Abaqus/CAE tutorial
3D Elasticity

2D Plane Stress vs 3D model

A cantilever beam is made of steel with a modulus of elasticity E = 200 GPa and a Poisson's ratio $\nu = 0.3$ and is subjected to a distributed normal traction on the top surface with magnitude 10MPa. The beam has dimensions 600mm (length) and 200mm (height)

Create **3 different models** for this problem.

- 3D with thickness 20mm
- 3D with thickness 100mm
- 2D plane stress model

Make conclusions about your results.

Part Module

- Create part: Cantilever-3D-t20: 3D Part, deformable, Solid, Extrusion. Create a rectangle with starting corner (0,0) and opposite corner (600,200). Click done and enter depth equal to 20

Property Module

- Create material (Steel), select Mechanical tab, Elasticity – Elastic. Select the material Type as Isotropic and define Young's modulus = 200e3 (MPa) and Poisson's ratio = 0.3
- Click Create Section (Solid, Homogeneous) and select material (Steel). Click OK.
- Click on Assign Section. Assign the section your created in the previous step to your rectangular part.

Assembly Module

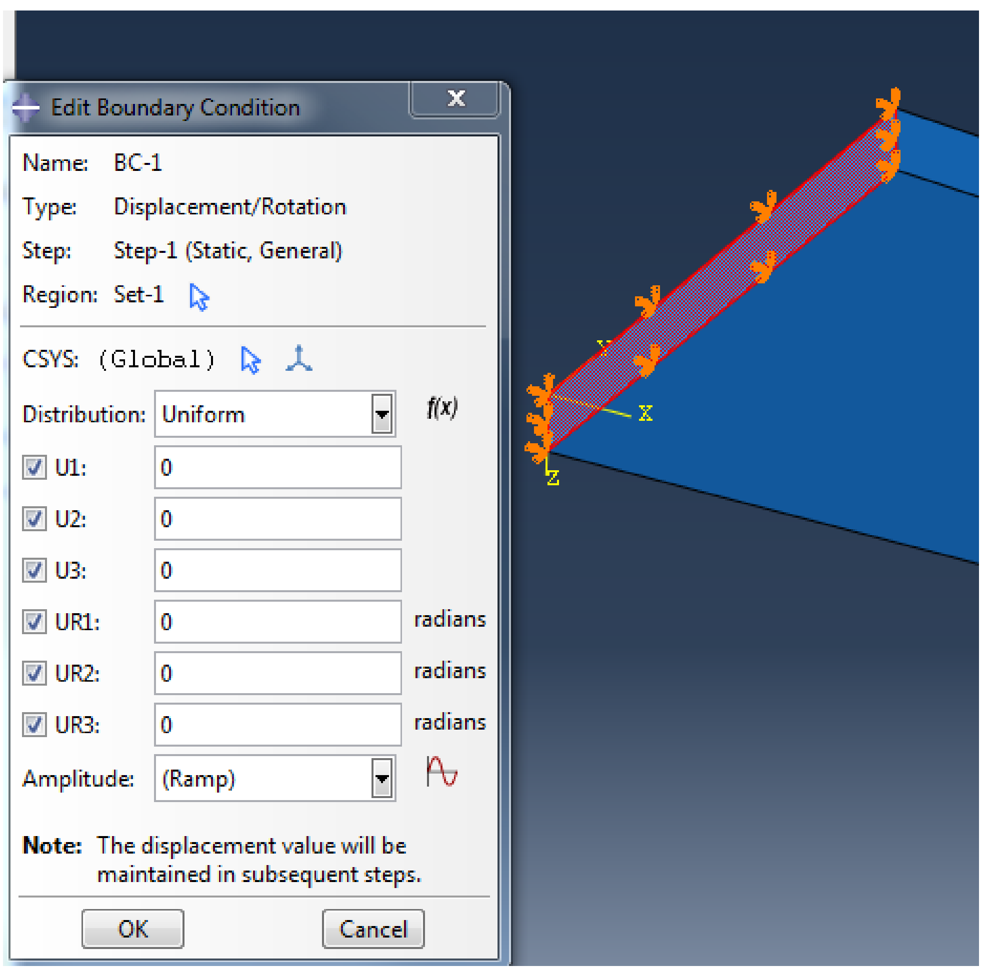
- Click on Create Instance
  - Create Instance dialog box appears, select Cantilever-3D-t20
  - Select Dependent for the instance type
  - Click OK

Step Module

- Click on the Create Step icon and the Create Step dialog box appears.
  - Name the step (e.g. Step-1), set the Procedure type to General and select Static, General
  - Click Continue
  - The Edit Step dialog box appear. Click OK

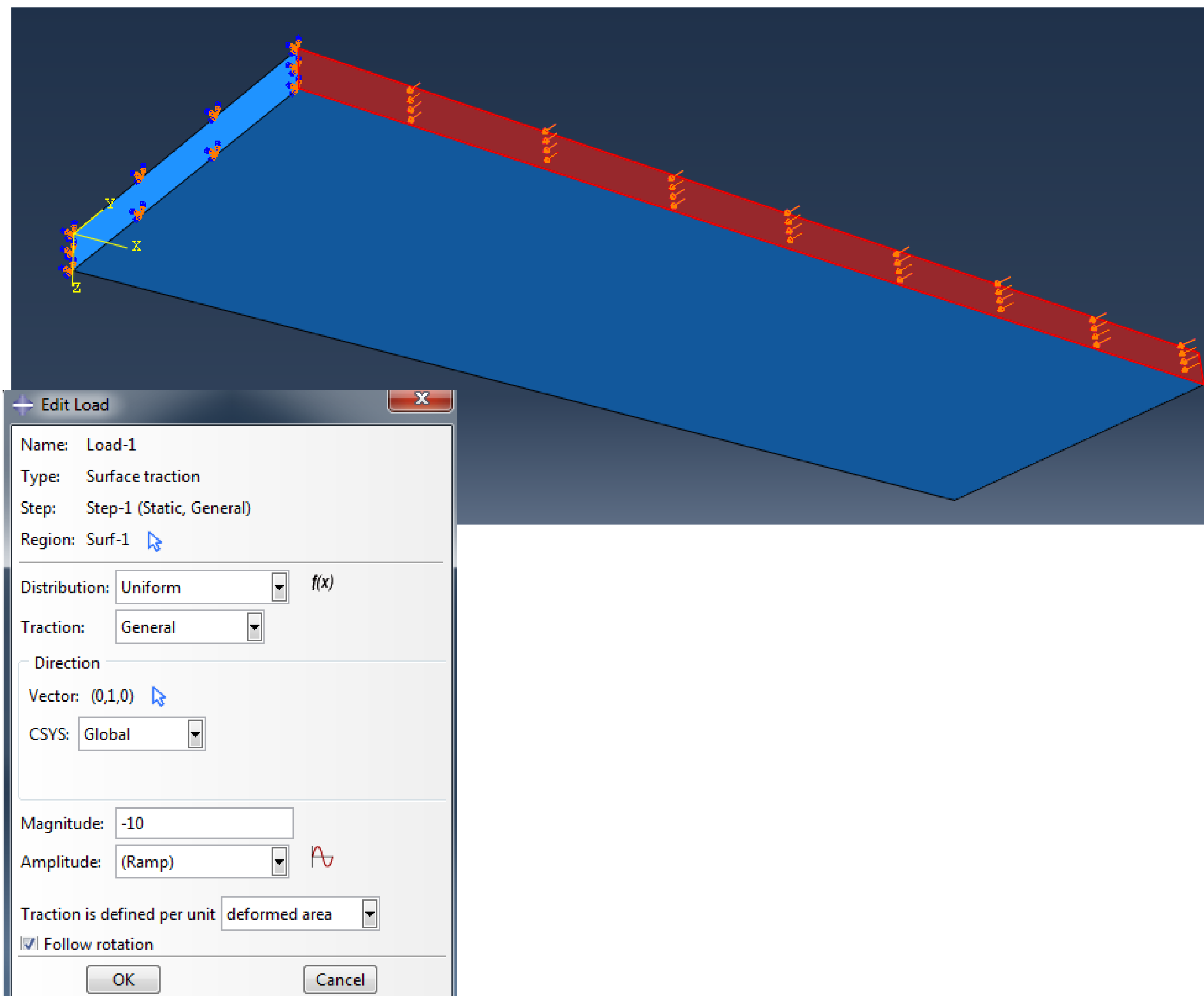
Load Module

- Click on Create Boundary Condition icon and the Create Load dialog box appears.
- Select Displacement/Rotation and click Continue.
- Select the edge that is clamped. The Edit Boundary Condition dialog box appears.
- Select Uniform distribution and select all degrees of freedom. Click OK



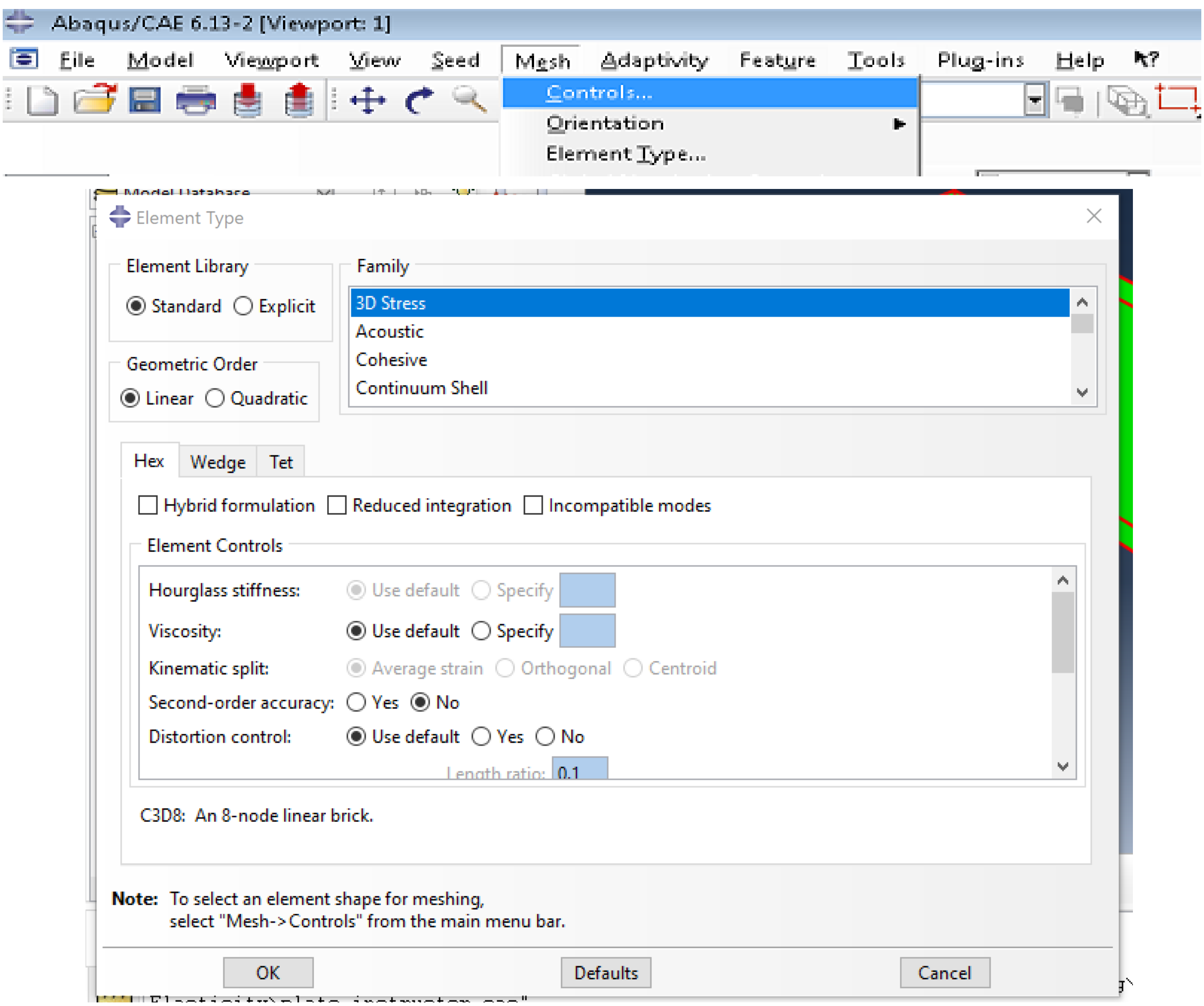
Load Module

- Click on Create Load icon and the Create Load dialog box appears.
- Select Surface traction and click Continue. Select the surface where load is applied and click Done. The Edit Load dialog box appears. Select Uniform distribution and general traction
  - Click on the arrow button next to “Vector: Required” to enter the direction of the normal vector (perpendicular to the surface)
  - Write the first point of the normal vector: 0,0,0. Press Enter.
  - Write the second point of the normal vector: 0,1,0. Press Enter..



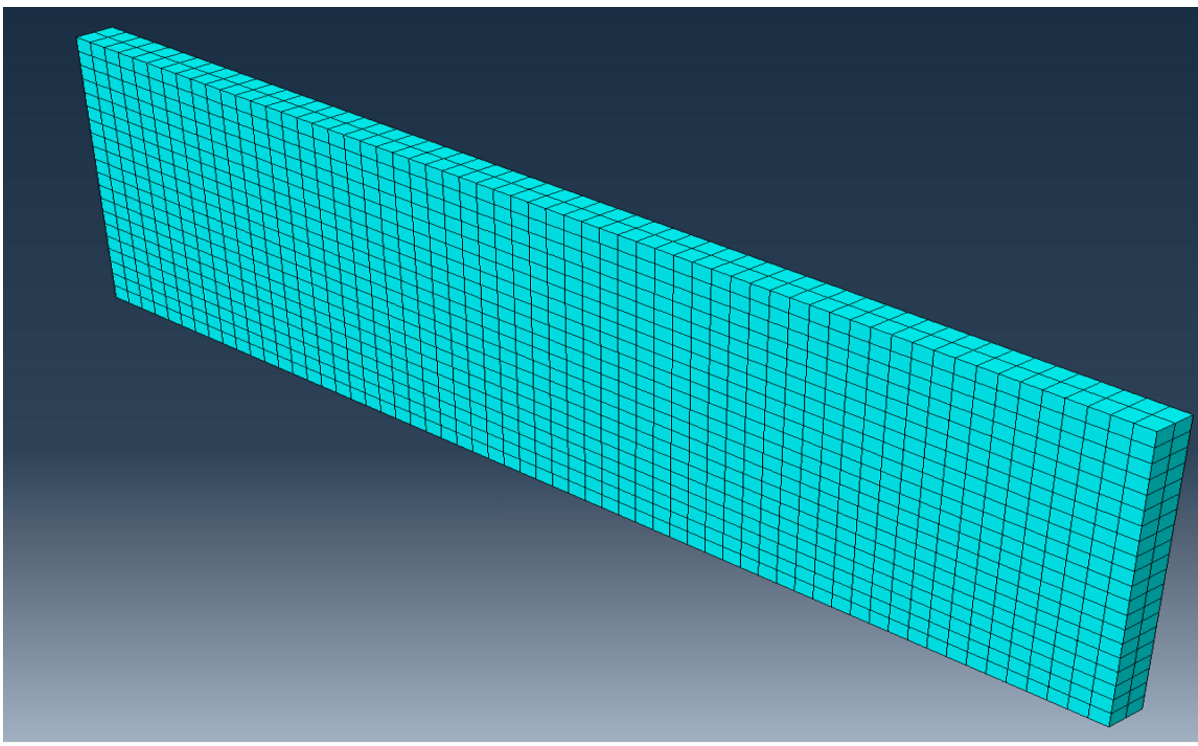
Mesh Module

- Go To Module Mesh
  - From the top toolbar, go to “Mesh” and select “Controls”
  - Select Element Shape: Hex
  - From the top toolbar, go to “Mesh” and select “Element Type”. Select each part in the viewport and click Done in the prompt area. Element Type dialog box appears.
  - Select Standard for Element Library, Linear for Geometric Order and 3D Stress for Family
  - Select Hex C3D8; uncheck Hybrid and Reduced integration
    - Read more on hybrid elements [here]
    - Read more on reduced integration [here]



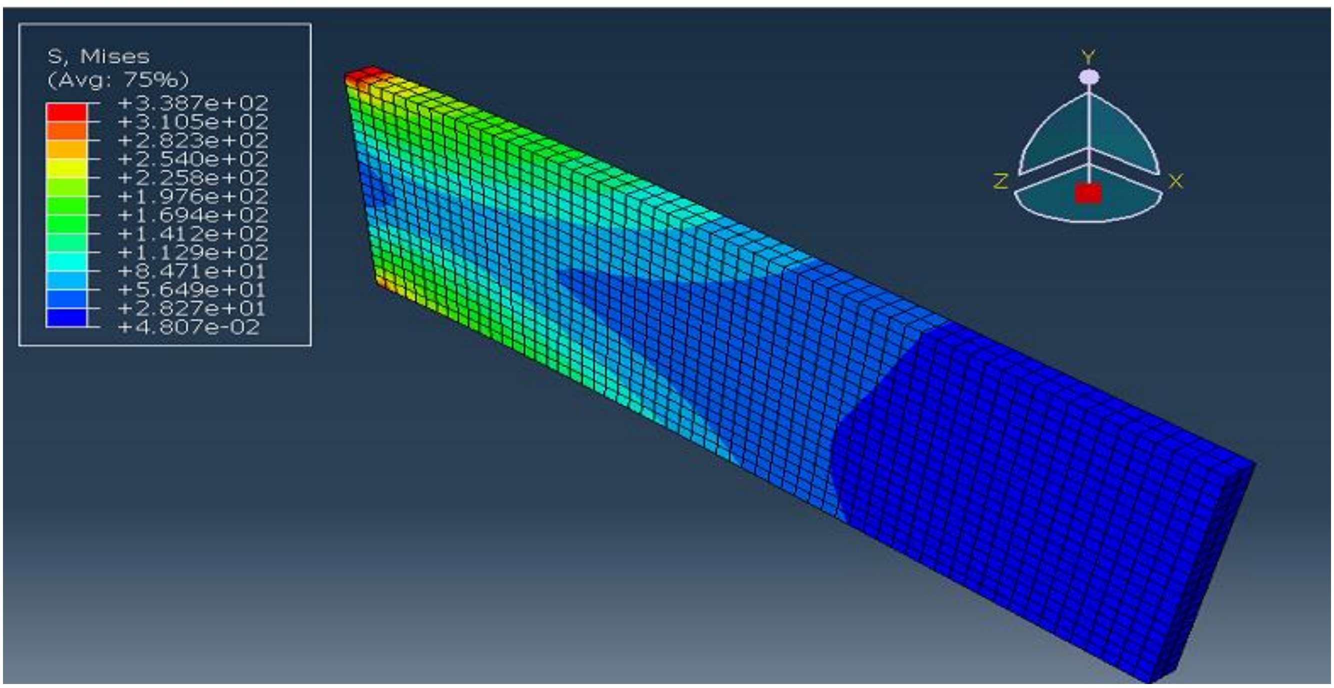
Mesh Module

- In the toolbox area click on the Seed Part
- Give approximate global size = 10
- In the toolbox area click on the Mesh Part
  - Click Yes in the prompt area



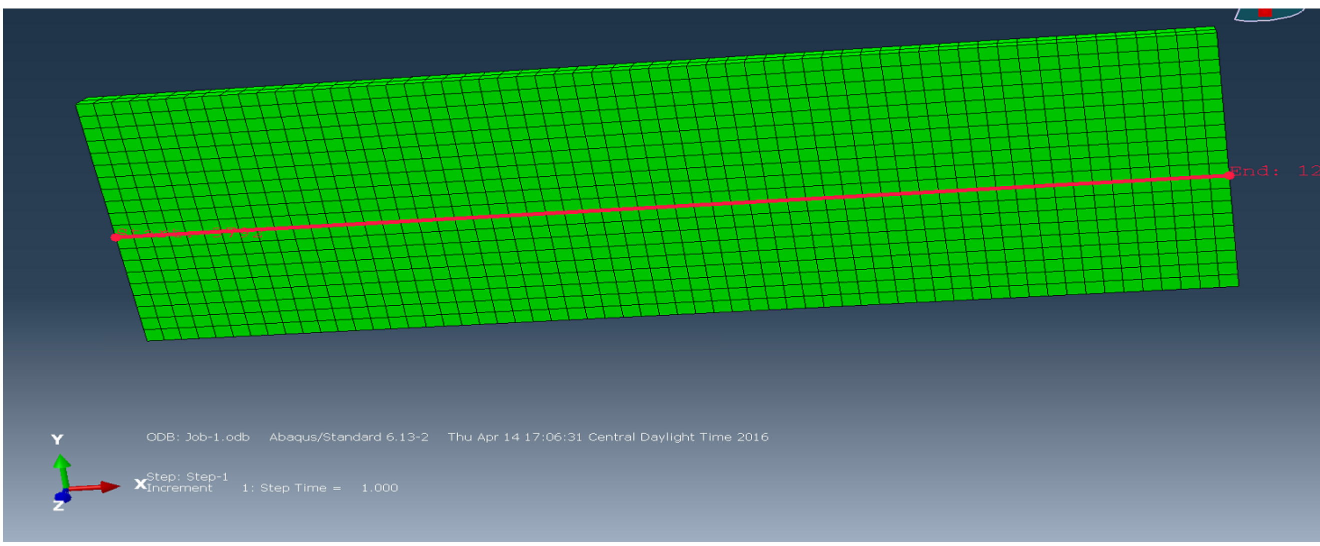
Job and Visualization Module

- Click on Job Manager icon and the Job Manager dialog box appears.
- Click Create and the Create Job dialog box appears.
- Give a name to the job (Cantilever-3D-t20) and click Continue
- Click OK on the Edit Job dialog box
- Click Submit on the Job Manager
- Once the job is completed (check status column), click Results. This will take you to the Visualization Module

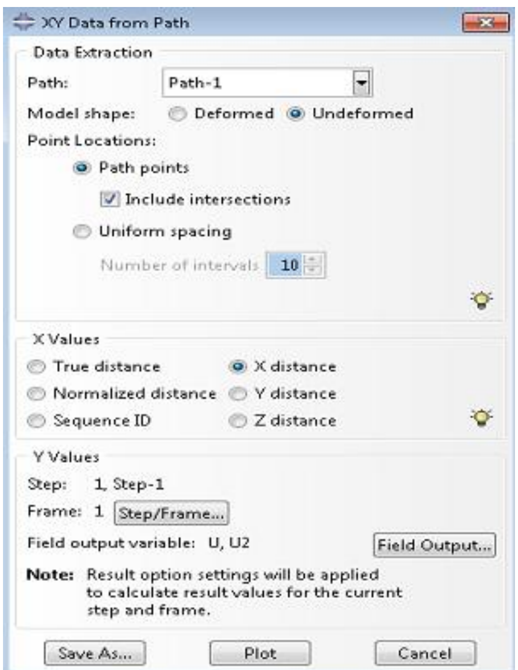


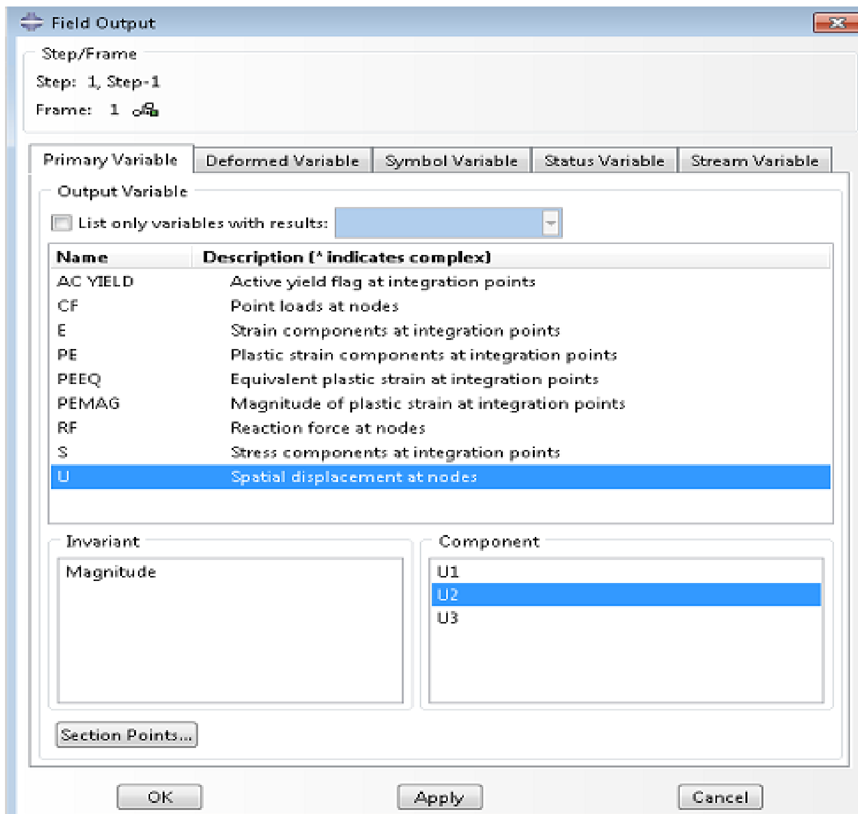
Displacement curves

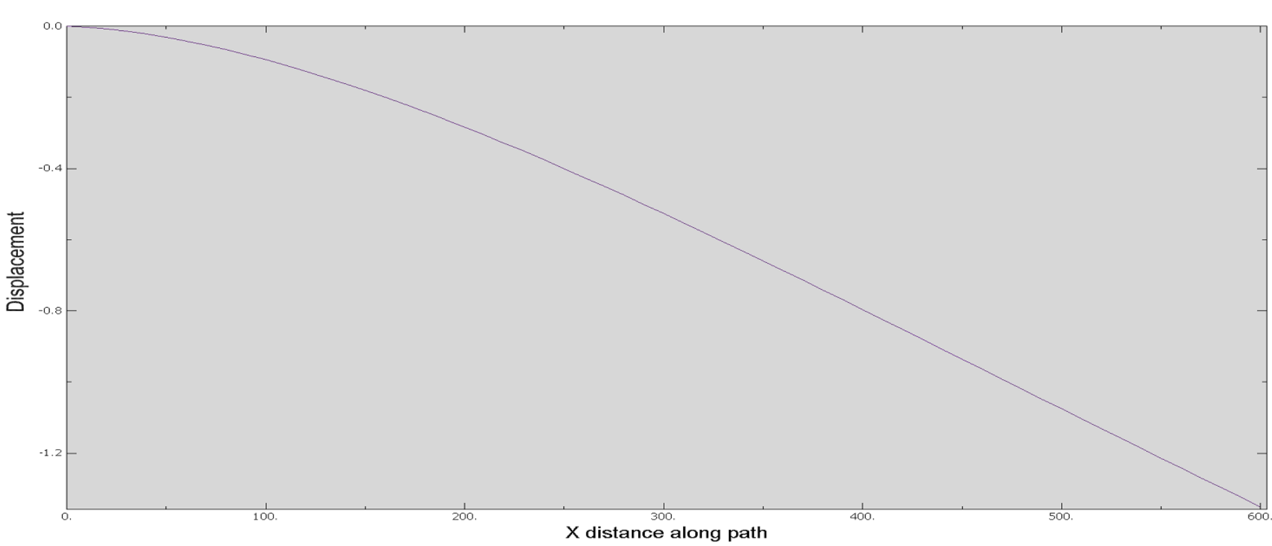
- From the toolbar menu, click Tools-Path-Create and select Node List
- The Edit Node List Path will appear. Click Add Before…
- Select nodes to be inserted in the path. For example, you may choose the start as (0,100) and the end as (100,100) as indicated below (z coordinate is 20)
- Click Done and OK



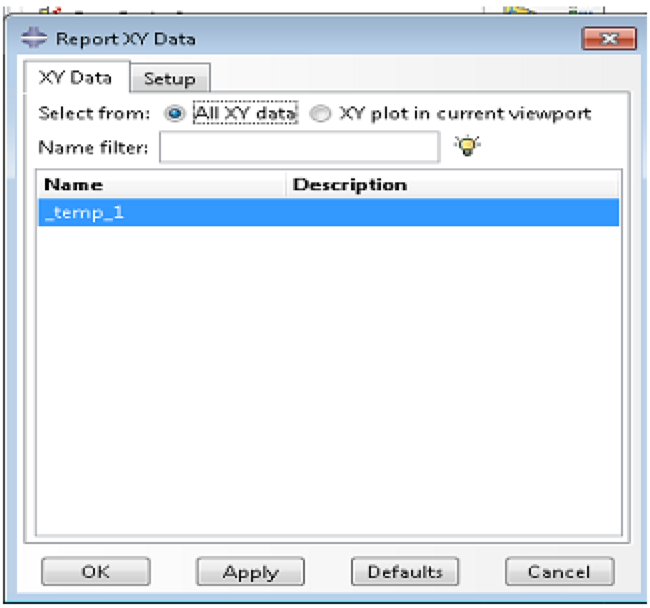
- From the toolbar menu, click Tools-XY Data-Create and Select Path
- Select the path your created (Path-1) and mark "Path points" and "Include interserctions)
- X values = x distance
- For Y values, click on the field output button and select displacement (U), component U2 (y-direction). You may want to try different fields too.
- Click OK and Plot







- From the toolbar menu, click Report-XY. The Report XY Data dialog box appears.
- Go to the Setup tab, give a name to the file “Cantilever-3D-t20.rpt” and remove the selection from the option “Append to file”. Click OK.
- Open the file to see your displacement results.



Comparison to other models

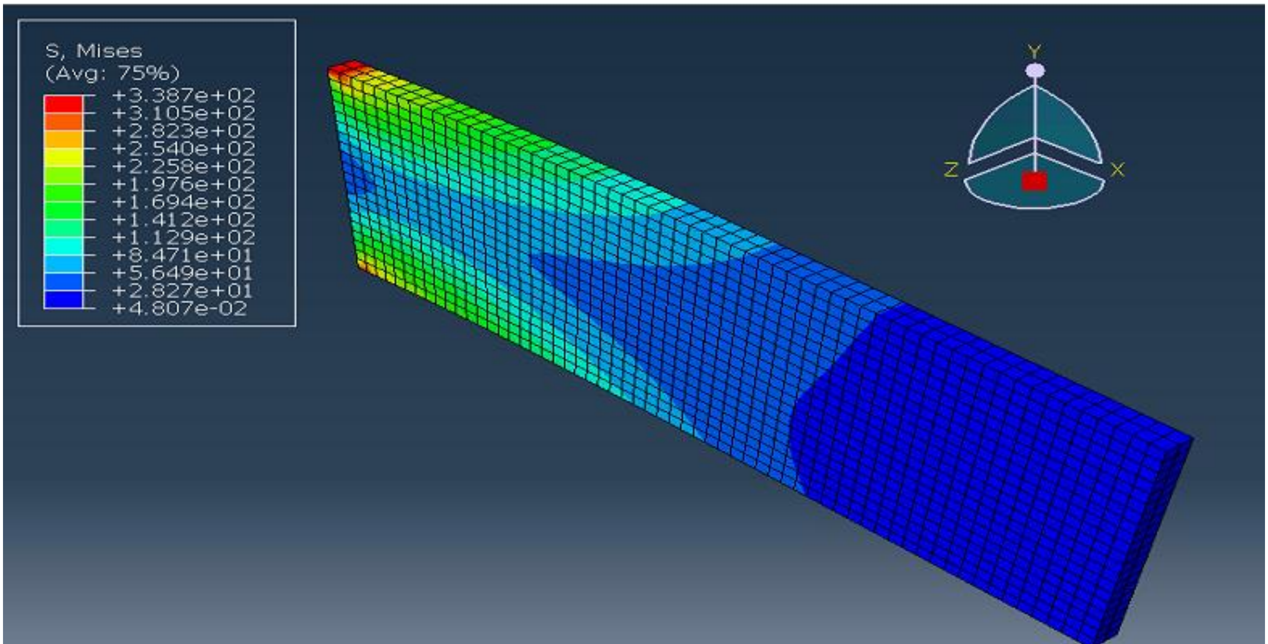
Now create two more models of the same problem; we'll compare the solution results

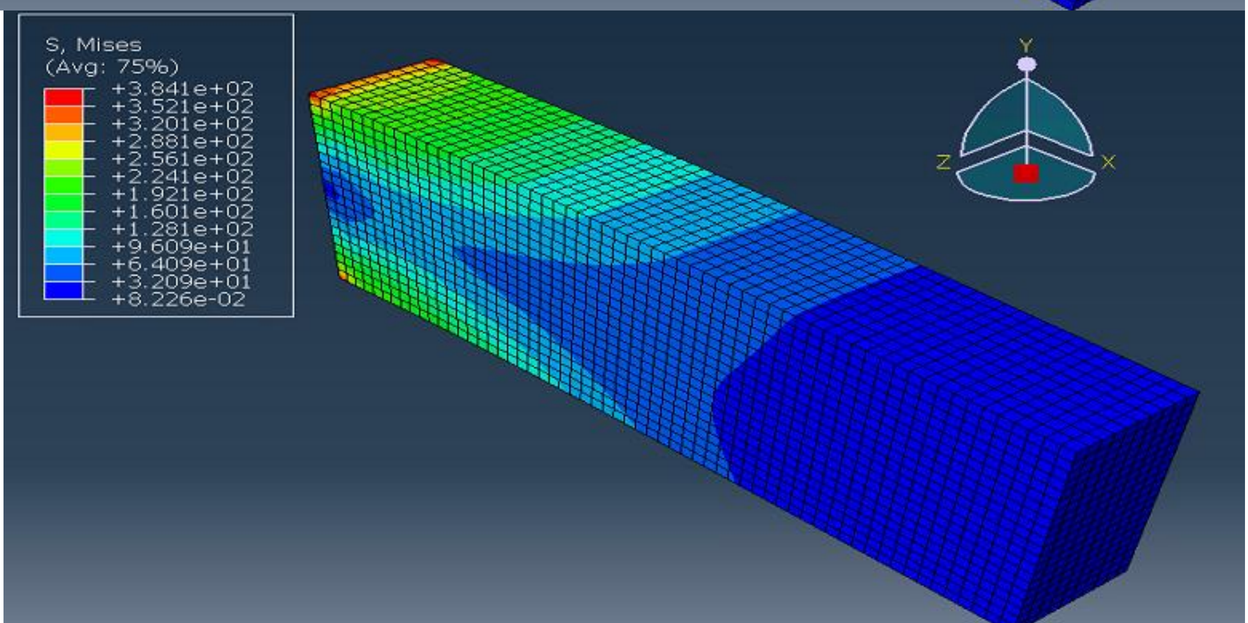
Model 2: 3D model with thickness 100

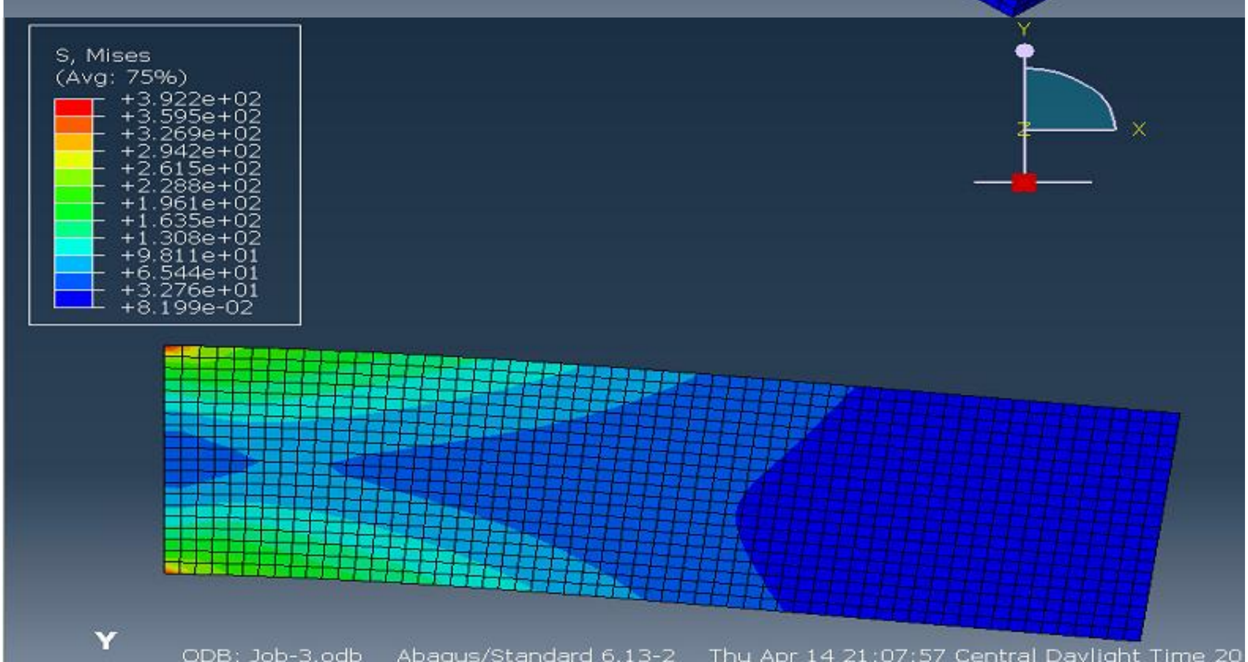
- Either copy the first model and modify the extrusion thickness, or repeat the previous steps using an extrusion thickness of 100
- You do not need to modify the traction load magnitude for the boundary condition

Model 3: 2D plane stress model

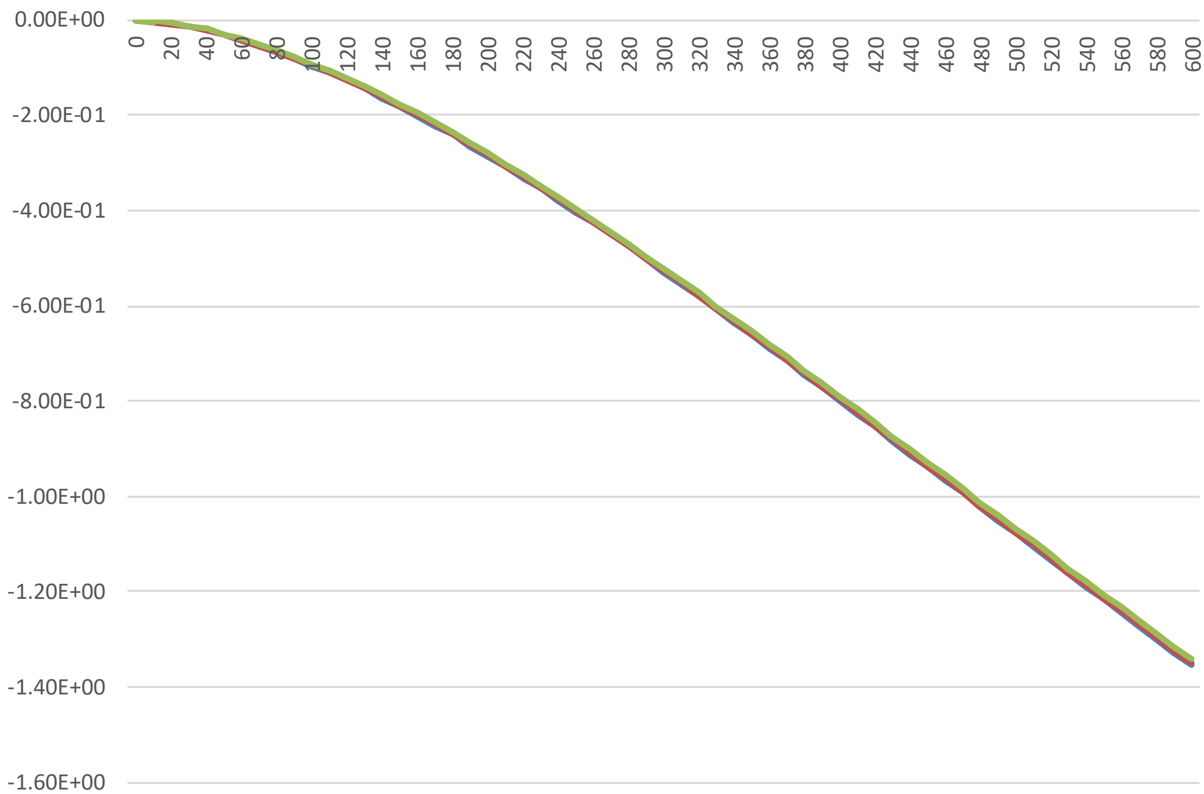
- Recreate the model using the same size mesh but with 4 node linear quadrilaterals. Use Plane stress linear 4 node quadrilateral elements (uncheck hybrid and reduced integration boxes)


S, Mises
(Avg: 75%)
+3.387e+02
+3.105e+02
+2.823e+02
+2.540e+02
+2.258e+02
+1.976e+02
+1.694e+02
+1.412e+02
+1.129e+02
+8.471e+01
+5.649e+01
+2.827e+01
+4.807e-02
Y
Z
X


S, Mises
(Avg: 75%)
+3.841e+02
+3.521e+02
+3.201e+02
+2.881e+02
+2.561e+02
+2.241e+02
+1.921e+02
+1.601e+02
+1.281e+02
+9.609e+01
+6.409e+01
+3.209e+01
+8.226e-02
Y
Z
X


S, Mises
(Avg: 75%)
+3.922e+02
+3.595e+02
+3.269e+02
+2.942e+02
+2.615e+02
+2.288e+02
+1.961e+02
+1.635e+02
+1.308e+02
+9.811e+01
+6.544e+01
+3.276e+01
+8.199e-02
Y
Z
X
Y
ODB: Job-3.odb Abaqus/Standard 6.13-2 Thu Apr 14 21:07:57 Central Daylight Time 20

- Displacement curves for the selected path are basically the same for the three different models.
- Plane stress is a good assumption for this cantilever beam model
- Results will start to deviate when the thickness becomes larger (comparable with beam length)



- The use of 2D models reduces the computational power