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)

The Pennsylvania State University

Contents

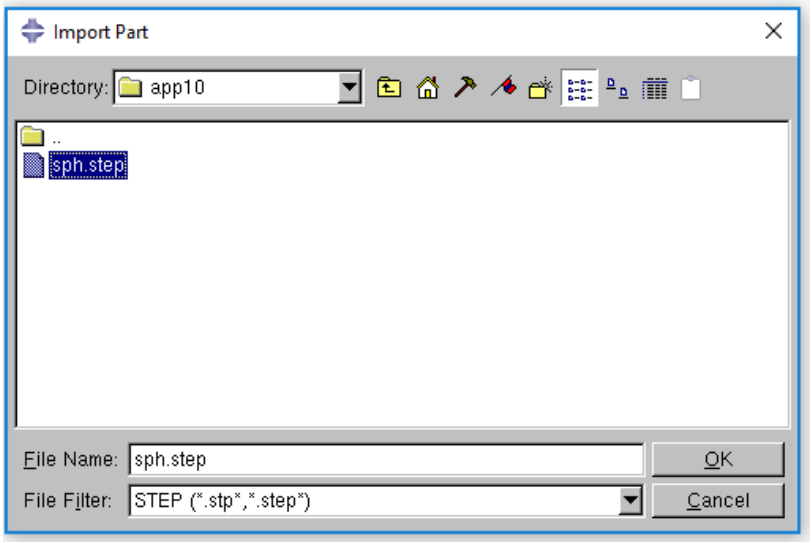
1. Part Creation and Modification 3
2. Material and Section Properties 6
3. Creating the steps 10
4. Contacts 12
5. Applying Loads and BCs 14
6. Field output requests 17
7. Meshing 17
8. Editing Keywords 20
9. Create job and submit it 21
10. Post-Processing 21

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.

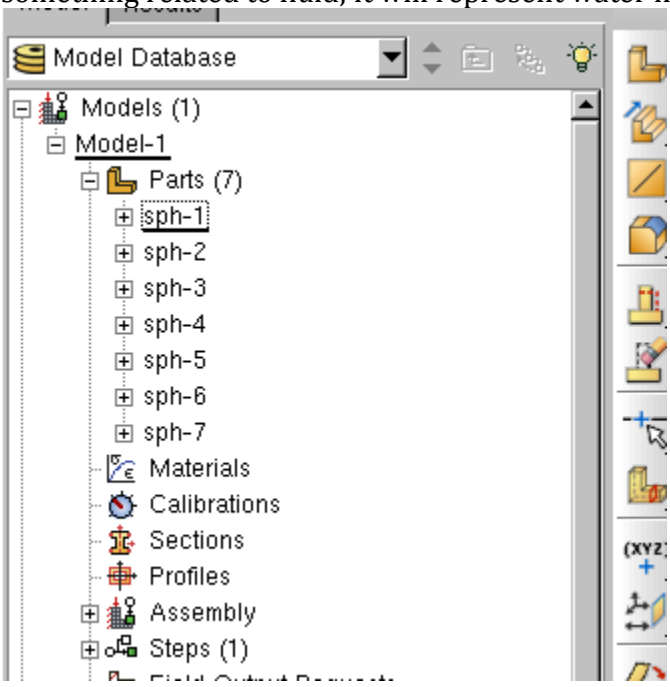
1. Part Creation and Modification

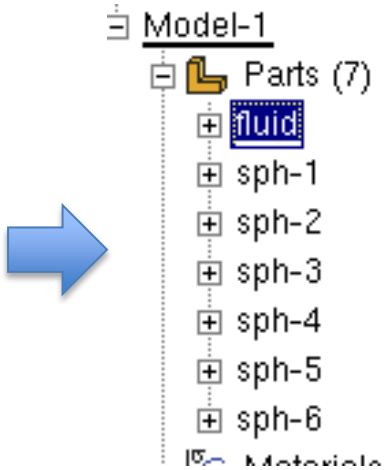
Obtain files from Github:
git clone https://github/com/rhk12/sph

Import part:

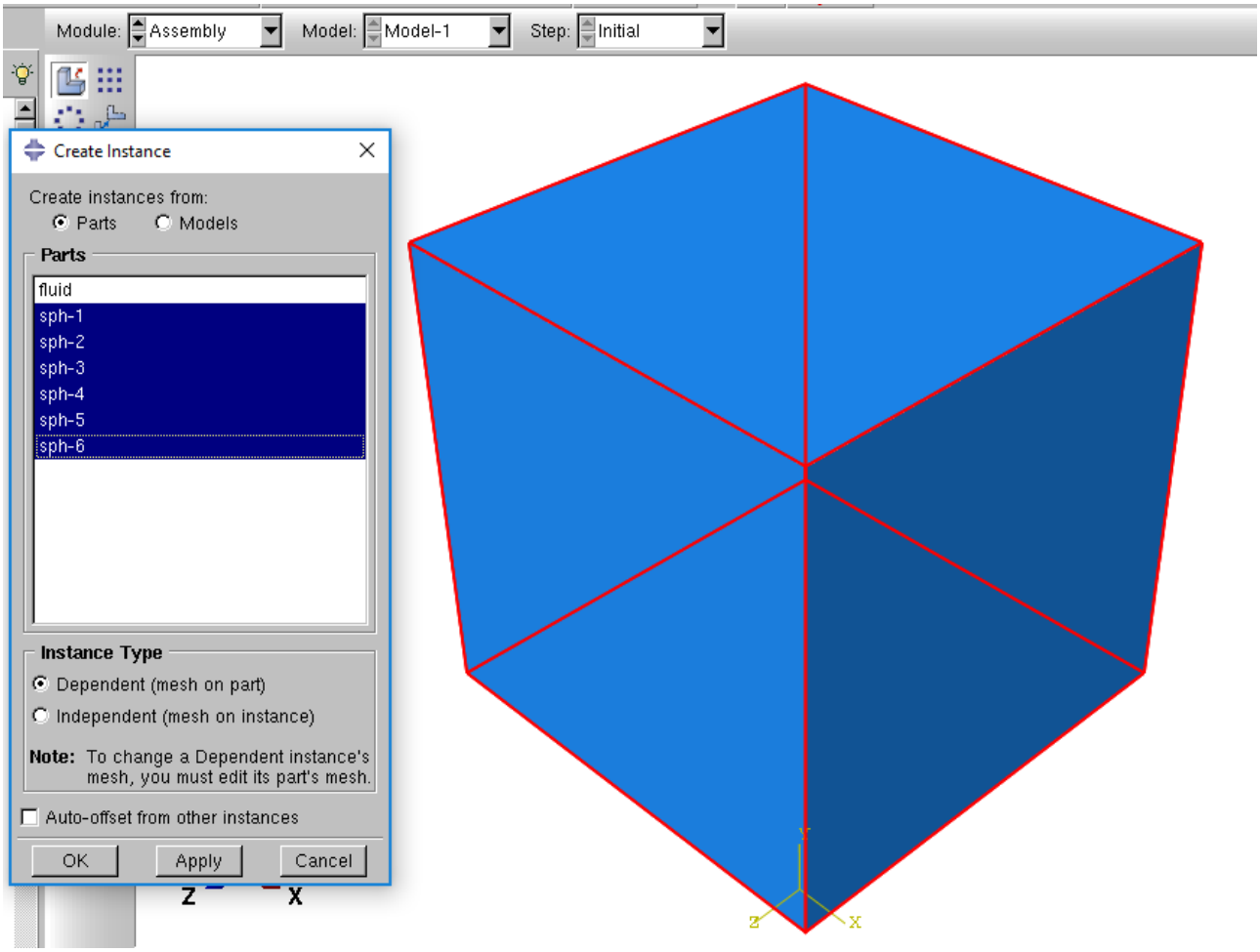


There are 7 different parts. This contains a block and cube made up six faces, rename the block to something related to fluid; it will represent water in this example.

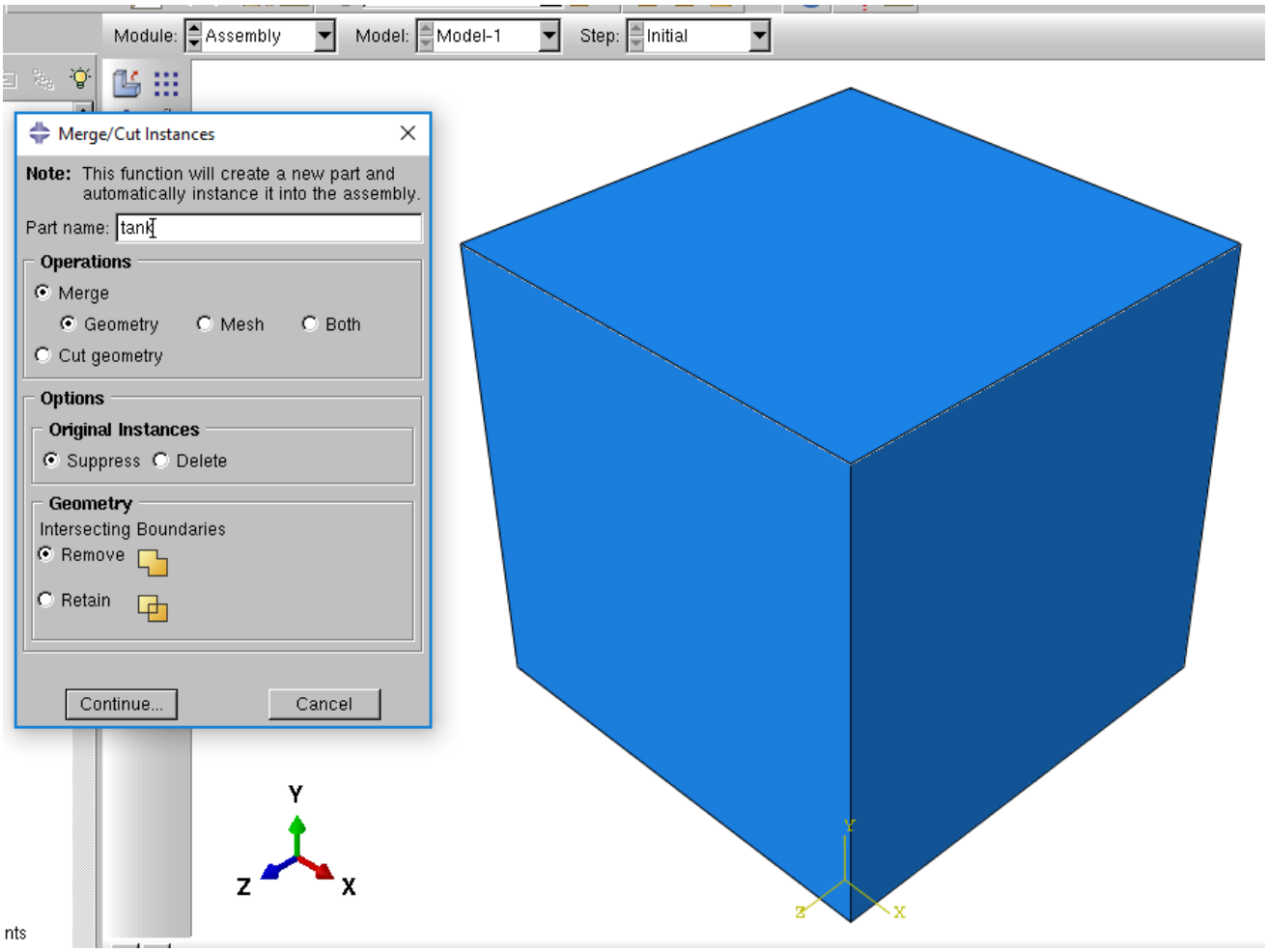




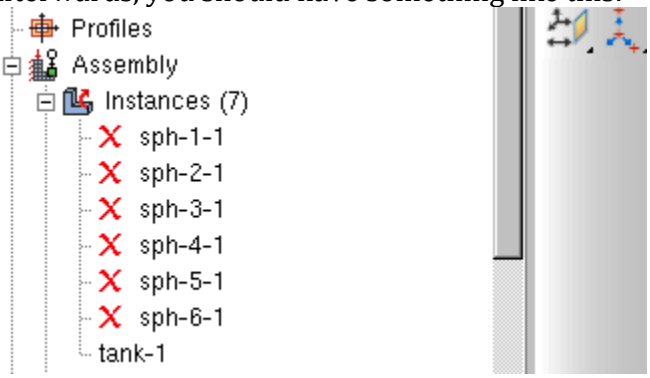
The next step is to combine the faces into a new part to form a box.



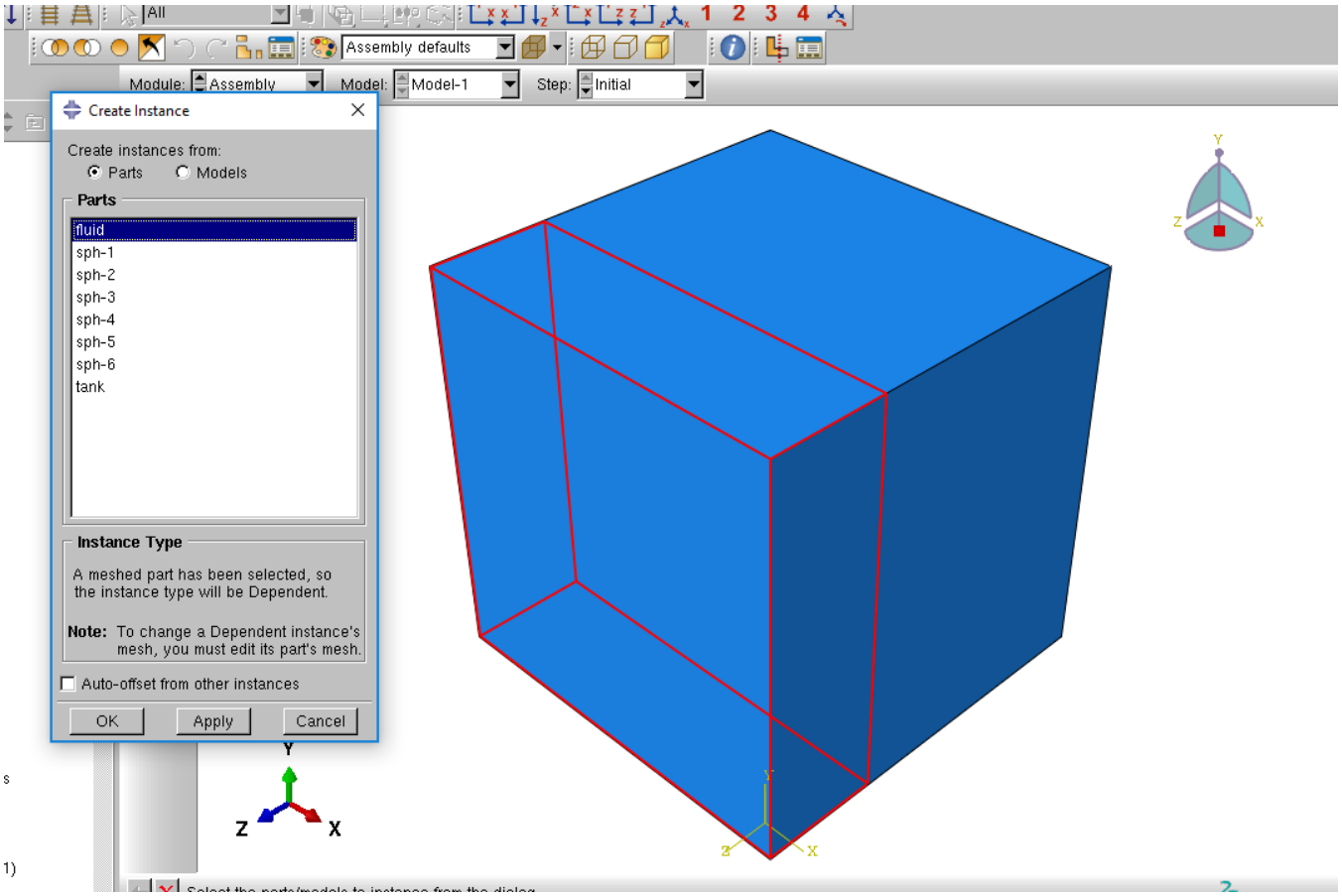
First instance the parts in the **assembly module**, and then once the parts are instanced select the **merge/cut** from the instance drop down menu



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

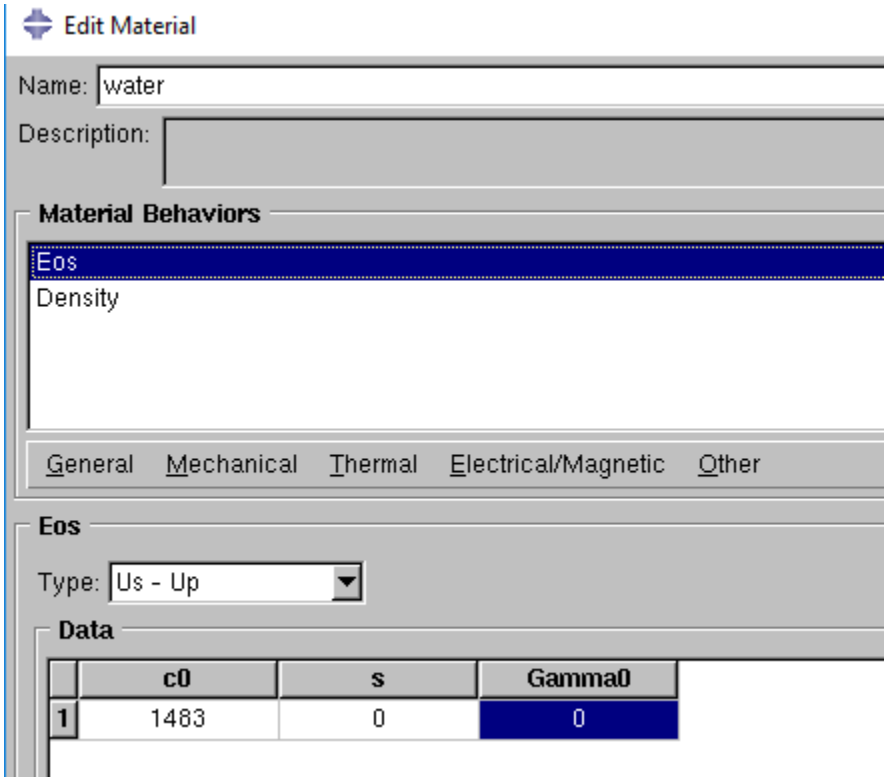
Aluminum:

Mechanical properties:

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

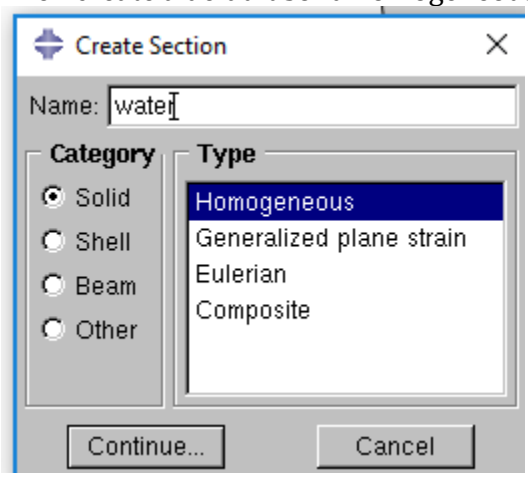
General properties:

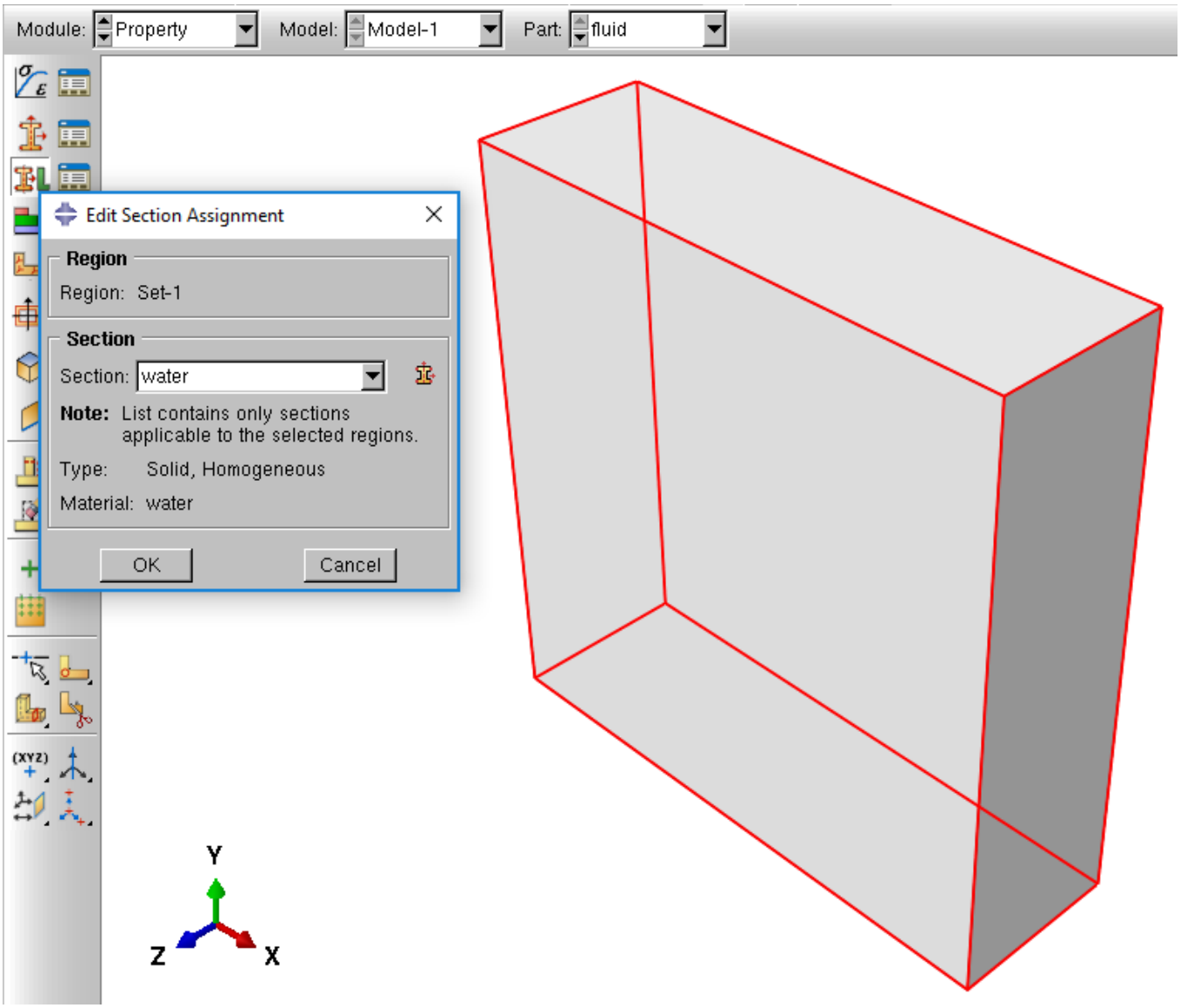
- Density = 2500



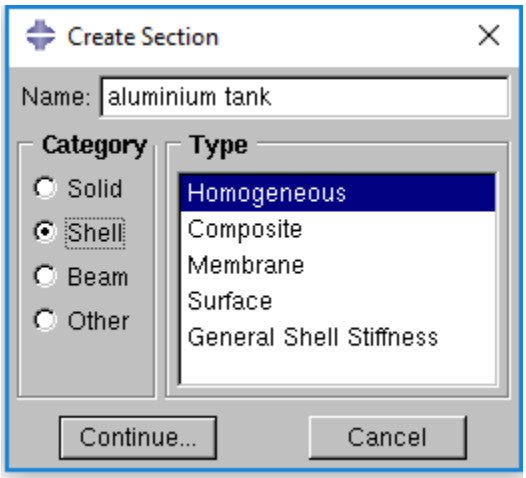
Note: When it comes to assigning sections ignore the 2D squares that merged into the box as they are no longer counted as part of the assembly.

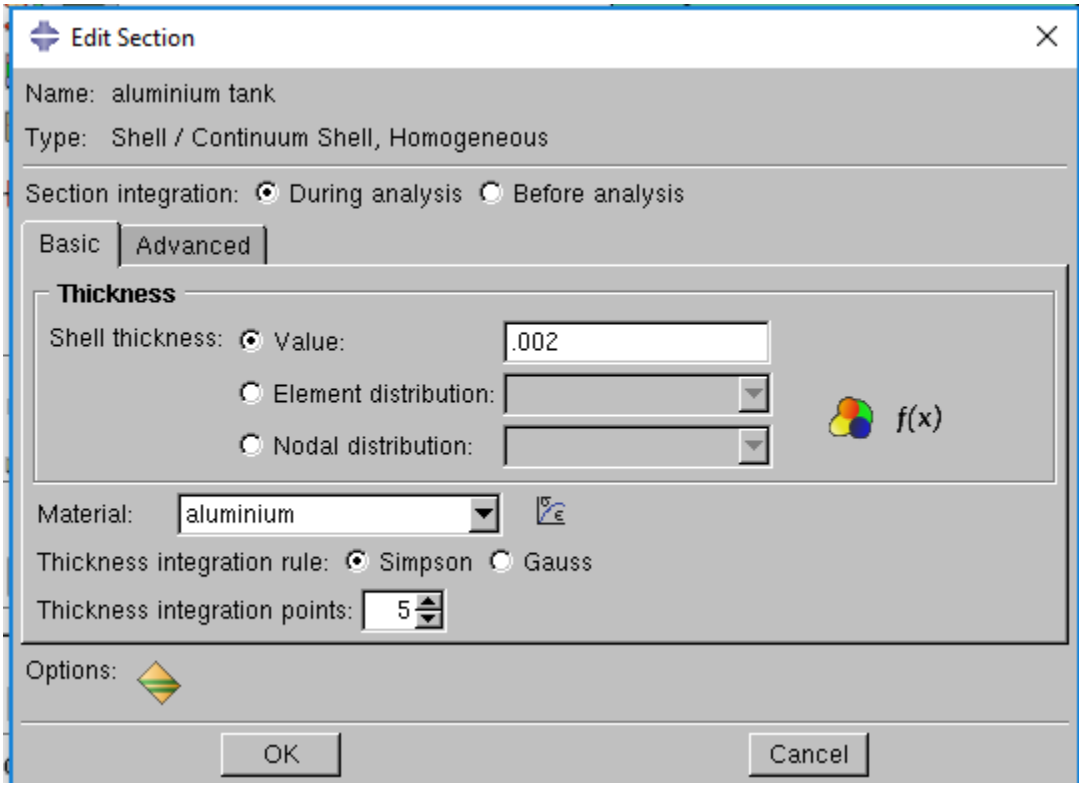
Then create a default solid homogeneous section for water and assign the section to the block.





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

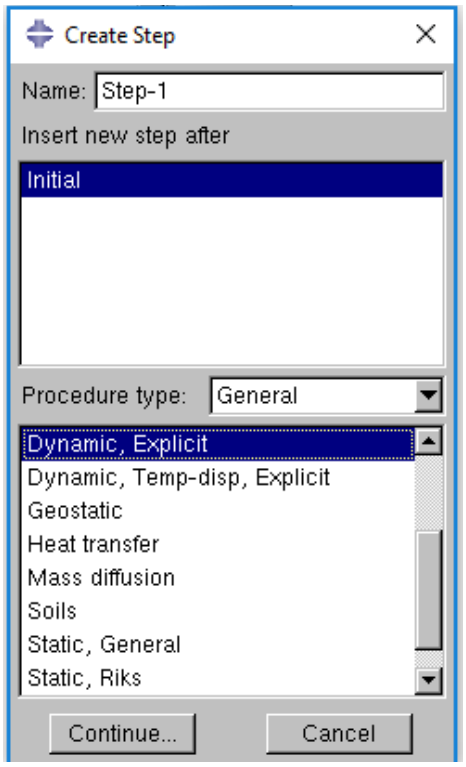


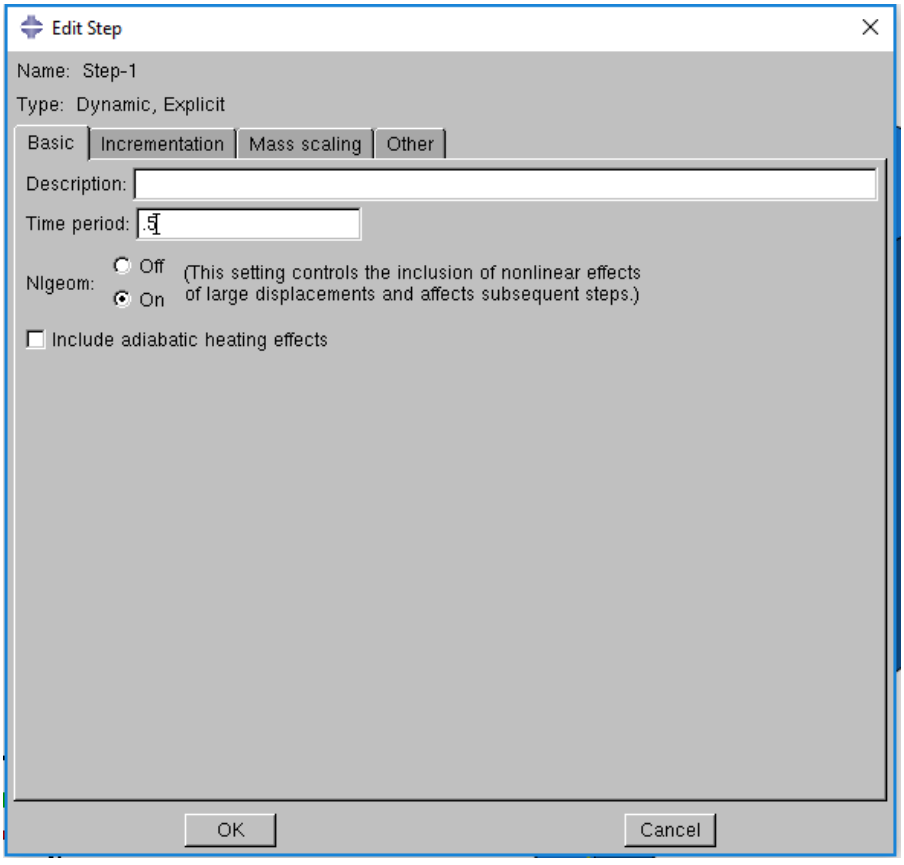

Edit Section
Name: aluminium tank
Type: Shell / Continuum Shell, Homogeneous
Section integration: During analysis Before analysis
Basic
Advanced
Thickness
Shell thickness: Value: .002
Element distribution:
Nodal distribution:
f(x)
Material: aluminium
Thickness integration rule: Simpson Gauss
Thickness integration points: 5
Options:
OK
Cancel

3. Creating the steps

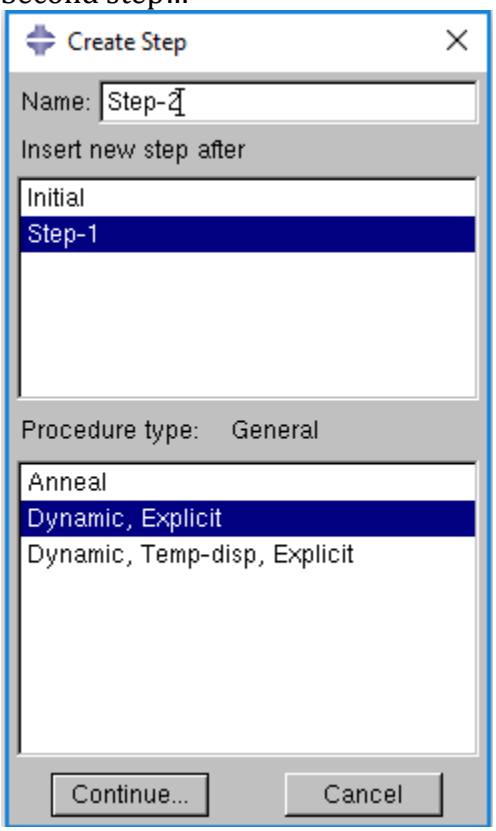
Go to the step module and create two new steps, for all steps in the model use the step option of '**dynamic, explicit**'

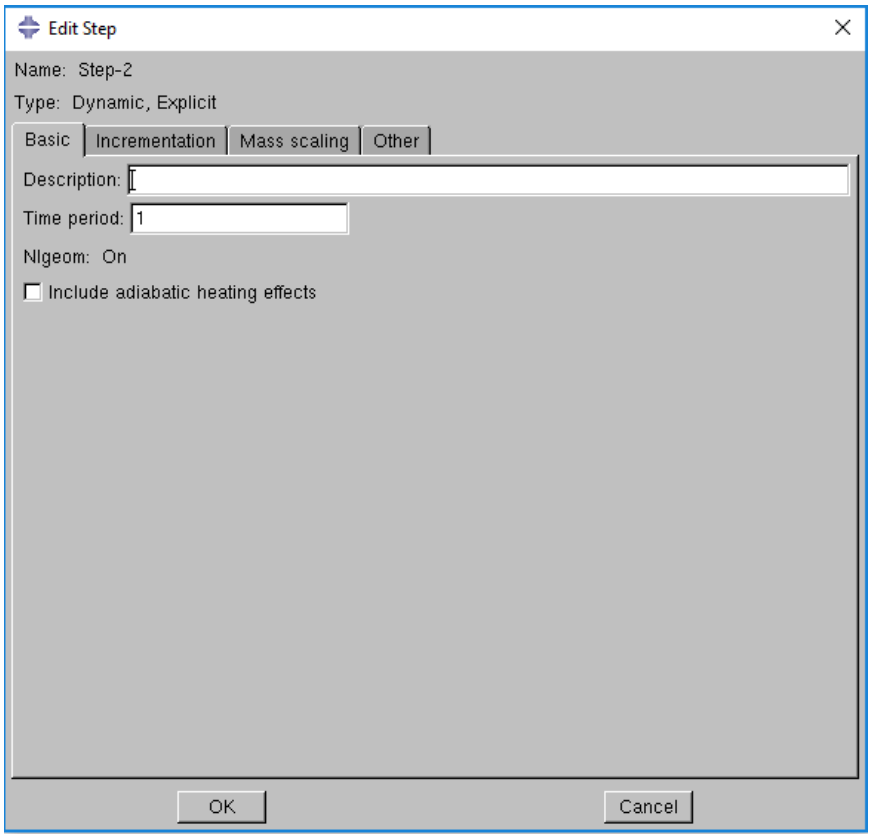
For the first step make the time period 0.5 seconds and 1 for the second step



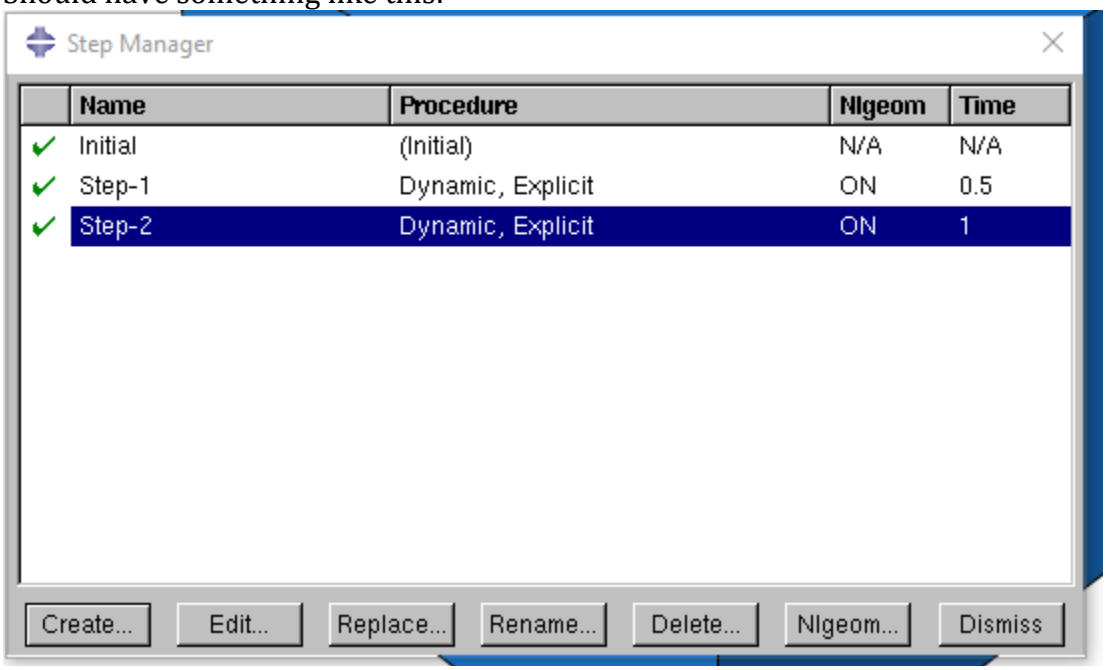


Second step…


Create Step
Name: Step-2
Insert new step after
Initial
Step-1
Procedure type: General
Anneal
Dynamic, Explicit
Dynamic, Temp-disp, Explicit
Continue...
Cancel

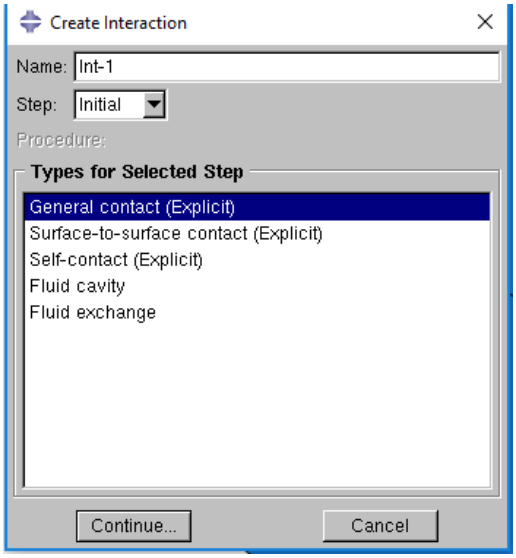

Edit Step
Name: Step-2
Type: Dynamic, Explicit
Basic
Incrementation
Mass scaling
Other
Description:
Time period: 1
Nlgeom: On
Include adiabatic heating effects
OK
Cancel

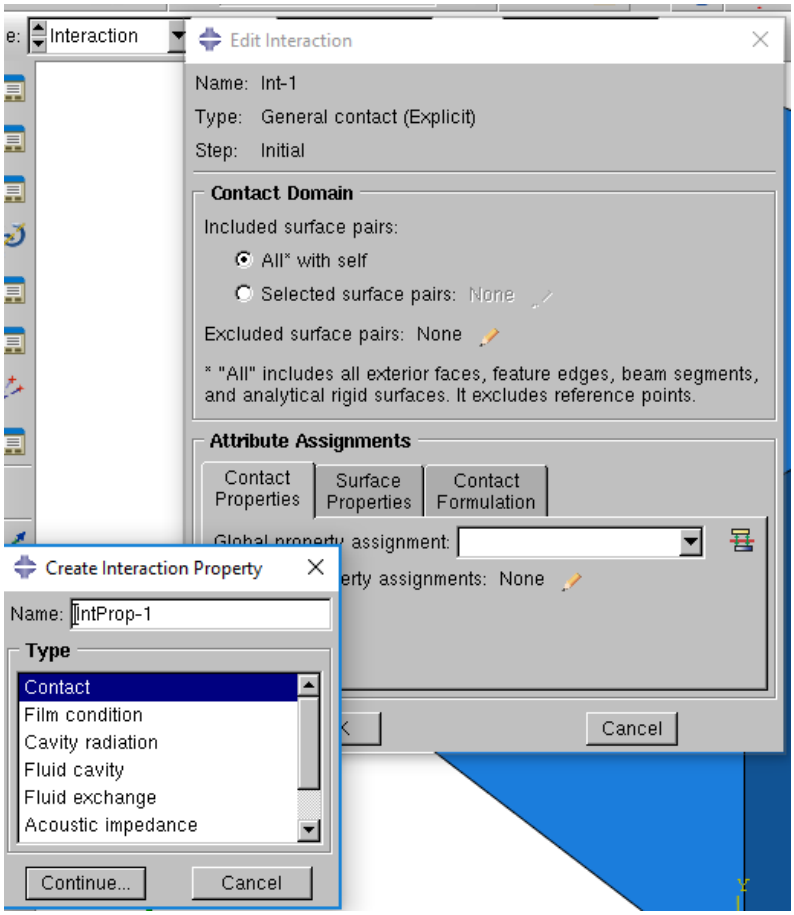
Should have something like this:

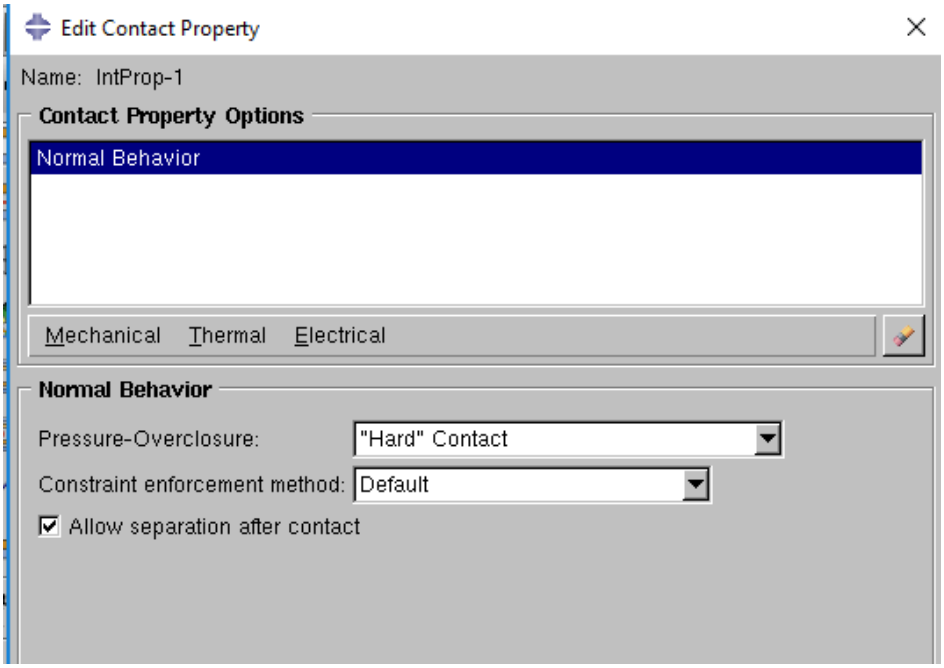


4. Contacts

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

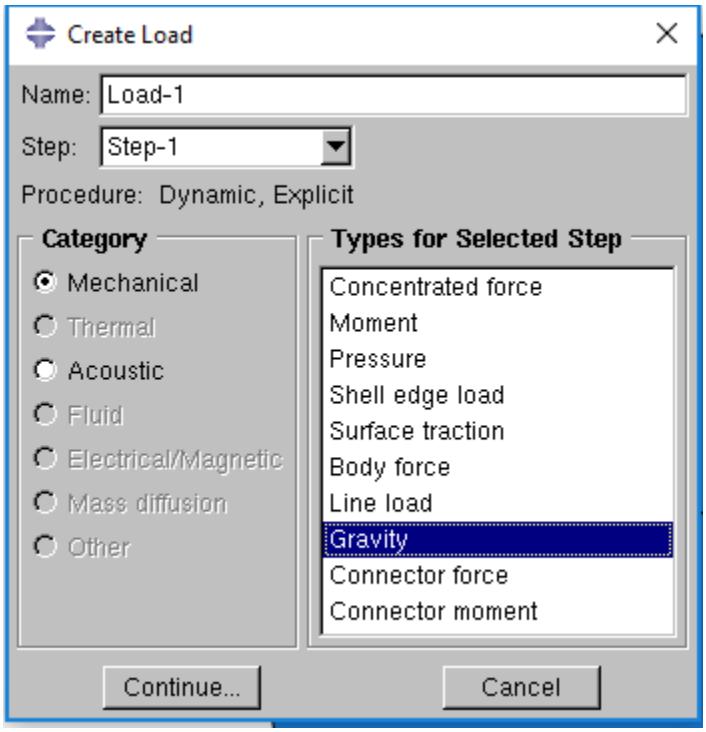



Interaction
Edit Interaction
Name: Int-1
Type: General contact (Explicit)
Step: Initial
Contact Domain
Included surface pairs:
All* with self
Selected surface pairs: None
Excluded surface pairs: None
* "All" includes all exterior faces, feature edges, beam segments, and analytical rigid surfaces. It excludes reference points.
Attribute Assignments
Contact Properties
Surface Properties
Contact Formulation
Global property assignment:
erty assignments: None
Cancel
Create Interaction Property
Name: IntProp-1
Type
Contact
Film condition
Cavity radiation
Fluid cavity
Fluid exchange
Acoustic impedance
Continue...
Cancel

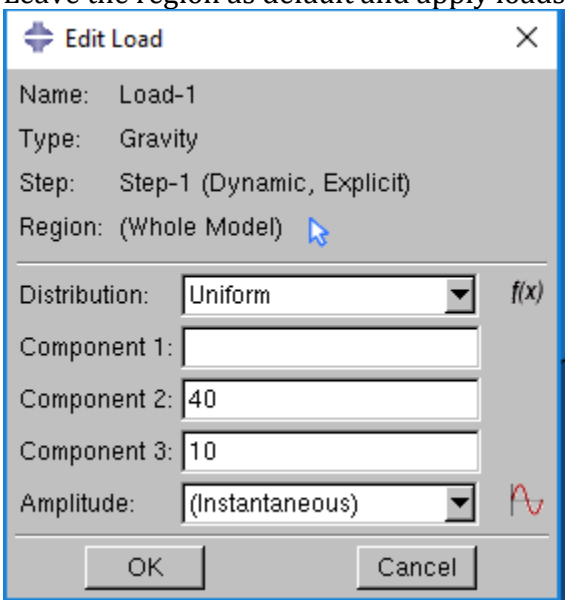

Edit Contact Property
Name: IntProp-1
Contact Property Options
Normal Behavior
Mechanical
Thermal
Electrical
Normal Behavior
Pressure-Overclosure: "Hard" Contact
Constraint enforcement method: Default
Allow separation after contact

5. Applying Loads and BCs

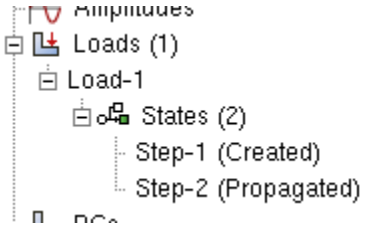
Go to the load module and create a new **Gravity** load



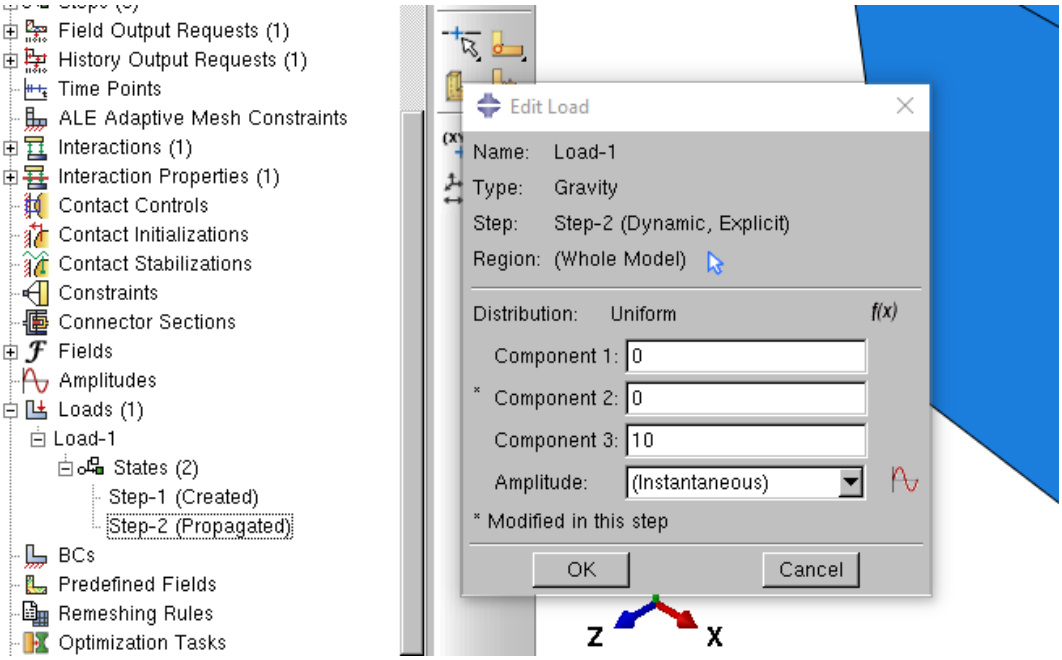
Leave the region as default and apply loads as shown



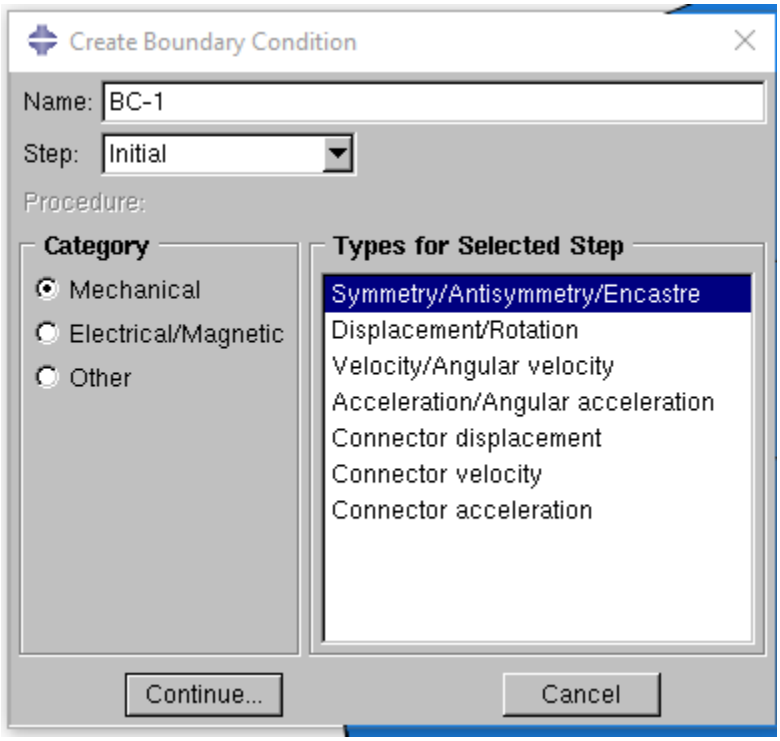
Then in the feature tree window expand the load states and select **step 2 propagated**

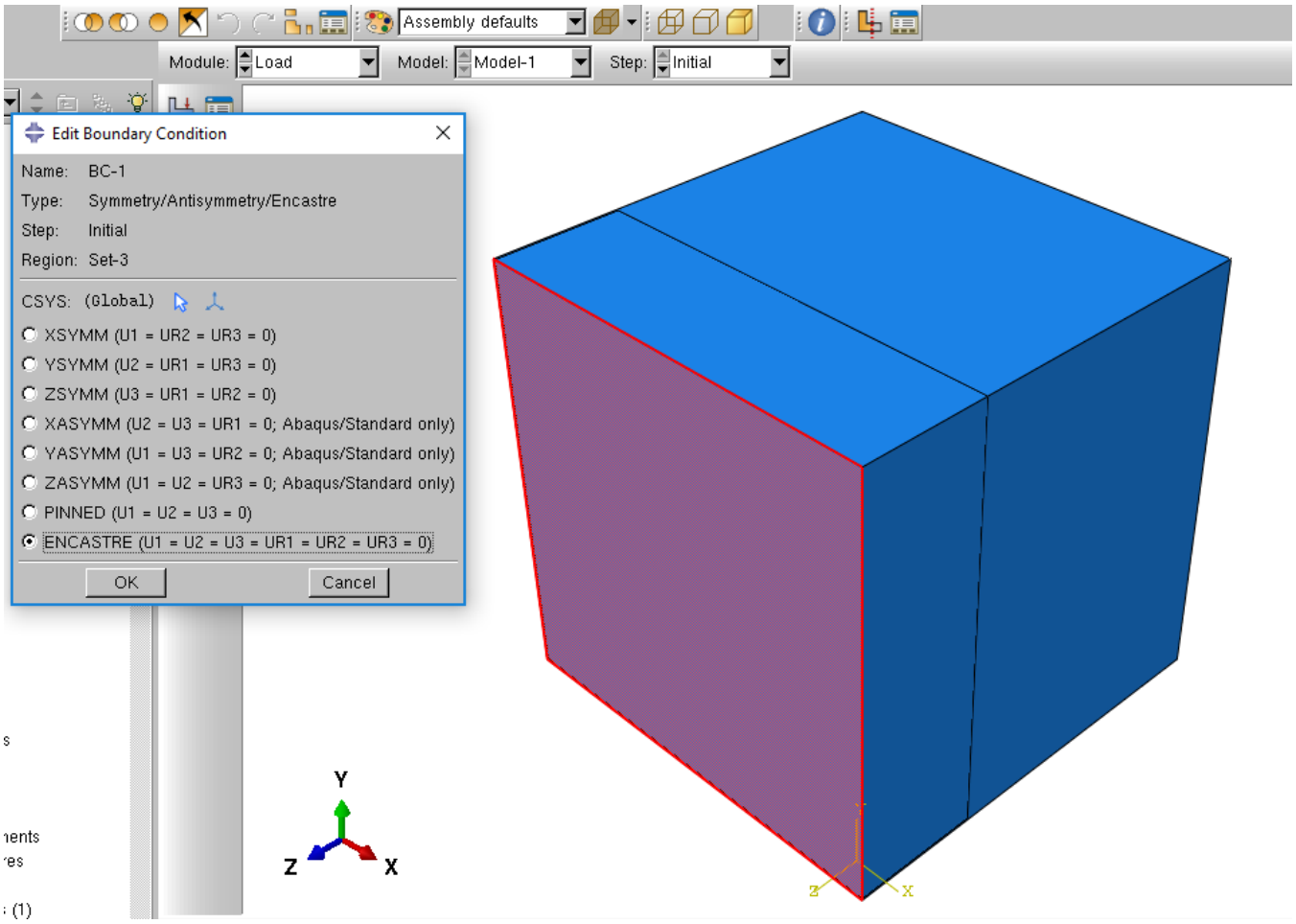


Double click on the Step-2 and reduce the 'component two' load to zero:



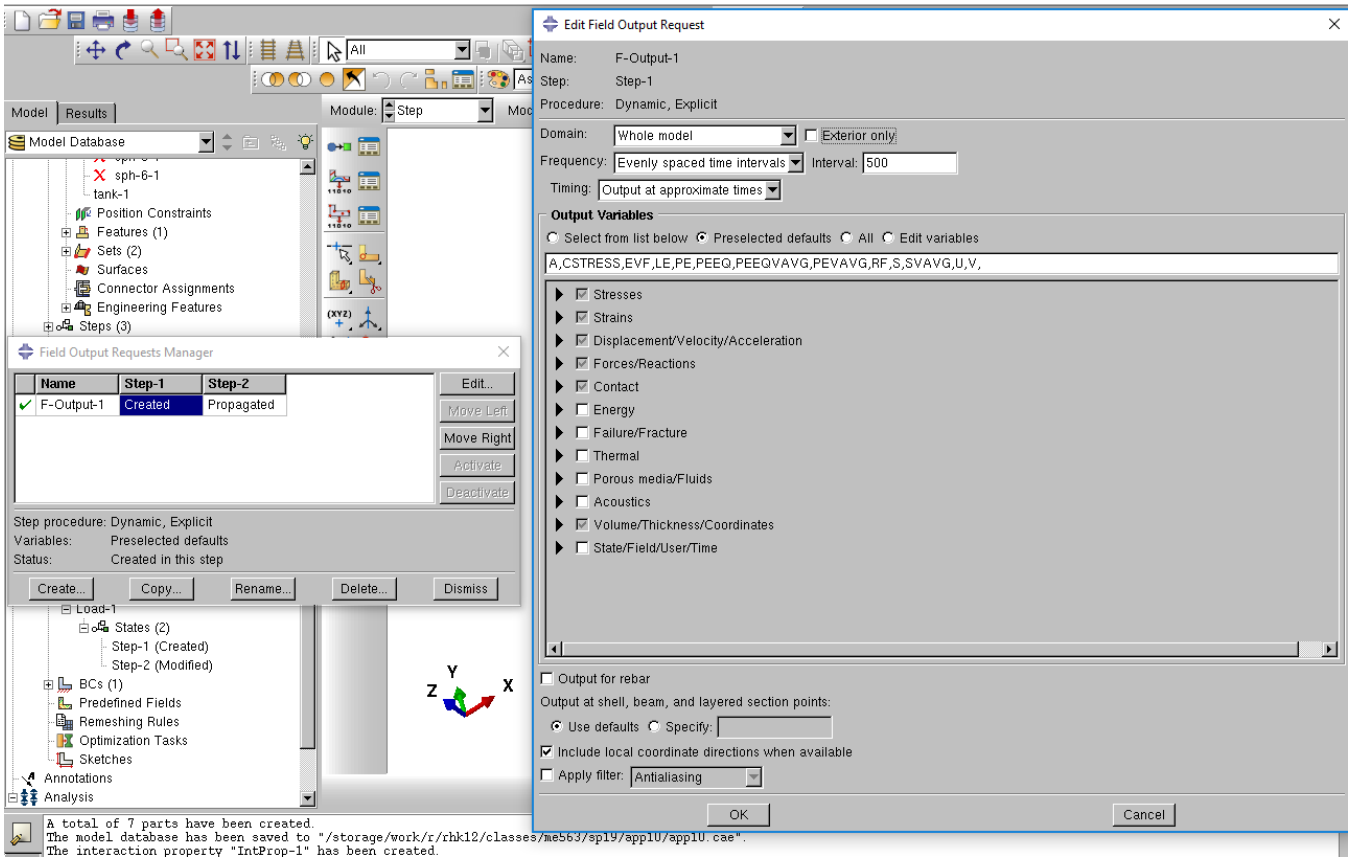
Lastly also apply an entcastre boundary conditions on the bottom four corners


Create Boundary Condition
Name: BC-1
Step: Initial
Procedure:
Category
Mechanical
Electrical/Magnetic
Other
Types for Selected Step
Symmetry/Antisymmetry/Encastre
Displacement/Rotation
Velocity/Angular velocity
Acceleration/Angular acceleration
Connector displacement
Connector velocity
Connector acceleration
Continue...
Cancel


Assembly defaults
Module: Load
Model: Model-1
Step: Initial
Edit Boundary Condition
Name: BC-1
Type: Symmetry/Antisymmetry/Encastre
Step: Initial
Region: Set-3
CSYS: (Global)
XSYMM (U1 = UR2 = UR3 = 0)
YSYMM (U2 = UR1 = UR3 = 0)
ZSYMM (U3 = UR1 = UR2 = 0)
XASYMM (U2 = U3 = UR1 = 0; Abaqus/Standard only)
YASYMM (U1 = U3 = UR2 = 0; Abaqus/Standard only)
ZASYMM (U1 = U2 = UR3 = 0; Abaqus/Standard only)
PINNED (U1 = U2 = U3 = 0)
ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)
OK
Cancel
Y
Z
X

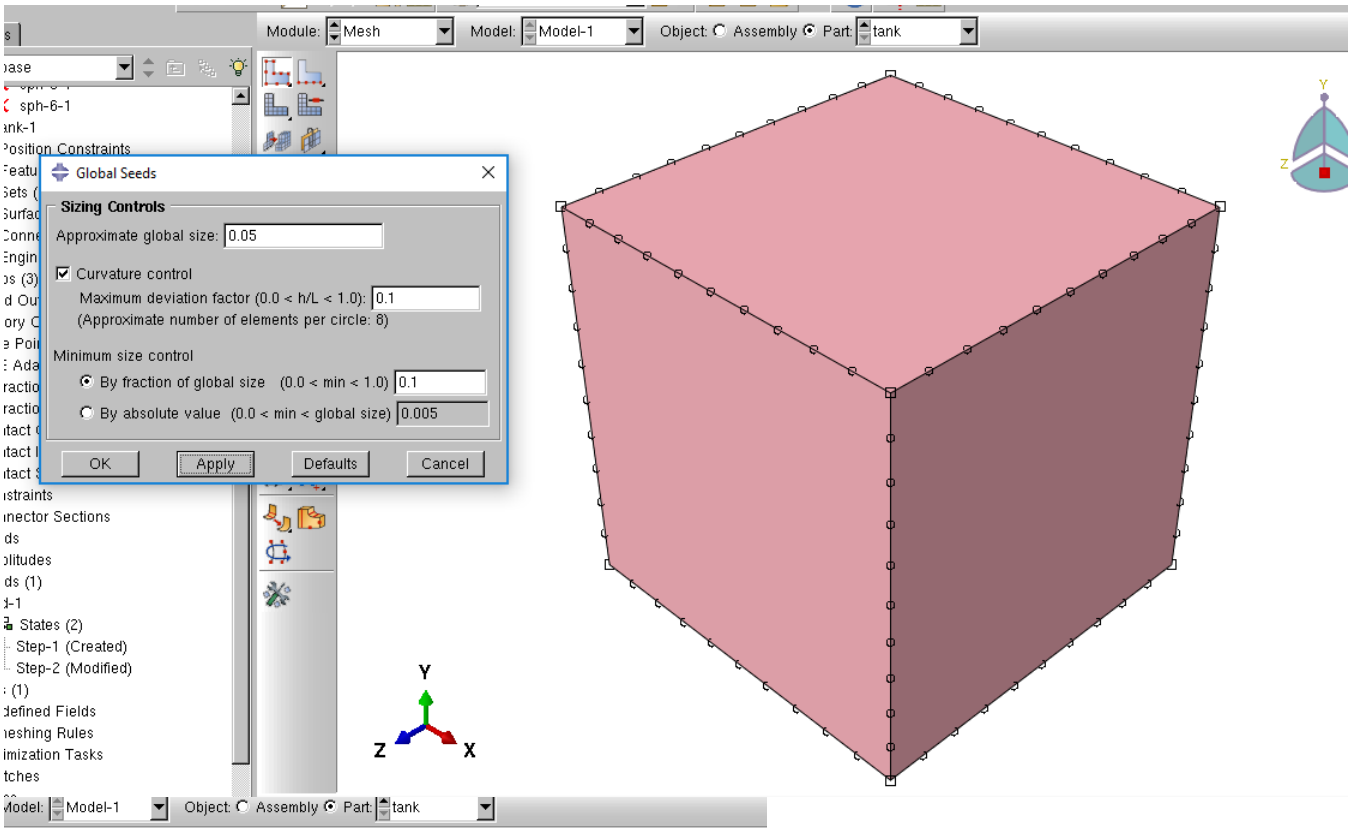
6. Field output requests

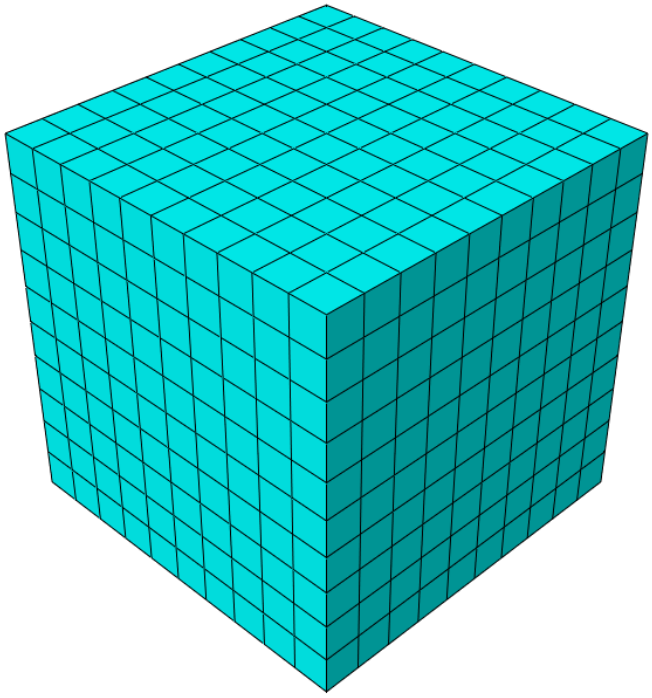
Before meshing select the **field output request** and increase the interval to 500.

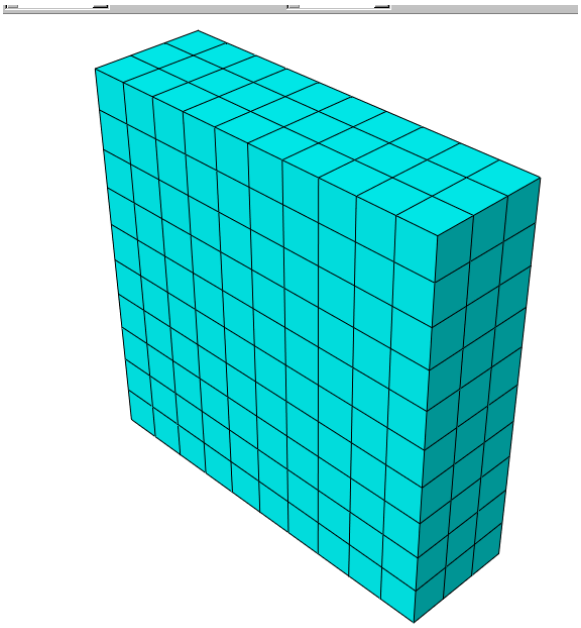


7. Meshing

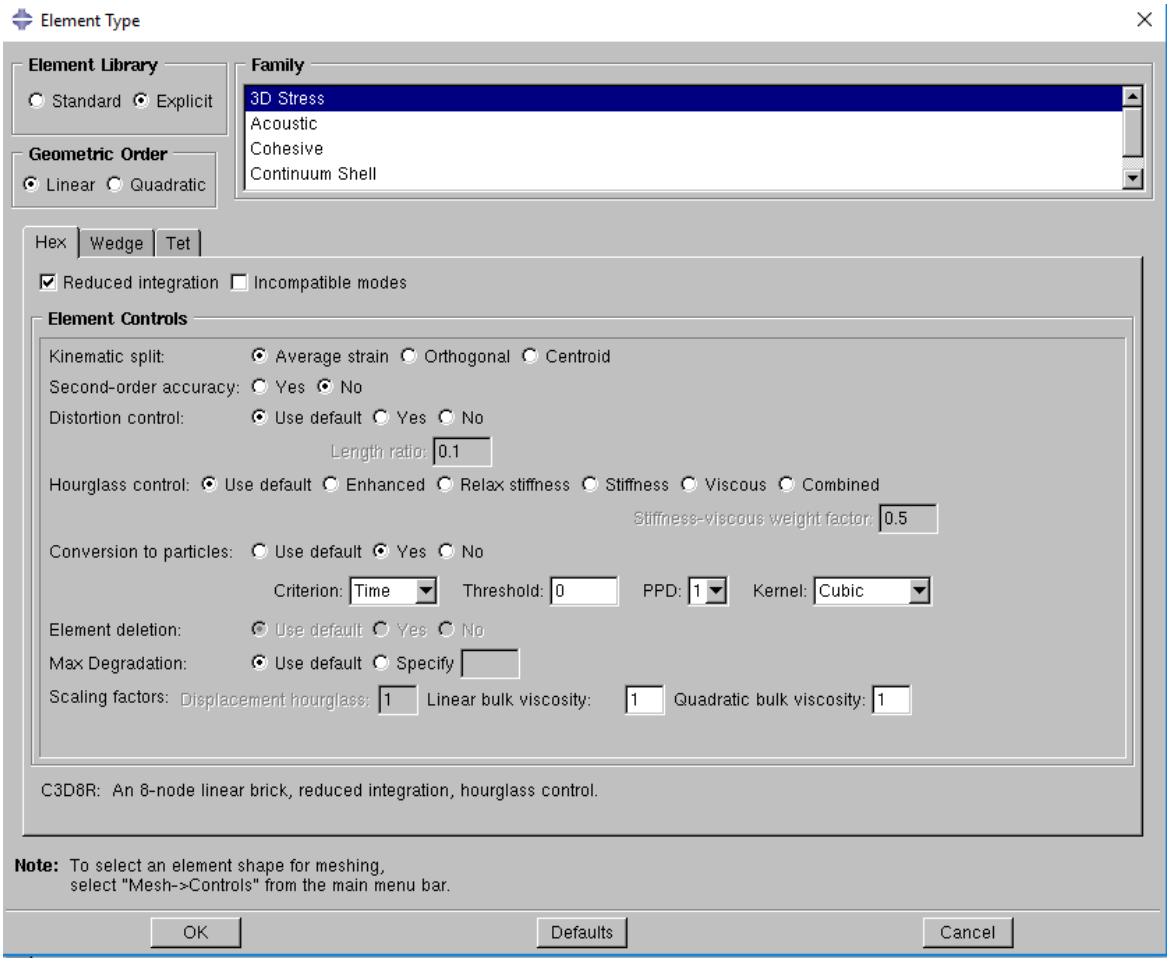
In the meshing module mesh the aluminum tank and water block with spacings of about 0.05. Now assign an element type to the water


Module: Mesh
Model: Model-1
Object: Assembly Part: tank
Global Seeds
Sizing Controls
Approximate global size: 0.05
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0) 0.1
By absolute value (0.0 < min < global size) 0.005
OK
Apply
Defaults
Cancel
Step-1 (Created)
Step-2 (Modified)
Y
Z
X



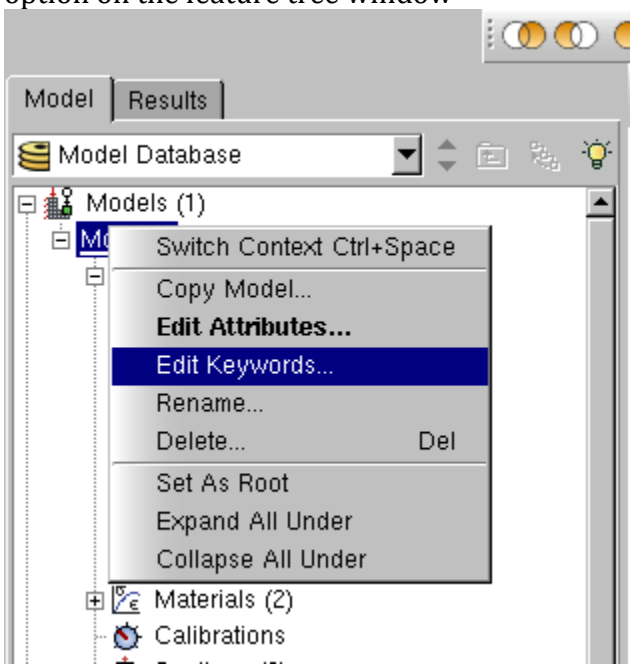


Now assign an element type to the water. Change the **element library** to explicit and Conversion to particle to yes, with time threshold = 0



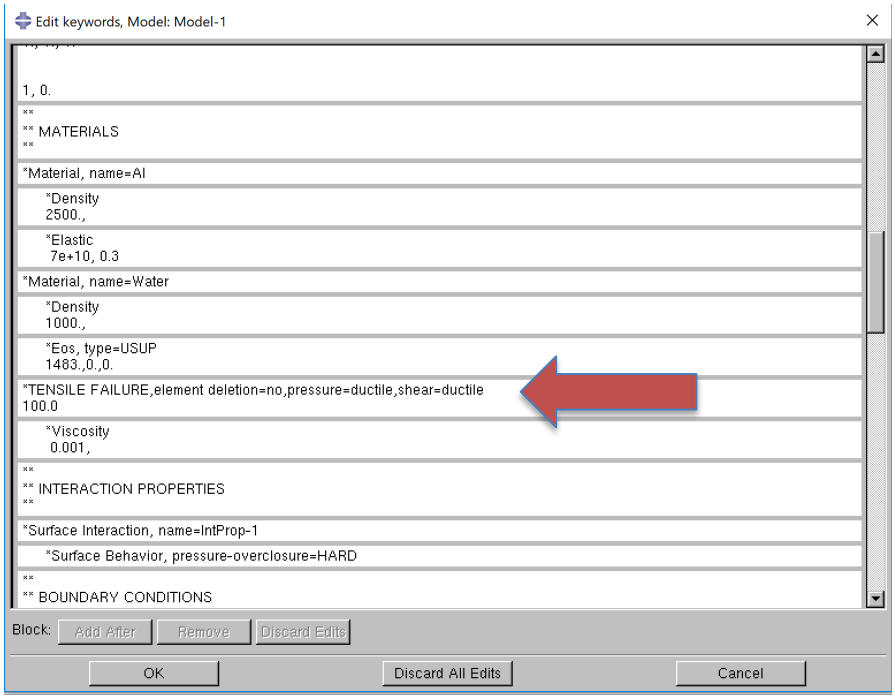
8. Editing Keywords

Before testing the model 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



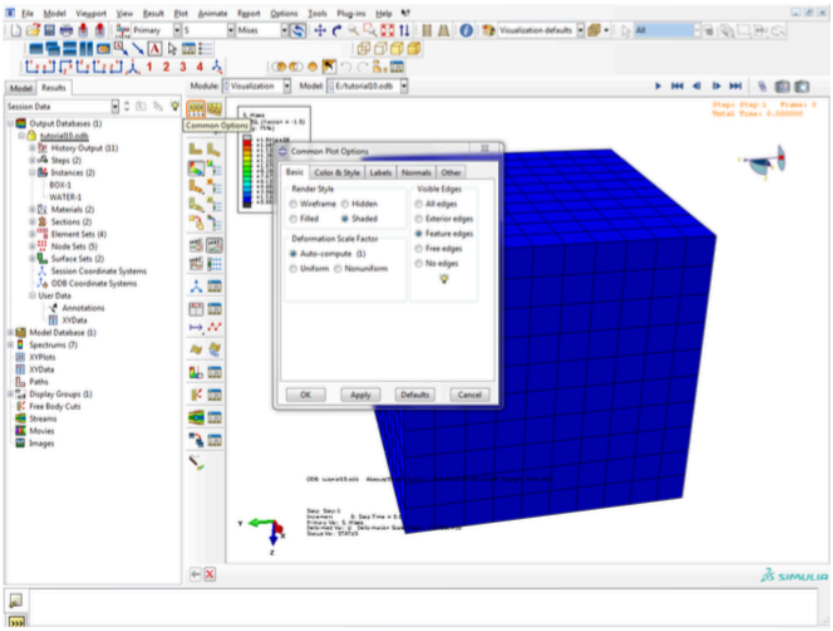
9. Create job and submit it

You can copy a pbs job from previous job or use the one provided.

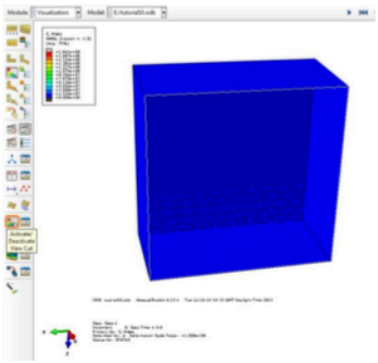
10. Post-Processing

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

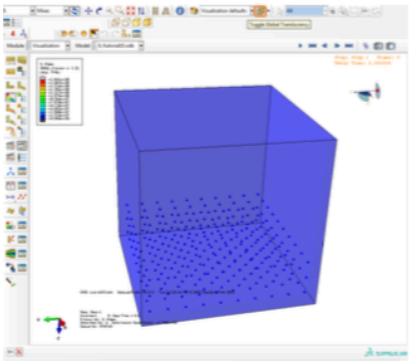
First remove the mesh lines; to do this select the **common options** tool and choose '**feature edges**'.



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

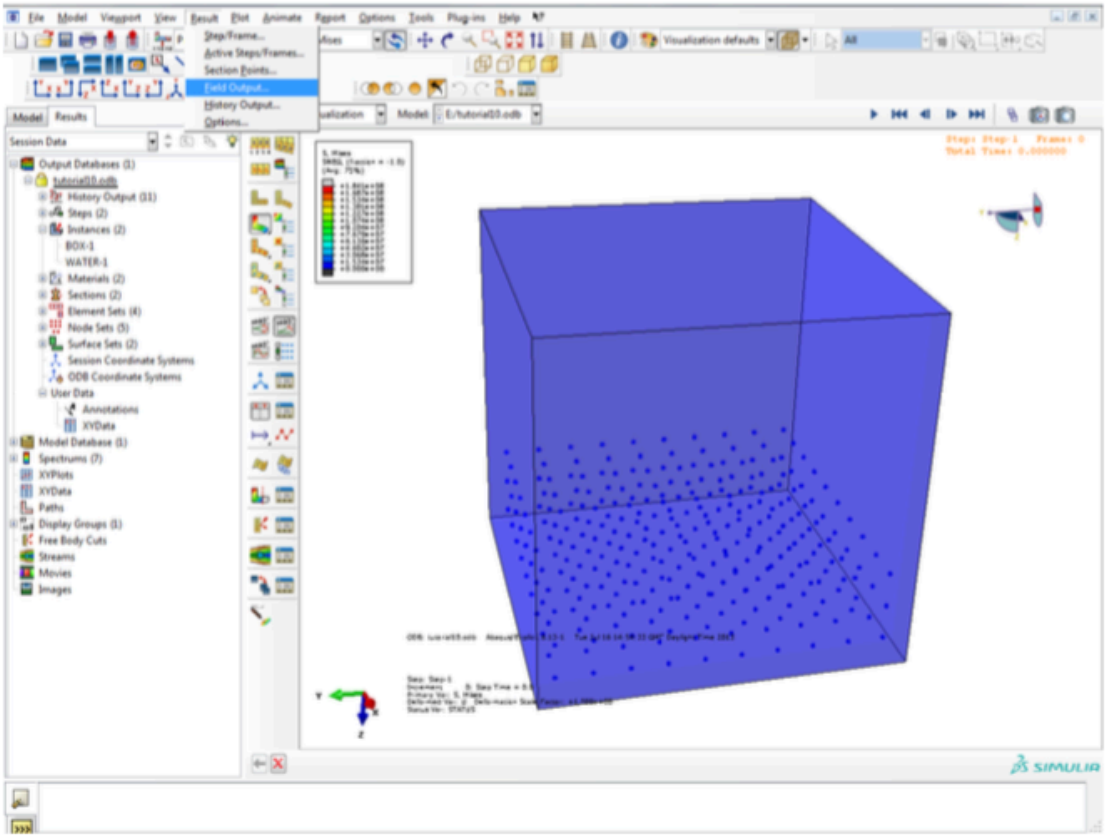


View cut cross-section



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**