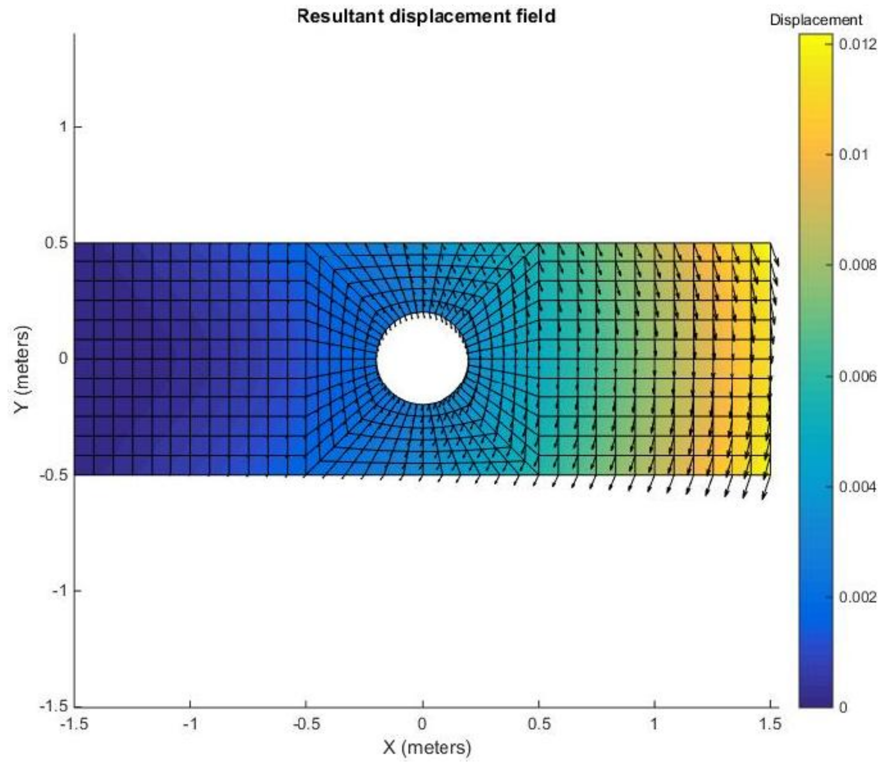
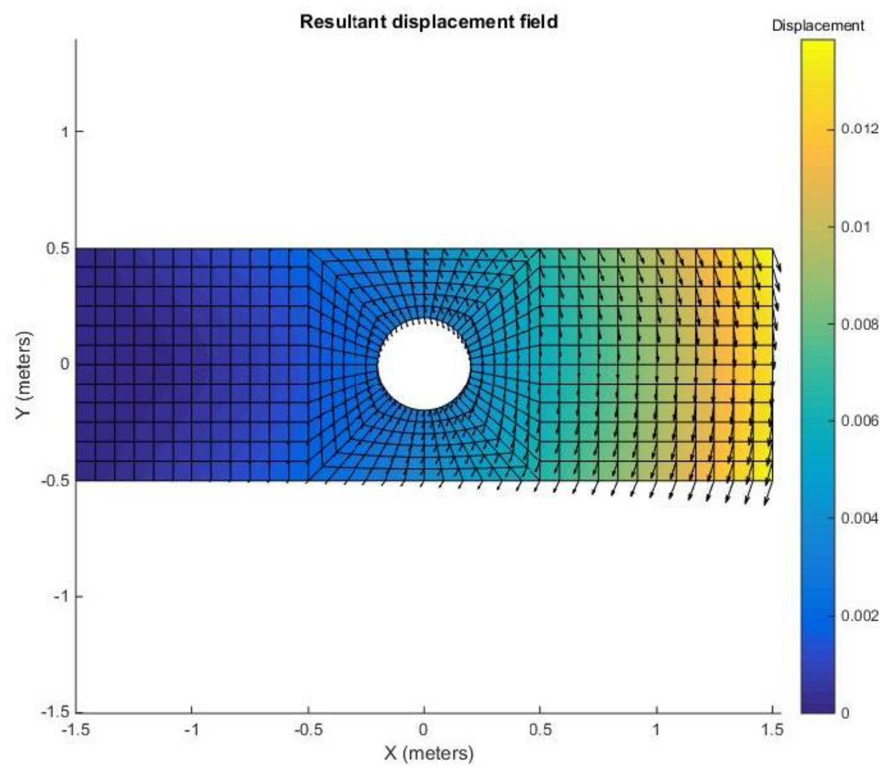
2. MATLAB SOLUTION

2.1) Contour plot of approximated displacement field under fine mesh

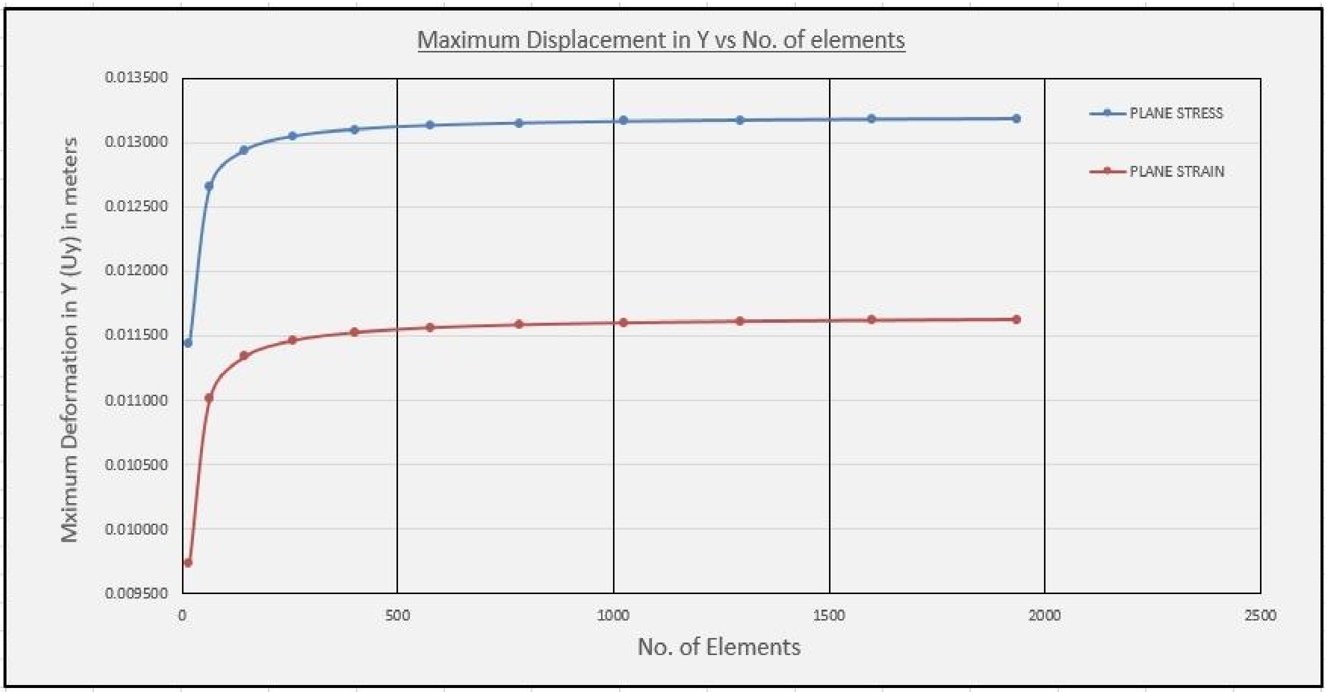


PLANE STRAIN MODEL (above): Maximum Displacement = 11.9mm



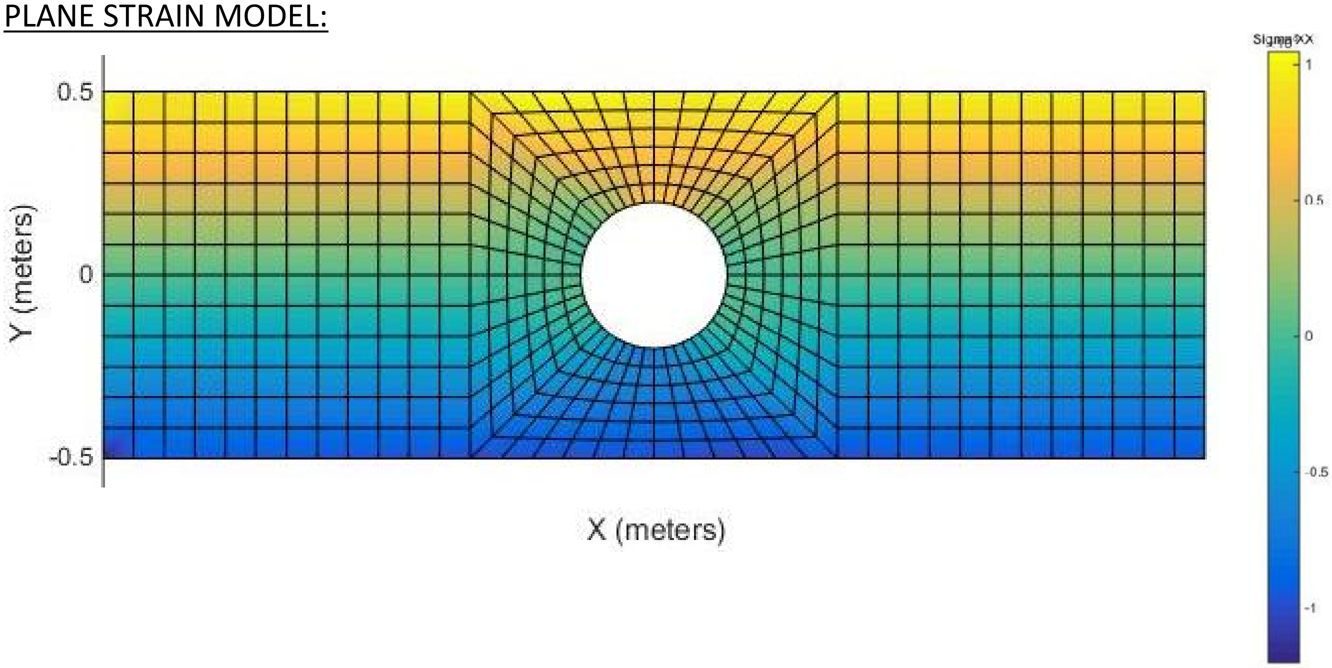
PLANE STRESS MODEL: Maximum Displacement = 13.8mm

2.2) Plot of maximum $|u_y|$ versus the number of elements using both plane strain and plane stress model in the same figure

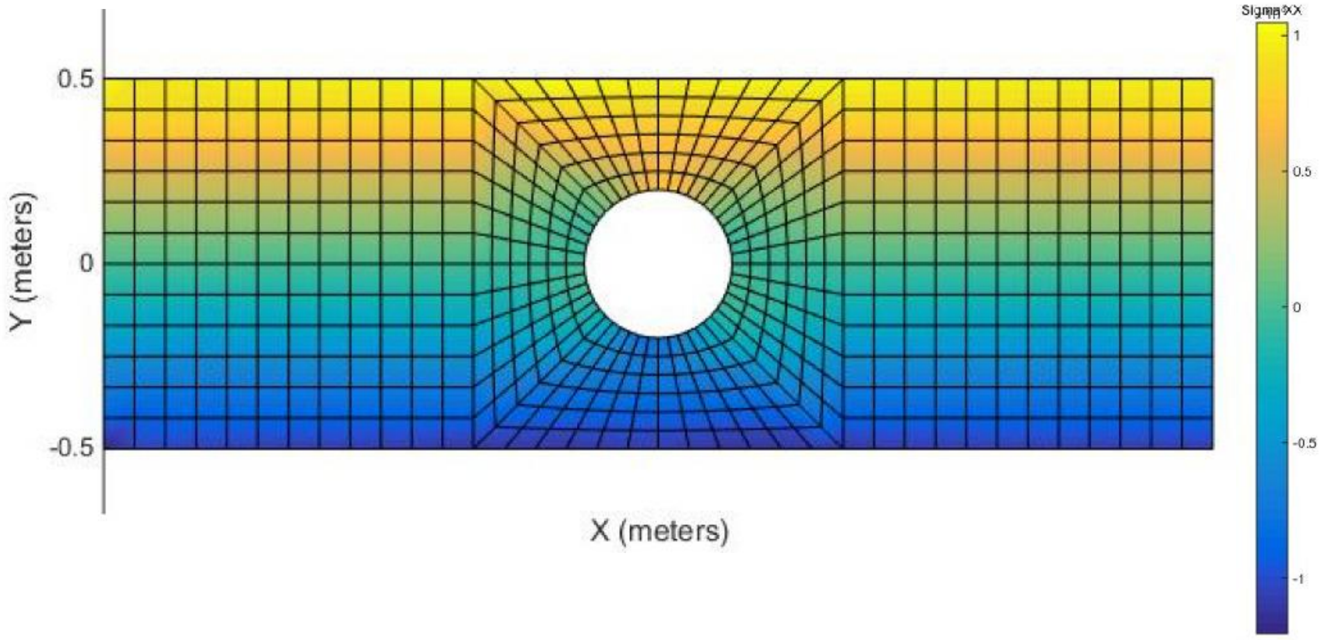


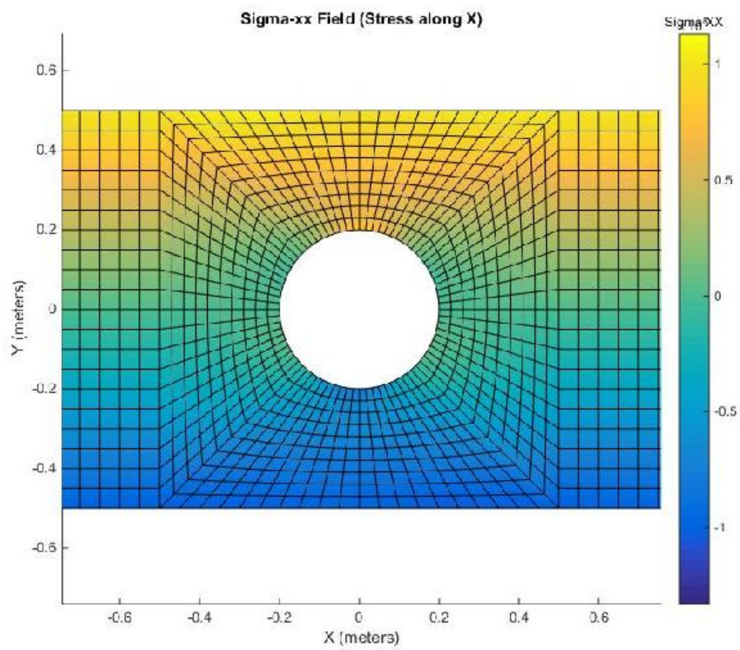
2.3) Contour plot of σ_{xx} using plane stress and plane strain model

PLANE STRAIN MODEL:

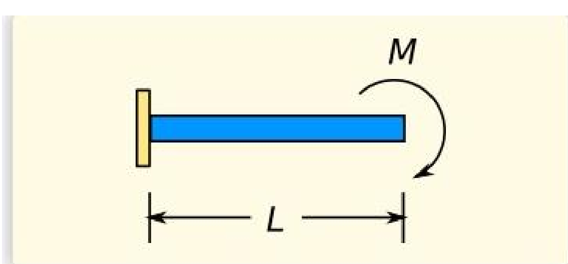


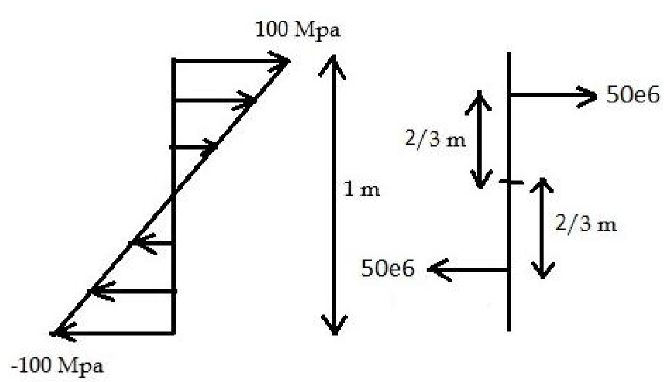
<u>PLANE STRESS MODEL:</u>





<u>Explanation for the results:</u>

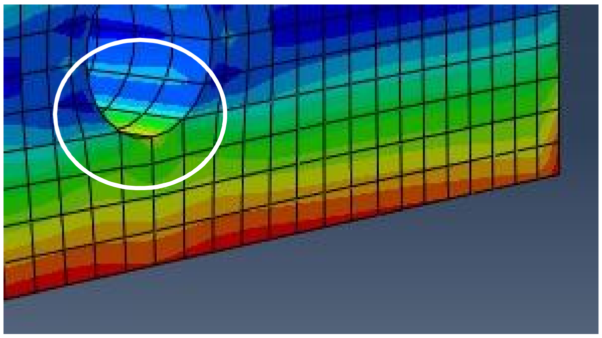


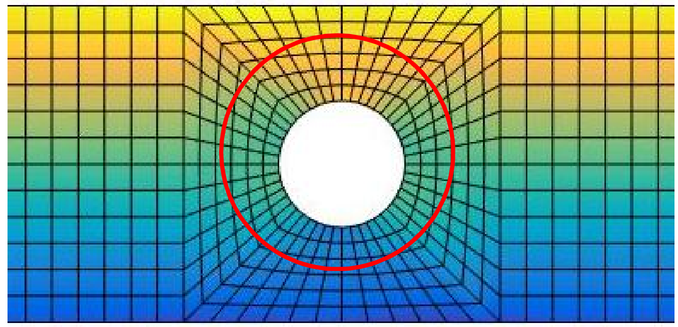


2.5 Where is the most likely location for failure if the material is ductile? Does the predicted failure depend on the assumption of plain stress or plain strain?

We will take the help of contour plots taken in Abaqus and those taken in Matlab for better depiction of stress values. In case of ductile materials (assuming that this is a ductile material), VonMises failure theory (which is an adaptation of Distortion Energy Theory) is used to predict failure location.

The region with the maximum values of the Von-Mises Stresses is likely to fail first.





- As it can be seen from the images, the regions (highlighted as white and red respectively) which develop the highest VonMises stresses are above and below the hole. These regions are most likely to fail first in this type of loading scenario. The failure will be along the Y axis through the hole in the center.
- The type of failure would be elastic plastic failure assuming that the material is a metal. Moreover, for most of the metals, the compressive strength is often less than the tensile strength. Hence, the beam is more likely to fail in compression at the bottom side of the hole.
- Another important fact to keep in mind that failure is never because of Stress. It is always because of Strain, Stress is a factitious kind of concept. So simple answer is where you are finding maximum strain that is the point of failure. The material will be strained most where there is highest stress. Hence, finding the region with maximum stress is likely to give you the location of failure which in this case is along the vertical axis of the hole in the beam.
- Does the predicted failure depend on the assumption on plane stress or plane strain? No, it does not depend on the assumption of plane stress/plane strain. As this is an isotropic material, it exhibits same mechanical properties in all directions. The failure location depends on the loading conditions and boundary conditions.

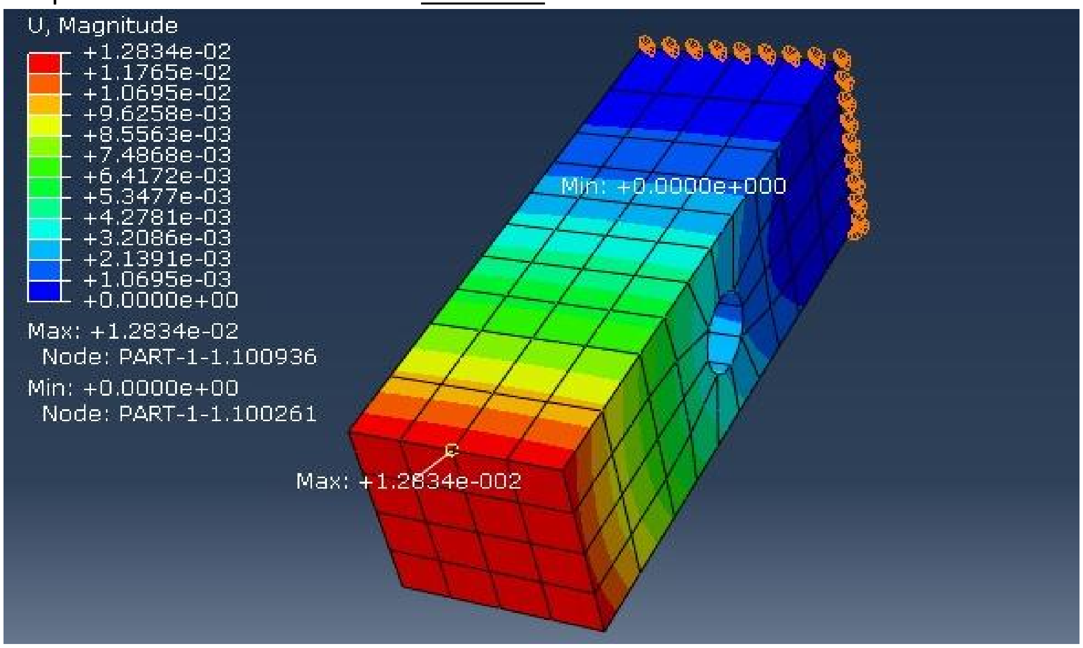
  The VonMises stresses observed at the location of hole are slightly different in values but the maximum stress is always found at the same location.

3. ABAQUS Solution

3.1. Plots of the displacements for two different element sizes

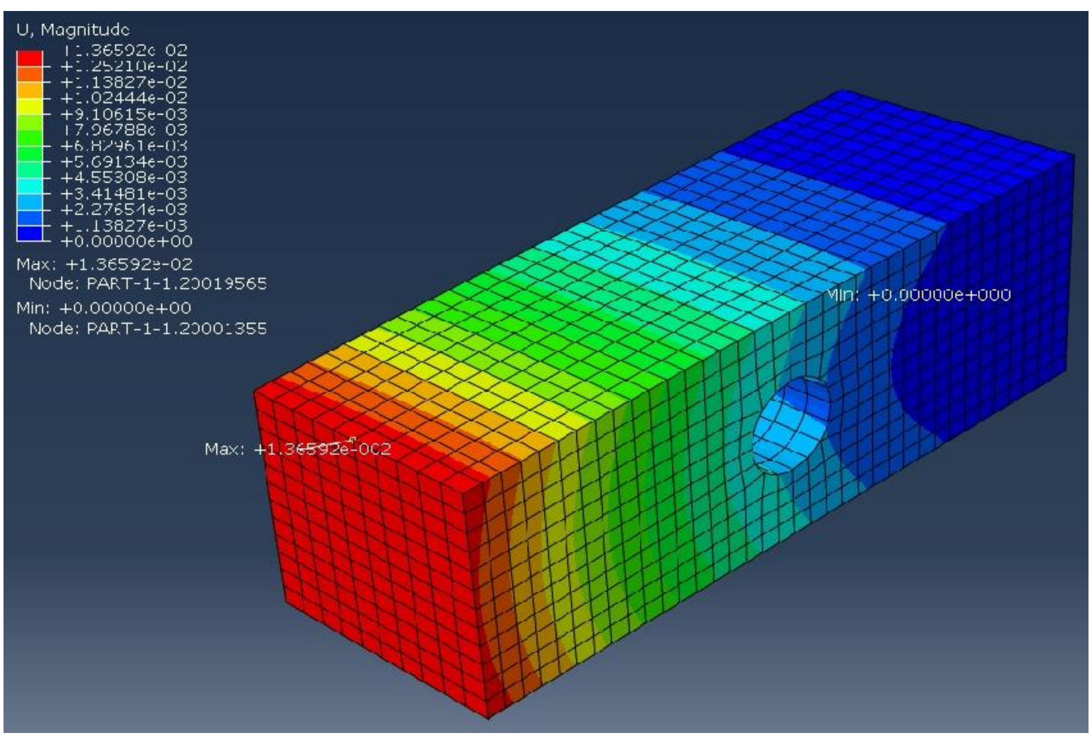
In the first case, the geometry is meshed with only 960 nodes and 144 hexahedron 2nd order elements. For any type of stress analysis problem, it is always better to use 2nd order elements as they yield much better results compared to 1st order elements. The boundary conditions are also shown here.

Maximum displacement for coarse mesh: 12.83mm



For the second case, the geometry is meshed with 4160 elements and 19860 nodes for 2nd order hexahedral elements. Maximum displacement is found at the same location.

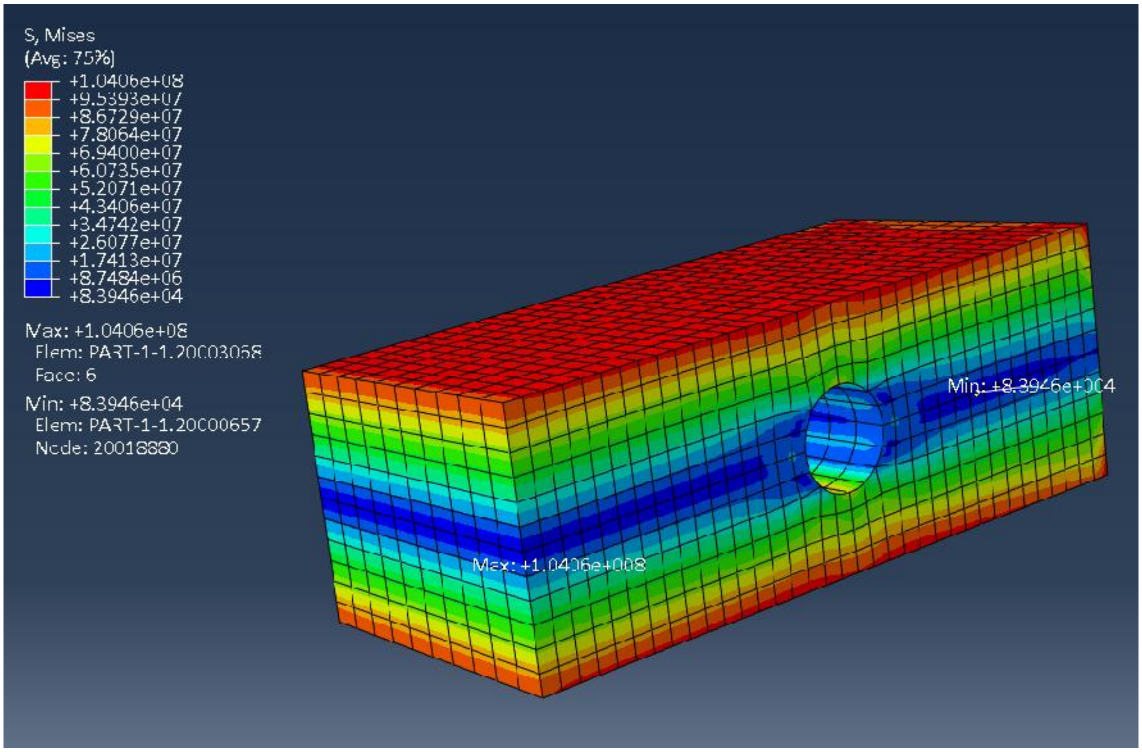
Magnitude for maximum displacement = 13.528mm



3.2. Plots of the stresses for two different element sizes

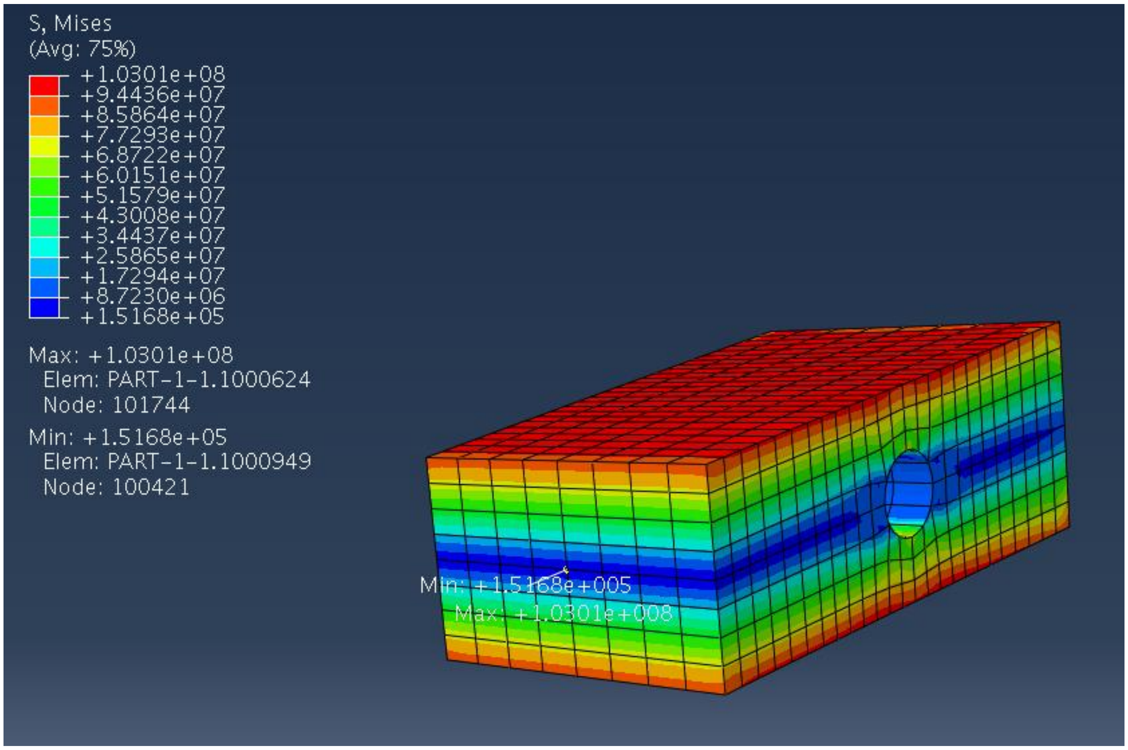
Von Mises Stresses for Quadratic Fine Mesh Size

Maximum Von Mises Stress = 1.0406e+08 N/m^2



Von Mises Stresses for Quadratic Coarse Mesh Size

Maximum Von Mises Stress = 1.0301e+08 N/m^2



3.3 Comparison of solutions between linear and quadratic elements

In order to compare solutions with different types of elements to check which elements yield better results for same number of elements, we need to mesh the model using the exactly the same mesh pattern with same no. of elements, however in the first case as 1st order elements (linear) while in the second case as 2nd order elements (quadratic). The following images show the results.

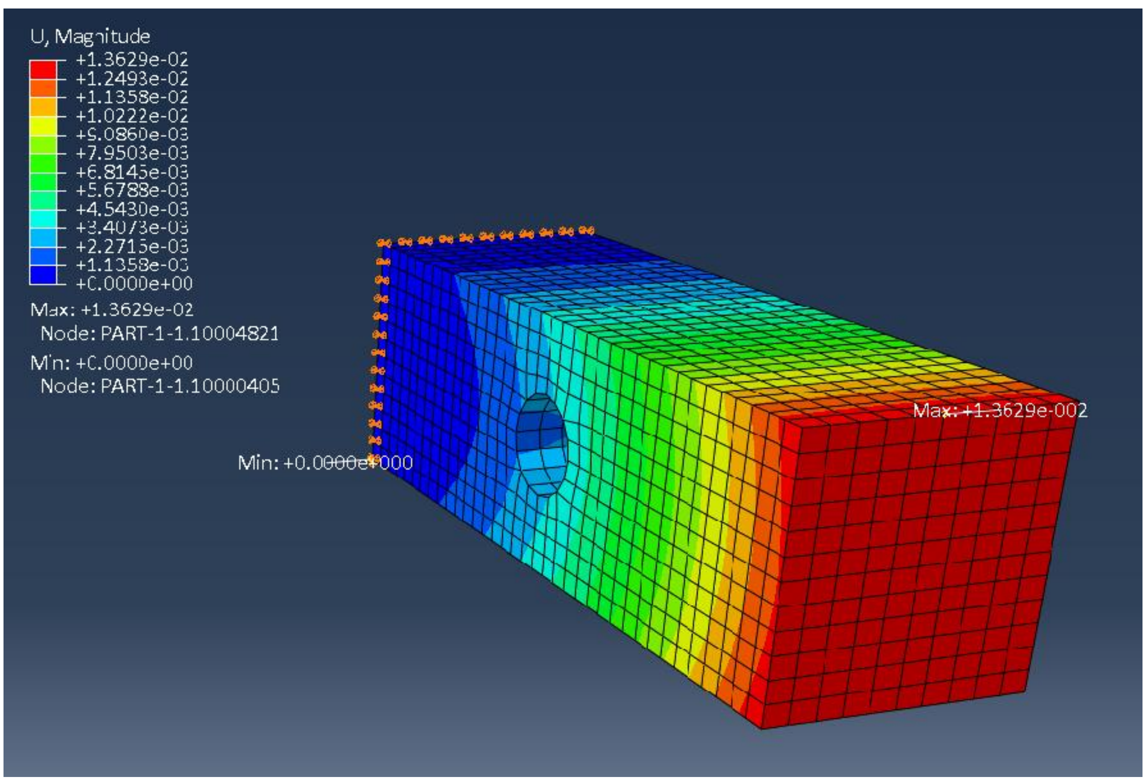
For hexahedral elements, even linear elements yield fairly good results. However, in case of a restriction on number of elements, we should always prefer 2nd order (quadratic) elements because of their higher accuracy in analyzing stress analysis problems.

There is one more point I will make here. If we compare the results using tetrahedral elements, we find that the quadratic tetra elements give a far better result compared with linear tetra elements. However, linear tetrahedral elements don't give a very reliable result and lack the accuary.

1st Order (Linear) Elements:

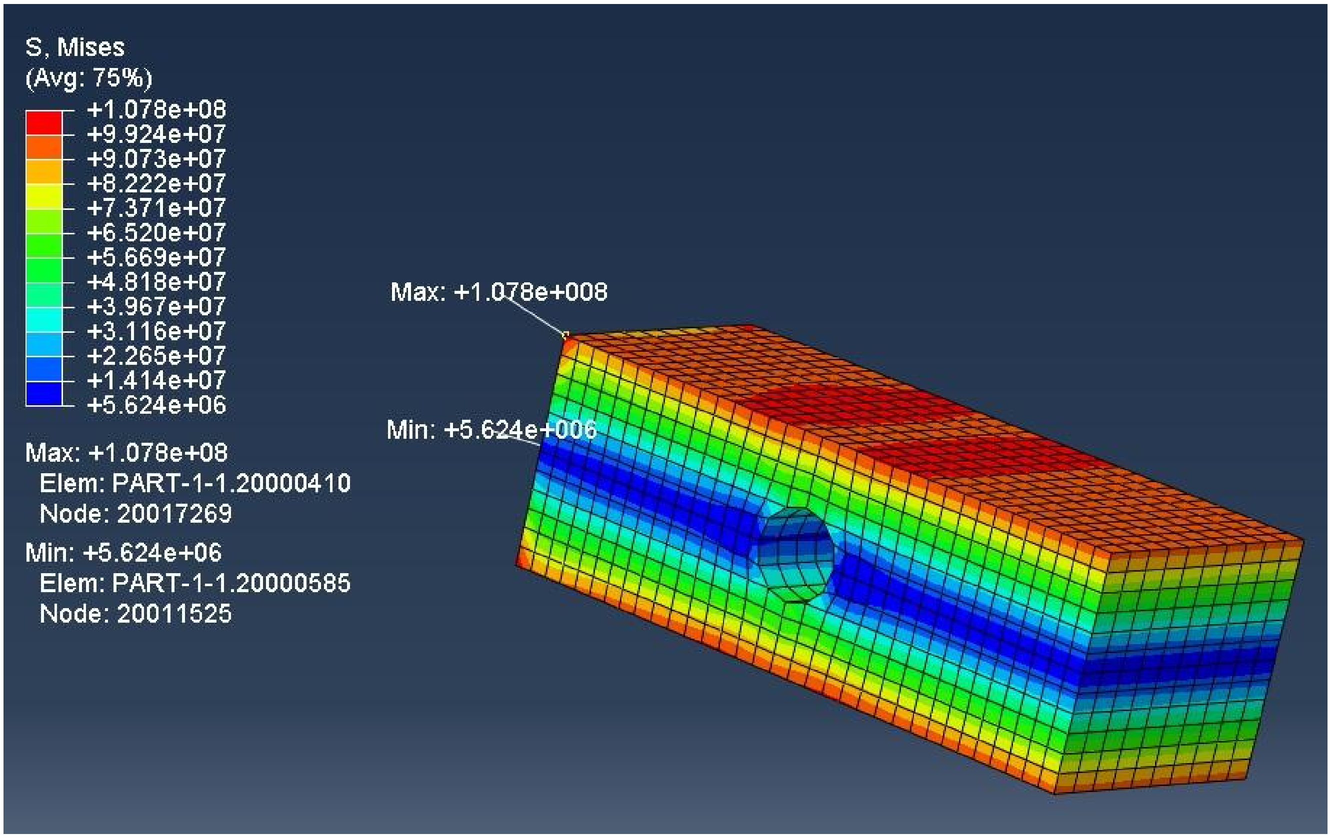
In the first case, the model was meshed with 4160 hexahedral 1st order elements having a total of 5192 nodes. The load traction was applied in the form of pressure normal to the surface and varying as t = 200y, with maximum values of 100Mpa and -100Mpa at y=0.5 and y=-0.5 respectively. The boundary conditions fixed support at y = -0.5 on the left edge and roller supports on the YZ plane above the fixed supports so that these nodes are free to move in y direction.

The following image shows the displacement contours obtained.

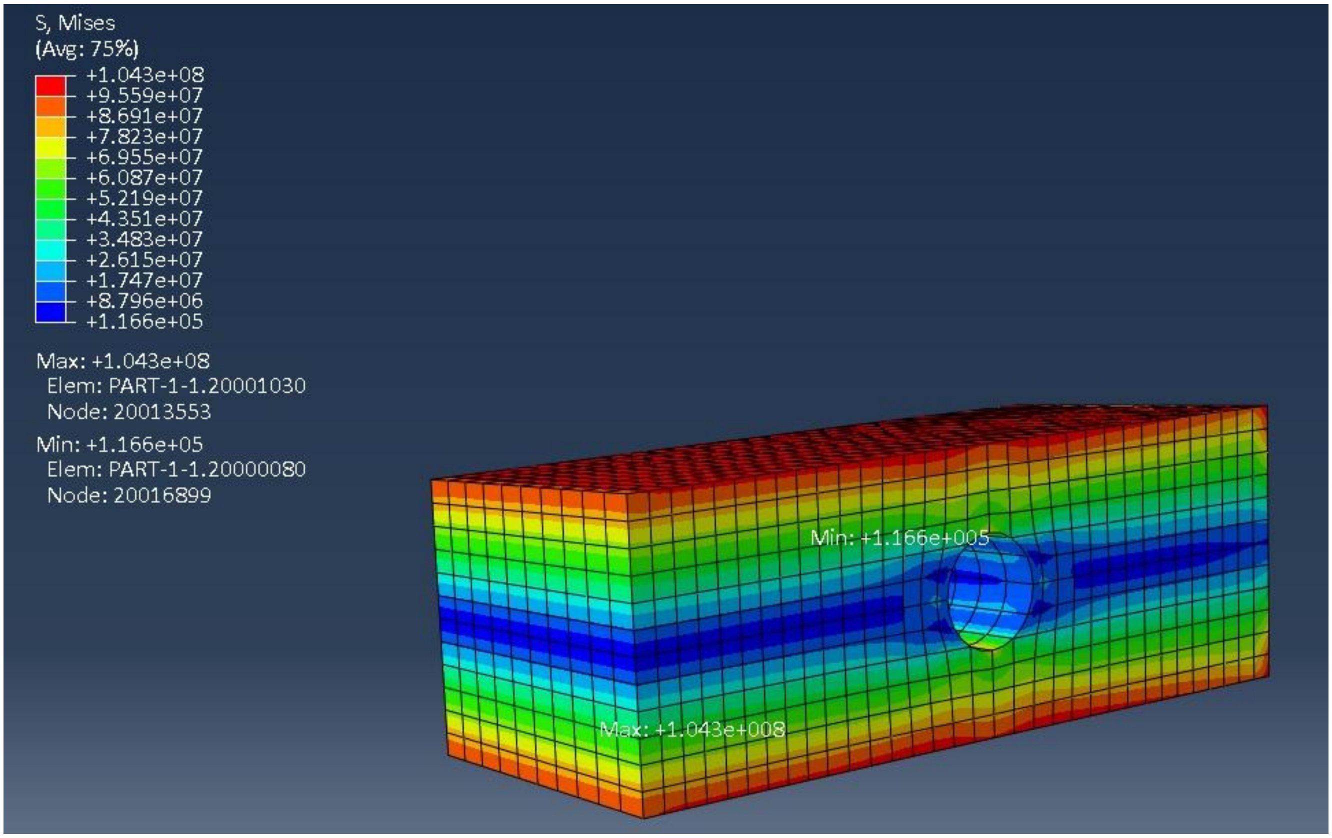


Maximum displacement for linear hexa elements = 1.3629e-02 m = 13.62 mm

The maximum displacement obtained for linear hexa elements is 13.62 mm which is fairly accurate.



2nd Order (Quadratic) Elements:

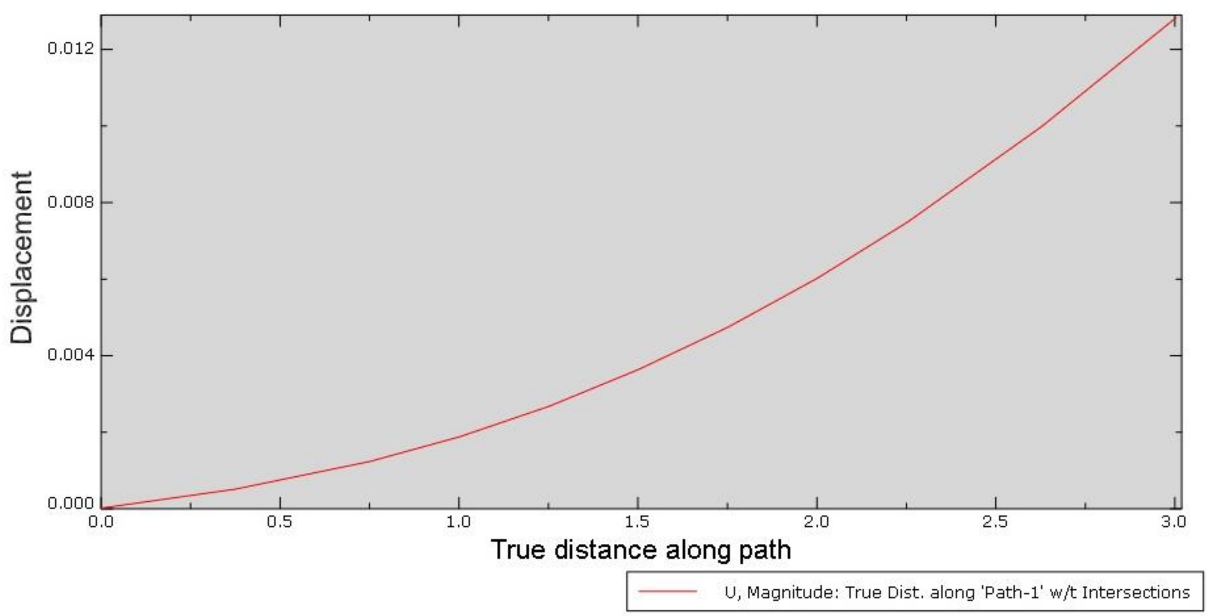


In the 2nd case, the model was meshed with 4160 hexahedral 2nd order elements with 19680 nodes.

It can be seen that the Quadratic solution converges faster than linear solution and provides more accurate results than linear solution. Quadratic solution should always be preferred in case of geometries with curvatures because of their higher rate of convergence than linear elements because geometry can be captured in a much better way.

3.4 Abaqus Solution Justification

1. The maximum displacement shown by the Abaqus solution is 13.62 mm. This value of displacement converges perfectly with both Analytical and Matlab solutions.
2. The problem is similar to that of cantilever beam with moment applied at the end point and the solution of displacement can be expected in the parabolic form along the length of the beam. The following plot shows the variation of displacement along the length of the beam. It can be seen that the graph follows the parabolic curve exactly and hence this justifies our solution.



All these points justify our Abaqus solution.