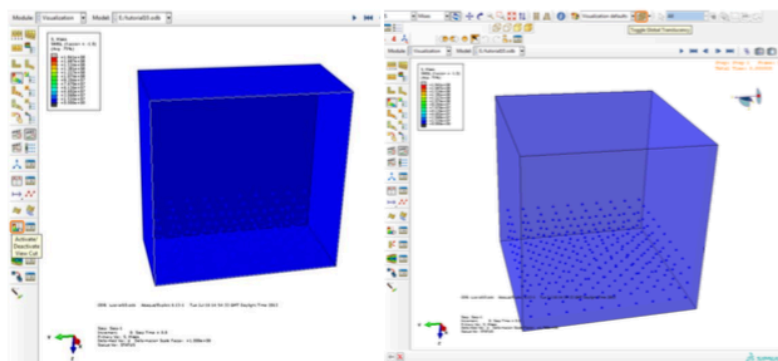
10. Post-Processing

To get the best results from the model a bit of post processing is necessary.

First remove the mesh lines; to do this select the **common options** tool and choose '**feature edges**'.



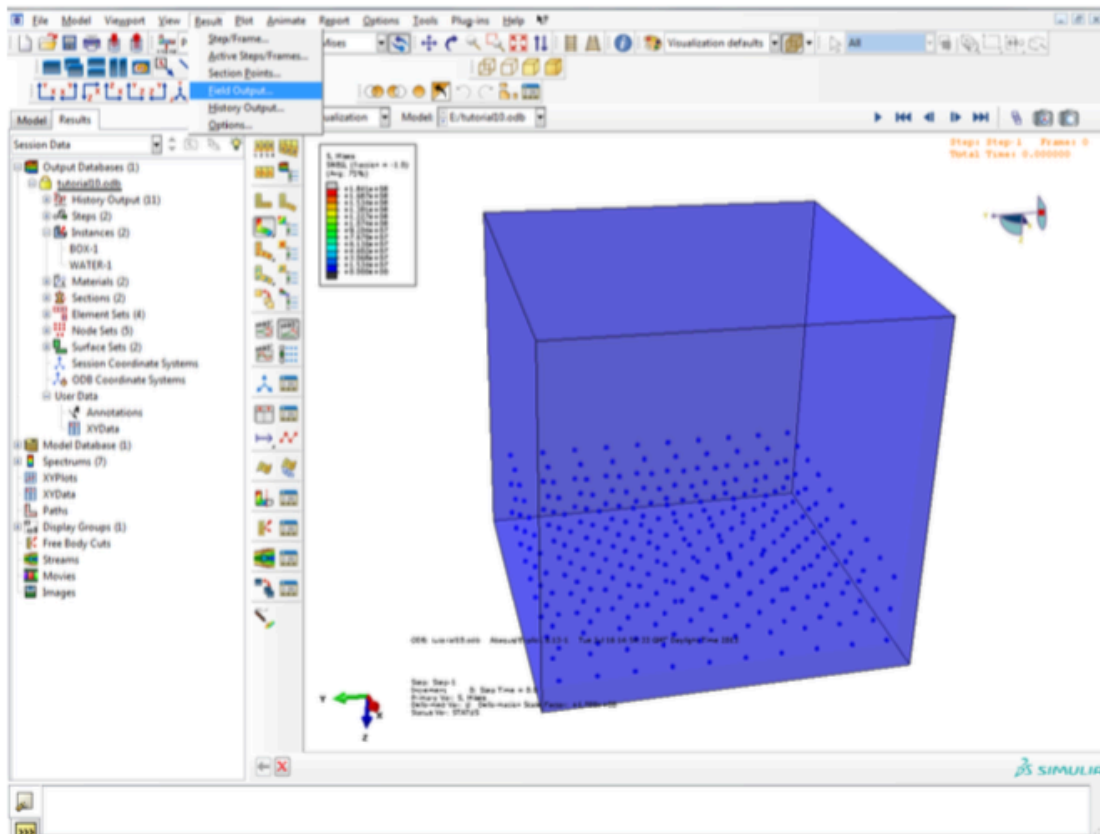
The water particles within the cube should be visible. This can be attained in one of two ways; using the **view cut** tool to cross-section the box or using **transparency**



View cut cross-section

Global transparency

You can also change the variables plotted on the geometry to, for example velocity at nodes. To do so select '**field output**' from the result drop down menu and choose from the list of variables.



To get the best demonstration of the movement of the particles choose either:

U spatial displacement at nodes

V spatial velocity at nodes