PennState
College of Engineering

ME 563: Nonlinear Finite Elements

Application and Exploration of Nonlinear Finite Elements

Hyperelasticity, Part 2

Table of Contents

1. Introduction 3
2. Define Geometry 3
3. Define Materials 6
4. Create and Assign Sections 8
5. Assembly 9
6. Create Step 12
7. Define Contact Interactions 14
8. Define Loads and BCs 17
9. Mesh the Parts 19
10. Create a simulation job 20
11. Visualize Results 21
12. Other items to consider: 22

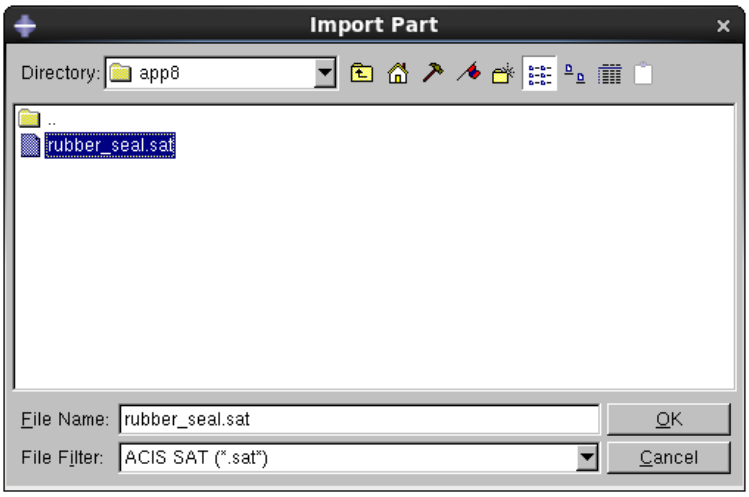
1. Introduction

In this application, we will learn about and apply the follow concepts:

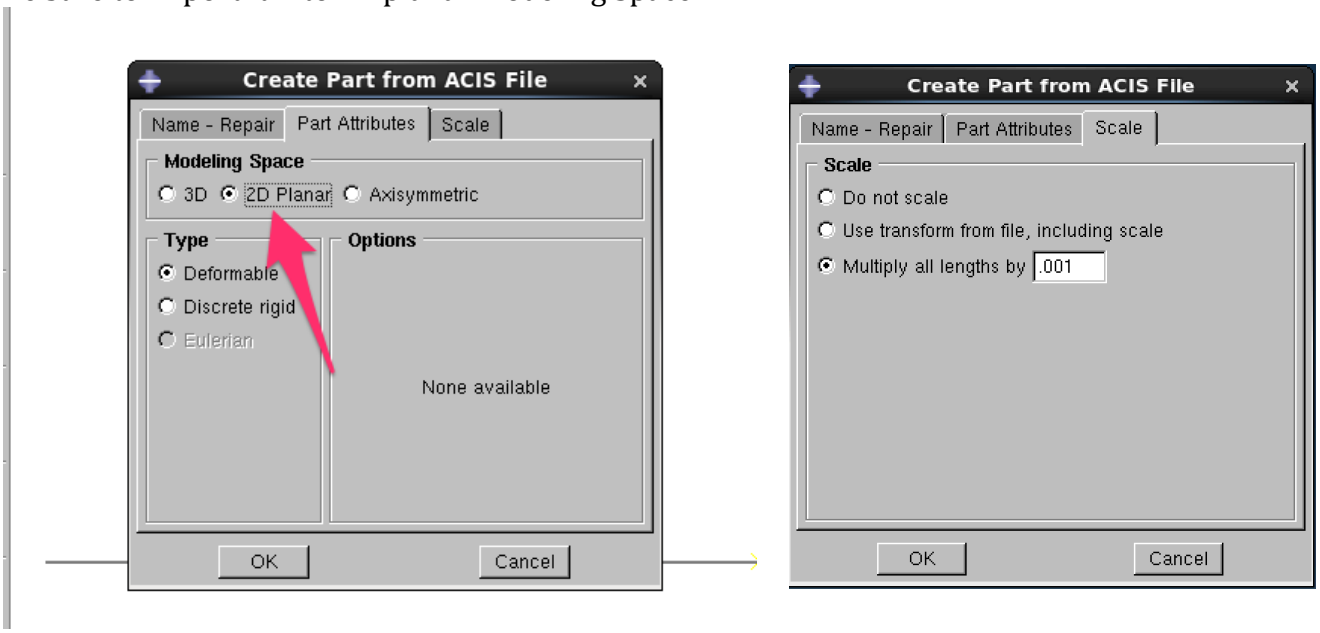
- Contact
- Rigid body
- Hyperelasticity
- PBS job scripts?

2. Define Geometry

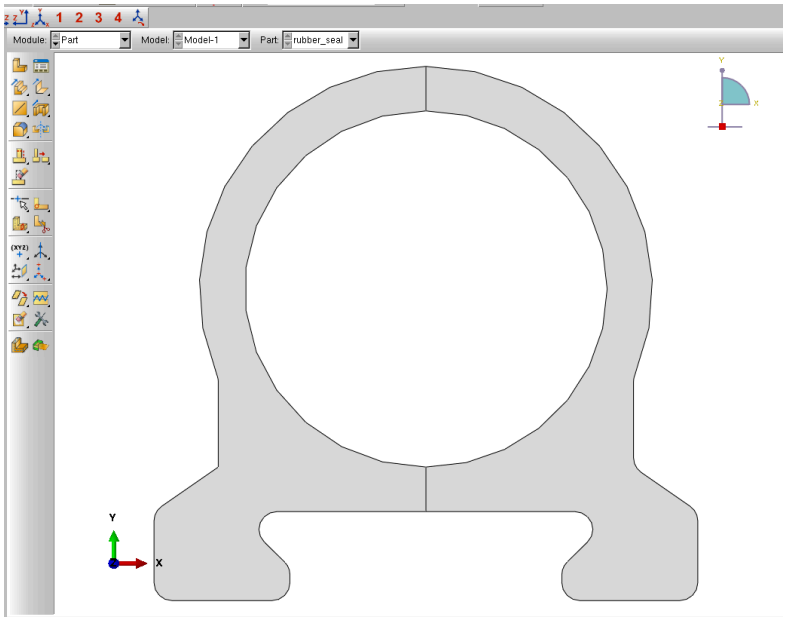
Clone github directory and import rubber_seal.sat into Abaqus:
> git clone https://github.com/rhk12/RubberSeal



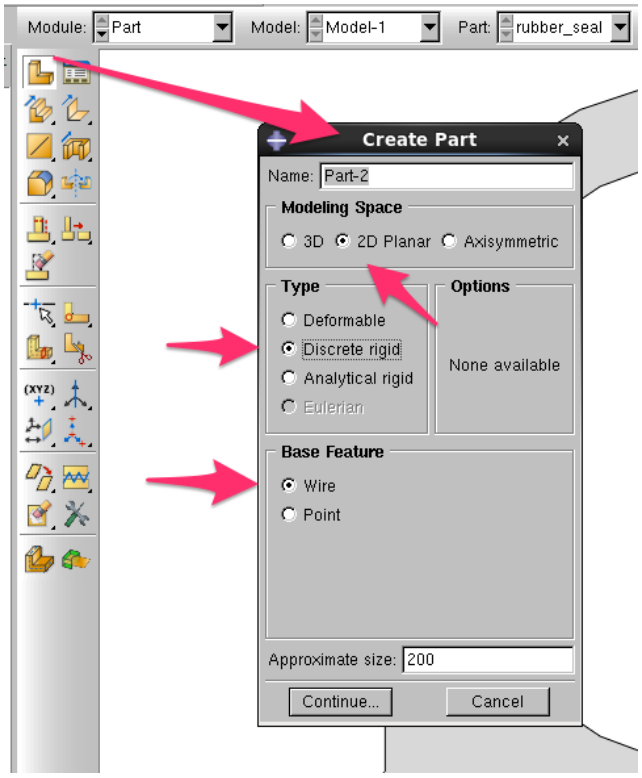
Be sure to import it into 2D planar Modeling Space:



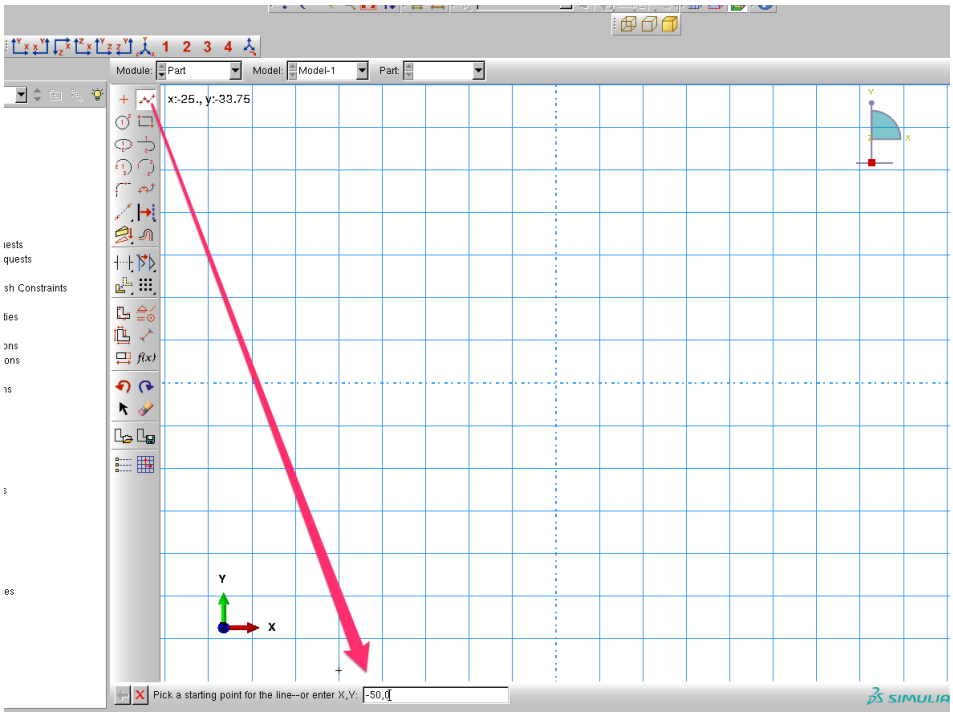
You should have this:

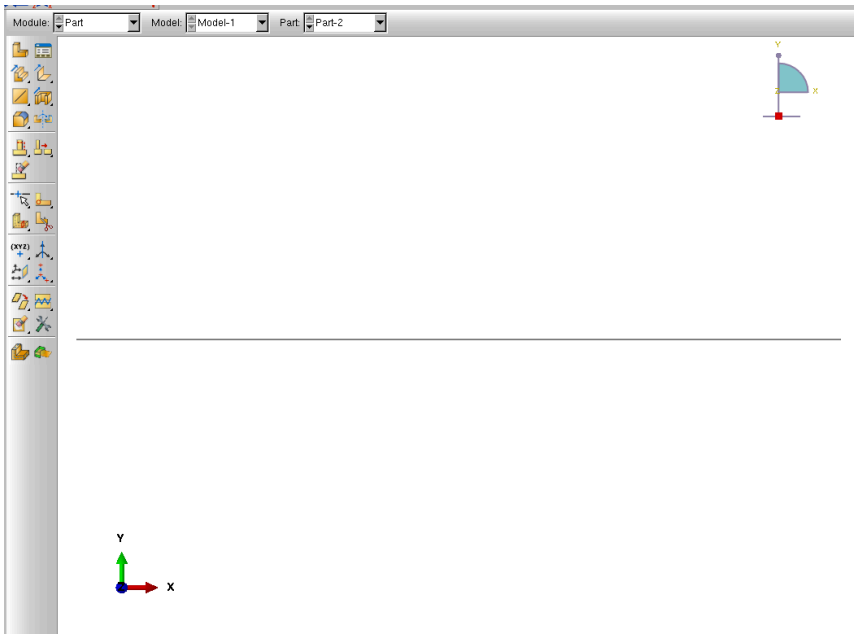


Create another part, choose 2d planar, discrete rigid:

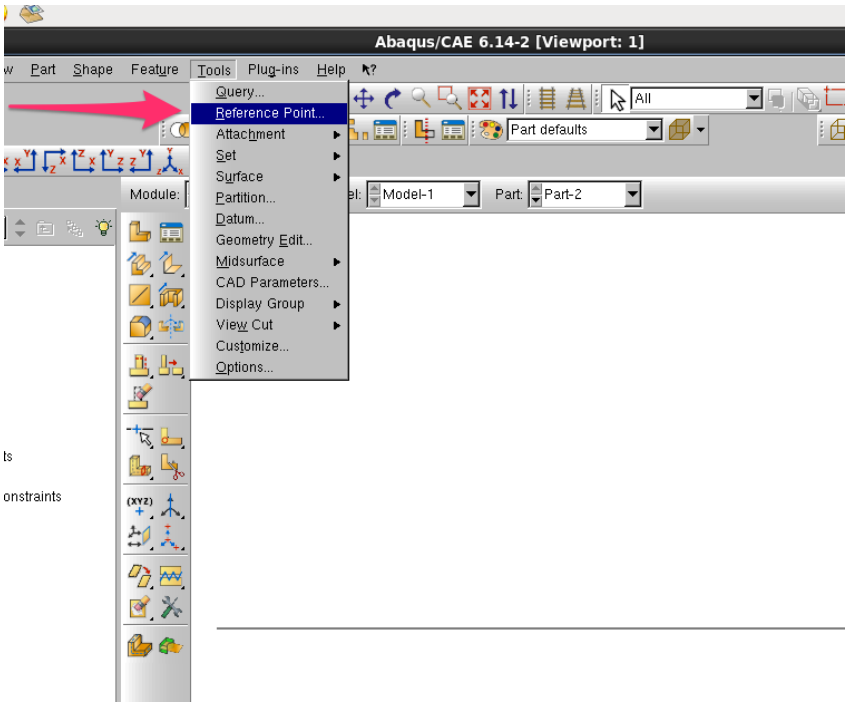


Create a wire part at the top and bottom: (-0.050,0) to (0.050, 0)

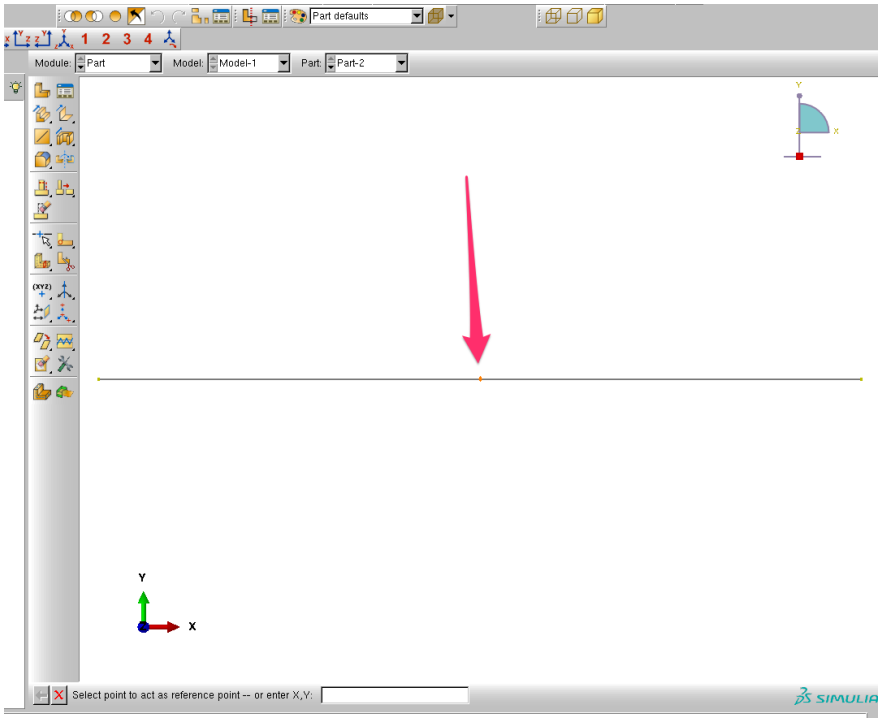




Use Tools->Reference Point to assign a ref. point to the rigid body:



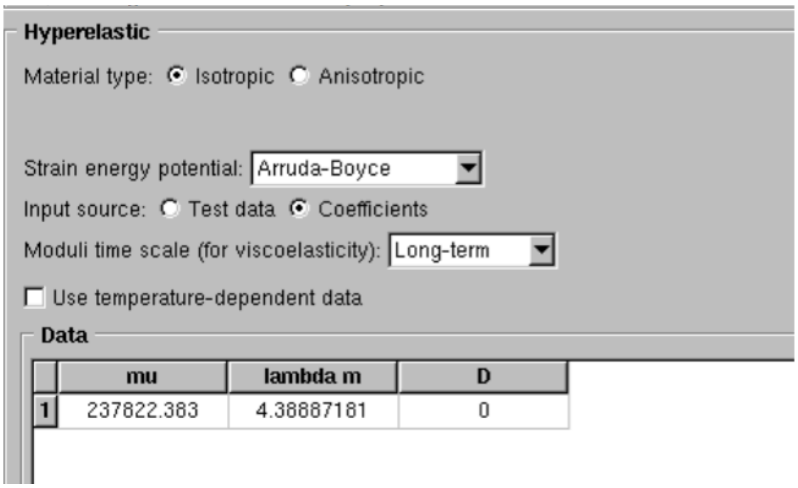
Select middle vertex:

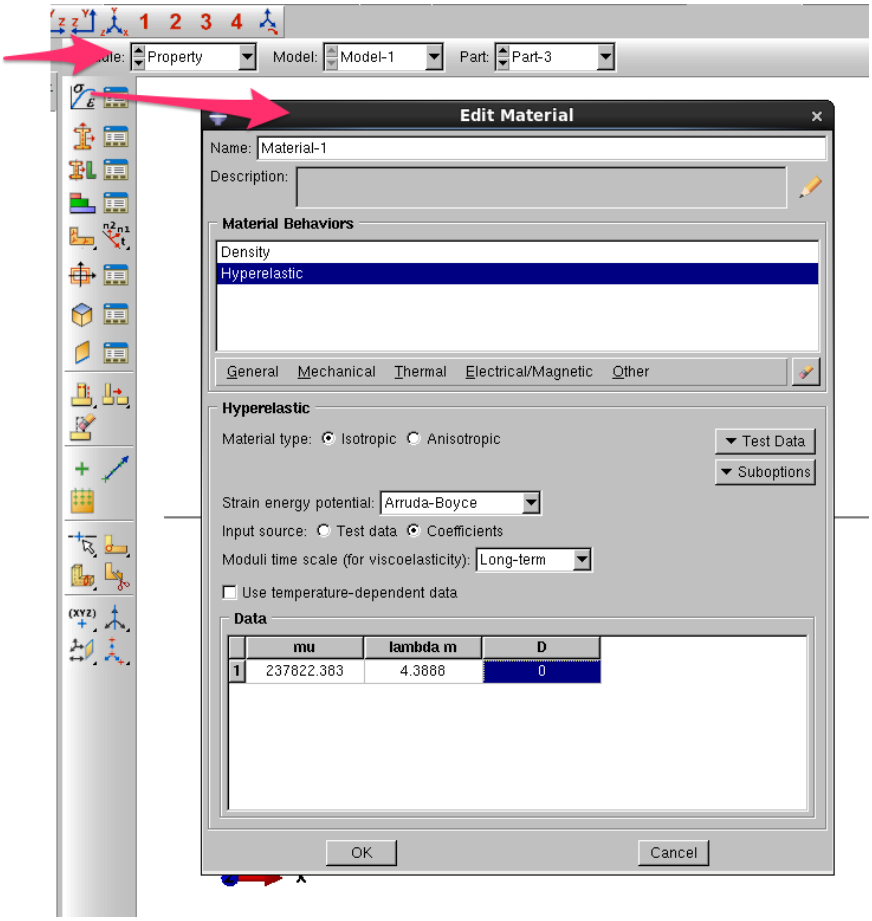


Do this another time so we have two wire parts.

3. Define Materials

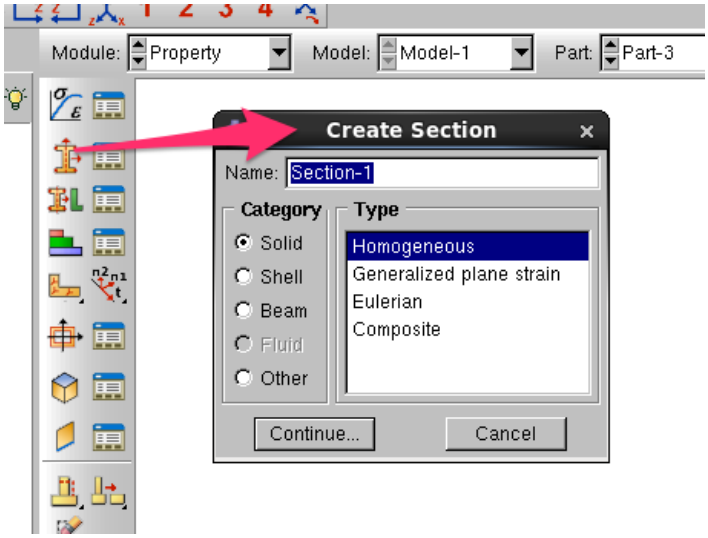
Create material using the Arruda Boyce constants we obtained last time.

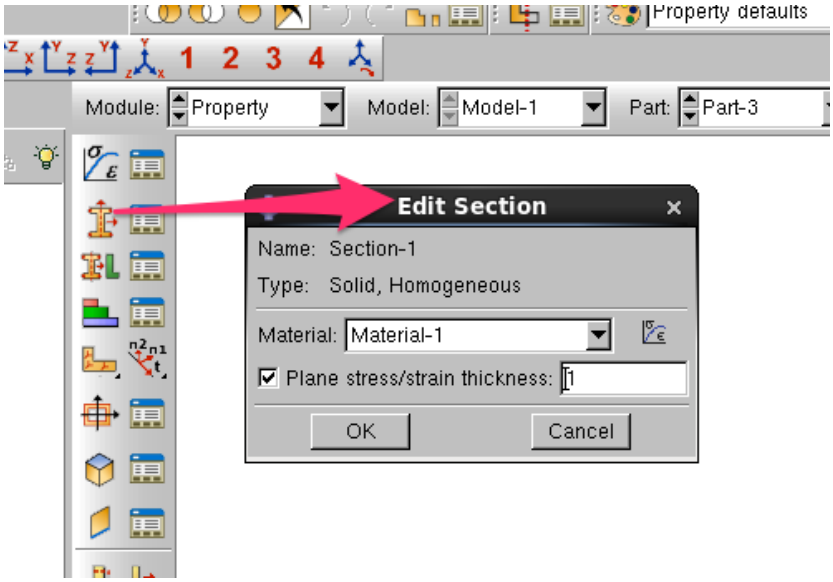

Hyperelastic
Material type: Isotropic Anisotropic
Strain energy potential: Arruda-Boyce
Input source: Test data Coefficients
Moduli time scale (for viscoelasticity): Long-term
Use temperature-dependent data
Data
mu
lambda m
D
1
237822.383
4.38887181
0


Module: Property
Model: Model-1
Part: Part-3
Edit Material
Name: Material-1
Description:
Material Behaviors
Density
Hyperelastic
General Mechanical Thermal Electrical/Magnetic Other
Hyperelastic
Material type: Isotropic Anisotropic
Test Data
Suboptions
Strain energy potential: Arruda-Boyce
Input source: Test data Coefficients
Moduli time scale (for viscoelasticity): Long-term
Use temperature-dependent data
Data
mu
lambda m
D
1
237822.383
4.3888
0
OK
Cancel

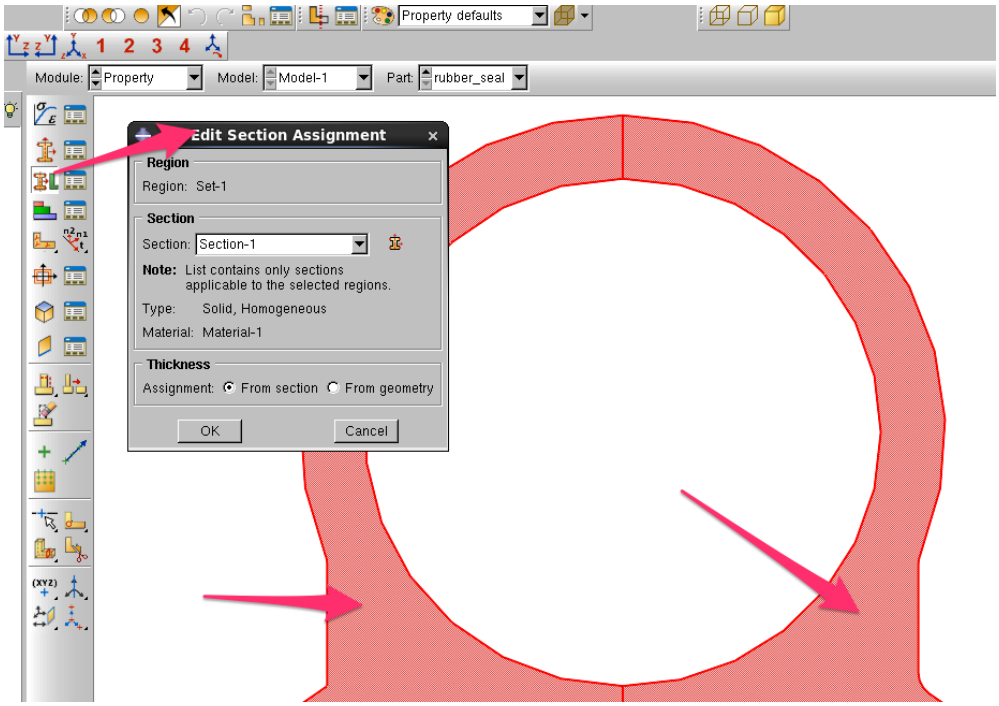
4. Create and Assign Sections

Create a solid homogeneous section



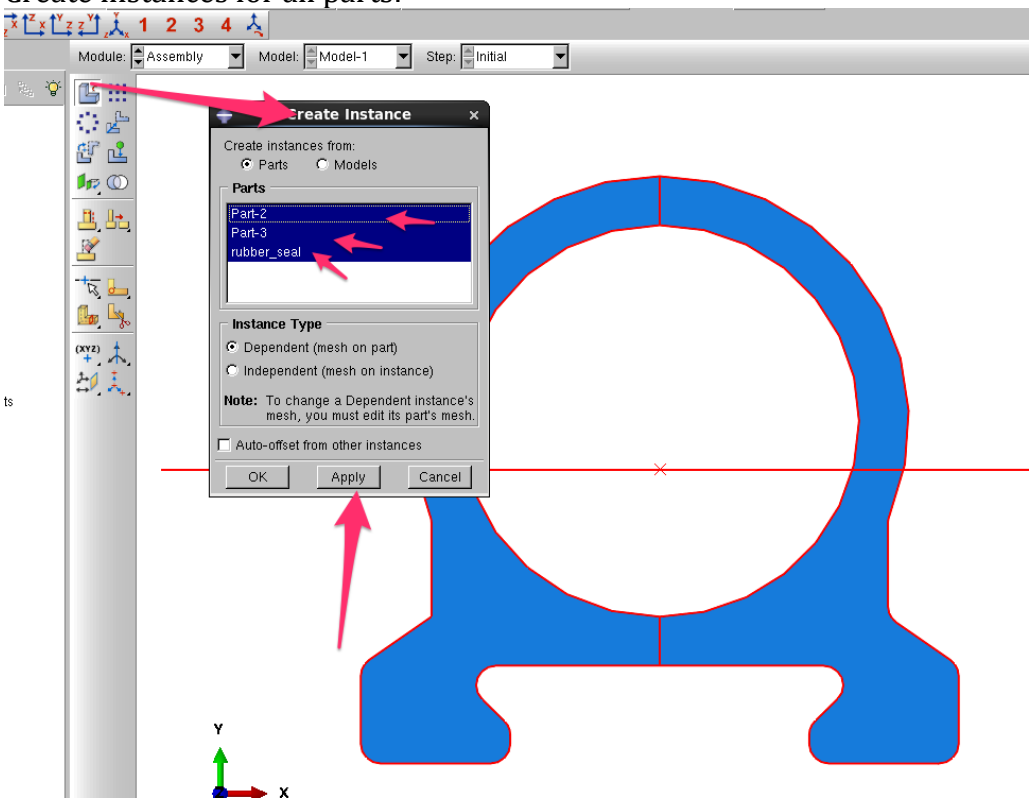


Assign section:

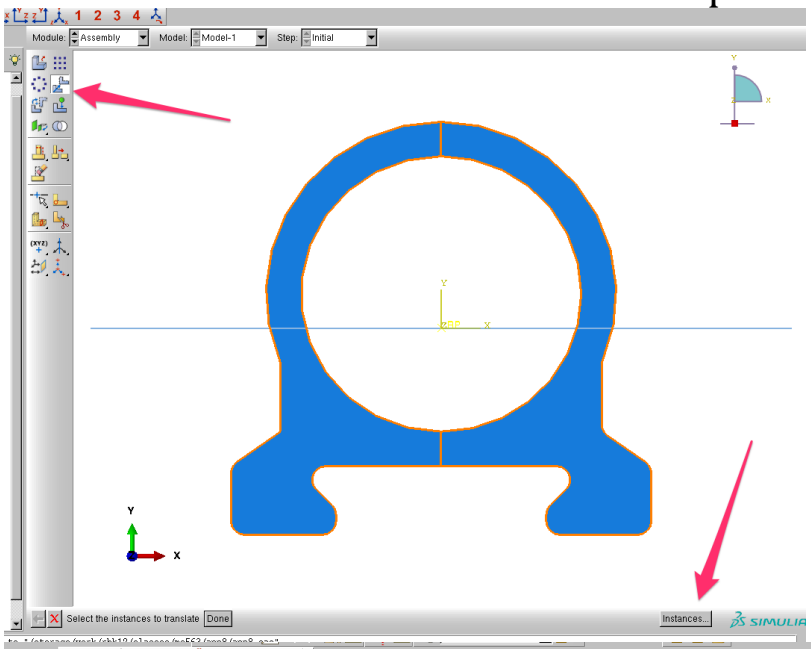


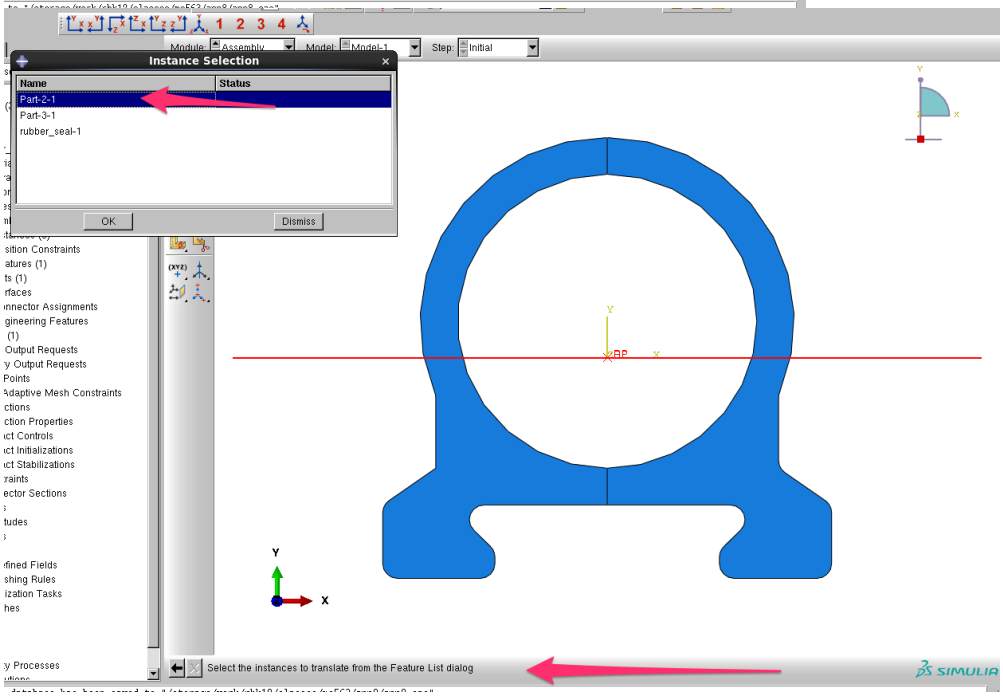
5. Assembly

Create instances for all parts:

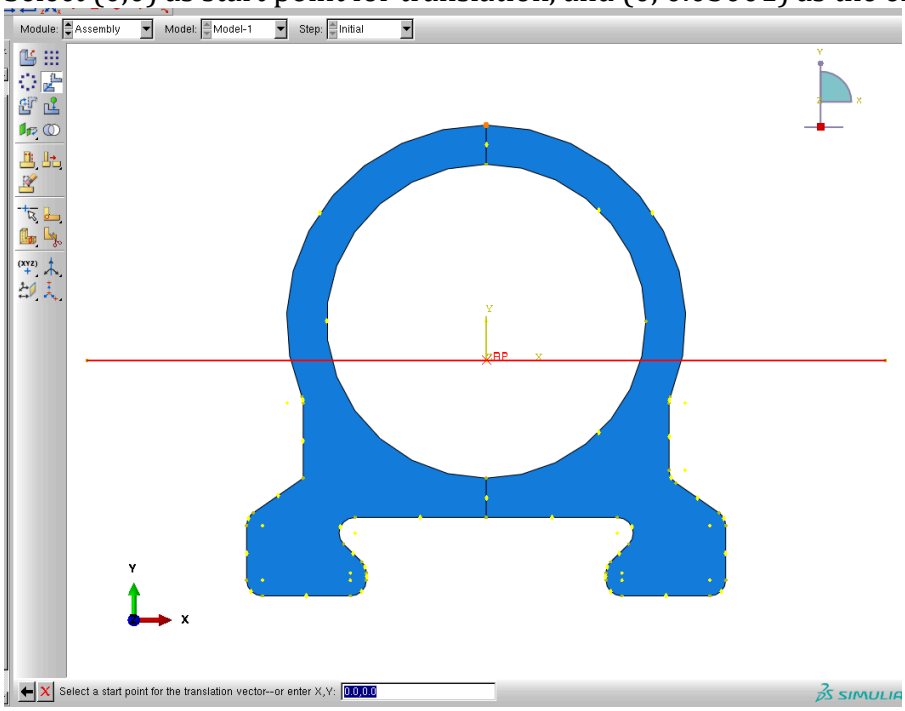


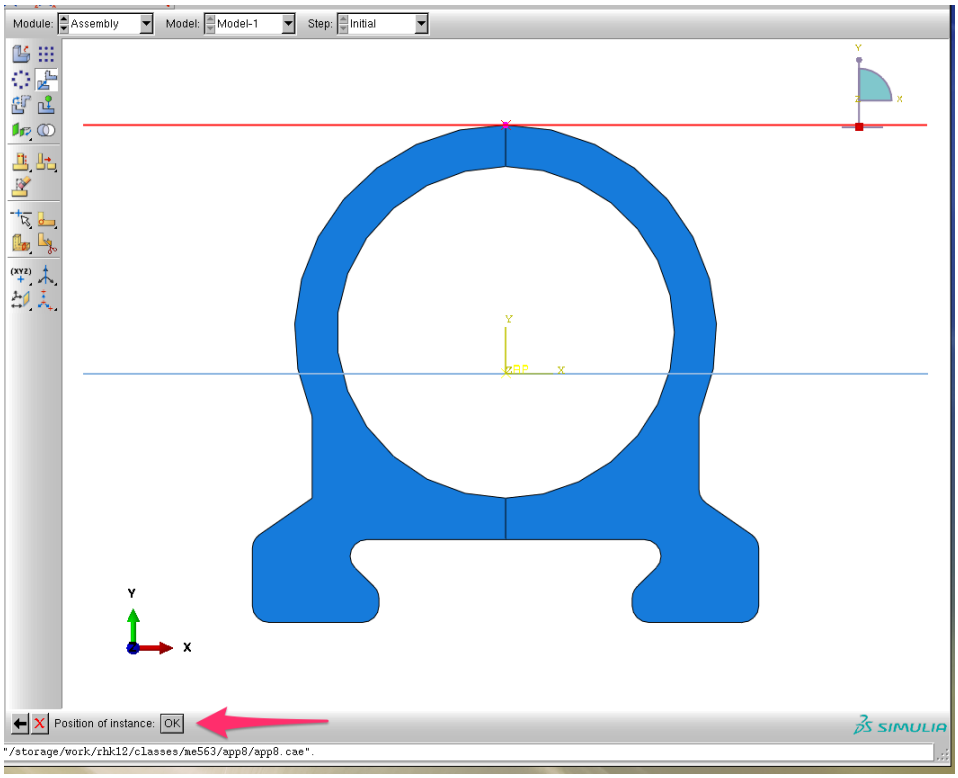
But now we need to translate the wire parts to top and bottom:



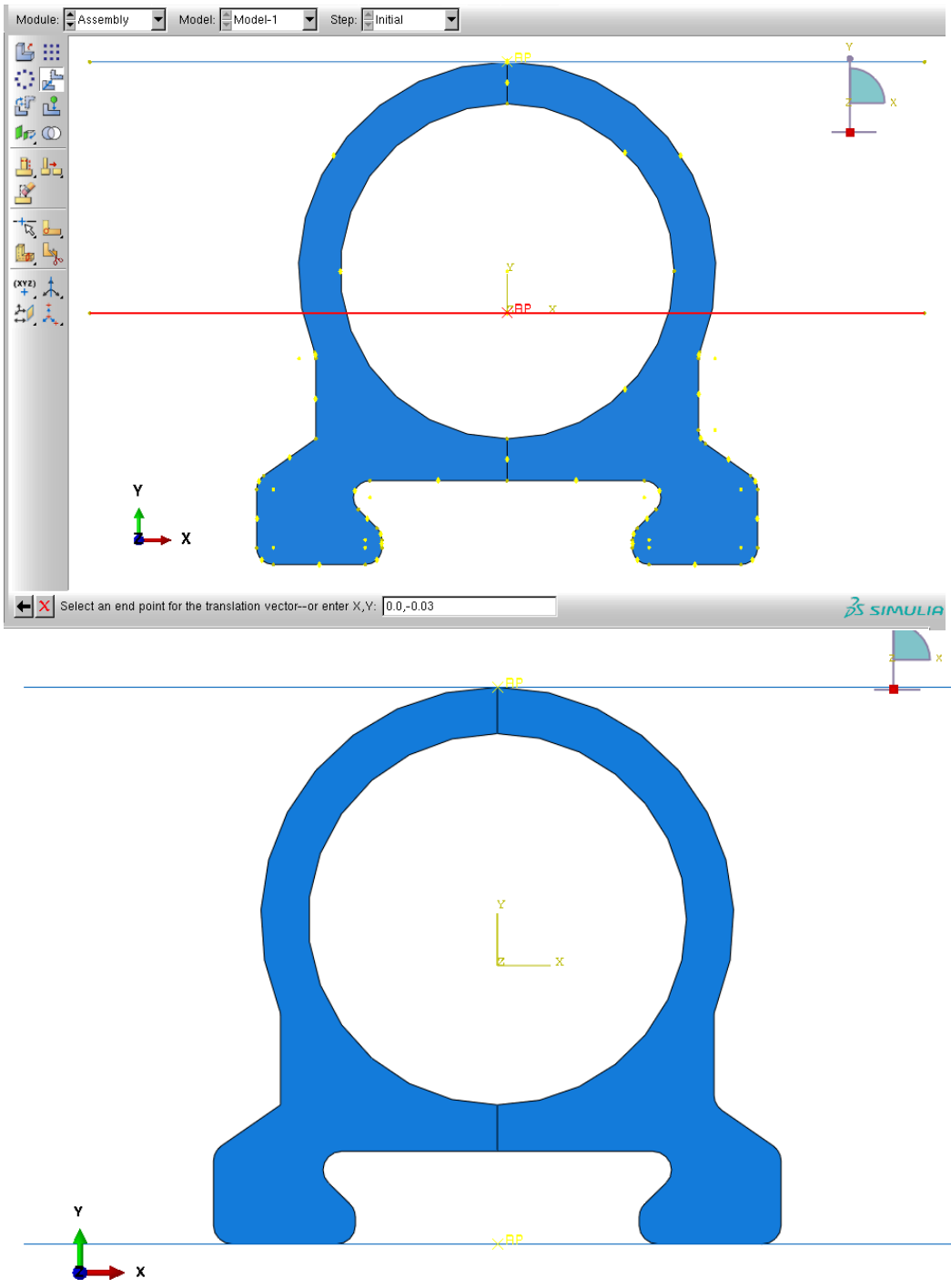


Select (0,0) as start point for translation, and (0, 0.03001) as the end point.



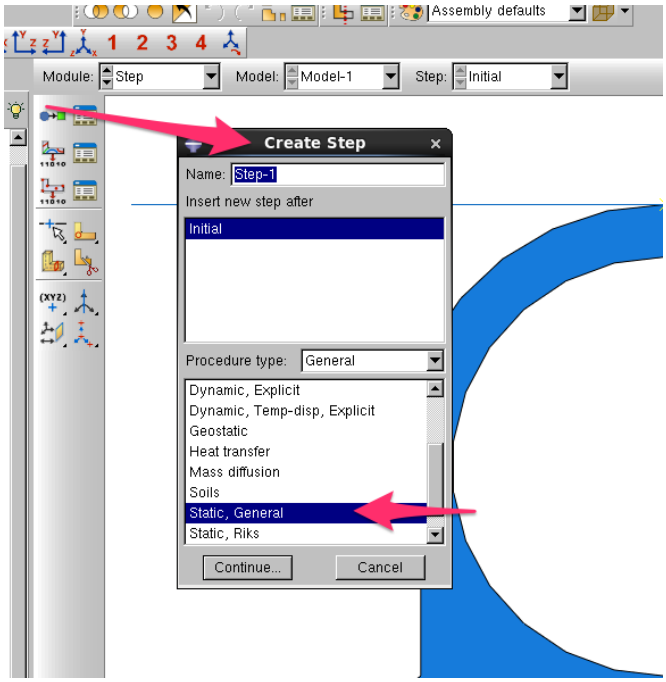


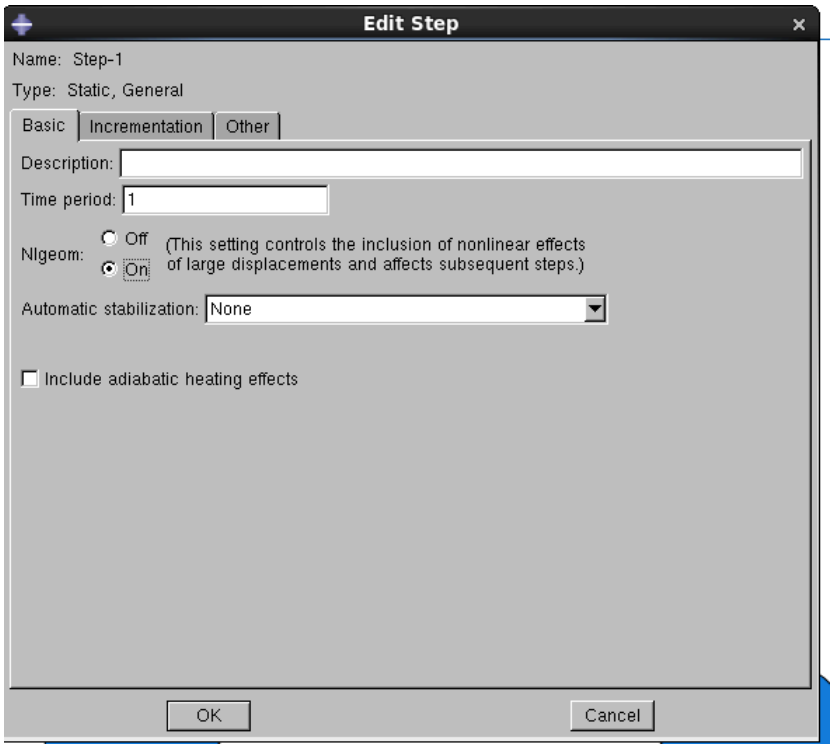
now do similar procedure to move the other wire to the bottom. You will want to translate the part -0.030 in y-direction:



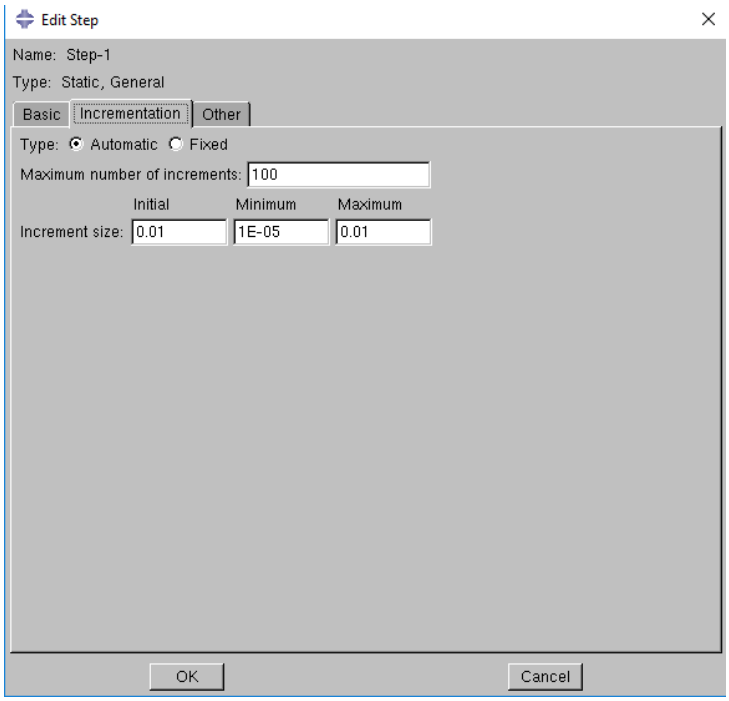
6. Create Step

Make a static step with nlgeom on.



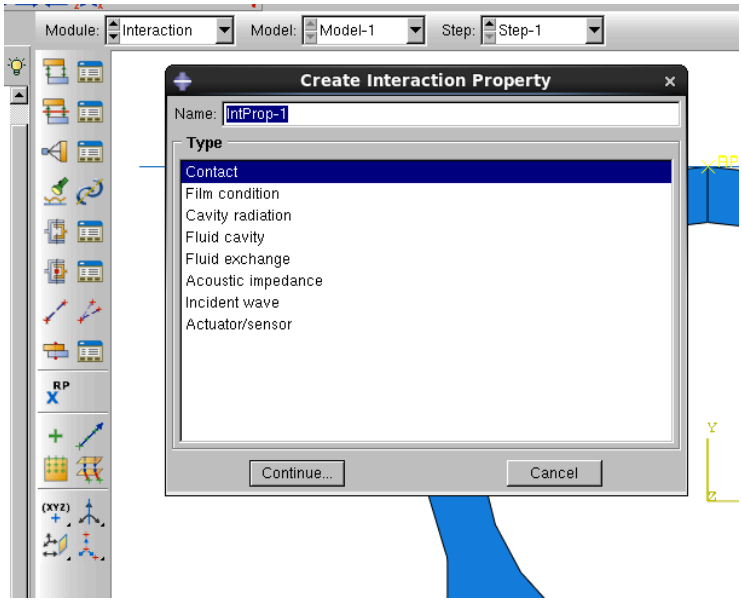


Adjust increment size to increase our number of plot steps and help with the NR convergence:

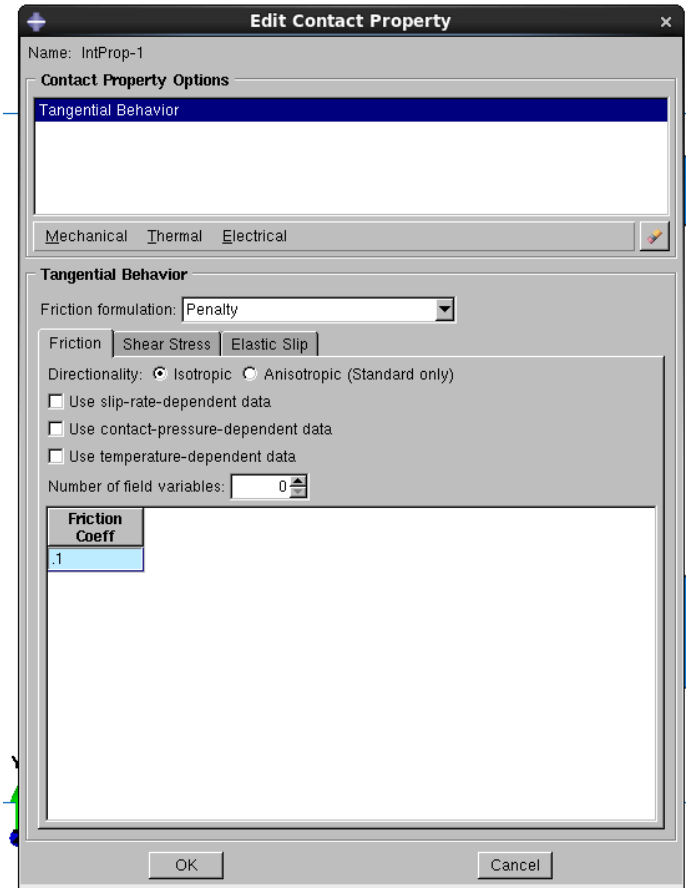


7. Define Contact Interactions

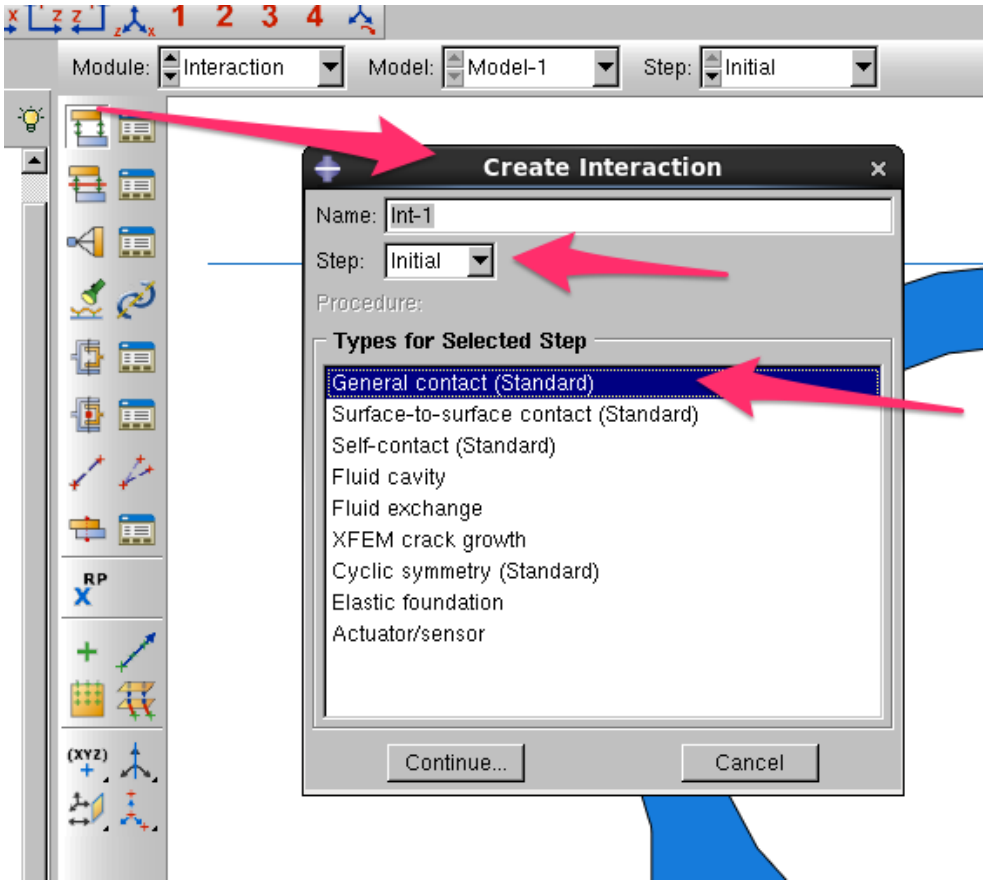
First define contact interactions:

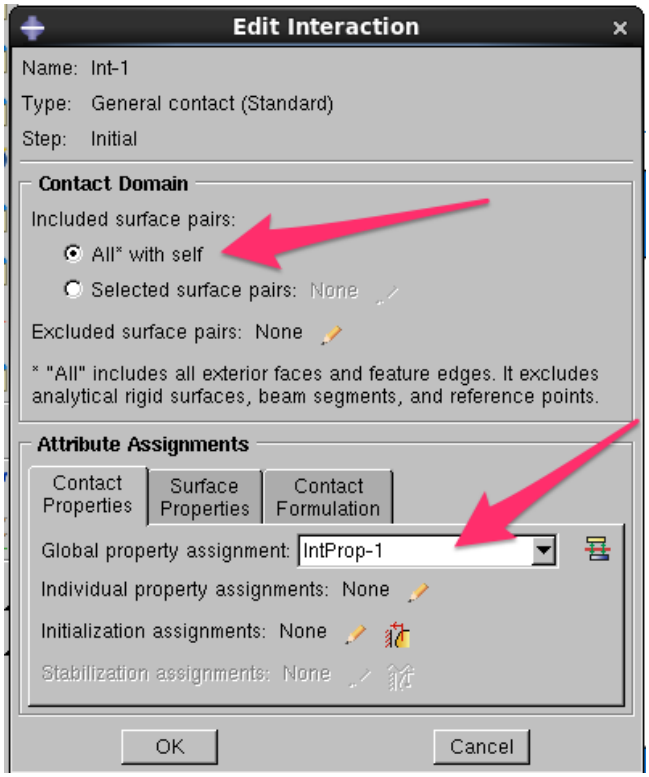


Define a tangential friction behavior using a penalty formulation:



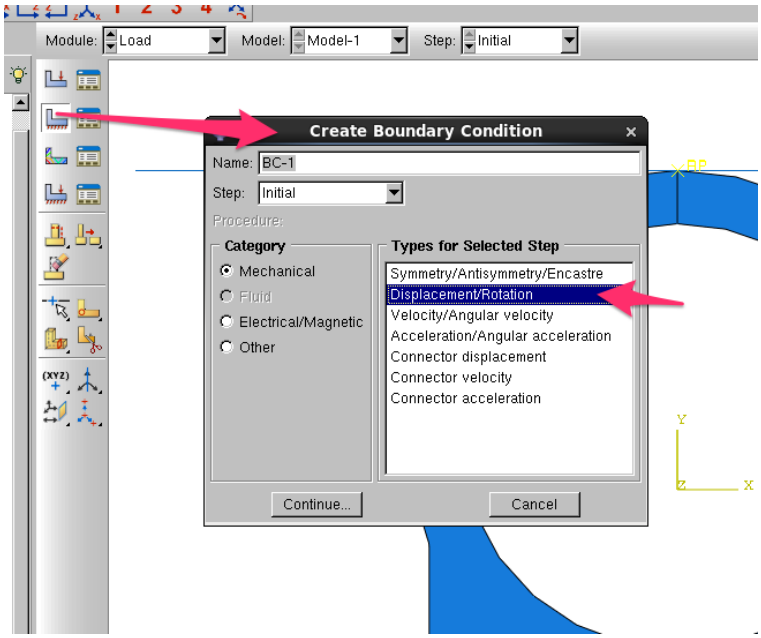
Create the interaction:

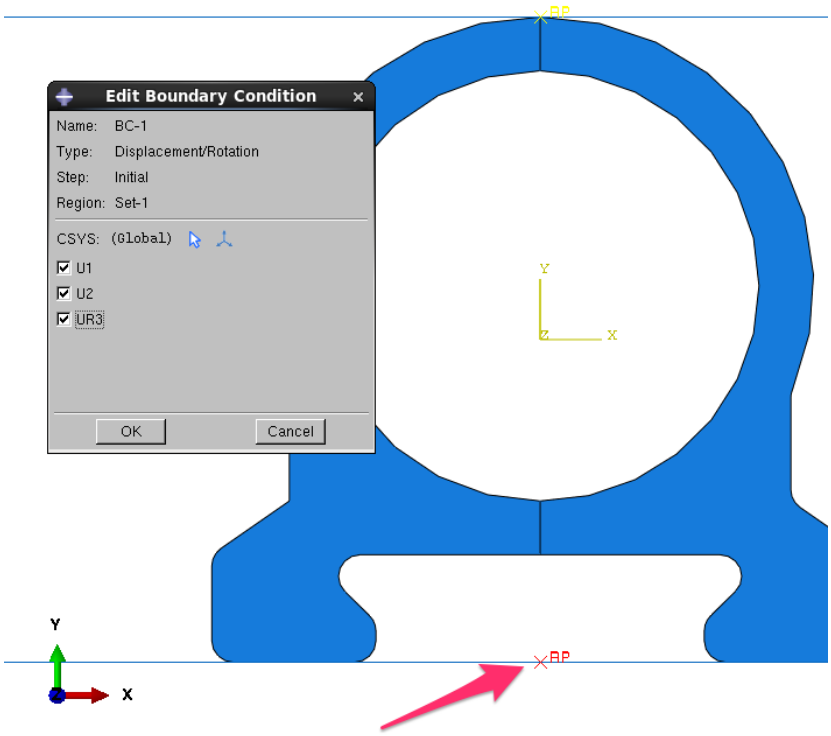

Module: Interaction
Model: Model-1
Step: Initial
Create Interaction
Name: Int-1
Step: Initial
Procedure:
Types for Selected Step
General contact (Standard)
Surface-to-surface contact (Standard)
Self-contact (Standard)
Fluid cavity
Fluid exchange
XFEM crack growth
Cyclic symmetry (Standard)
Elastic foundation
Actuator/sensor
Continue...
Cancel


Edit Interaction
Name: Int-1
Type: General contact (Standard)
Step: Initial
Contact Domain
Included surface pairs:
All* with self
Selected surface pairs: None
Excluded surface pairs: None
* "All" includes all exterior faces and feature edges. It excludes analytical rigid surfaces, beam segments, and reference points.
Attribute Assignments
Contact Properties
Surface Properties
Contact Formulation
Global property assignment: IntProp-1
Individual property assignments: None
Initialization assignments: None
Stabilization assignments: None
OK
Cancel

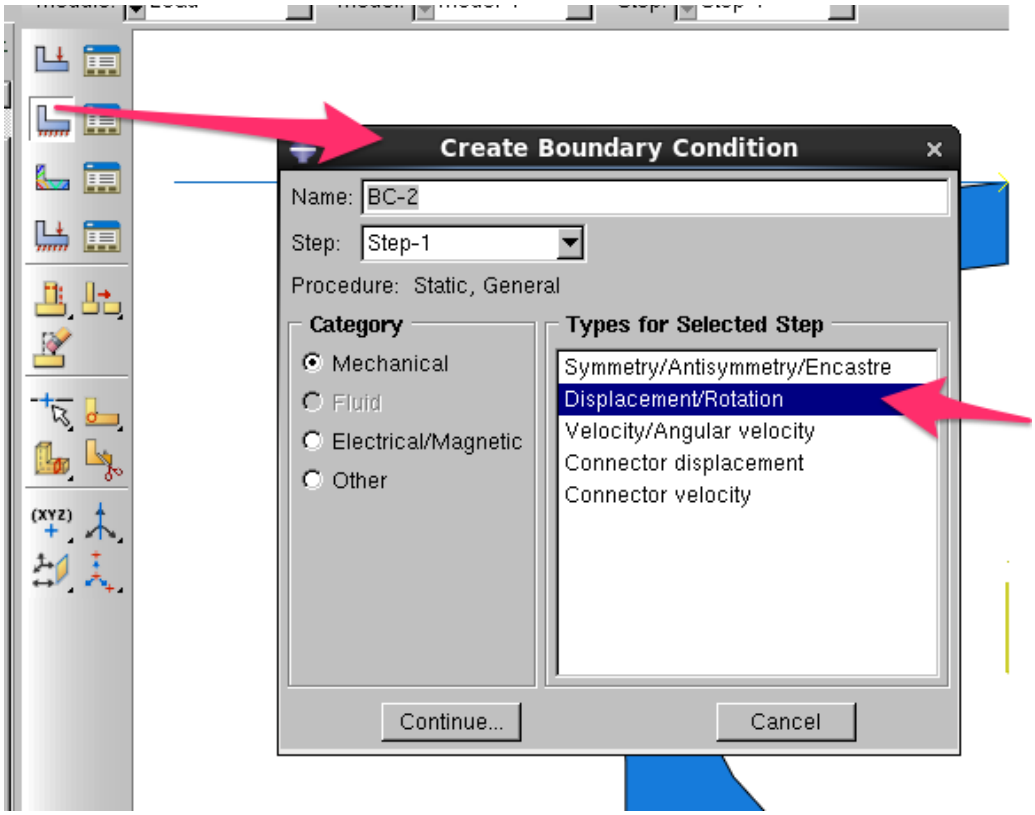
8. Define Loads and BCs

Create a boundary condition fixed on bottom wire:

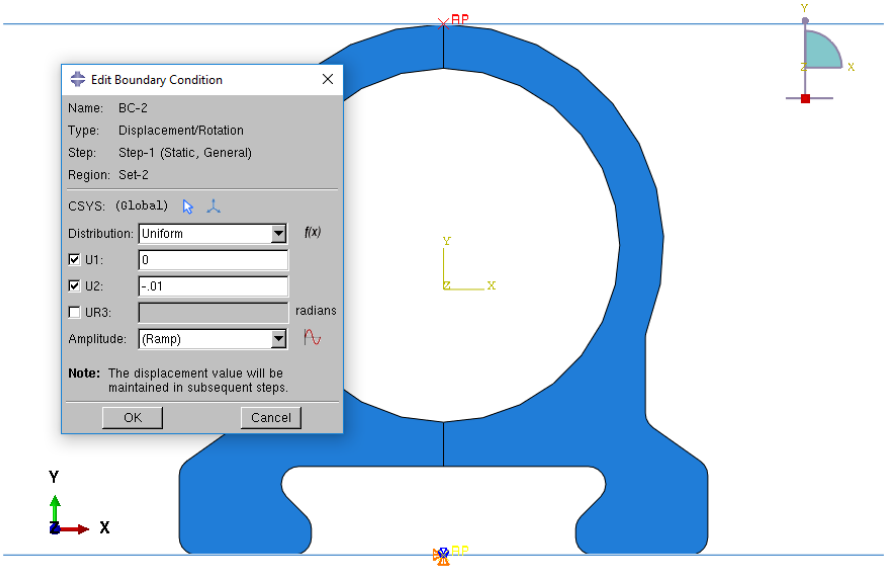




Apply negative displacement and **constrain in the x-direction** to top wire:

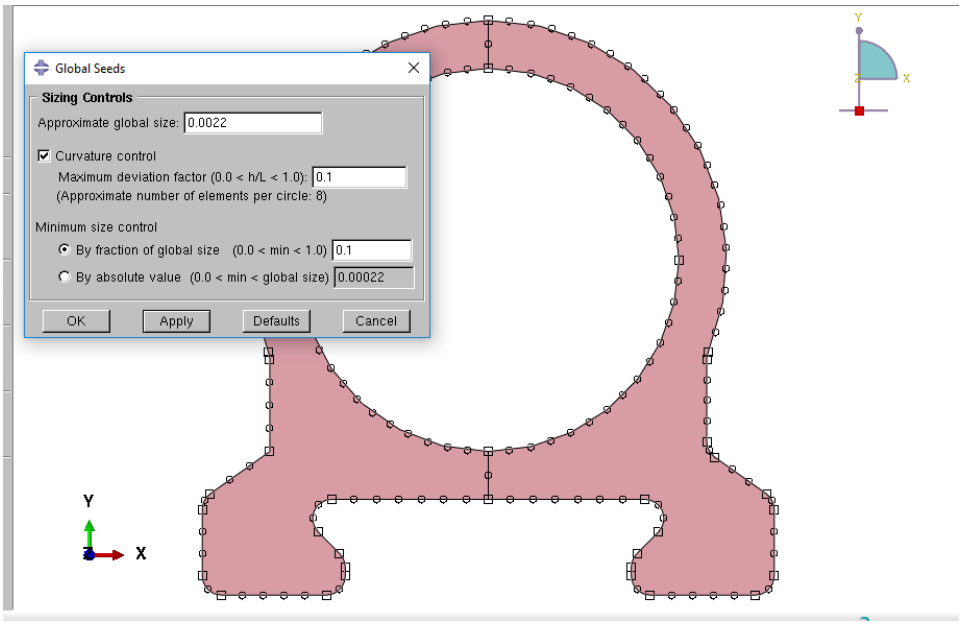


Apply -0.01 in the y-direction:

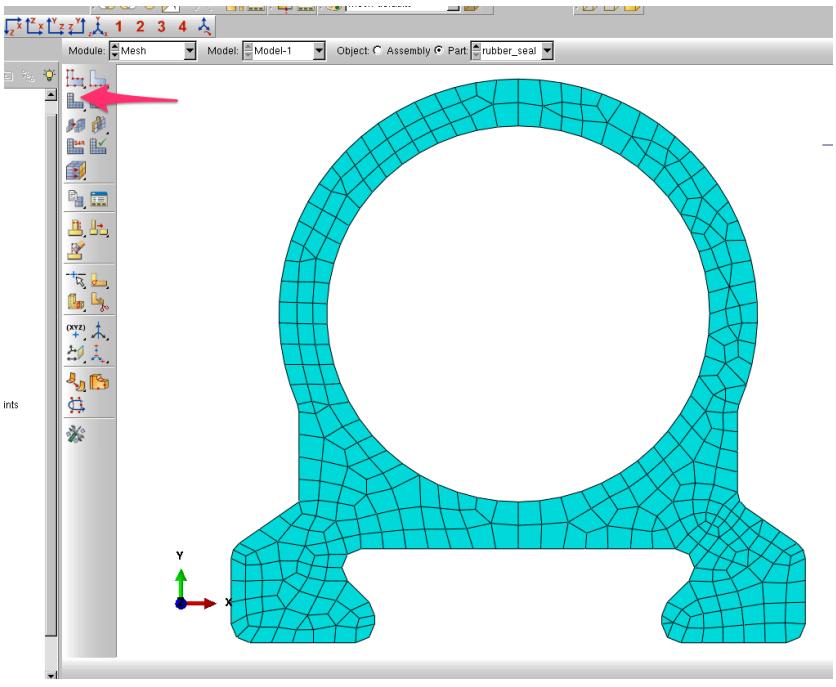


9. Mesh the Parts

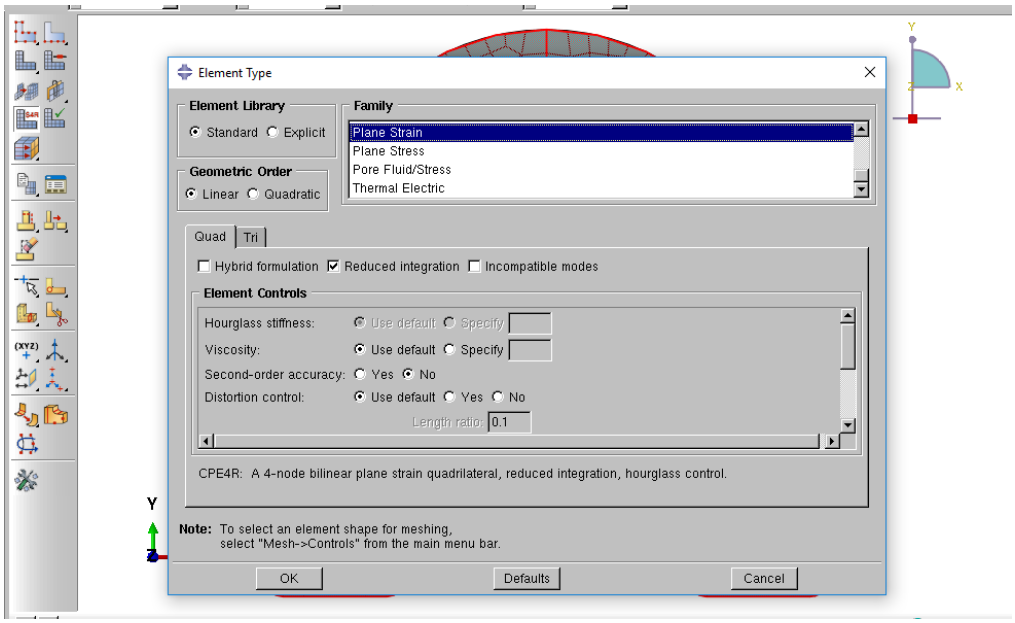
Take the defaults for the mesh:



And mesh:

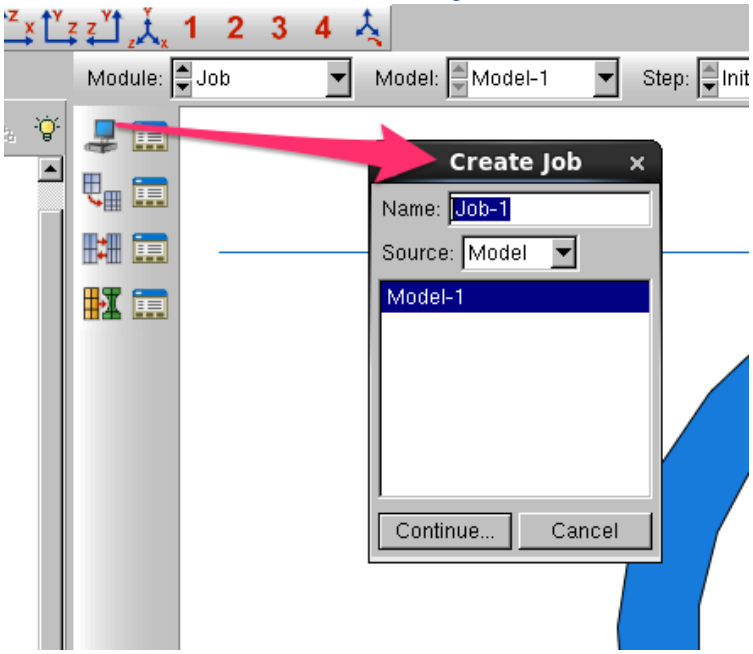


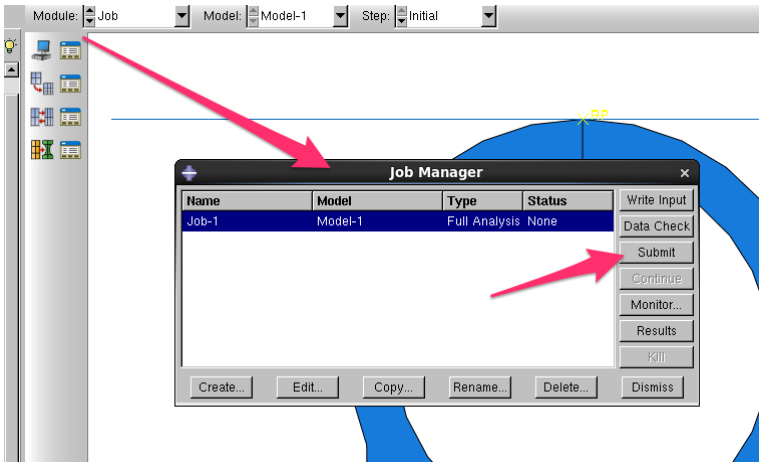
Be sure to select plane strain elements at element type:



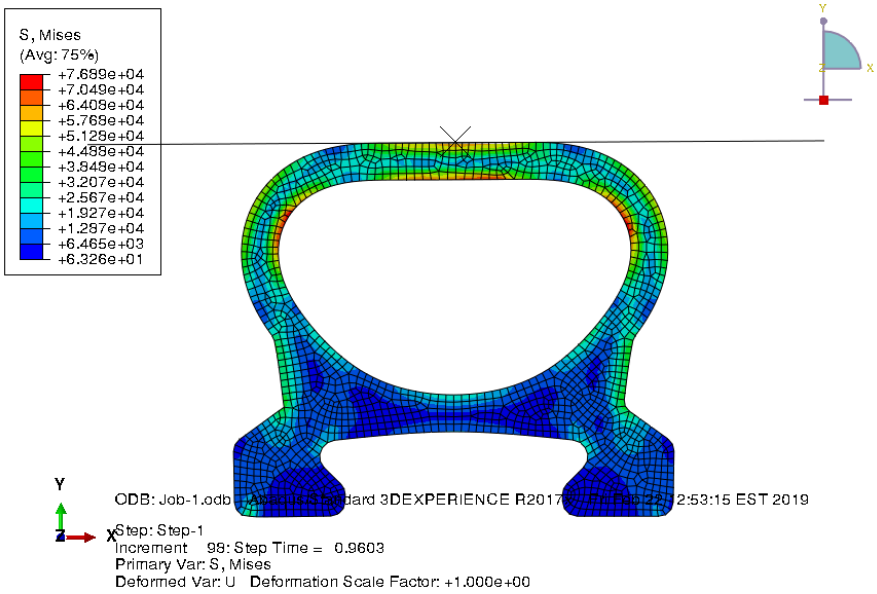
Mesh the wires with the defaults as well.

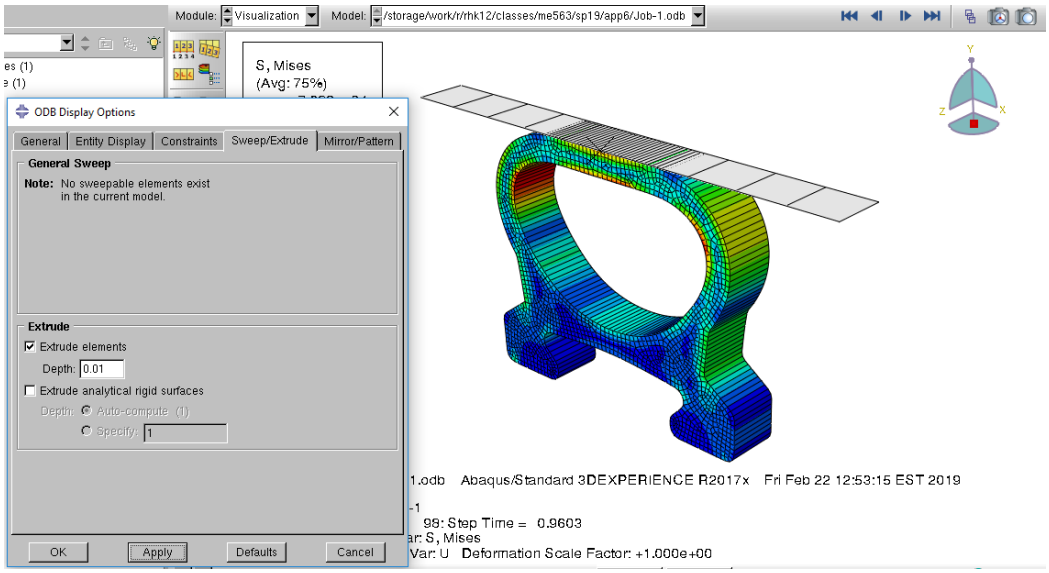
10. Create a simulation job





11. Visualize Results





12. Other items to consider:

- **Energy of system**
  - Can you plot it and show that energy is conserved?
- **Modify mesh density**
  - Do we have a converged solution?
  - Are there certain element types that exhibit locking or hourglassing in this problem?