PennState
College of Engineering

ME 563: Nonlinear Finite Elements

Application of the Finite Element Method to Real World Problems

Coupled Lagrangian-Eulerian Simulations

The Pennsylvania State University

In this tutorial, you will setup a Coupled-Eulerian-Lagrangian (CEL) model of the molding process of a polymeric bottle.

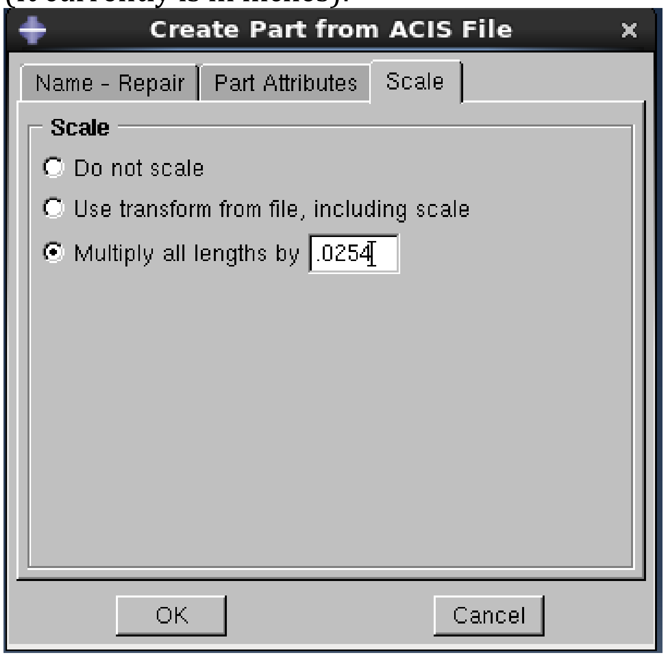
Plastic bottles might be created by molding melt polymer within rigid frames.

Main assumptions and limitations. In order to keep the tutorial simple and fast to solve, the external and internal frames will be considered as rigid bodies while the melt polymer will be modelled as a simple elastic material with very low stiffness. Also, no 2D modeling is allowed with CEL so a thin layer of 3D elements is here considered.

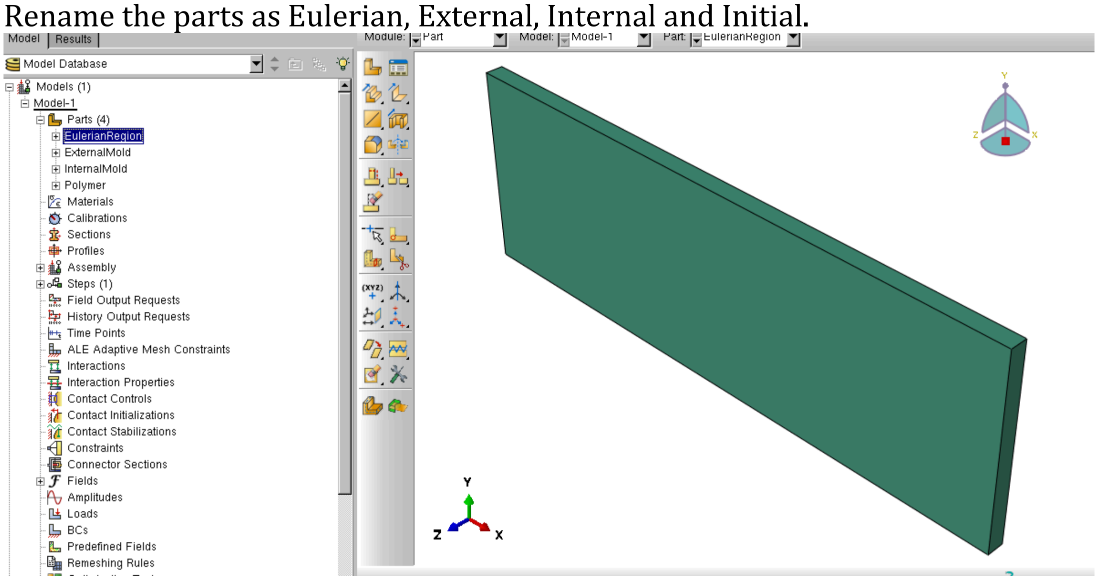
1. Geometry Import

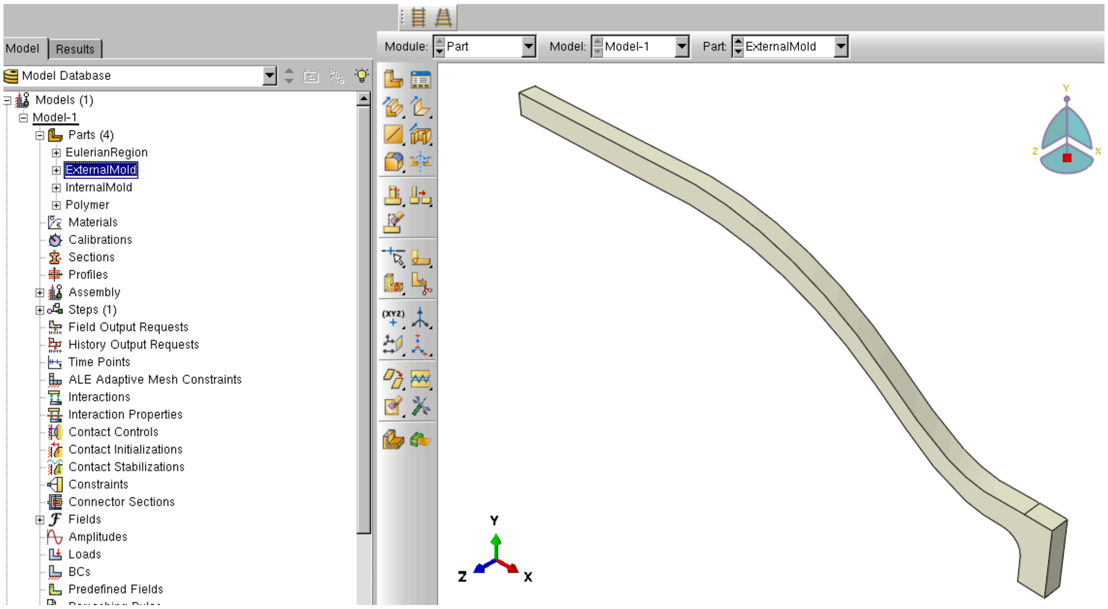
Obtain geometry from GitHub:
git clone https://github.com/rhk12/cel

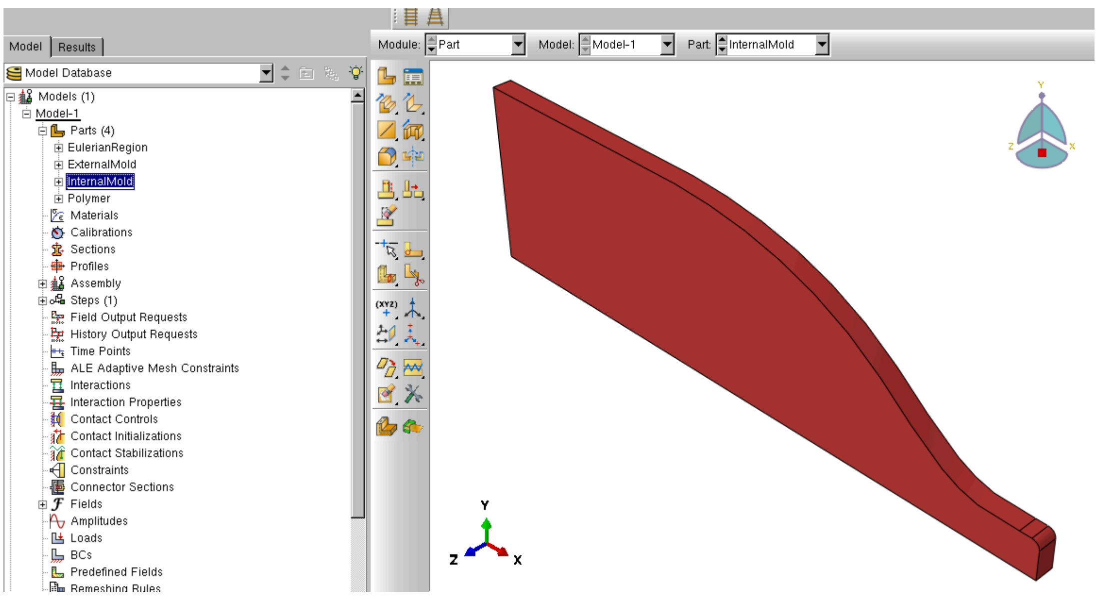
Import the file bottle.sat (File → Import → Part). During the import scale the geometry to meters. (It currently is in inches):

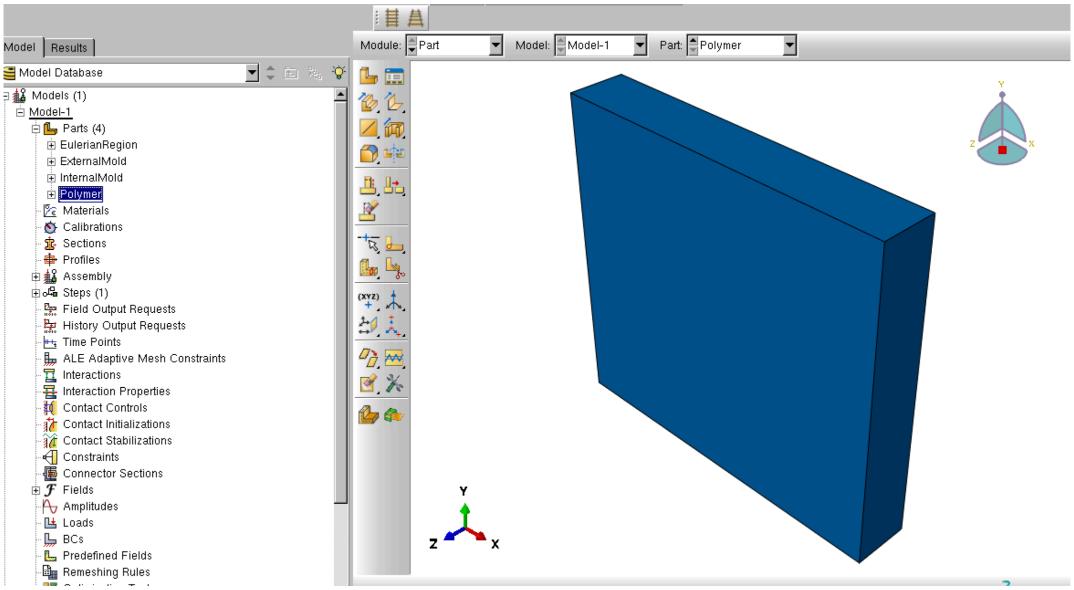


Rename the parts as Eulerian, External, Internal and Initial.

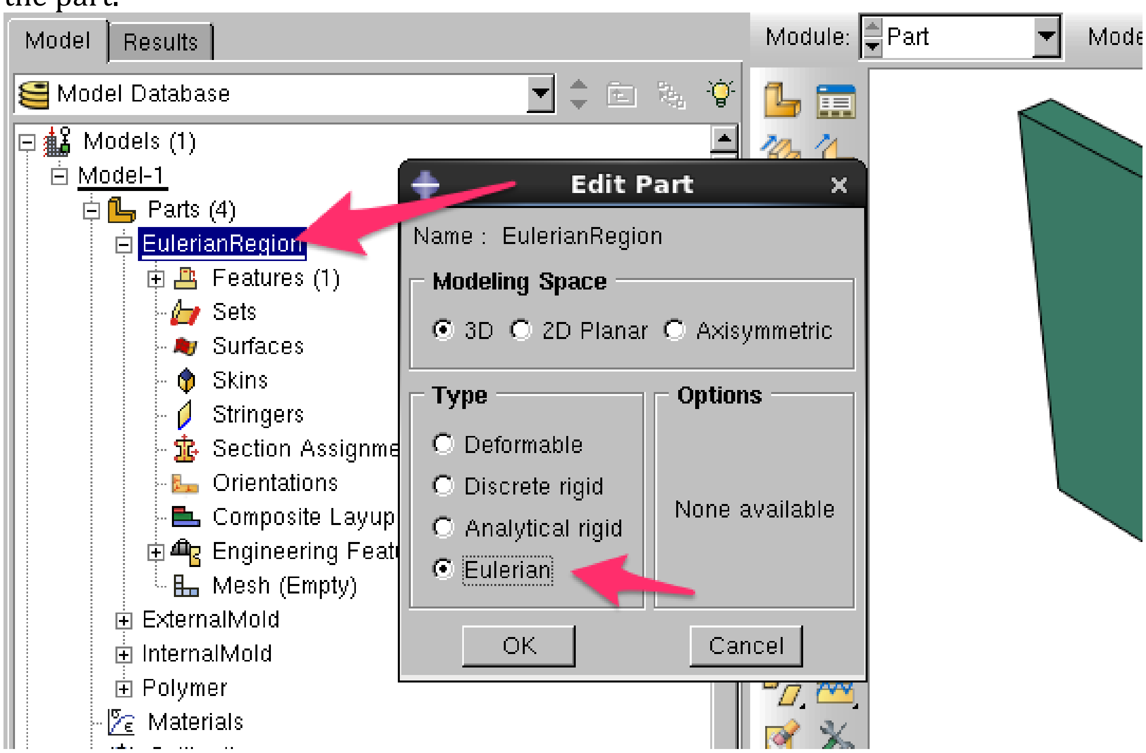



Model
Results
Module: Part
Model: Model-1
Part: ExternalMold
Model Database
Models (1)
Model-1
Parts (4)
EulerianRegion
ExternalMold
InternalMold
Polymer
Materials
Calibrations
Sections
Profiles
Assembly
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Y
Z
X


Model
Results
Module: Part
Model: Model-1
Part: InternalMold
Model Database
Models (1)
Model-1
Parts (4)
EulerianRegion
ExternalMold
InternalMold
Polymer
Materials
Calibrations
Sections
Profiles
Assembly
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Y
Z
X



In the model tree, click with the right button on Eulerian, select edit and choose Eulerian as type of the part.



2. Material and section properties

Start to define the material properties – go into the properties module

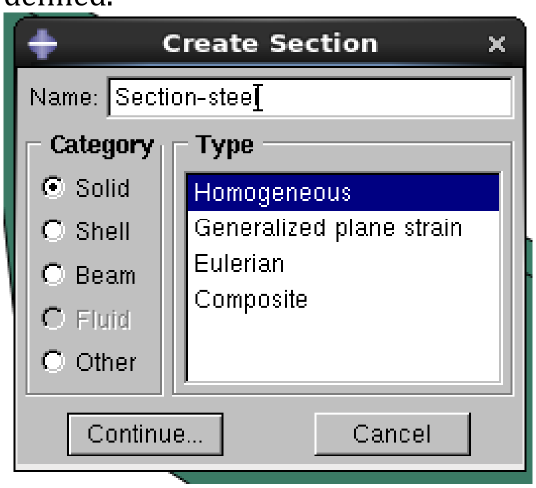
Define two material models called ***Polymer*** for the bottle and steel for the internal and external molds.

1. In the Edit Material dialog box, name the material Polymer.
2. From the material editor's menu bar, select Mechanical → Elasticity →Elastic
3. Enter a Young modulus equal to **E = 0.21 x 10⁹ Pa** and a Poisson ration equal to ***ν*=0.35**.
4. From the material editor's menu bar, select General → Density
5. Enter a density value equal to **ρ = 1455 kg/m³**.
6. Click OK to exit the material editor

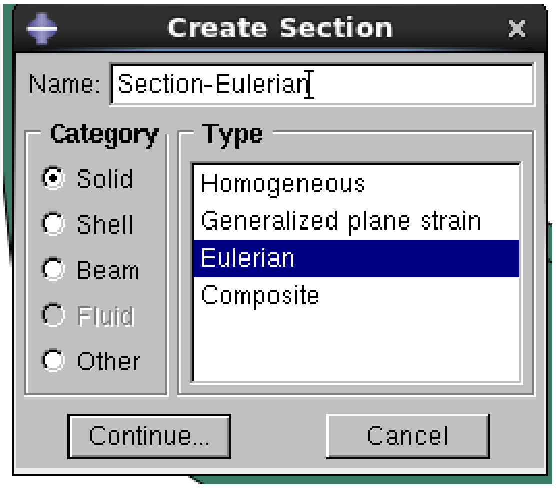
Due to the rigid body definitions, the steel mechanical properties in terms of Young's modulus and Poisson ratio will not have any influence on the actual solution while the density chosen will drive the computational cost of the analysis due to the explicit solver used.

Repeat the same as above for a material called ***Steel*** using the following data:
Young's modulus=210 x 10⁹ Pa
Poisson Ration = 0.35
Density = 7800 kg/m³

Create a solid homogeneous section called Section-Steel using the Steel material previously defined.

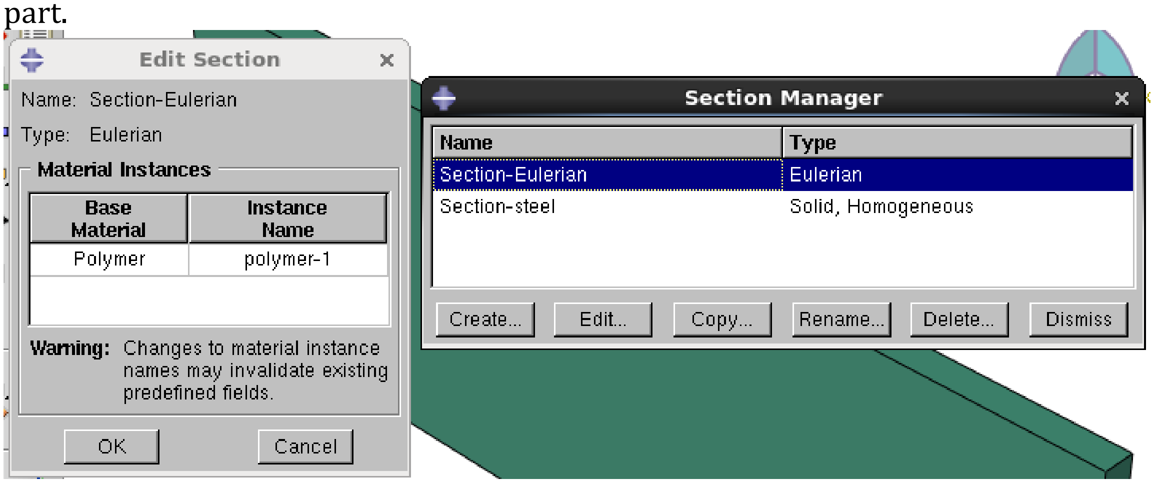


Create another solid section called Section-Eulerian selecting Eulerian as type and polymer as base material.



Assign the sections:
Go into the Property Module and click the Assign Section icon.

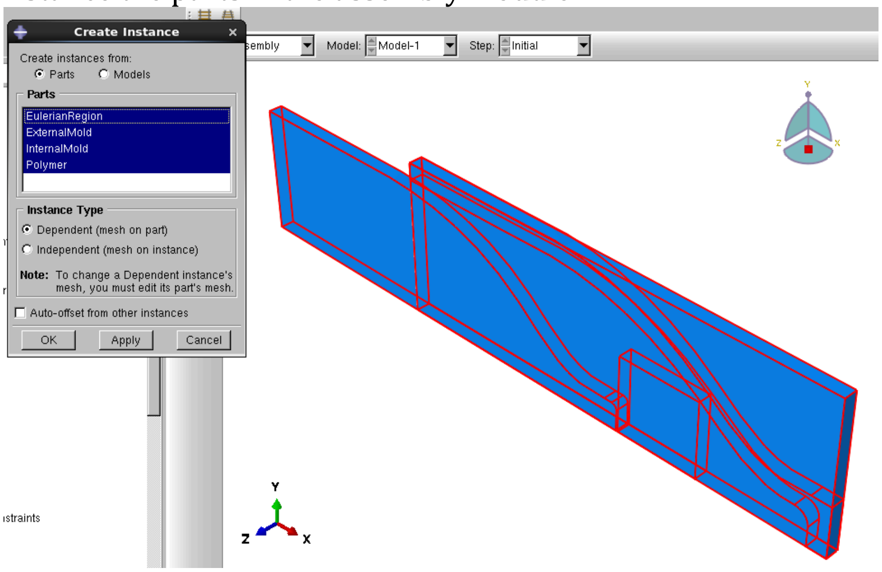
Assign Section-Steel to the internal and external molds and the Eulerian section to the Eulerian part.



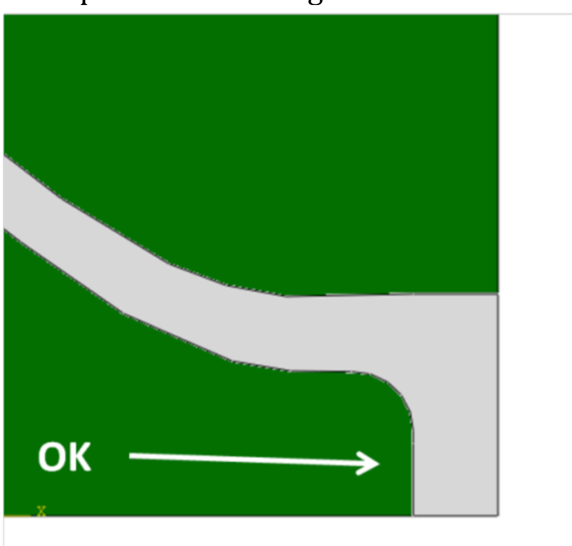
The initial polymer part does not need any section assignment since it will not be part of the actual simulation but only used to define the initial conditions

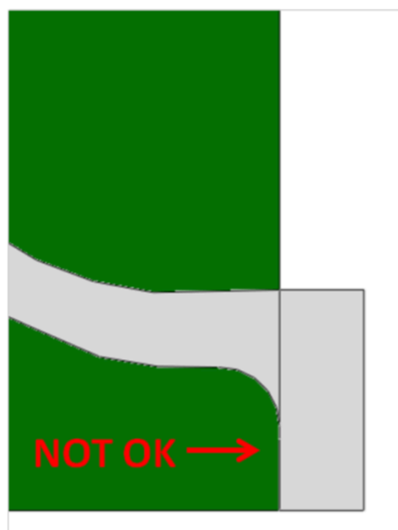
3. Create an assembly

Instance the parts in the assembly module.



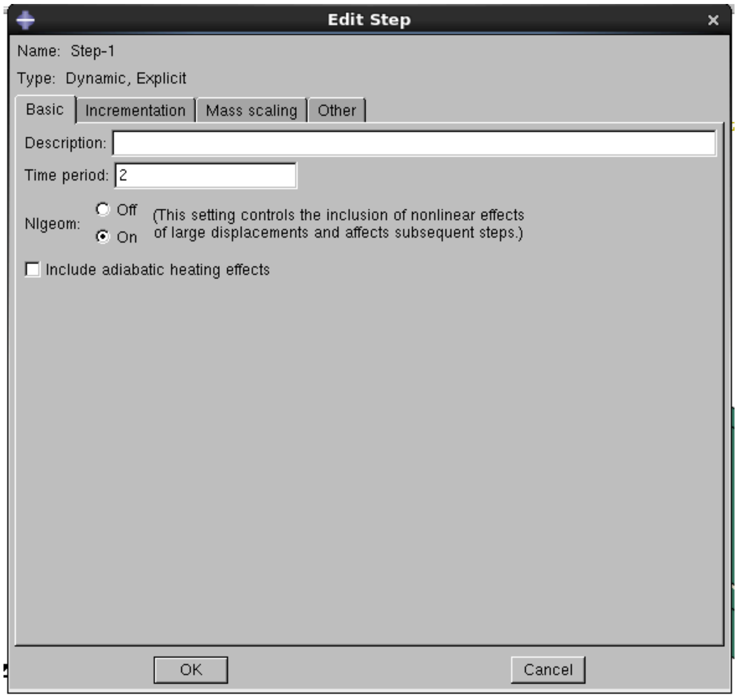
It is important that in every ALE analysis, all the parts in contact with the Eulerian material overlap the Eulerian region at the contact surface.





4. Define the analysis step

Go into the Step module and select the Create Step button. Create a Dynamic Explicit step choosing 2 seconds as Time step and making sure that the Nlgeom parameter is on.



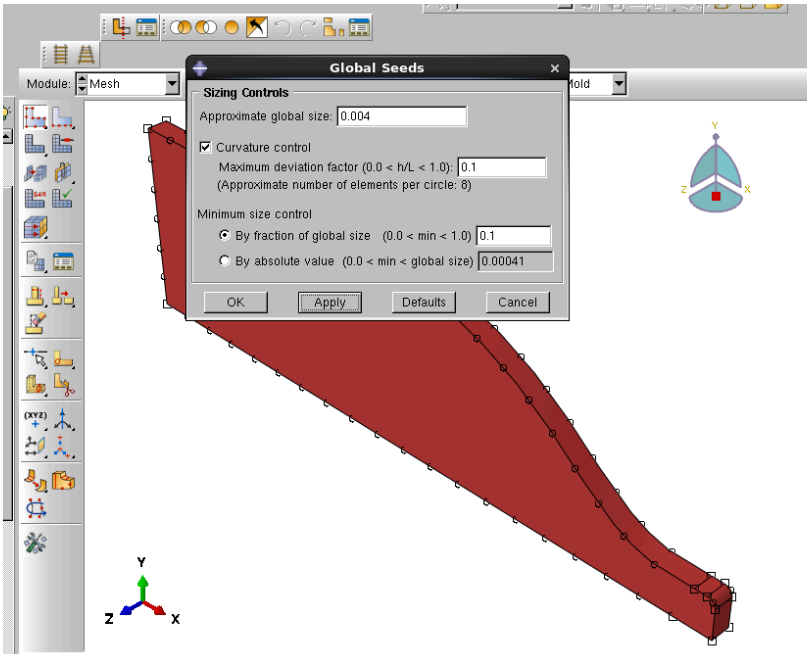
In the model tree, explode the Field Output requests container and double click on F-Output-1. Set the number of interval as 50, add EVF (Eulerian Volume Fraction) as new element output in the Volume/thickness/coordinates container and click OK.

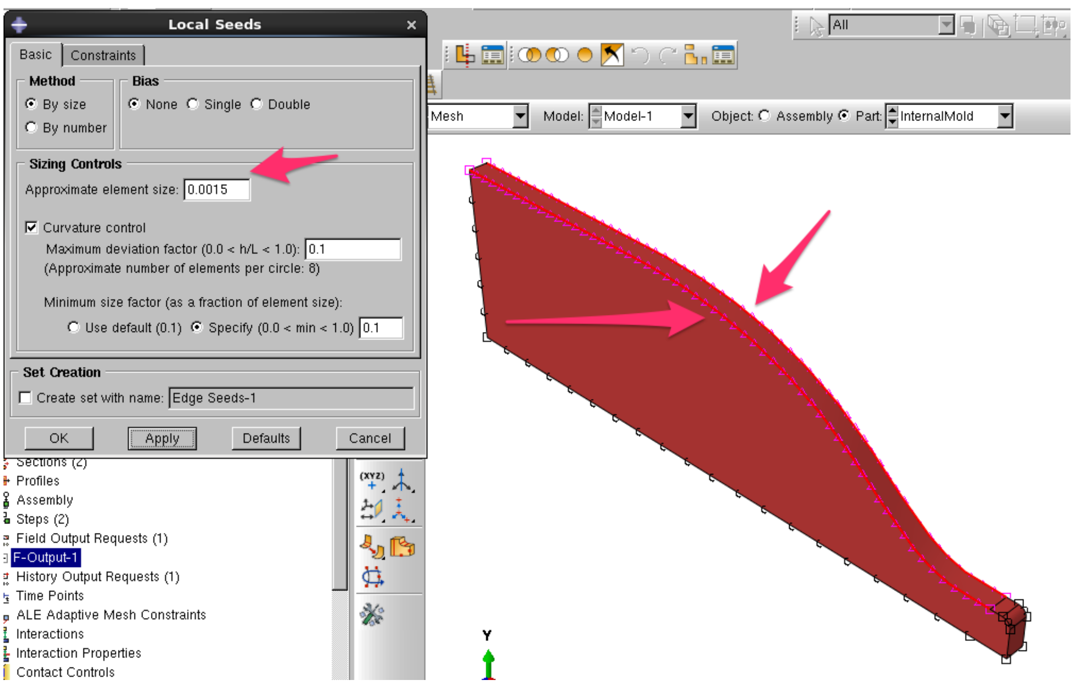
5. Meshing

Go into the Mesh module to discretize the molds and the Eulerian part.

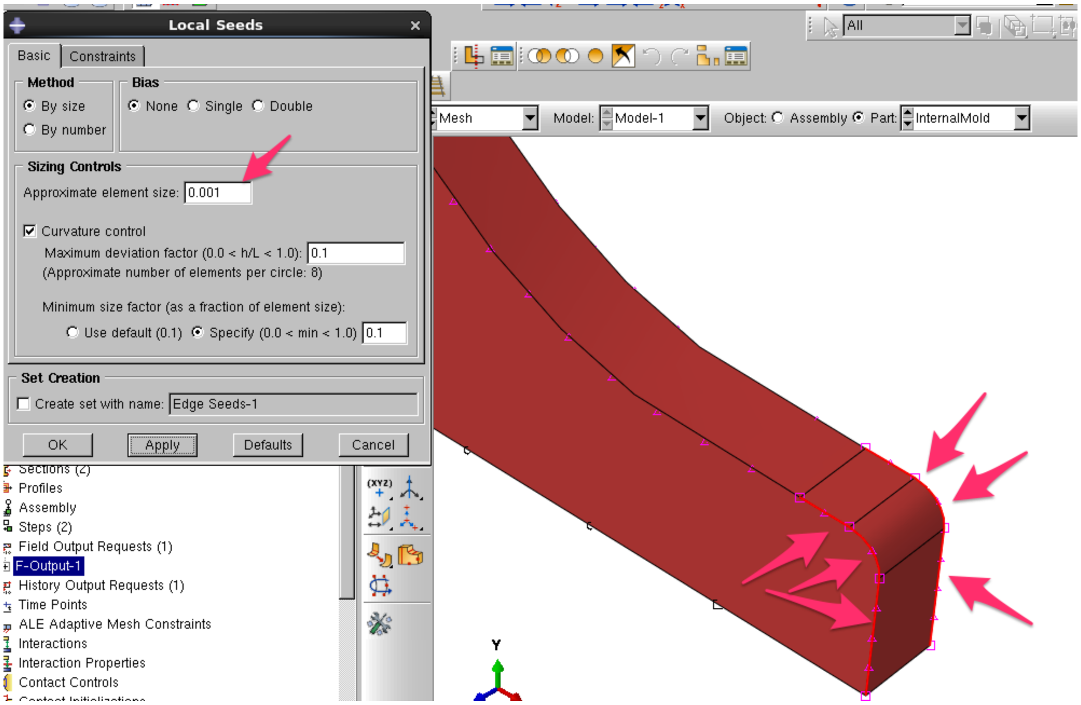
Select hexahedral swept elements for all the parts.

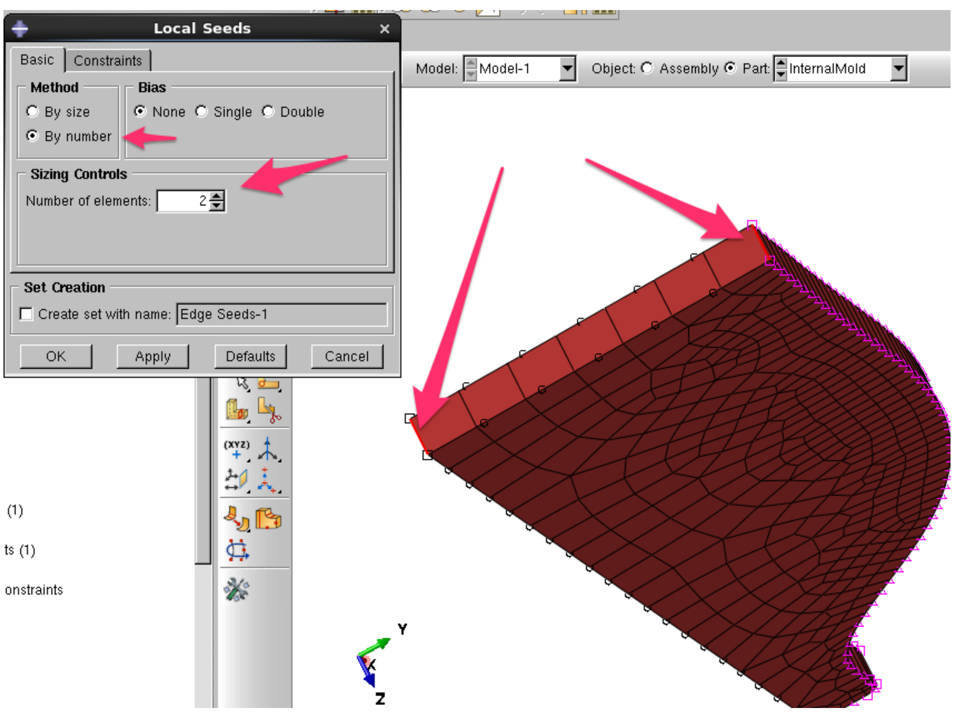
1. Mesh of the molds.
For each mold select 0.004 as global element size and 0.0015 as local element size for the edges shown in the following picture.

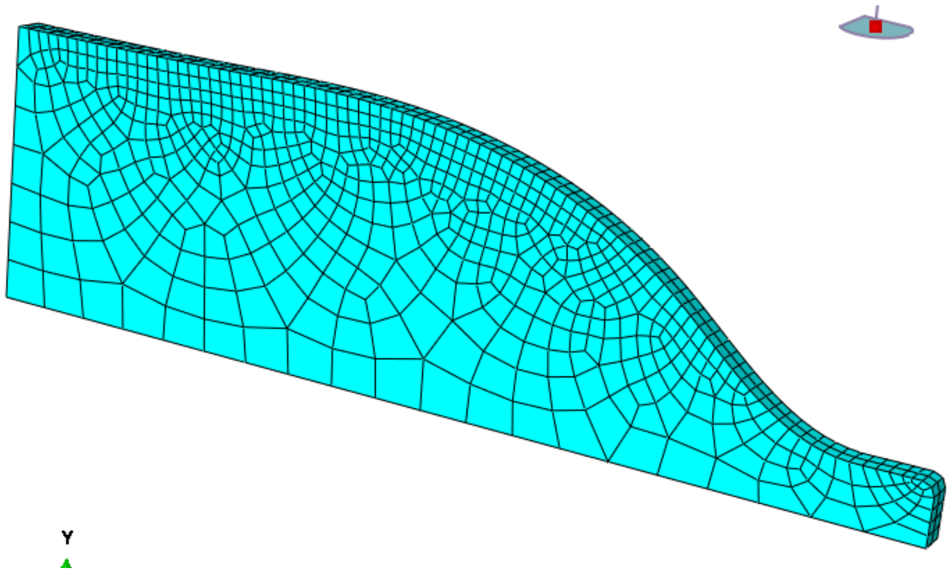


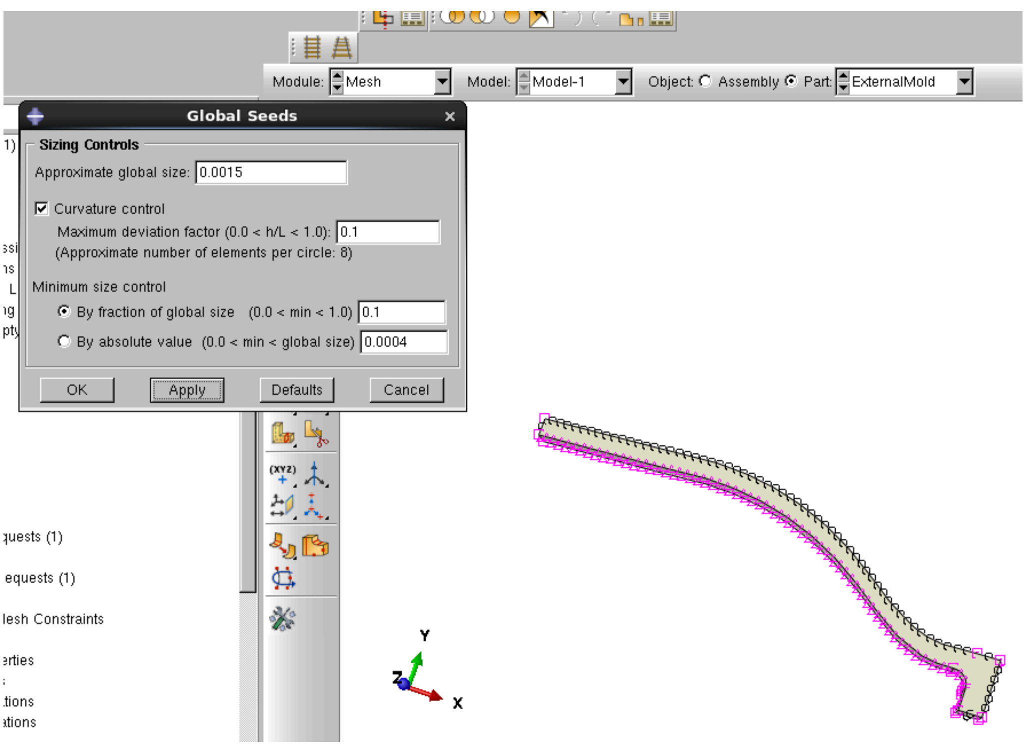


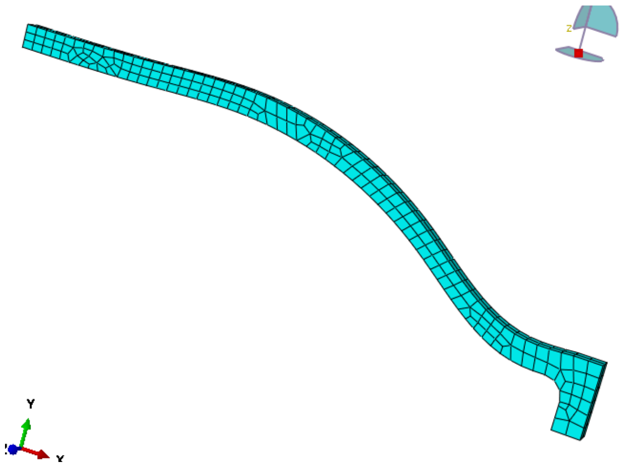
Lastly, set 2 as the number of elements in the thickness of the molds and 0.02 as local element size at the top of the internal mold.


Local Seeds
Basic
Constraints
Method
By size
By number
Bias
None
Single
Double
Sizing Controls
Approximate element size: 0.001
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size factor (as a fraction of element size):
Use default (0.1)
Specify (0.0 < min < 1.0) 0.1
Set Creation
Create set with name: Edge Seeds-1
OK
Apply
Defaults
Cancel
Mesh
Model: Model-1
Object: Assembly Part: InternalMold
Profiles
Assembly
Steps (2)
Field Output Requests (1)
F-Output-1
History Output Requests (1)
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Y

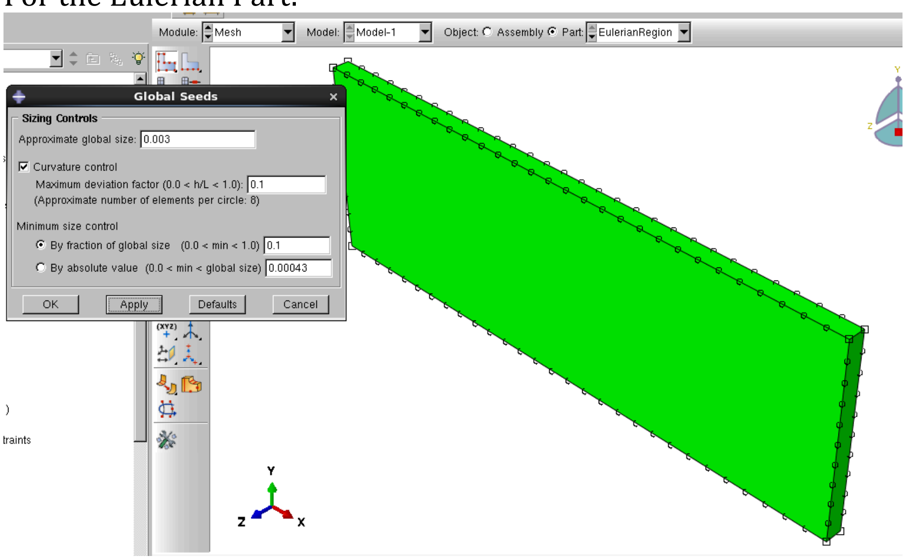

Local Seeds
Basic
Constraints
Method
By size
By number
Bias
None
Single
Double
Sizing Controls
Number of elements: 2
Set Creation
Create set with name: Edge Seeds-1
OK
Apply
Defaults
Cancel
Model: Model-1
Object: Assembly Part: InternalMold
Y
Z

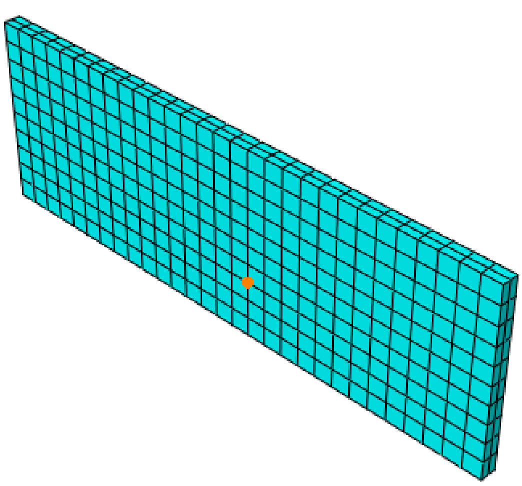

Y


Module: Mesh
Model: Model-1
Object: Assembly Part: ExternalMold
Global Seeds
Sizing Controls
Approximate global size: 0.0015
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) 0.0004
OK
Apply
Defaults
Cancel
Y
Z
X



For the Eulerian Part:



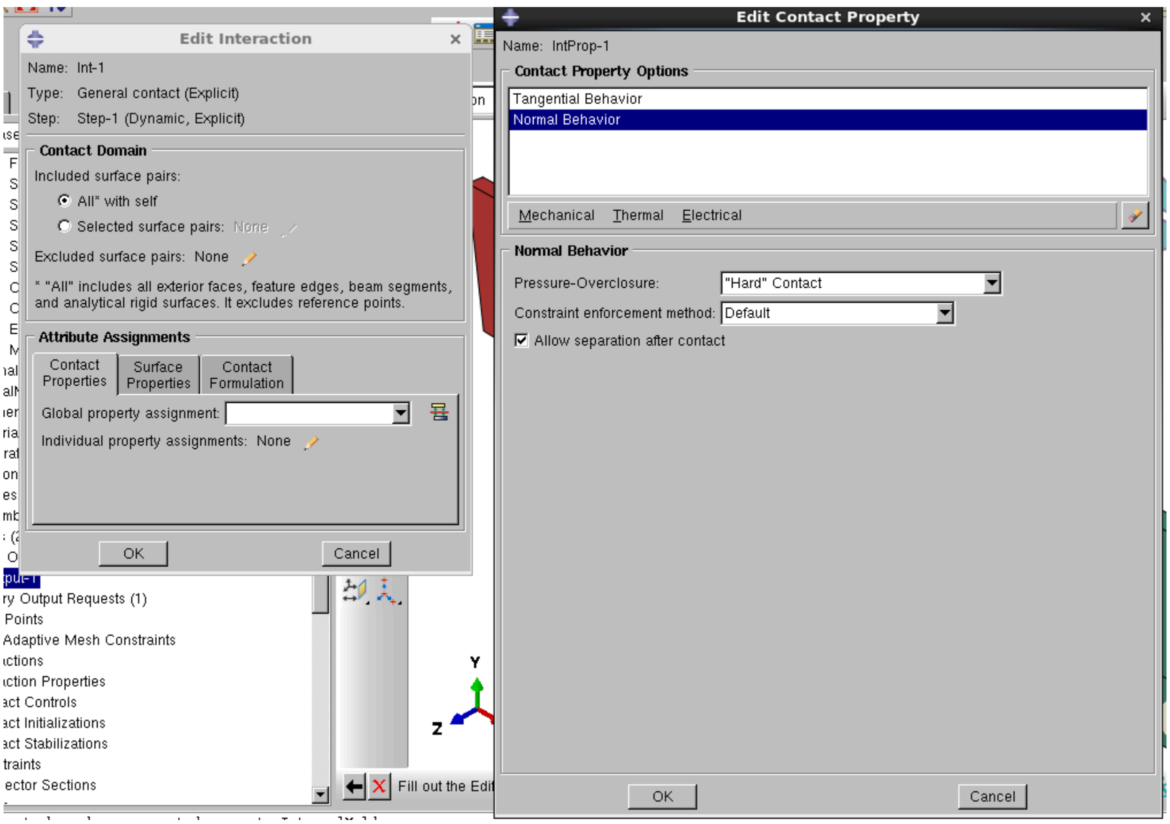


6. Interactions/Contact

Go into the Interaction modules. **Three main tasks have to be performed here.**

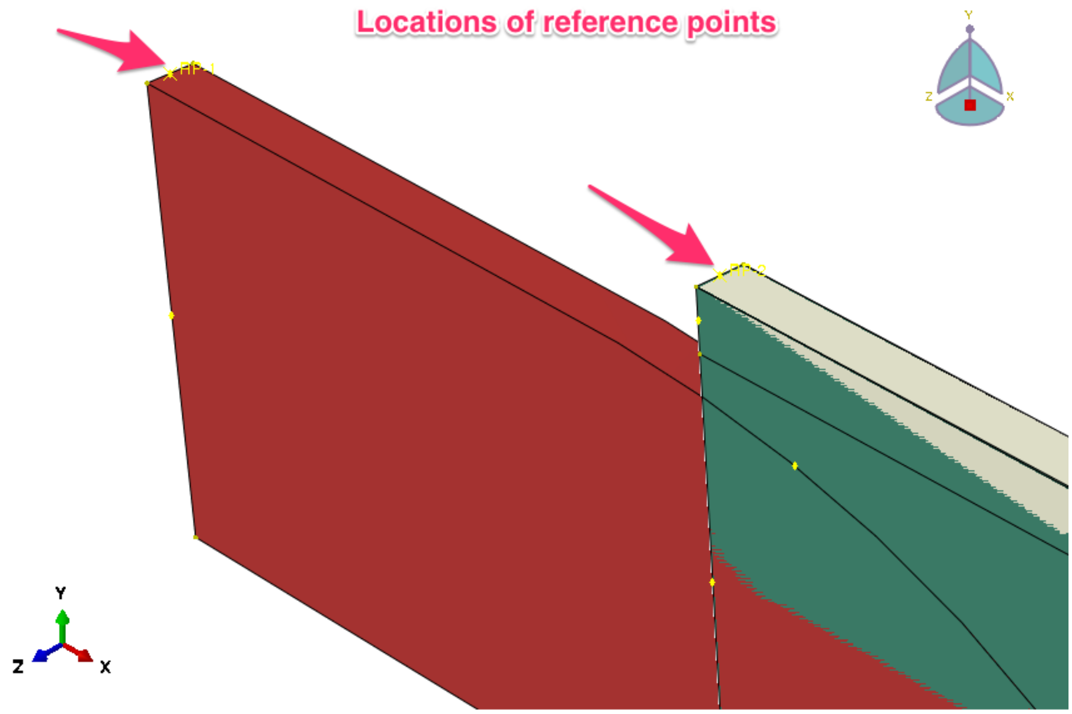
1. Create a new General Contact Interaction.
Click on the first button of the toolbox to create a new interaction. Select Initial as step and General contact as contact type.

In the General Contact window, select "All* with self" as contact domain and click on the Create Interaction property button close to the Global Property Assignment. Select Contact in the new window that appears and create a contact property with a **Hard Contact normal behavior** and a **frictionless tangential**

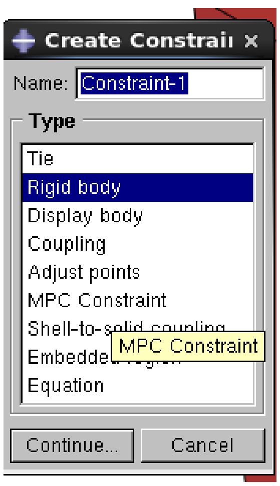


2. Create two new rigid bodies constraints.

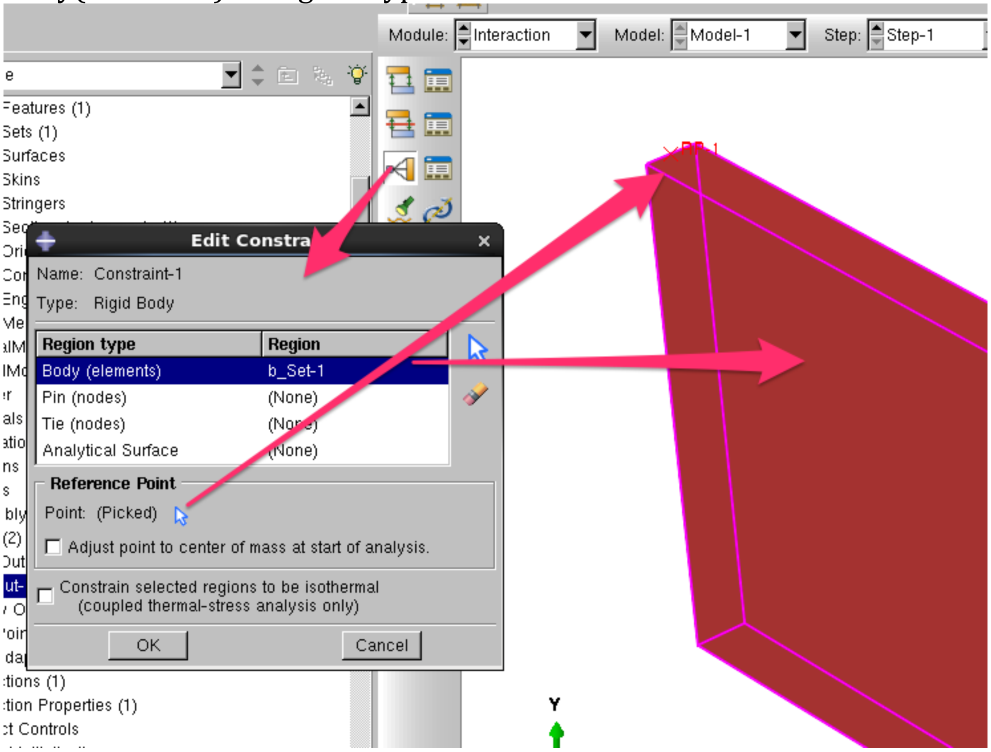
First, click on Tool → Reference Point from the menu toolbar to create a new reference point for each mold.



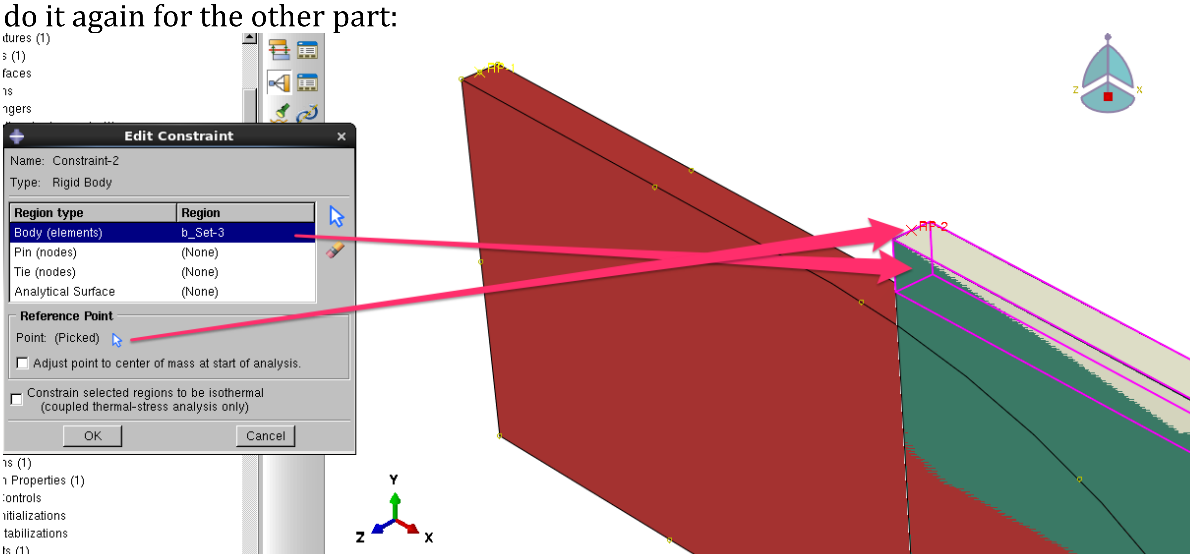
Click on the third button of the toolbox's left column to create two new rigid body constraints for the internal and external molds. Select Rigid Body as Constraint Type,



Body(Elements) as region Type and the new Reference Points as the control points.

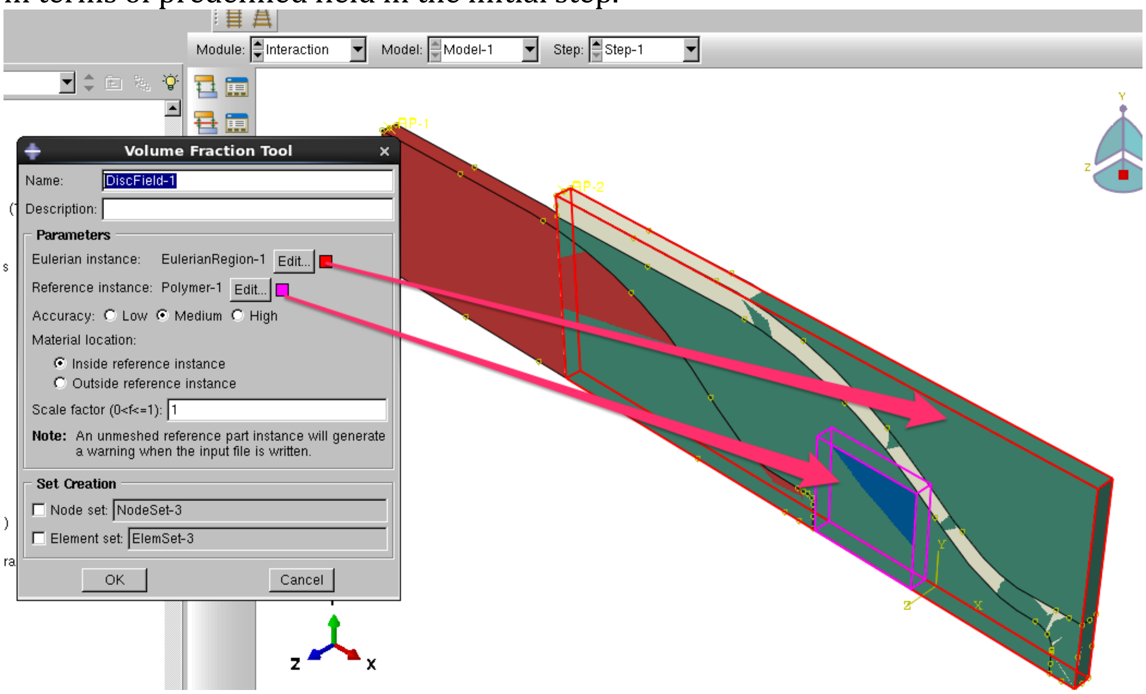


do it again for the other part:

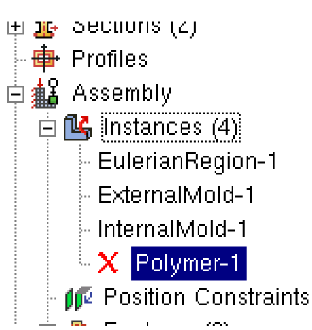


3. This step is fundamental for <u>any CEL analysis where the Eulerian mesh does not conform to the initial material boundaries</u> and aims at identifying which part of the Eulerian region initially contains the material.

To achieve this goal, you have to first click on **Tools→Discrete Field→Volume Fraction Tool**. Follow the instruction and select the Eulerian part first and then the Initial part as the reference instance. This operation will create a discrete field that will then be associated to the Eulerian part in terms of predefined field in the initial step.



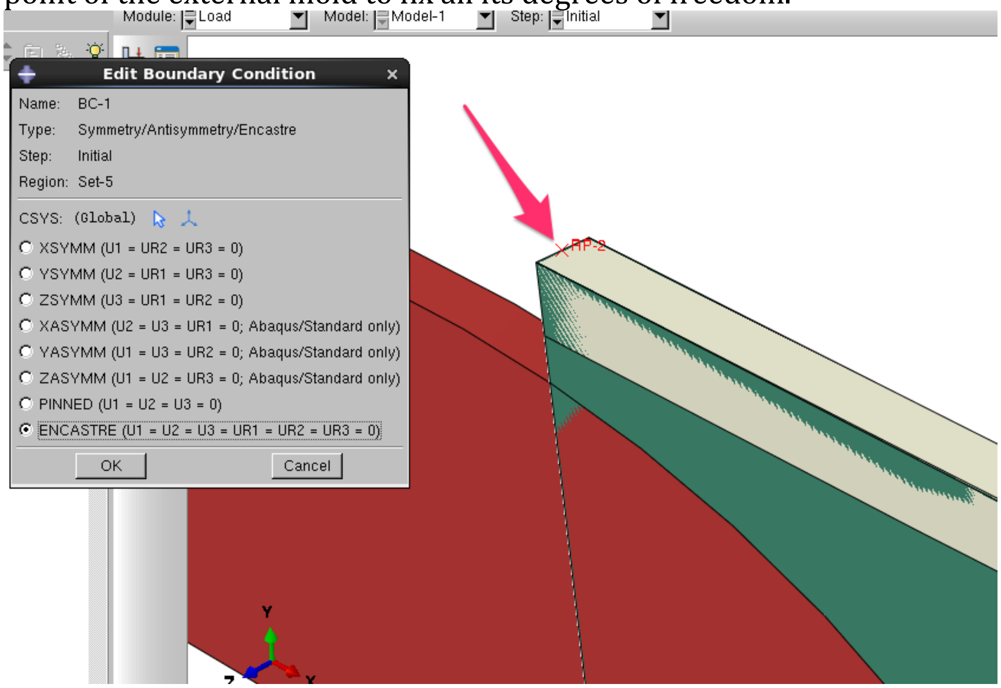
At this point you have to remove the instance of the Initial part from the assembly. It might be useful to only suppress it since every change in the mesh of the Eulerian body will require a new calculation of the initial discrete field.



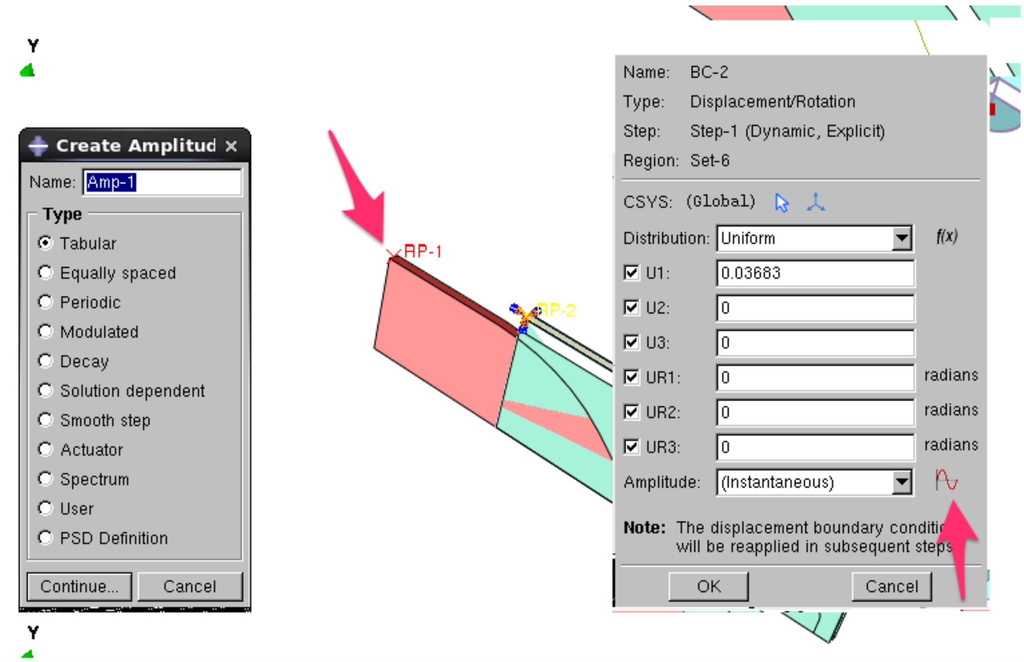
7. Loads and Boundary Conditions

Create the boundary conditions for the external and internal molds. Because of the rigid boy constraints previously defined, the boundary conditions have to be applied to the Reference points.

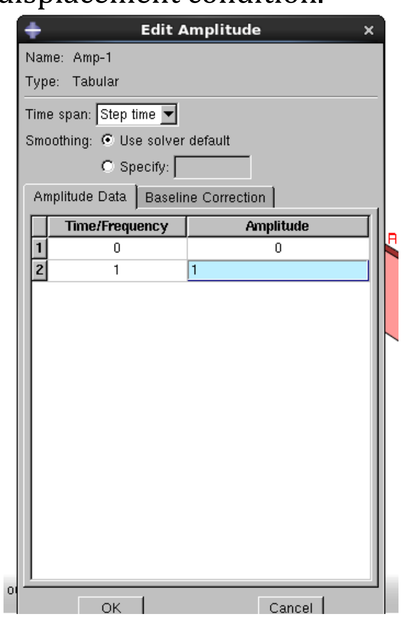
a) Click on the **Create Boundary condition** Icon, and create an **encastre** at the reference point of the external mold to fix all its degrees of freedom.



b) Create a displacement boundary condition at the reference point of the internal mold. Click on the Create **Boundary condition Icon**, select Step-1 as the step and displacement/rotation as type.

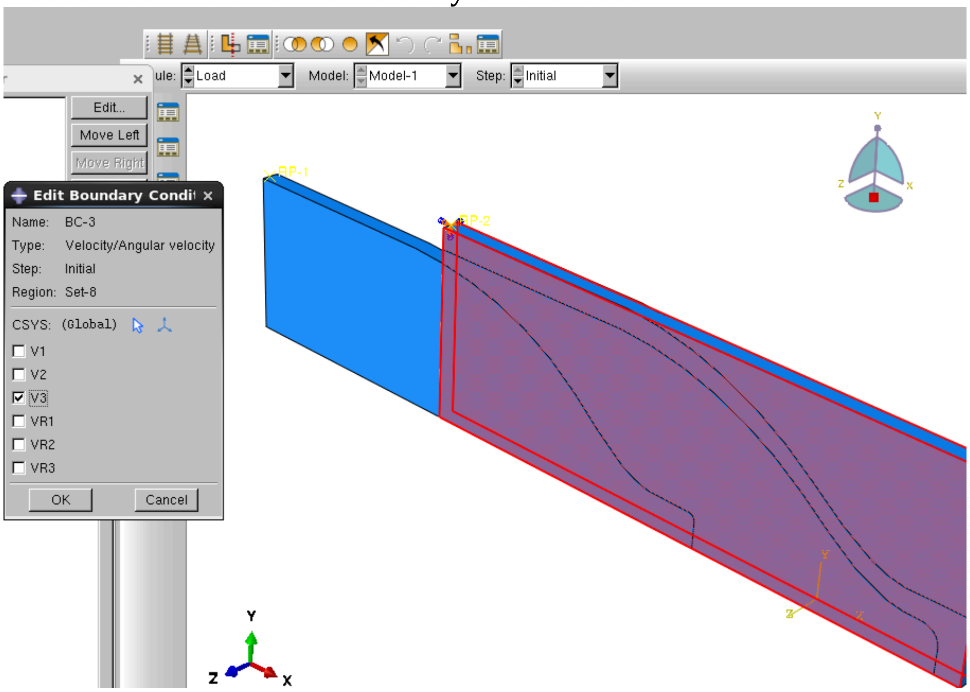


c) Select the Reference point of the internal mold, enter 0.03683 in the U1 degree of freedom and 0 in all the others. Then, click on the Create Amplitude Icon on the right and create a tabular amplitude replicating a standard ramp and apply it to the displacement condition.



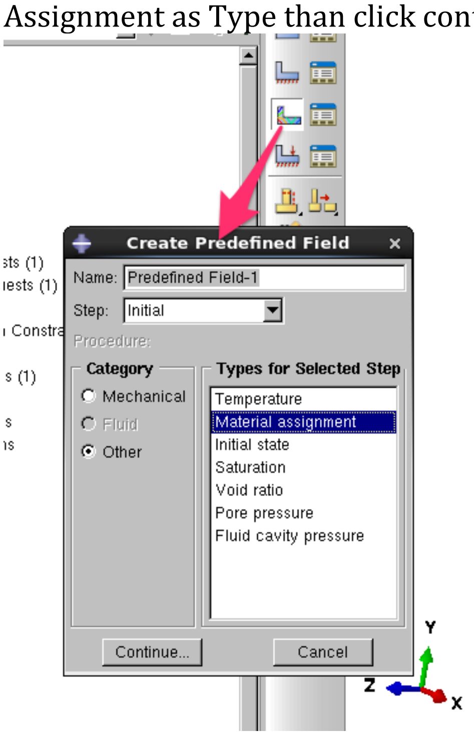
Create the **boundary conditions** for the Eulerian Region.

Click on the Create **Boundary condition Icon**, select Velocity/Angular velocity as Type, select the two faces of the Eulerian region with face normal to the Z axis and select the v3 degree of freedom to fix the normal velocity on this faces.

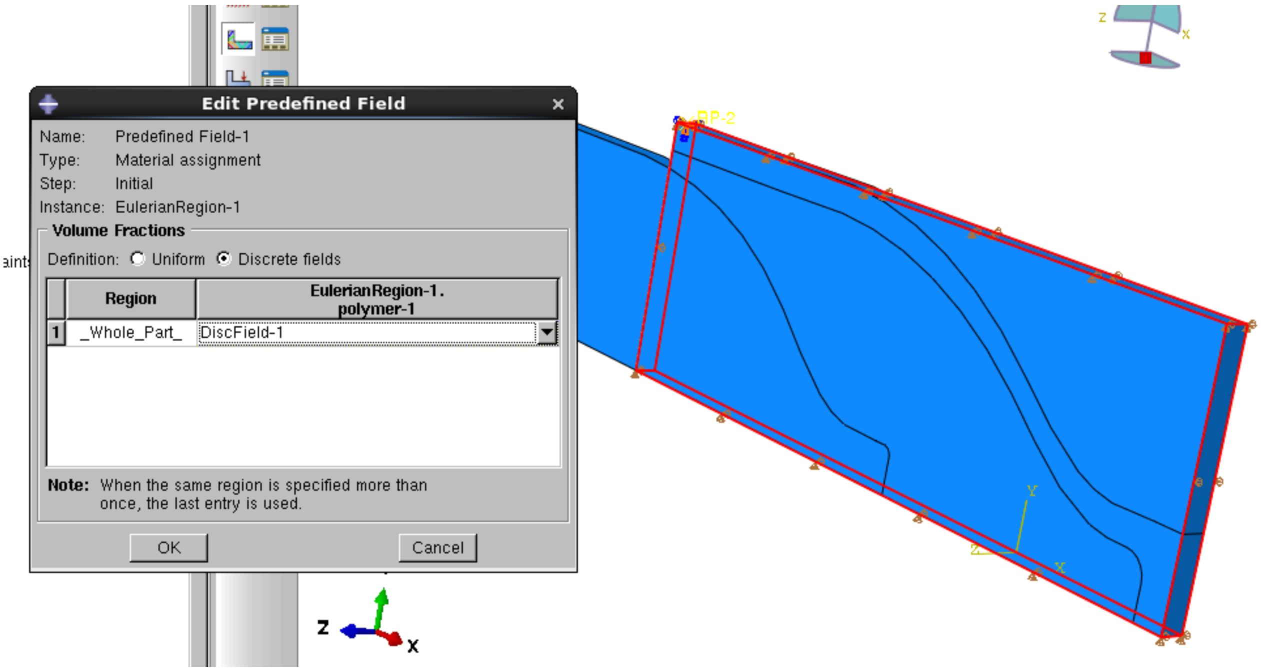


b. repeat the same operation by constraining the normal velocity (V2) at the surfaces normal to the Y axis.

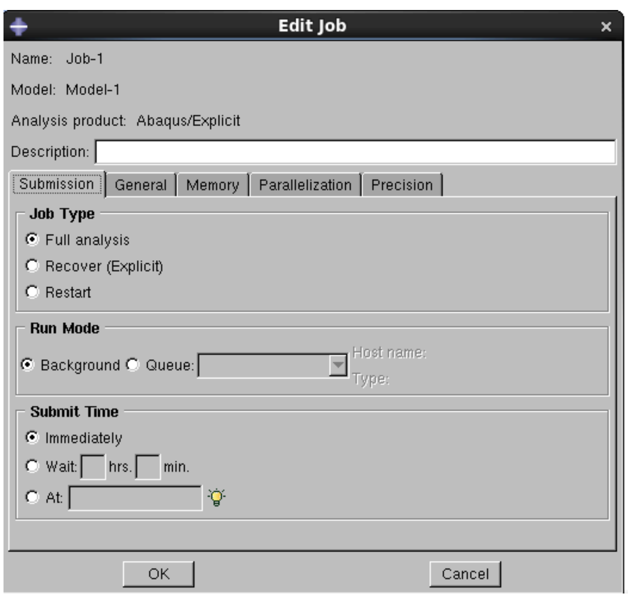
Click on the Create Predefined Field Icon, select the Initial step, Other as category and Material Assignment as Type than click continue.



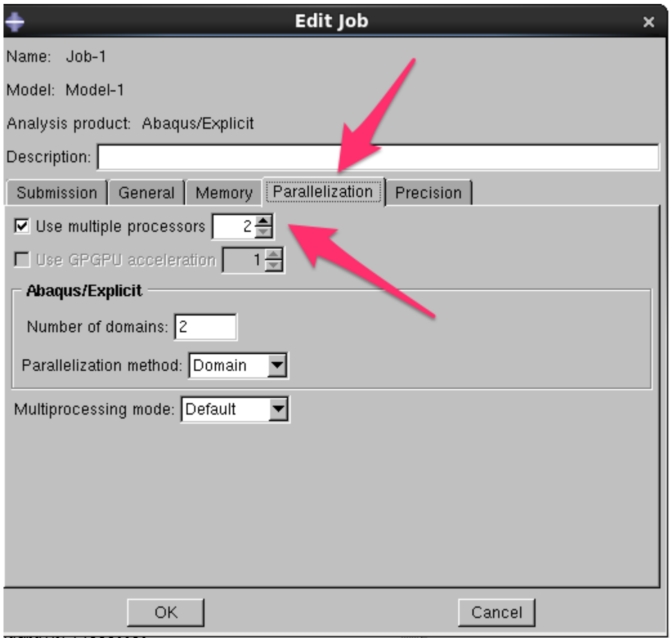
Select the Eulerian part in the viewport and the option Discrete Fields in the Volume fraction definition. At this point Select the whole Eulerian part and the Discrete Field previously created from the menu option.



8. Create a job and then submit it



Set the parallelization tab for 2 processors:

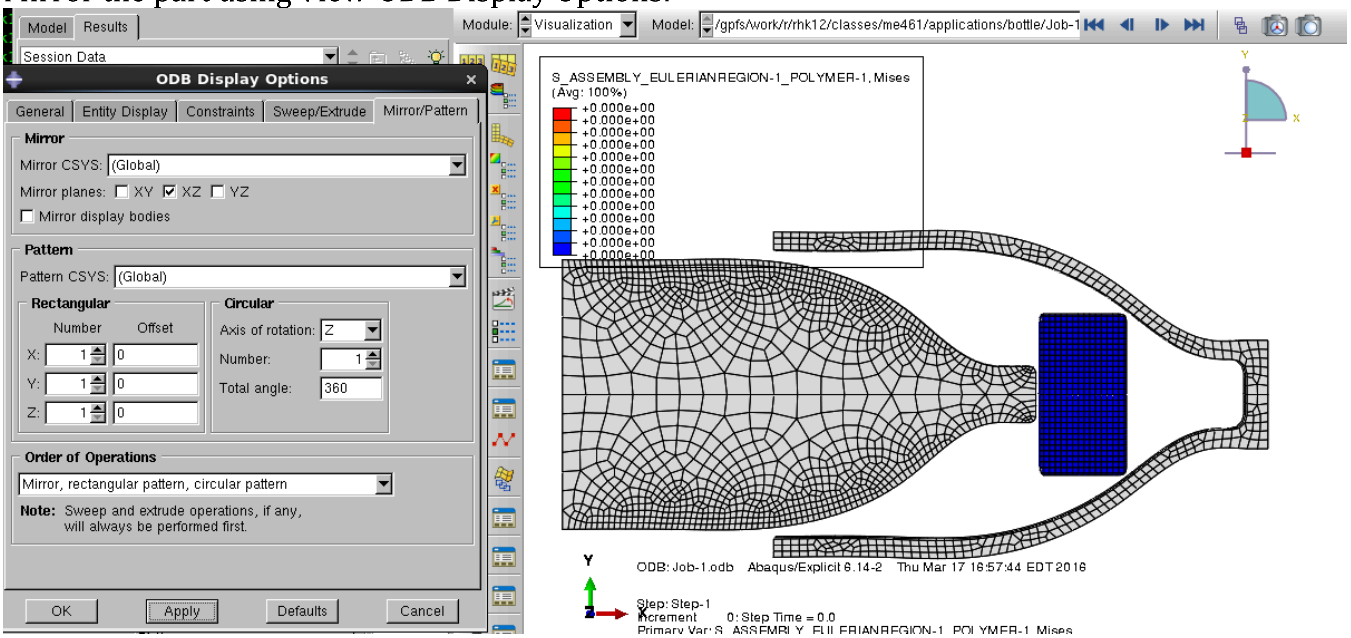


Use lion-xg to submit a simulation through qsub

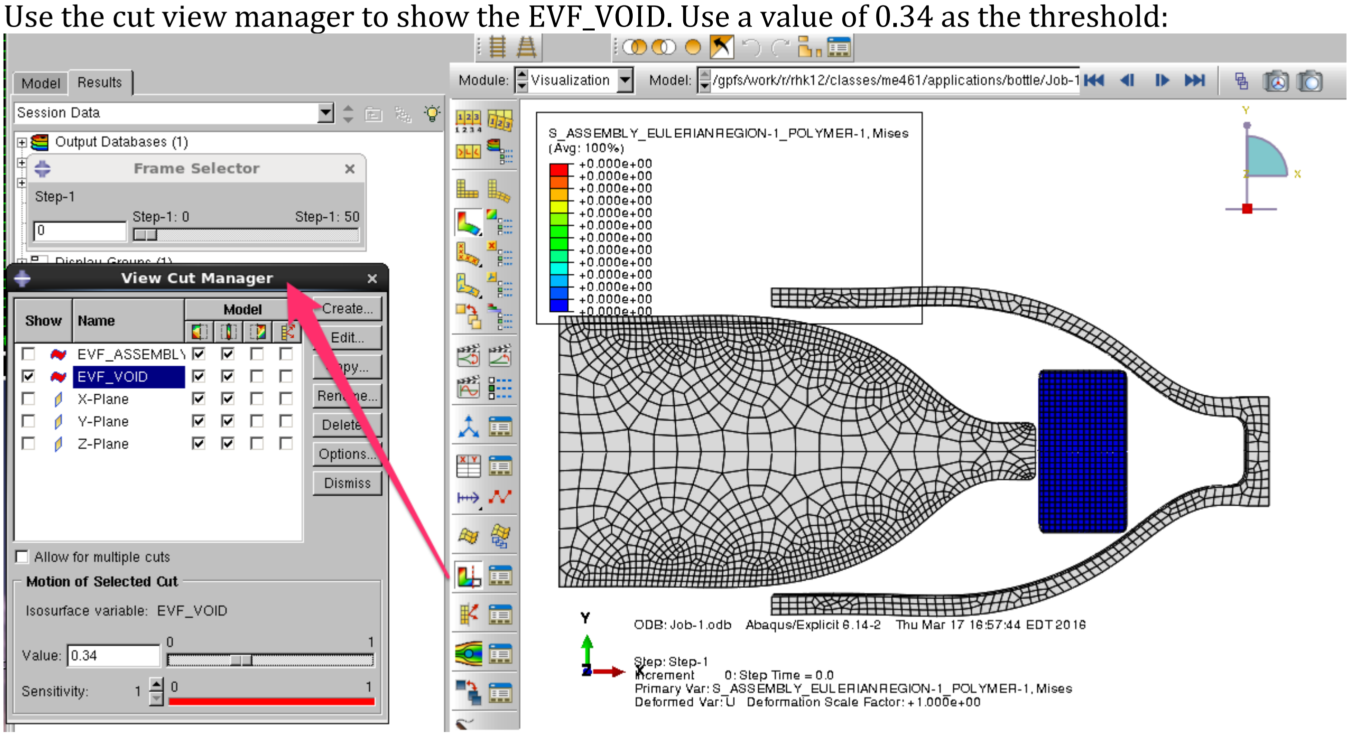
9. Open the results file and plot the results

Open the odb file in the visualization tab.

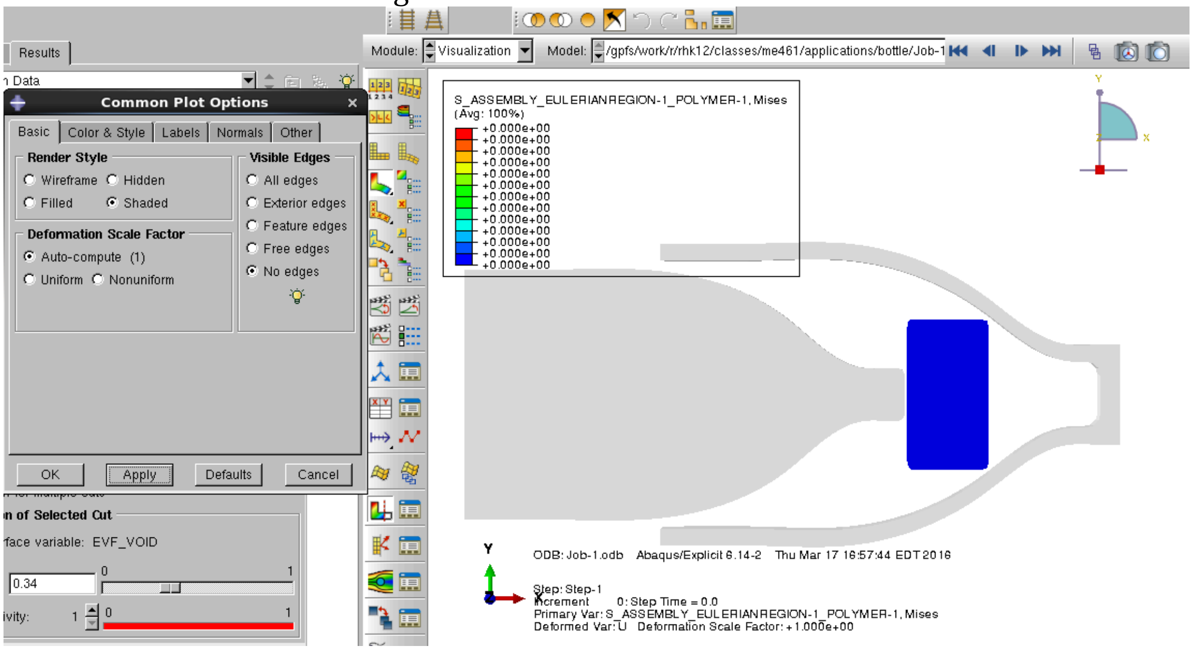
Mirror the part using View-ODB Display Options:



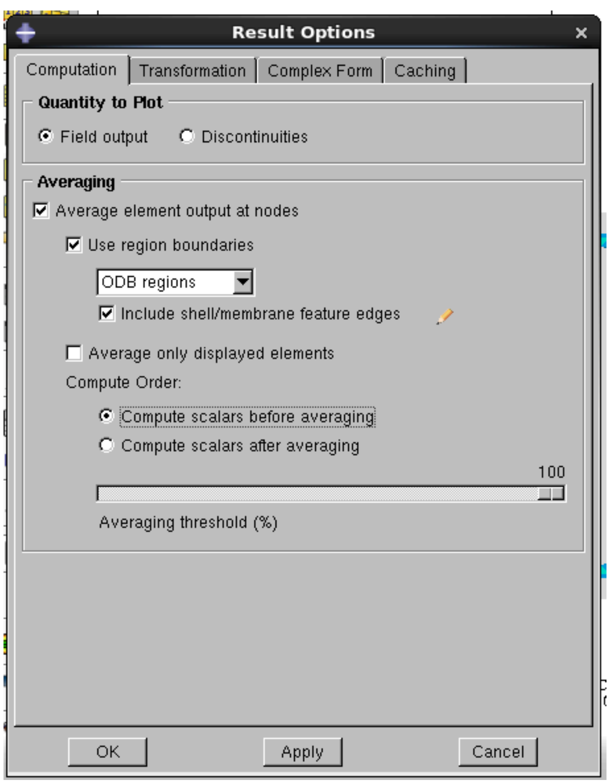
Use the cut view manager to show the EVF_VOID. Use a value of 0.34 as the threshold:

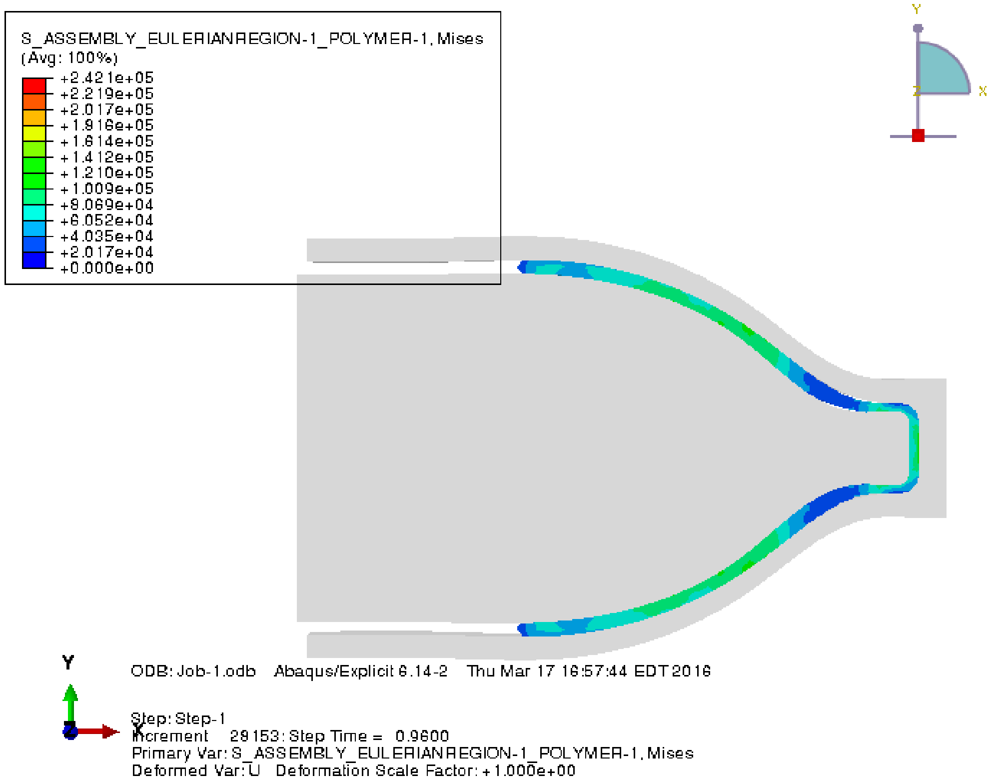


You can turn off the mesh edges:



From the Result-> options menu select the option "Compute scalars before averaging" and then put 100% the averaging threshold.




S_ASSEMBLY_EULERIANREGION-1_POLYMER-1, Mises
(Avg: 100%)
+2.421e+05
+2.219e+05
+2.017e+05
+1.816e+05
+1.614e+05
+1.412e+05
+1.210e+05
+1.009e+05
+8.069e+04
+6.052e+04
+4.035e+04
+2.017e+04
+0.000e+00
ODB: Job-1.odb Abaqus/Explicit 6.14-2 Thu Mar 17 16:57:44 EDT 2016
Step: Step-1
Increment 29153: Step Time = 0.9600
Primary Var: S_ASSEMBLY_EULERIANREGION-1_POLYMER-1, Mises
Deformed Var: U Deformation Scale Factor: +1.000e+00