PennState
College of Engineering

ME 563: Finite Elements in Engineering

Application of the Finite Element Method to Real World Problems

Hyperelasticity and non-linear self-contact

The Pennsylvania State University

1. Table of Contents

ME 563: FINITE ELEMENTS IN ENGINEERING 1

..... 1

1. TABLE OF CONTENTS 2
2. INTRODUCTION 3
3. PROCEDURE 3
4. ASSIGN MATERIALS 6
5. CREATE SECTION 7
6. ASSEMBLY 8
7. CREATE A STATIC NON-LINEAR STEP 12
8. CREATE CONTACT INTERFACES 13
9. CREATE TIE CONSTRAINTS 16
10. ADD BOUNDARY CONDITIONS 18
11. CREATE MESH 20
12. CREATE JOB 22
13. ANALYZE RESULTS 23

2. Introduction

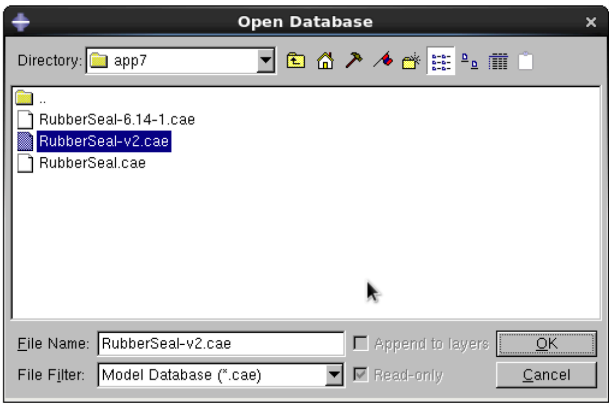
This exercise will show you the capability of Abaqus to solve for contact where an instance gets in contact with itself.

This highly non-linear event is solved by defining a **self-contact interaction**. General contact automatically already includes this kind of interactions in the 'All with self' contact domain default option so we are going to use the **contact pair approach** in this application.

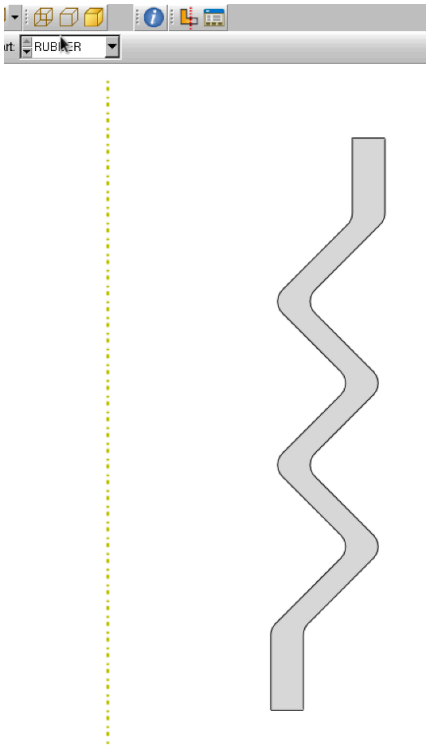
In this example, a **rubber seal** is modeled with a hyperelastic material will undergo an **axial compression** loading condition enforced through **tie constraints** with rigid analytical surface. This kinematic constrain is very useful to simulate contact conditions where no movement is allowed between two contacting instances.

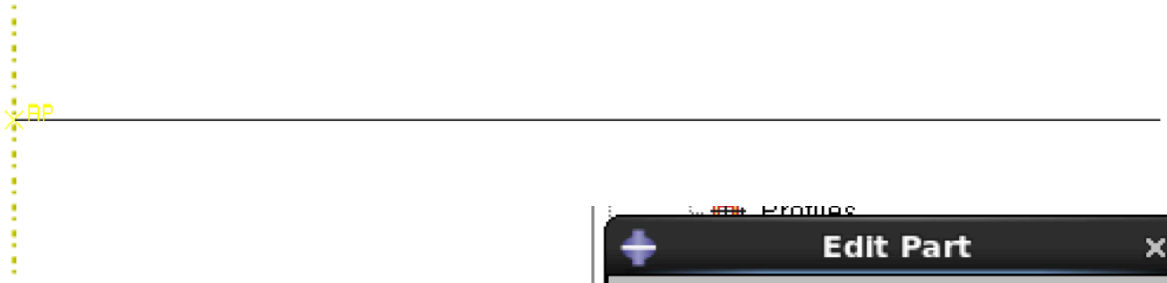
3. Procedure

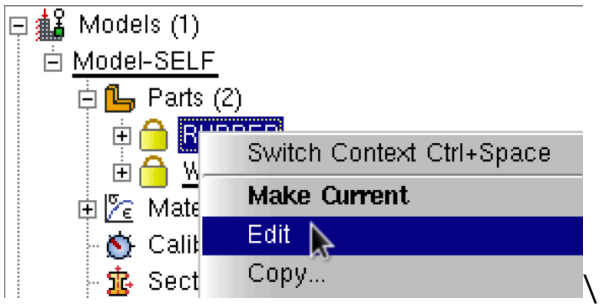
Obtain *RuberSeal-v2.cae* from github (git clone https://github.com/rhk12/RubberSeal2) this will provide an Abaqus CAE file where you will already find a model called *Model-SEAL.*

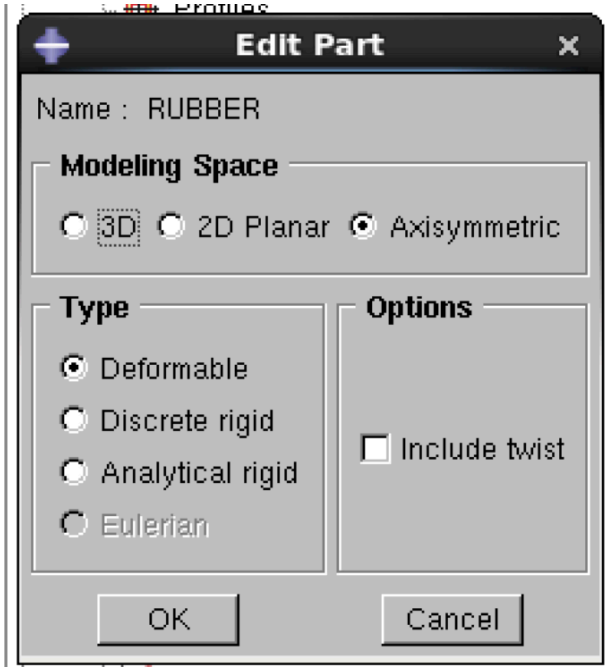


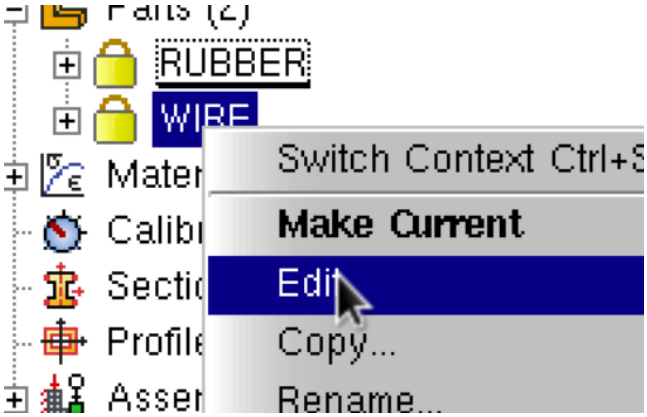
The model is a 2D axisymmetric assembly composed of the rubber seal and two rigid analytical wires.

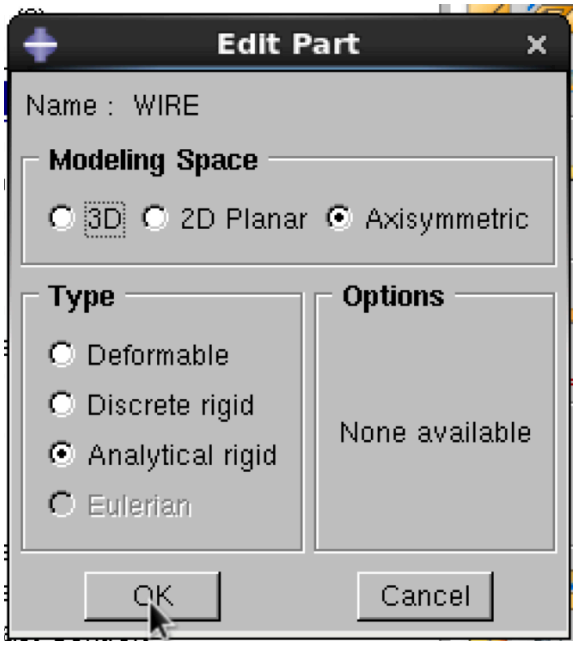

RUBBER


RP

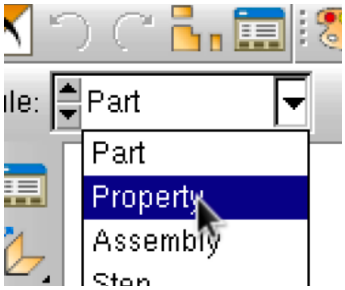

Models (1)
Model-SELF
Parts (2)
Switch Context Ctrl+Space
Make Current
Edit
Copy...

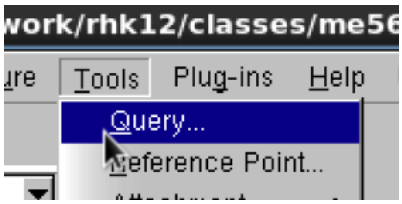

Edit Part
Name : RUBBER
Modeling Space
3D
2D Planar
Axisymmetric
Type
Deformable
Discrete rigid
Analytical rigid
Eulerian
Options
Include twist
OK
Cancel

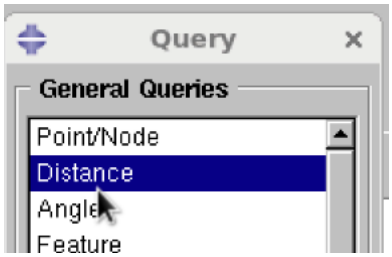




Let's check the dimensions of the problem





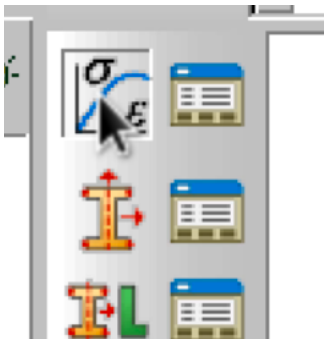


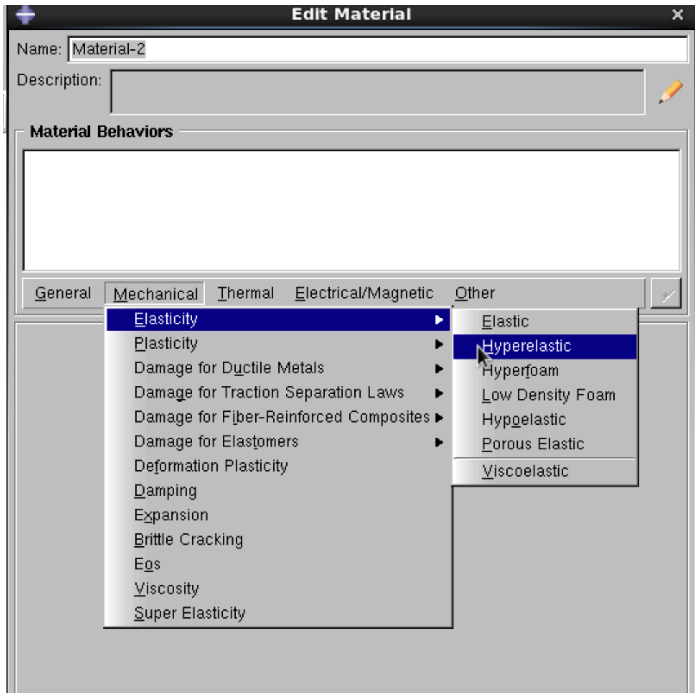
Units are in centimeters.

```
Point 1: 3., 7., 0.  Point 2: 2.2, 0., 0.
   Distance: 7.045566  Components: -800.E-03, -7., 0.
```

4. Assign Materials

Create a hyperelastic material called ***Rubber with a Mooney-Rivlin*** strain energy density function with parameters as shown below.



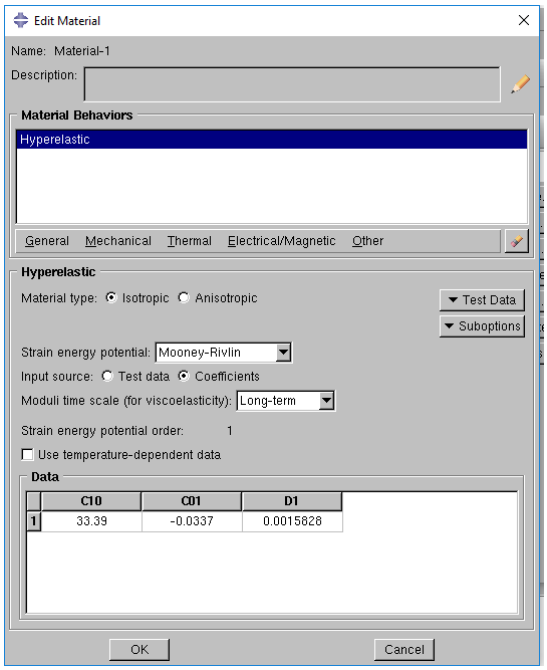


Based on Shahzad et al paper. (Table 3).

C_{10} = 0.3339 MPa or 333900 Pa or 33.39 N/cm^2

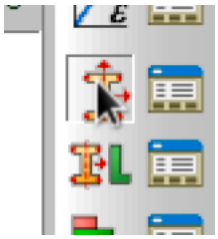
C_{01} = -3.37e^{-4} MPa or -337 Pa or -0.0337 N/cm^2

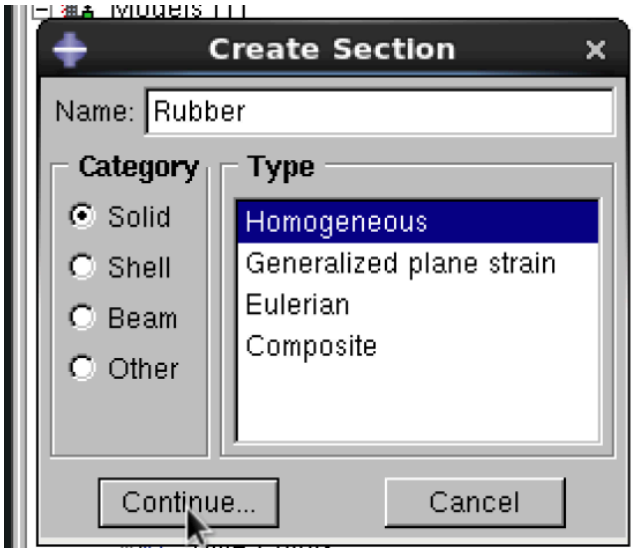
D_1 = 1.5828e^{-3}

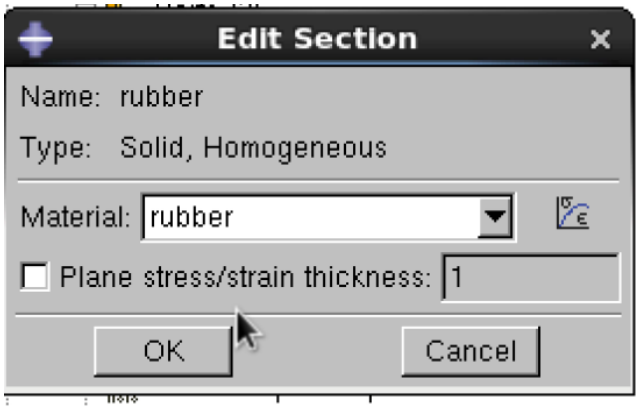


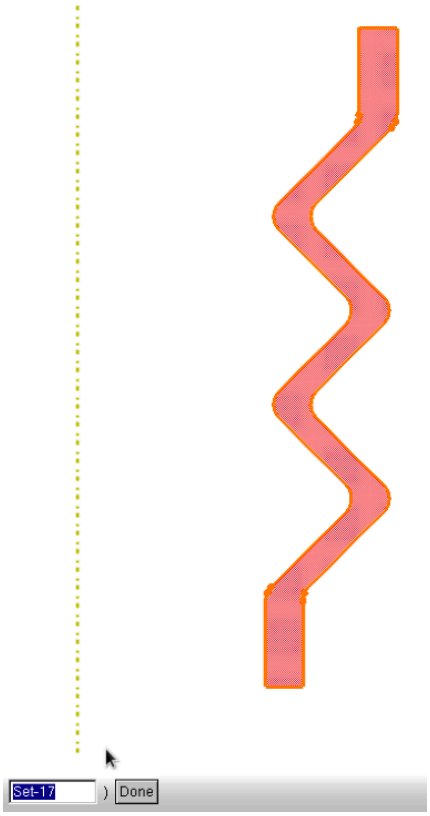
5. Create Section

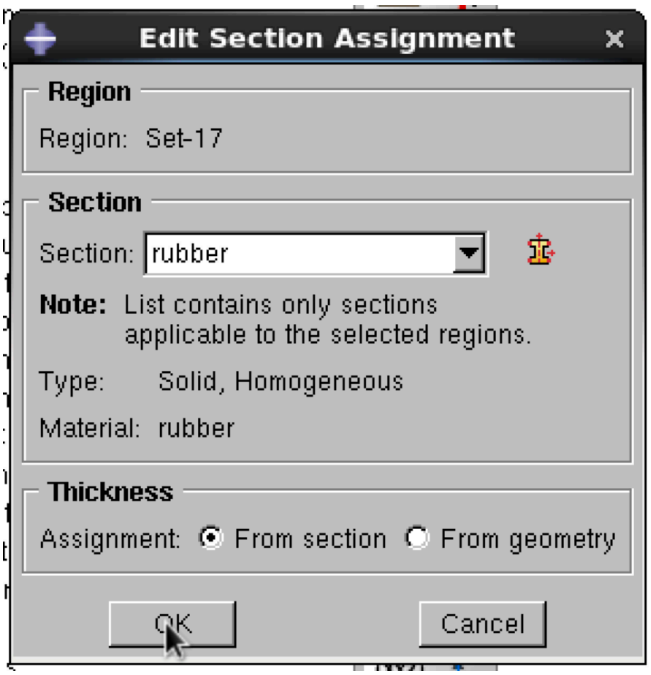
Create a section called ***Section-Rubber*** and assign to the Rubber Part.



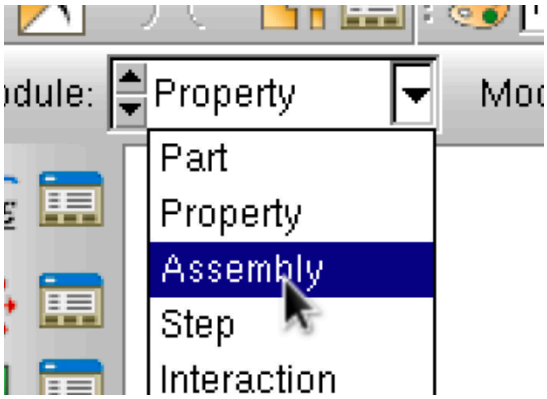


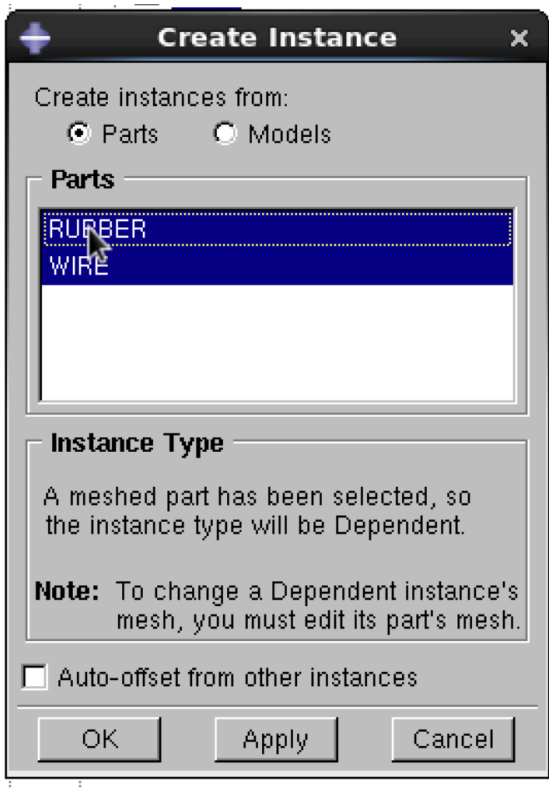



Set-17
Done

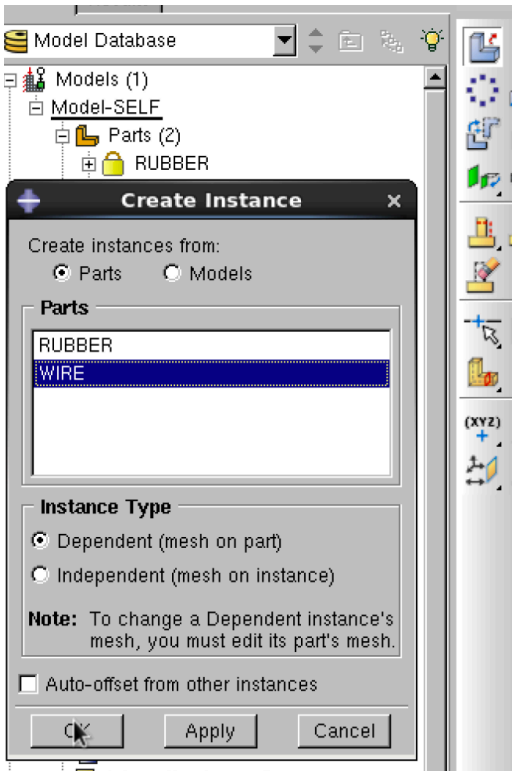

Edit Section Assignment
Region
Region: Set-17
Section
Section: rubber
Note: List contains only sections applicable to the selected regions.
Type: Solid, Homogeneous
Material: rubber
Thickness
Assignment: From section From geometry
OK
Cancel

6. Assembly

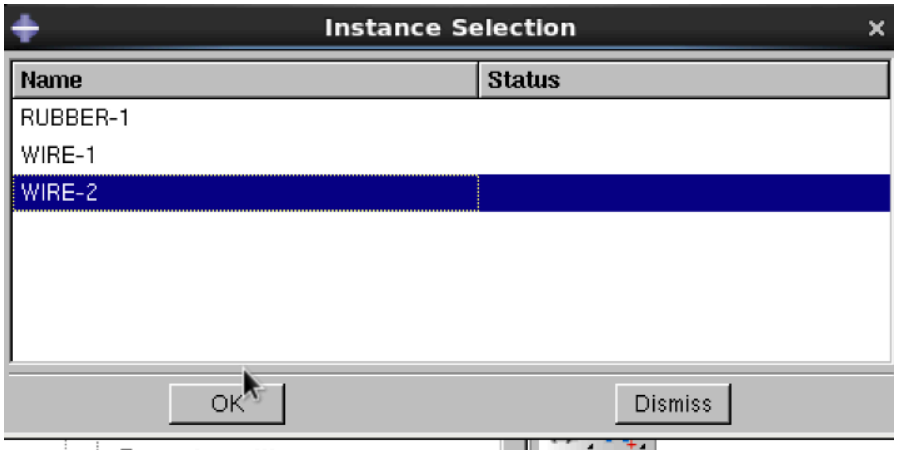


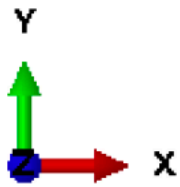


Now you need to instance the wire again (this will become the top wire)


Model Database
Models (1)
Model-SELF
Parts (2)
RUBBER
Create Instance
Create instances from:
Parts
Models
Parts
RUBBER
WIRE
Instance Type
Dependent (mesh on part)
Independent (mesh on instance)
Note: To change a Dependent instance's mesh, you must edit its part's mesh.
Auto-offset from other instances
OK
Apply
Cancel

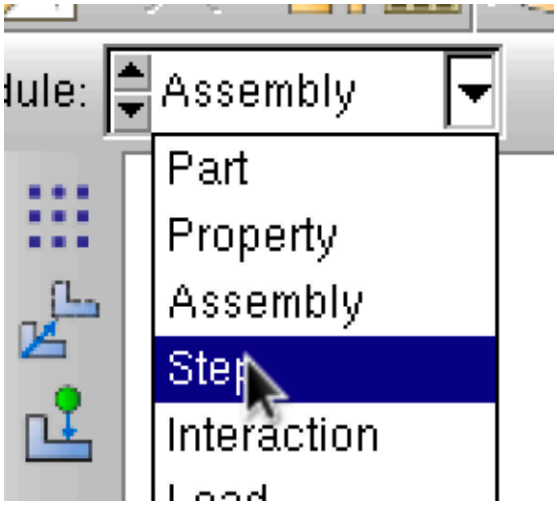
Instances...

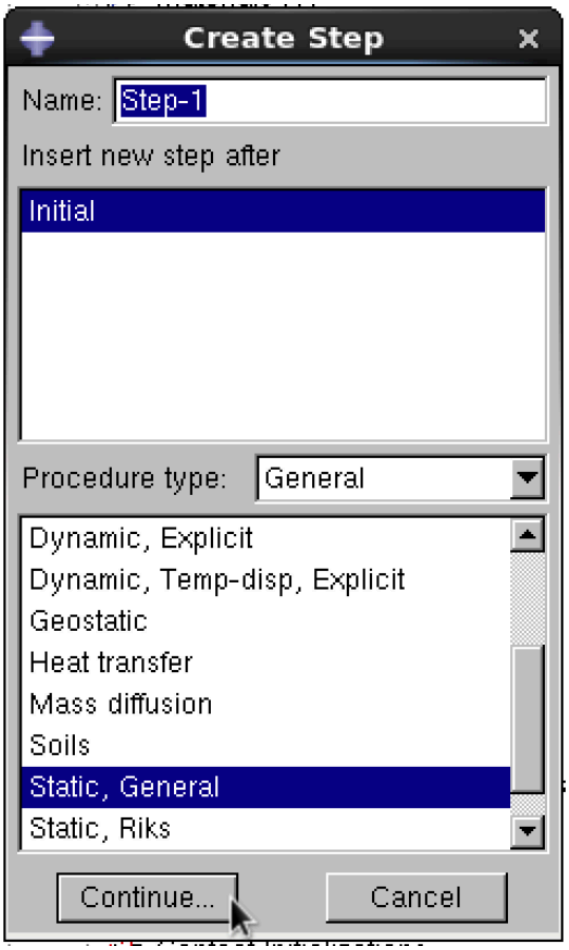




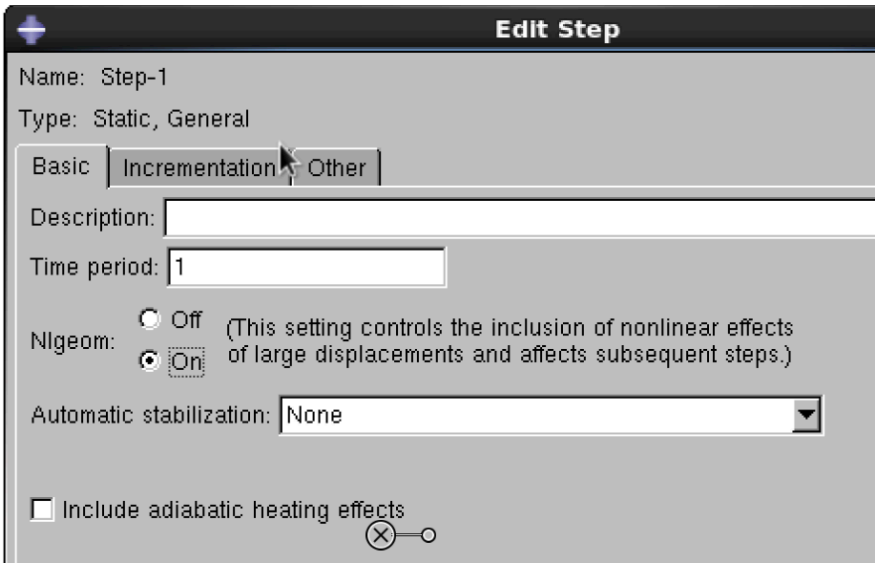
associated with this part or its instances

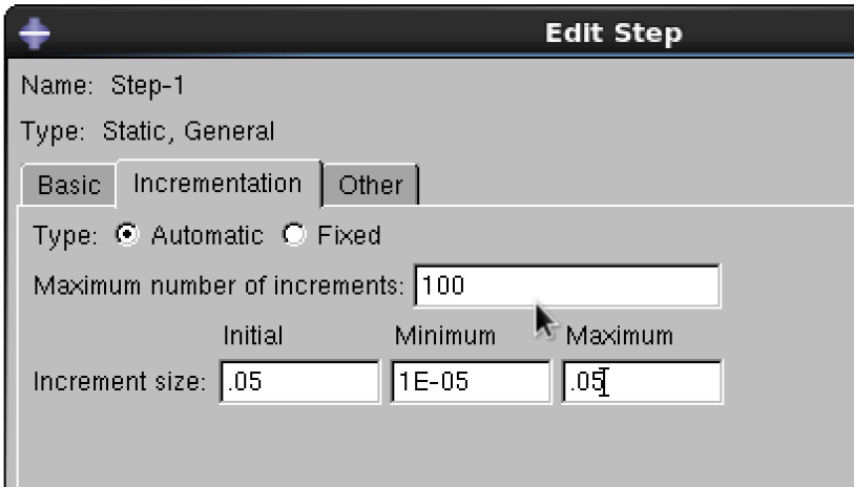
7. Create a Static Non-linear step



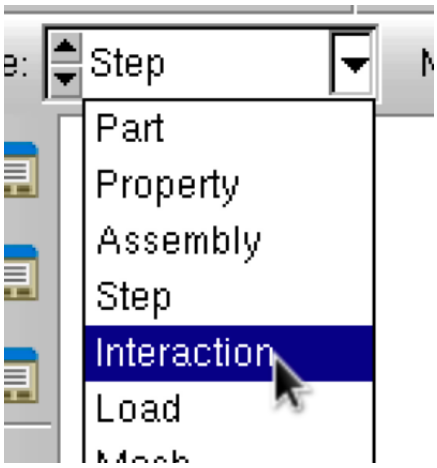


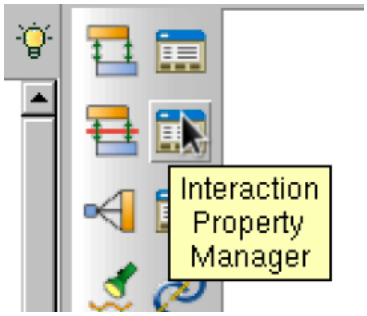
Assign parameters to the step as shown below

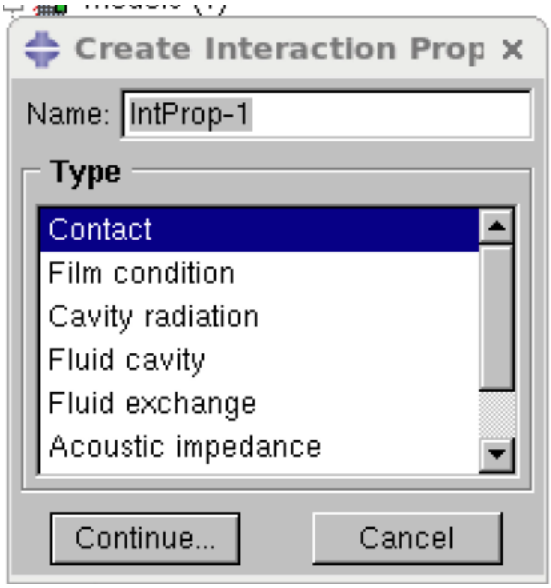


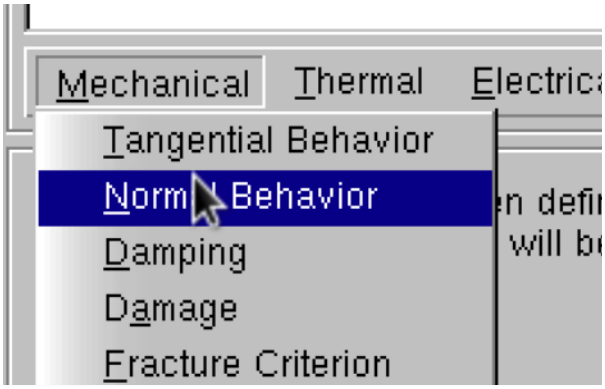


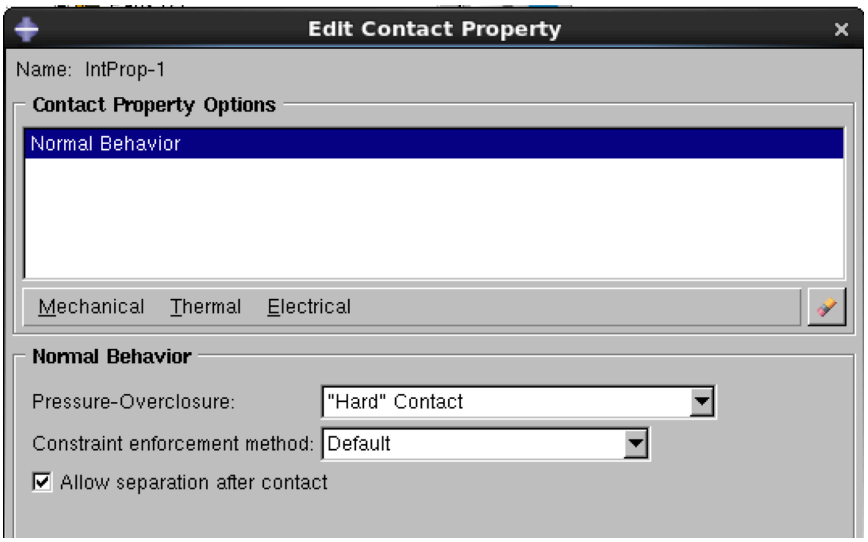
8. Create Contact Interfaces





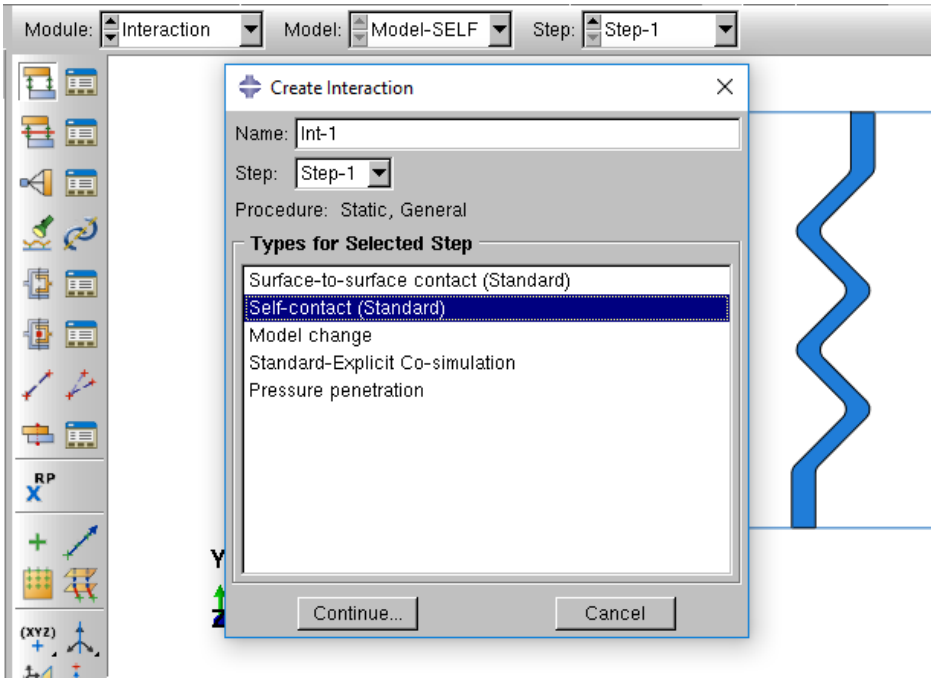

Create Interaction Prop
Name: IntProp-1
Type
Contact
Film condition
Cavity radiation
Fluid cavity
Fluid exchange
Acoustic impedance
Continue...
Cancel


Mechanical
Thermal
Electrica
Tangential Behavior
Norm Behavior
Damping
Damage
Fracture Criterion

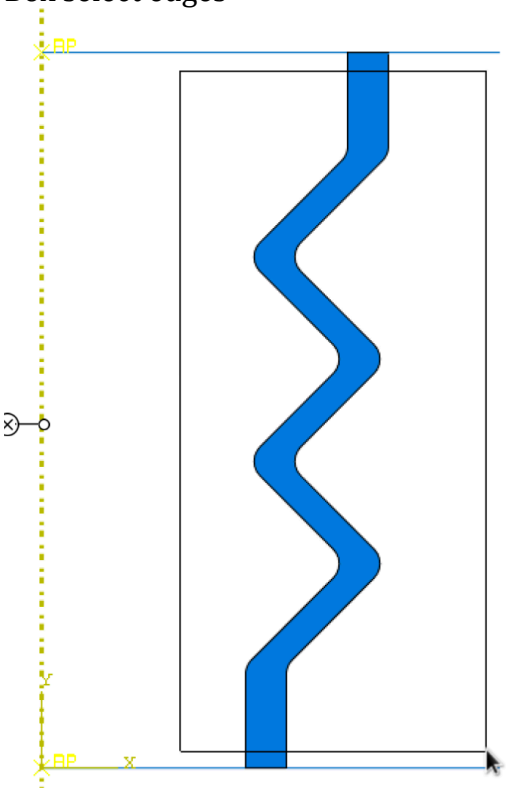

Edit Contact Property
Name: IntProp-1
Contact Property Options
Normal Behavior
Mechanical
Thermal
Electrical
Normal Behavior
Pressure-Overclosure:
"Hard" Contact
Constraint enforcement method:
Default
Allow separation after contact

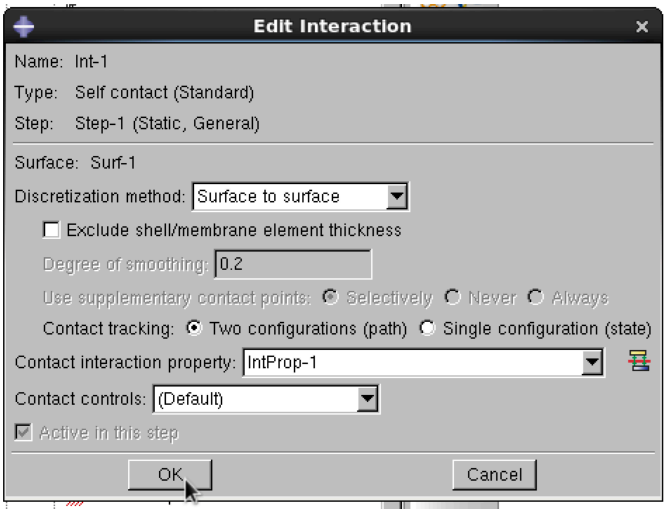
Select OK.

Now create an interaction:



Box select edges



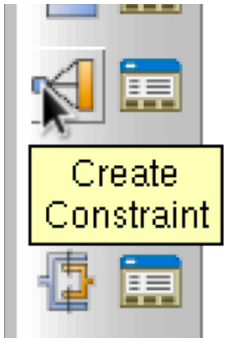


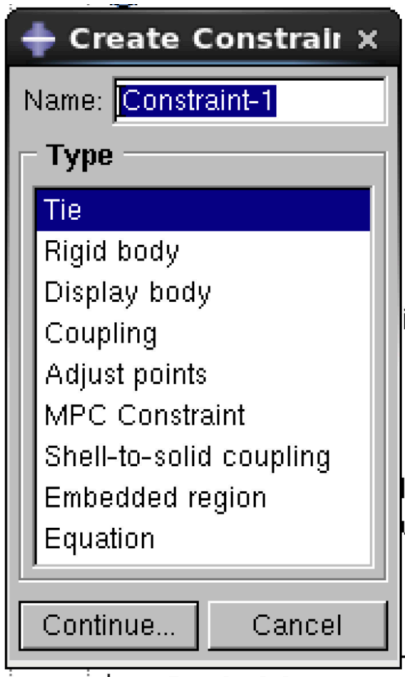
9. Create Tie Constraints

Define two new **Tie Constraints** to model the kinematic coupling between the rubber and the two wires where *no relative movement* is allowed between the instances.

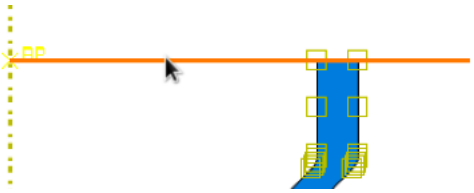
In the Model tree, double-click **Constraints**, select **Tie** as Type and name the Constraint as *Constraint-Up*. Select Surface as master type, the upper wire as master surface and choose the color of the shell side facing the rubber (yellow or magenta arrows). Select Surface as Slave type and click on Surfaces button in the bottom bar to directly select the predefined surface called Surf-TIE-UP as slave surface. Click Continue, then click OK.

- Repeat the same process to create the Tie constraint between the lower wire and the predefined surface called Surf-TIE-DOWN. Name the constraint as *Constraint-Down*.

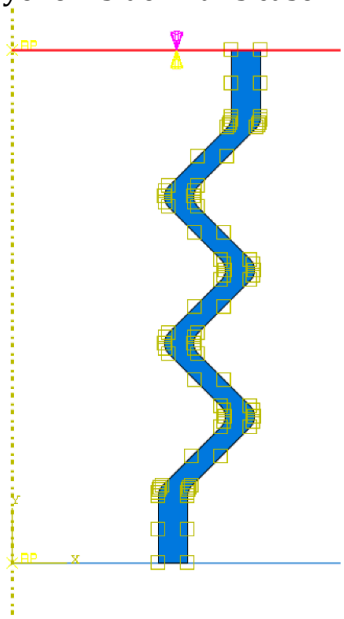


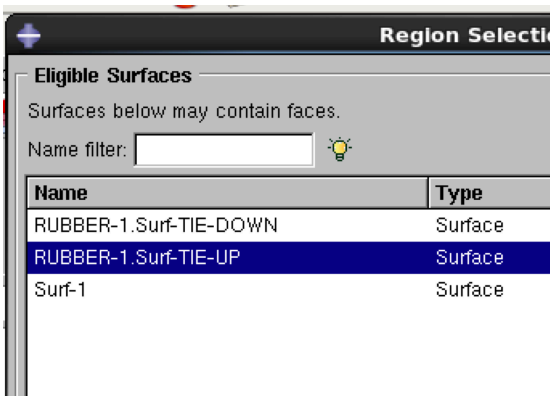


Select surface



Select top wire – yellow side in this case:



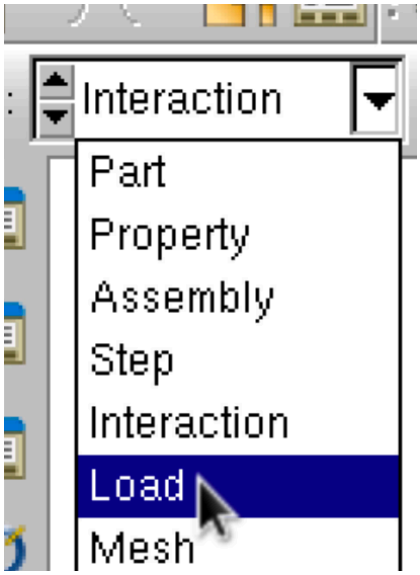


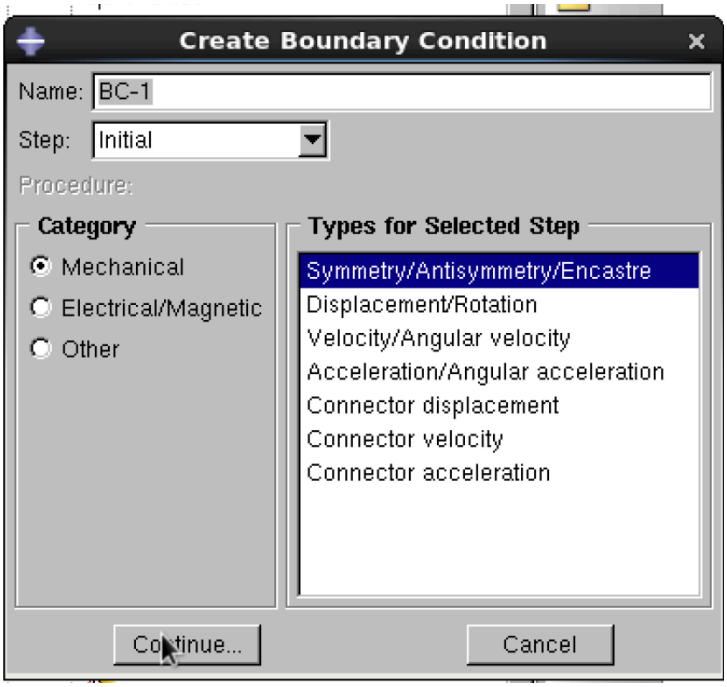
Repeat procedure for bottom surface.

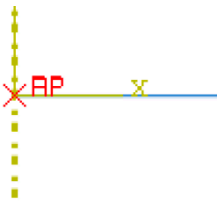
10. Add Boundary Conditions

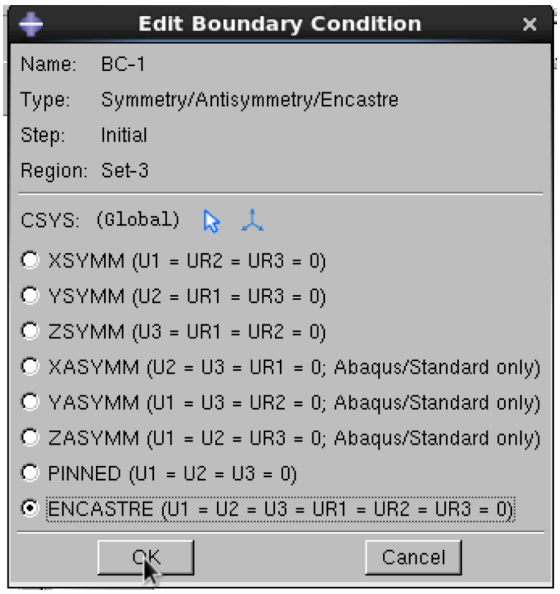
Two boundary conditions are applied to this model.

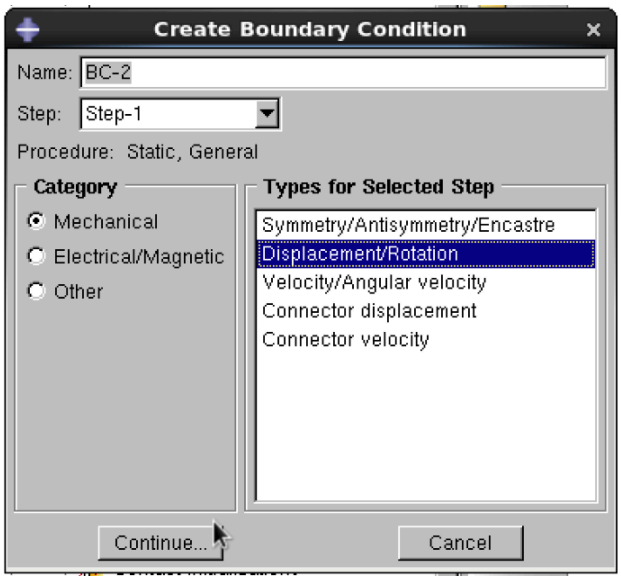
Create now a new Boundary condition called **BC-Encastre** to encastre the lower rigid tool. Select Initial as step and Symmetry/Antisymmetry/Encastre as type. Select the reference point of the lower wire as region (see figure) and then **ENCASTRE** as type. All the nodes of this rigid part will follow the movement of the reference node, thus the whole part will be fixed to its initial position.



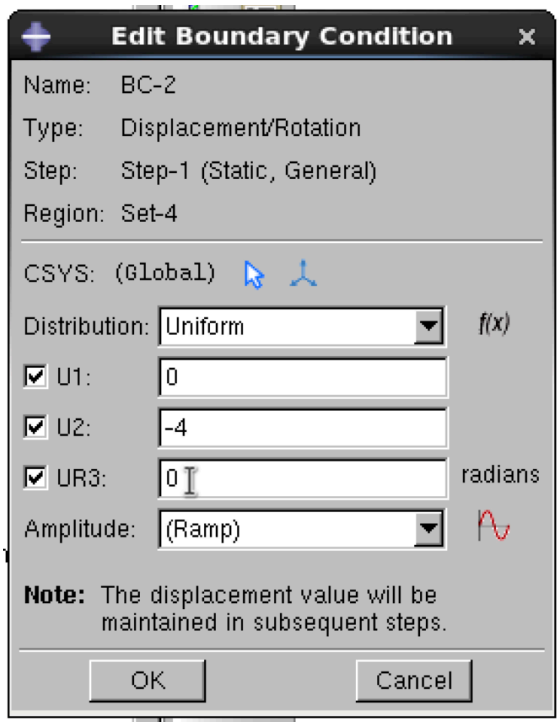




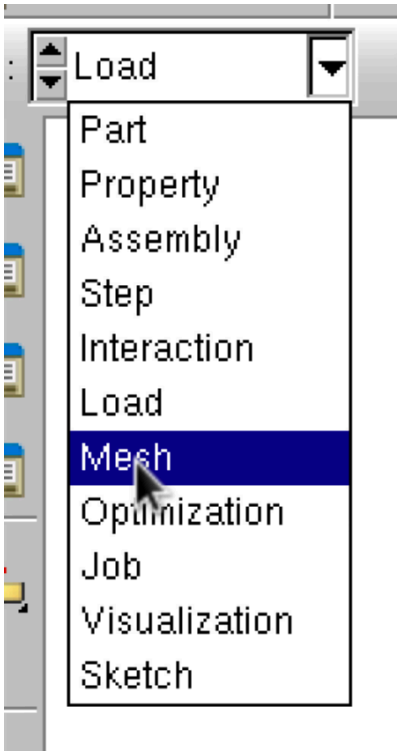


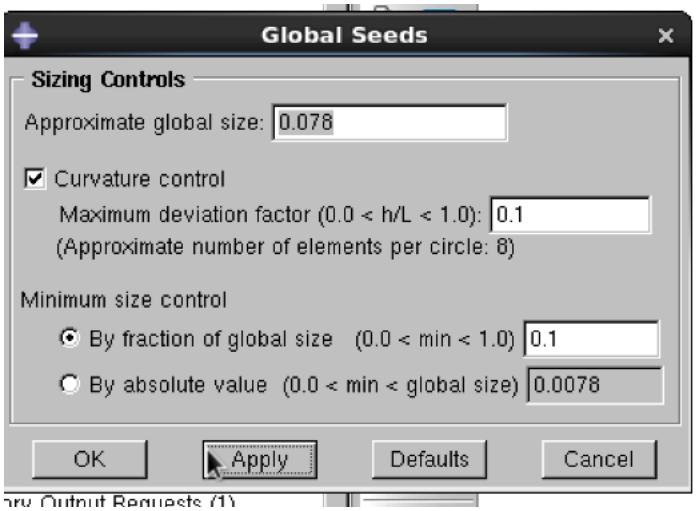

Create Boundary Condition
Name: BC-2
Step: Step-1
Procedure: Static, General
Category
Mechanical
Electrical/Magnetic
Other
Types for Selected Step
Symmetry/Antisymmetry/Encastre
Displacement/Rotation
Velocity/Angular velocity
Connector displacement
Connector velocity
Continue...
Cancel

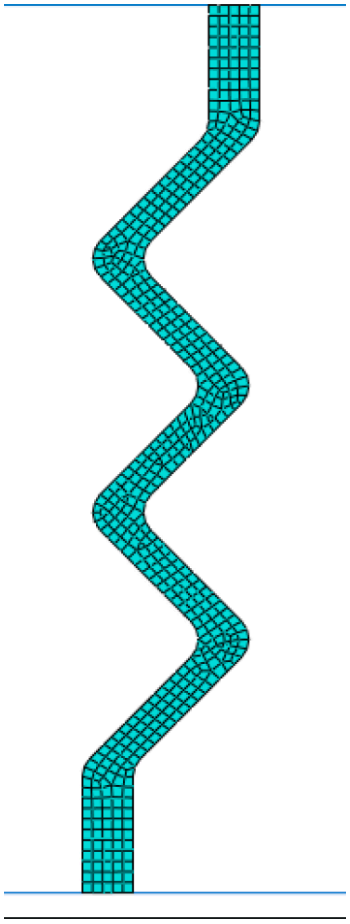
AP



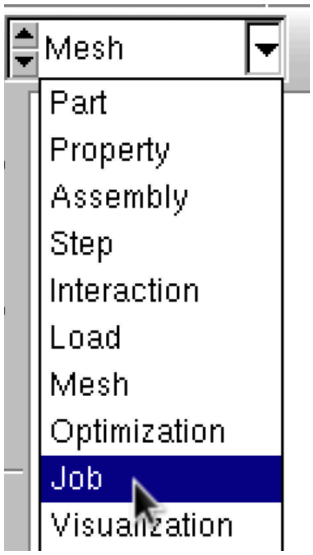
11. Create Mesh

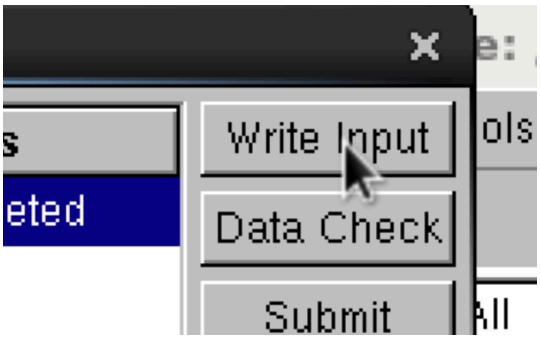


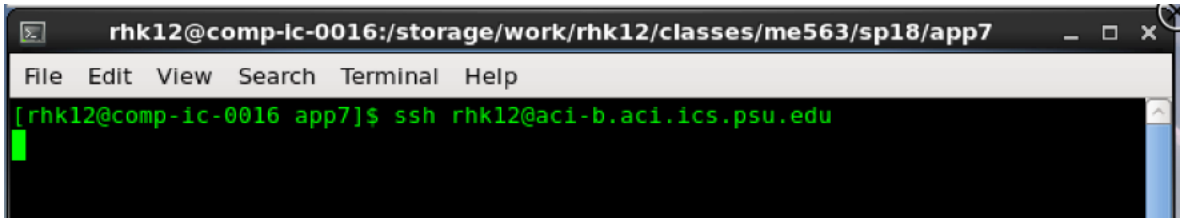

Global Seeds
Sizing Controls
Approximate global size: 0.078
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0) 0.1
By absolute value (0.0 < min < global size) 0.0078
OK
Apply
Defaults
Cancel

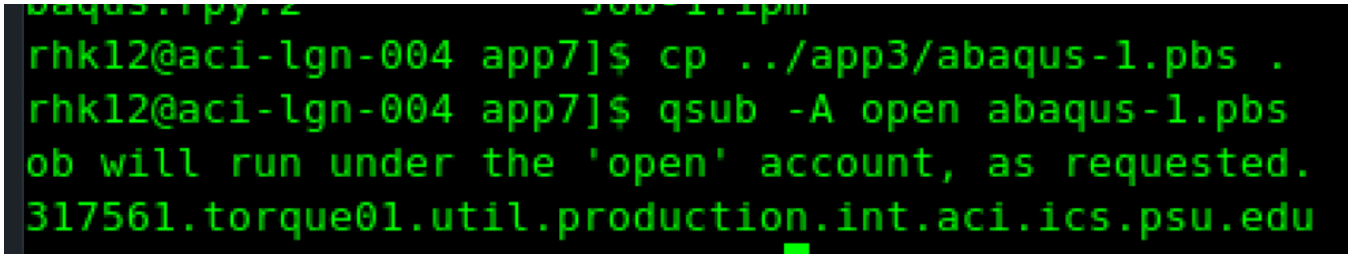


12. Create Job



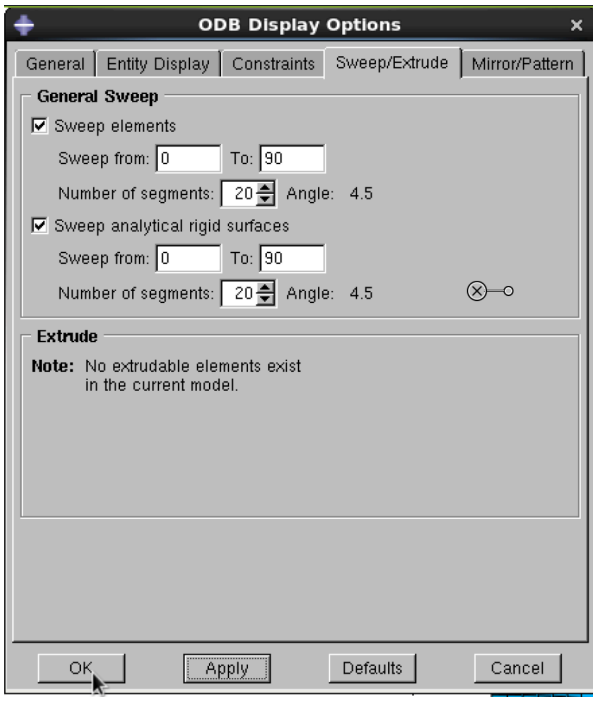


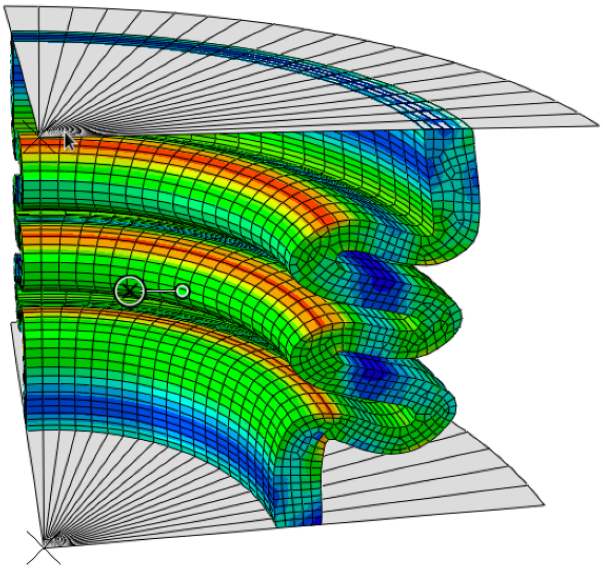
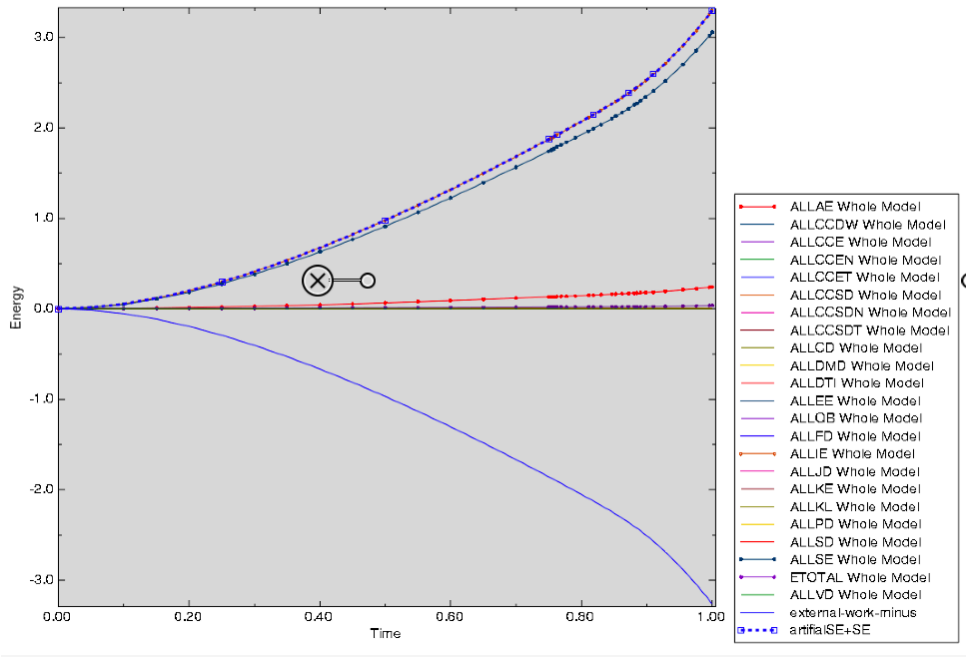




13. Analyze Results

Visualize the complete model. In the main menu bar, click on View Odb Display options and enter the **Sweep/Extrude** tab. Tick the Sweep Elements and Sweep analytical rigid surfaces options. In both cases, sweep from 0 to 90 degrees and enter 20 as number of segments Click OK.





Examine the energy response: