CITY COLLEGE
CITY COLLEGE OF NEW YORK

HOMEWORK 6:
FEM Project 1.0

ME 371 Computer Aided Design
Section: 3362
Dr. Savvas Xanthos

Submitted by: Pradip Thapa
Number # 23
Date: March 28, 2011

1. Introduction of FEM

The finite element method (FEM), sometimes referred to as Finite Element Analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. A boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called *Field* problems. The field is the domain of interest and most often represents a physical structure. The *Field Variables* are the dependent variables of interest governed by the differential equation. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity etc.

A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS

- Define the geometric domain of the problem.
- Define the material properties of the elements.
- Define the physical constraints (boundary conditions).
- Define the physical constraints (boundary conditions).
- Define the element connectivity (mesh the model).
- Define the element type(s) to be used.
- Define the geometric properties of the elements.
- Run the model
- Post processing the result.

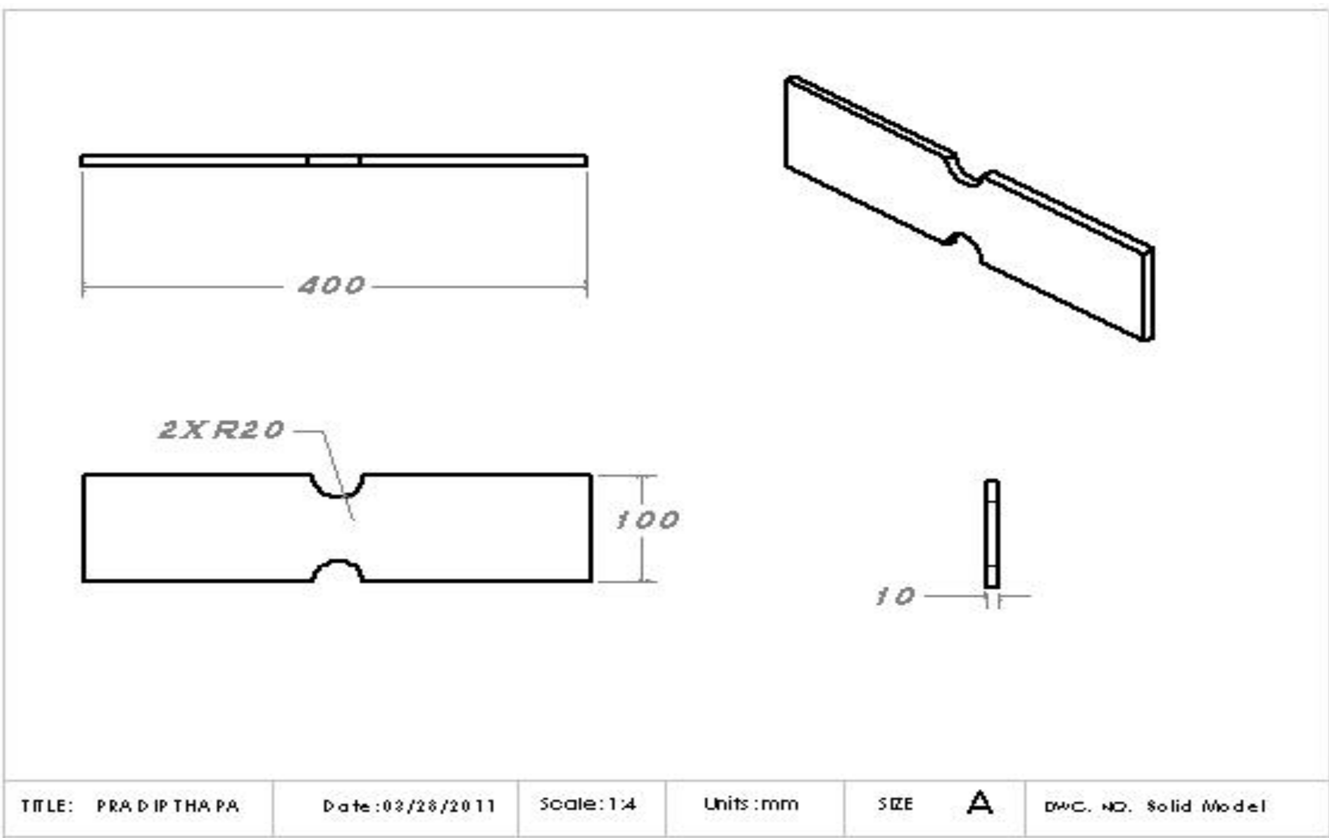
2. Overview of FEM in Stress Analysis:

The field variables for stress analysis is physical element nodal displacement and this objective of this project is to understand the application of FEM and its procedures to use in Solidwork 2006 SP3 COSMOS for linear stress analyses on solid models and simultaneously compare with its know analytical solution. This comparison checks the validity of the solution provided by the computer which is an approximation made by computer with finite computation. The convergence test is to quick method to check the validation of the result provided by the software.

The method involves cutting a structure into a several elements (meshing), which describes the behaviors of each element in simple way then reconnecting elements at nodes which results in a set of simultaneous algebraic equations. In stress analysis these equations are equilibrium equation of the nodes which are solved by computer in an iterative process. As describe the element size the order of polynomial is very important in the sophistication of the result that it. If the element size is changed this is h-refinement where the elements size are smaller and the truncation error in interpolation is reduce and approximation is more refine. Another method of refinement is P-refinement; this approximation is also called polynomial order refinements where the order of spline joining the nodes is not linear which also reduce the approximation error in interpolation.

3. Geometric and Material Parameter of Model:

Material Parameter	Value
Assign Material	Steel 1020
Young Modulus	200 GPa
Poisson ratio	0.29



4. Boundary Condition:

Model	Value
Restrain	Yes
Load	100 Newton

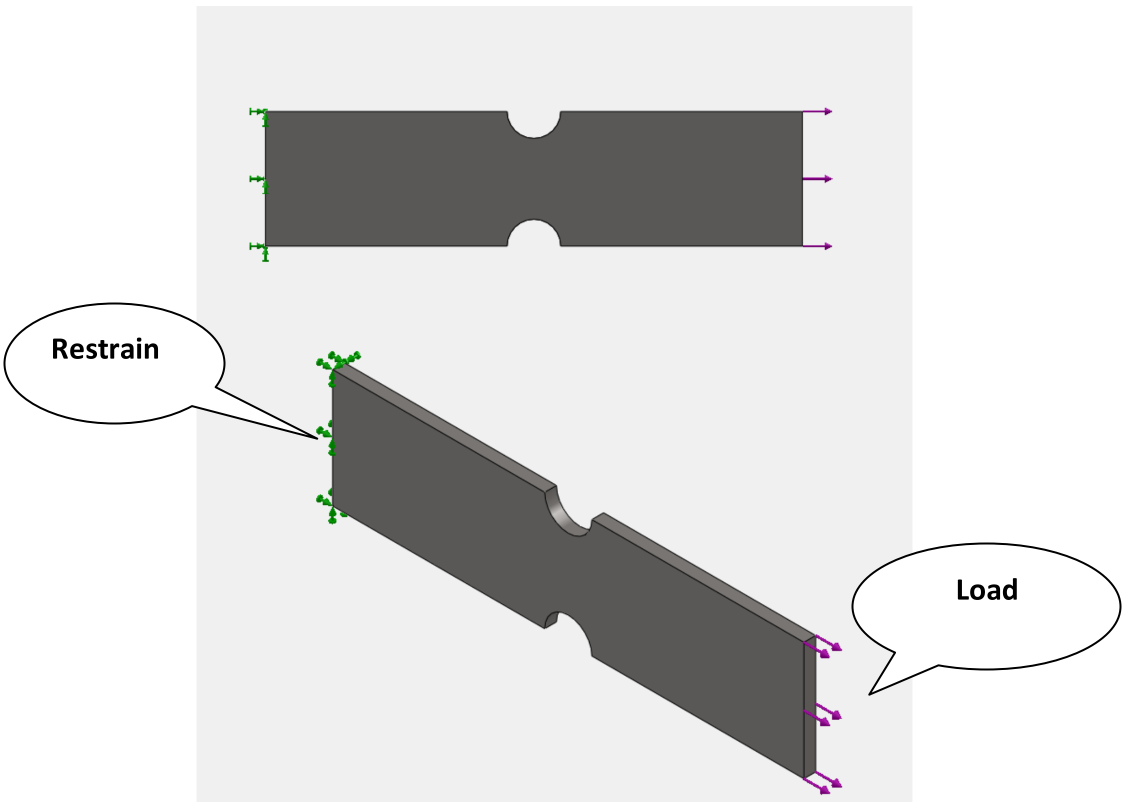


Figure 1 Boundary Condition

5. Analytical Solution:

The theoretical nominal stress was calculated by two different methods in order to compare the results obtained from Solidworks Simulation.

$$\sigma = \frac{F}{A} \rightarrow E * \varepsilon = \frac{F}{A} \rightarrow E * \frac{\delta}{L} = \frac{F}{A}$$

$$\delta = \frac{FL}{AE}$$

where,

$$L = \ total\ length\ of\ the\ plate, (m)$$

$$A = \ cross - sectional\ area, (m^2)$$

$$E = Young's\ Modulus\ of\ Elasticity\ \left(\frac{N}{m^2}\right)$$

$$F = Total\ Force\ applied\ (N)$$

According to stress concentration theory, the stress in the notches, is higher than the nominal stress predicted by the area reduction, therefore a new factor is required to consider compensating this value which is called stress concentration factor. The nominal stress in the reduced area where the notches present is given by:

$$\sigma_{nominal} = \frac{F}{A_{reduced}}$$

$where$

$$A_{reduced} = reduced\ area\ where\ the\ notches$$

$$= (H - 2 * R) * T$$

$The\ maximum\ stress\ at\ notches\ is\ given\ by$:

$$\sigma_{max} = K_e * \sigma_{nominal}$$

$where,$

$$stress\ concentration\ factor\ (K_e) = 3.065 - 3.370\left(\frac{2R}{H}\right) + 0.647\left(\frac{2R}{H}\right)^2 + 0.658\left(\frac{2R}{H}\right)^3$$

$H = total\ width\ across\ of\ the\ section\ (m)$

$R = radius\ of\ each\ notch\ (m)$

MATLAB code:

```
clear all
clc

L=input('Enter the length of Model (mm): ');
L=L/1000;
R=input('Enter the radius of Notch (mm): ');
R=R/1000;
t=input('Enter the thickness of the Plate (mm): ');
t=t/1000;
h=input('Enter the height of the Plate (mm): ');
```

```
h=h/1000;
F=input('Enter the Applied Force (N): ');
E=input('Enter the Youngs Modulus of Material (GPa): ');
E=E*10^9;

% Area
Area=h*t;
Area_reduce=(h-2*R)*t;

% Stress
K_e=3.065-3.37*(2*R/h)+0.647*(2*R/h)^2+0.658*(2*R/h)^3;
sigma=F/Area;
sigma_nominal=F/Area_reduce;
sigma_max=K_e*sigma_nominal;
deflection=((F*L)/(Area*E))*10^9;


disp('Calculated value')
disp('The stress concentration factor: '),disp(K_e)
disp('The stress far away from the notch: (N/m^2) '),disp(sigma)
disp('The nominal stress at the notch: (N/m^2) '),disp(sigma_nominal)
disp('The maximum stress  at the notch: (N/m^2) '),disp(sigma_max)
disp('The maximum deflection : (nm) '),disp(deflection)
```

Calculated Value

	Value	Unit
The stress concentration factor	1.8626	NA
The stress far away from the notch	100.00	KPa
The nominal stress at the notch	166.67	KPa
The maximum stress at the notch	310.44	KPa
The maximum deflection	200	nm

6. Procedure:

Step 1: Creating the model as the geometric dimension was provided.

Step 2: Material AISI 1020 was assigned as the material which has Young's Modulus of Elasticity 200GPa.

Step 3: A new study was created for static analysis.

Step 4: Then the boundary condition was defined as left end of the model was restrain with fixed support and the right end face was loaded with force 100N. As shown in Figure 1. The green arrow represents the fixed support and the pink arrow represents the face where force is applied and the arrow head represent the direction of force applied. Then, several studies were performed for creating the mesh and running the model for stress analysis. Studies performed:

Study 1: The first analysis was done by applying three different kind of mesh in draft quality (which means Polynomial Order, P=1), where three different mesh size was selected. Coarse, medium and Fine which represent the different mesh size in descending order also called h-refinement.

Study 2: The Second analysis was done by applying three different kind of mesh in high quality (which means Polynomial Order, P=2), where three different mesh size was selected Changing the order from P=1 to P=2 is called P-refinement. Coarse, medium and Fine which represent the different mesh size in descending order also called h-refinement.

Study 3: The Third analysis was done by control on the upper and lower notch. The study was perform for high quality (P=2) and global mesh was kept for coarse, medium and fine and for each case, local control on the notch was applied setting up the local mesh size for coarse and fine.

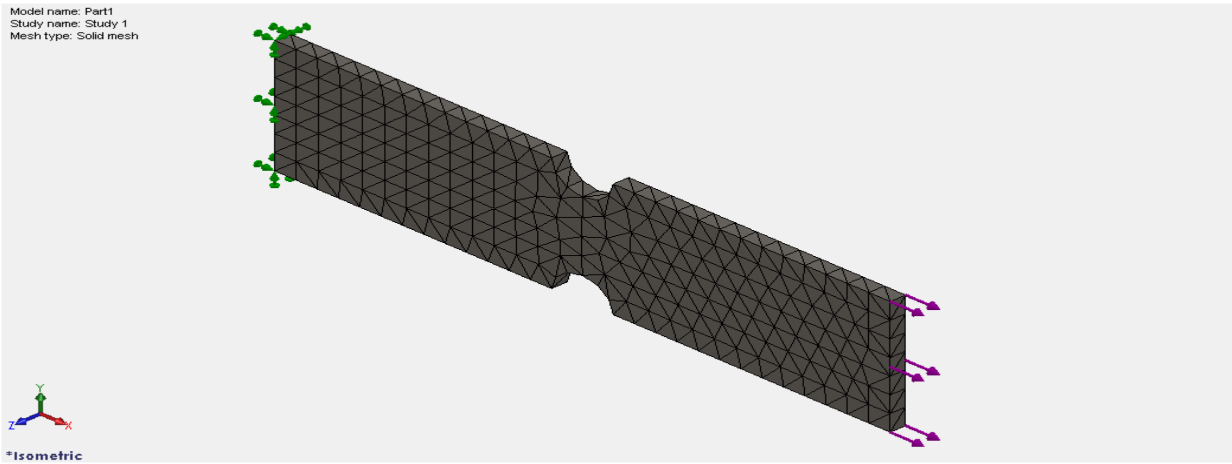
Study 4: The forth analysis was done by control on section of the upper and lower notch. The study was perform for high quality (P=2) and global mesh was kept for coarse, medium and fine and for each case, local control on the section on the notch by applied by setting up the local mesh size for coarse and fine. To create the section on the notch, the split part was created by splitting the line and projecting it to the curve of the notch.

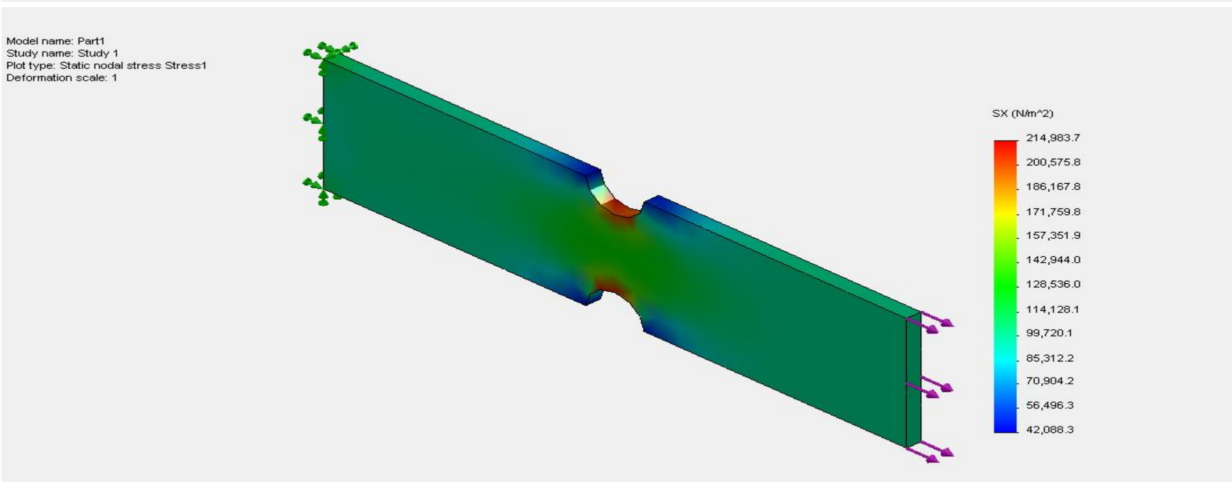
Study5: The fifth analysis is h-adaptive, the mesh size and the regions of the notch is selected to improve the results manually. The h-adaptive method identifies regions with high errors automatically and continues to refine them until the specified accuracy level or the maximum allowed number of iterations is reached. The sixth analysis is p-adaptive, the order of P was increases to P = 4, this increases the order of the polynomial to improve results in areas with high stress errors.

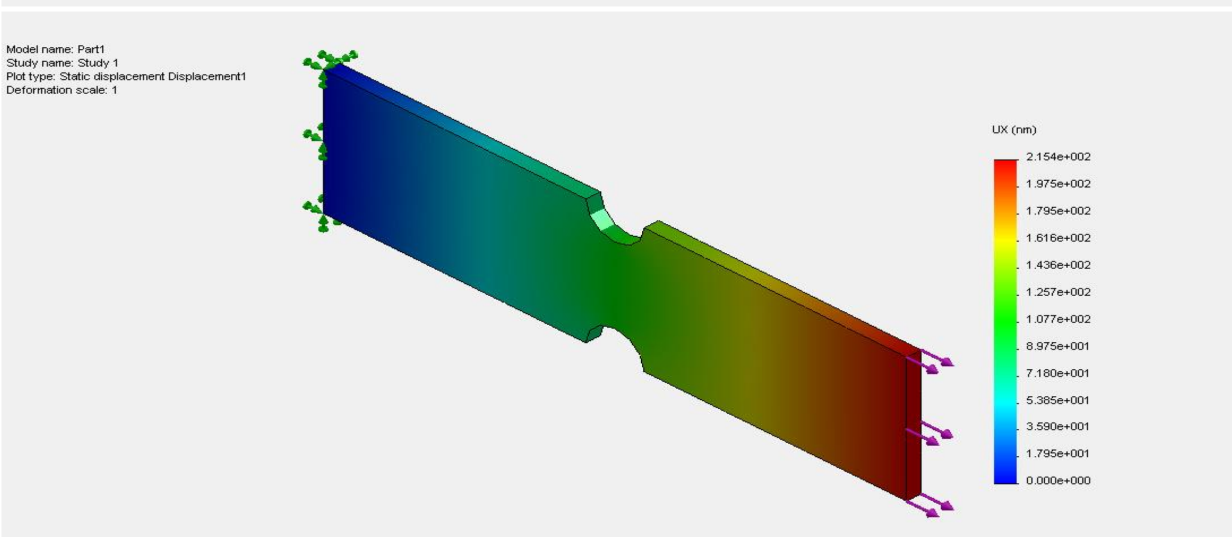
7. Cosmos Study:

7.1. Draft Quality

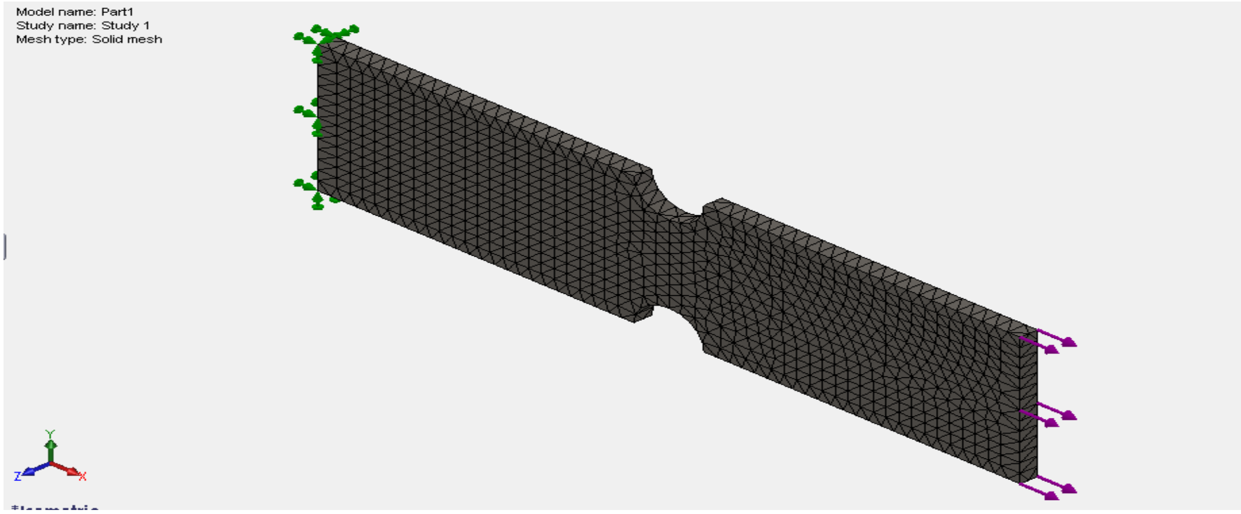
- Coarse

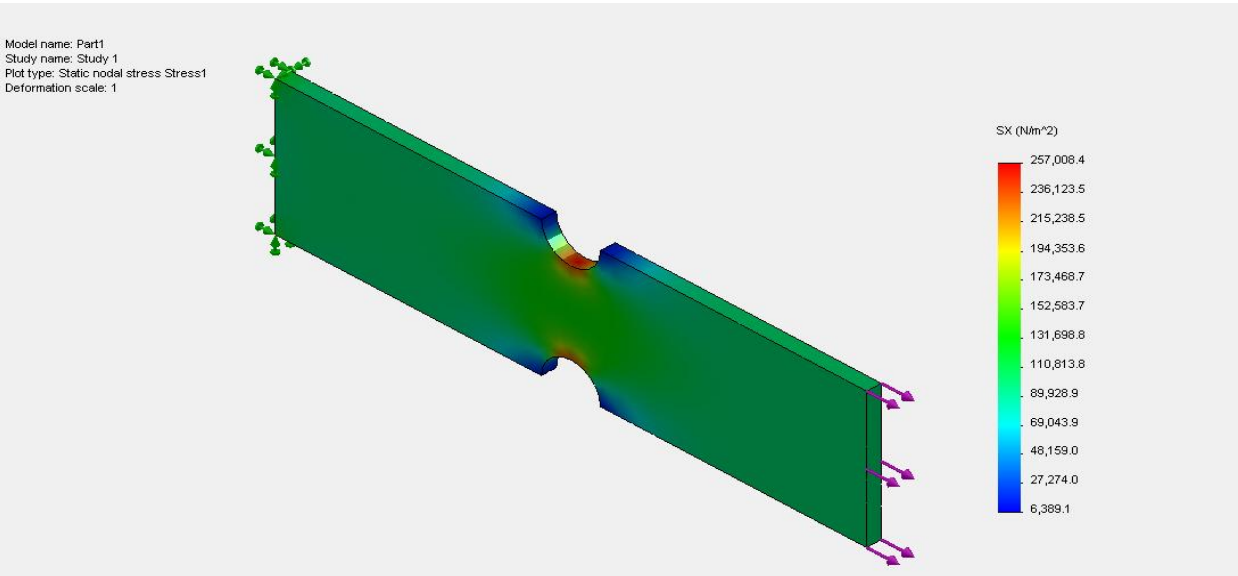


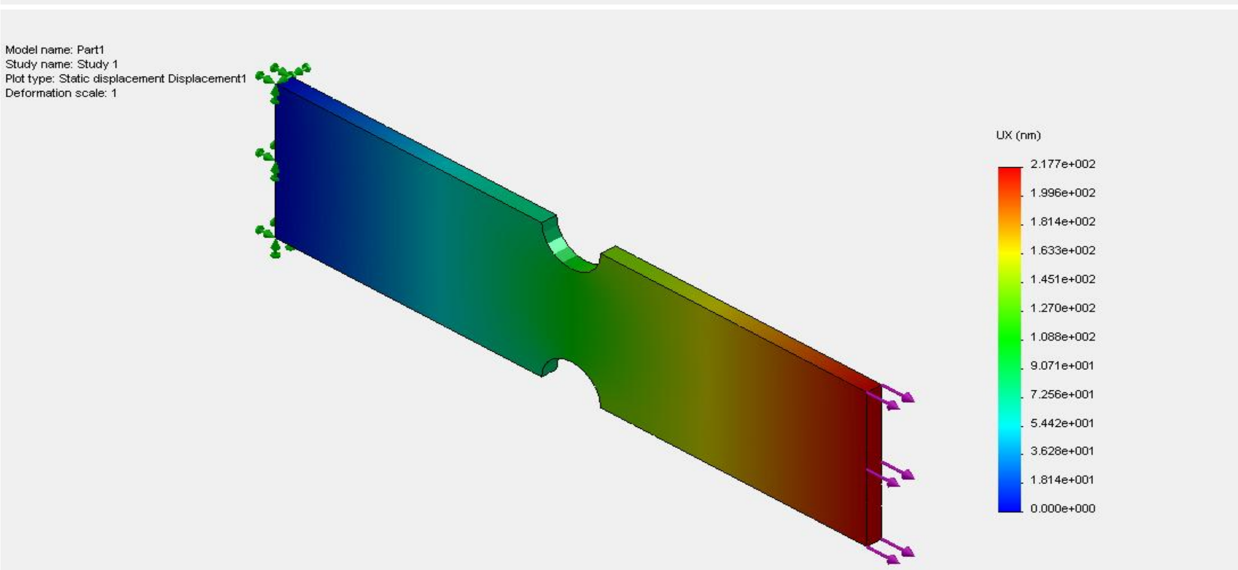




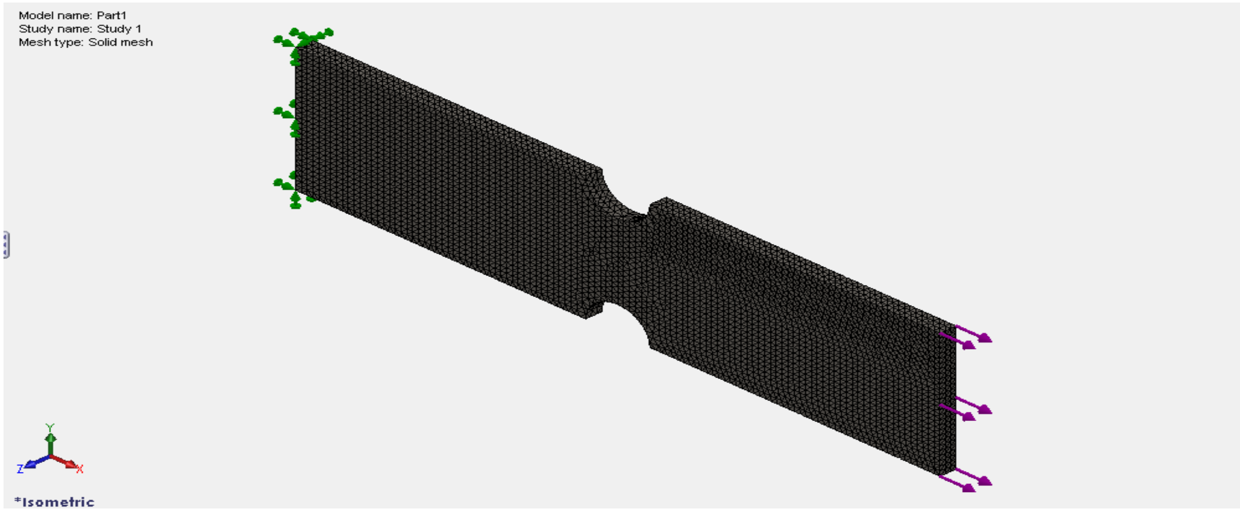
- Medium

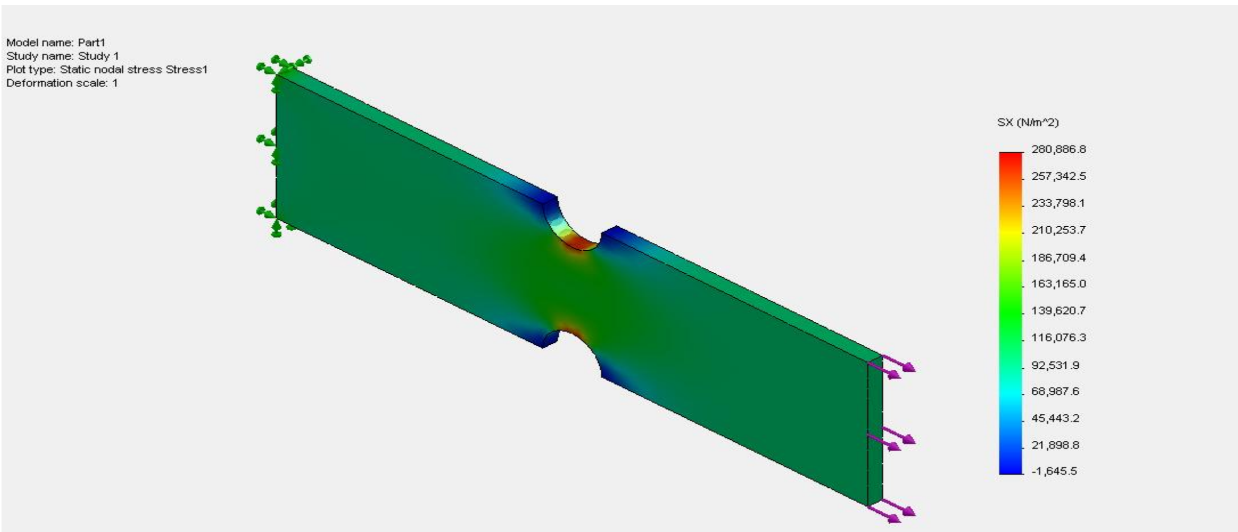

Model name: Part1
Study name: Study 1
Mesh type: Solid mesh

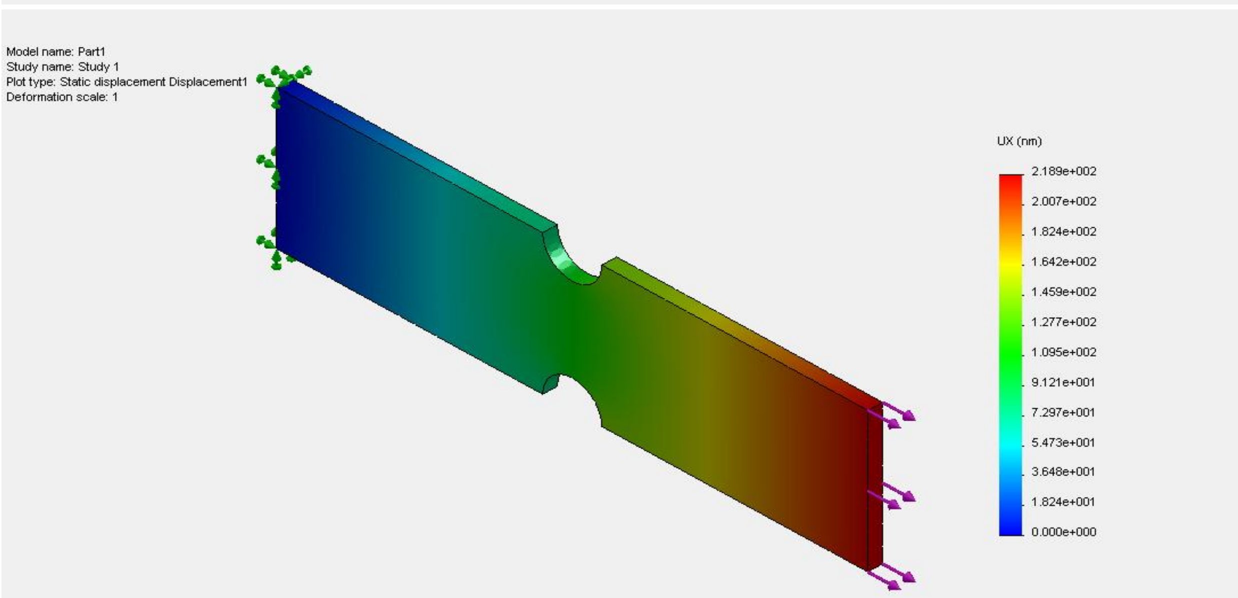

Model name: Part1
Study name: Study 1
Plot type: Static nodal stress Stress1
Deformation scale: 1
SX (N/m^2)
257,008.4
236,123.5
215,238.5
194,353.6
173,468.7
152,583.7
131,698.8
110,813.8
89,928.9
69,043.9
48,159.0
27,274.0
6,389.1


Model name: Part1
Study name: Study 1
Plot type: Static displacement Displacement1
Deformation scale: 1
UX (nm)
2.177e+002
1.996e+002
1.814e+002
1.633e+002
1.451e+002
1.270e+002
1.088e+002
9.071e+001
7.256e+001
5.442e+001
3.628e+001
1.814e+001
0.000e+000

- Fine

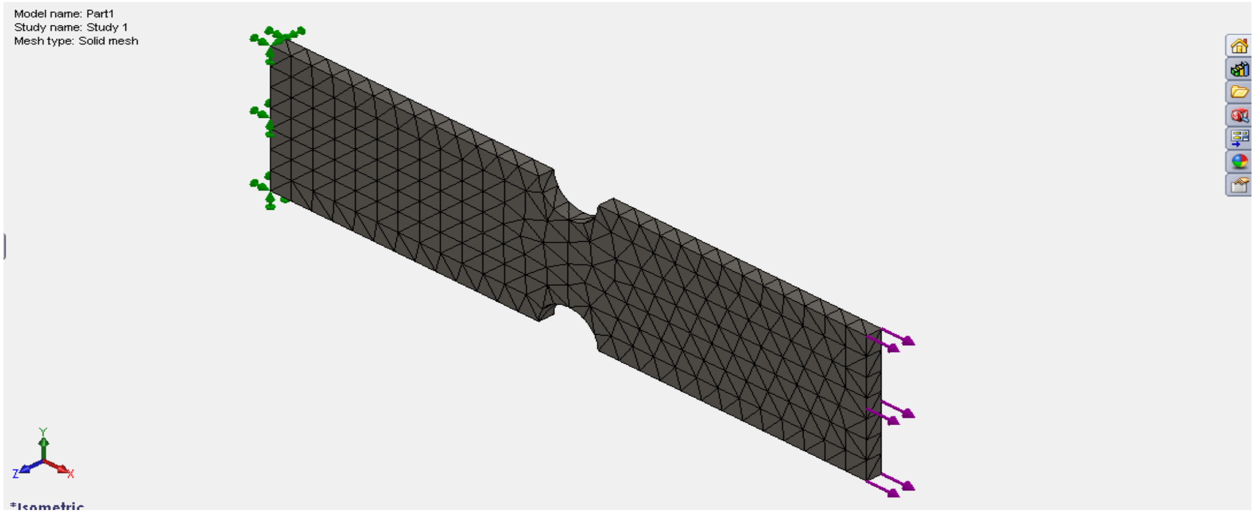

Model name: Part1
Study name: Study 1
Mesh type: Solid mesh
*Isometric

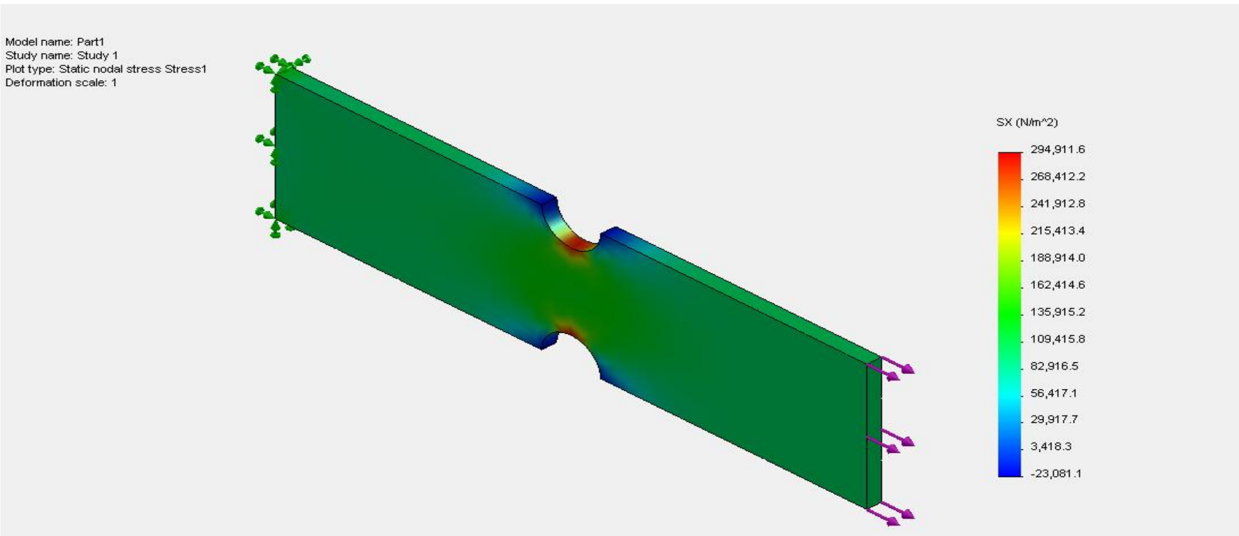

Model name: Part1
Study name: Study 1
Plot type: Static nodal stress Stress1
Deformation scale: 1
SX (N/m^2)
280,886.8
257,342.5
233,798.1
210,253.7
186,709.4
163,165.0
139,620.7
116,076.3
92,531.9
68,987.6
45,443.2
21,898.8
-1,645.5

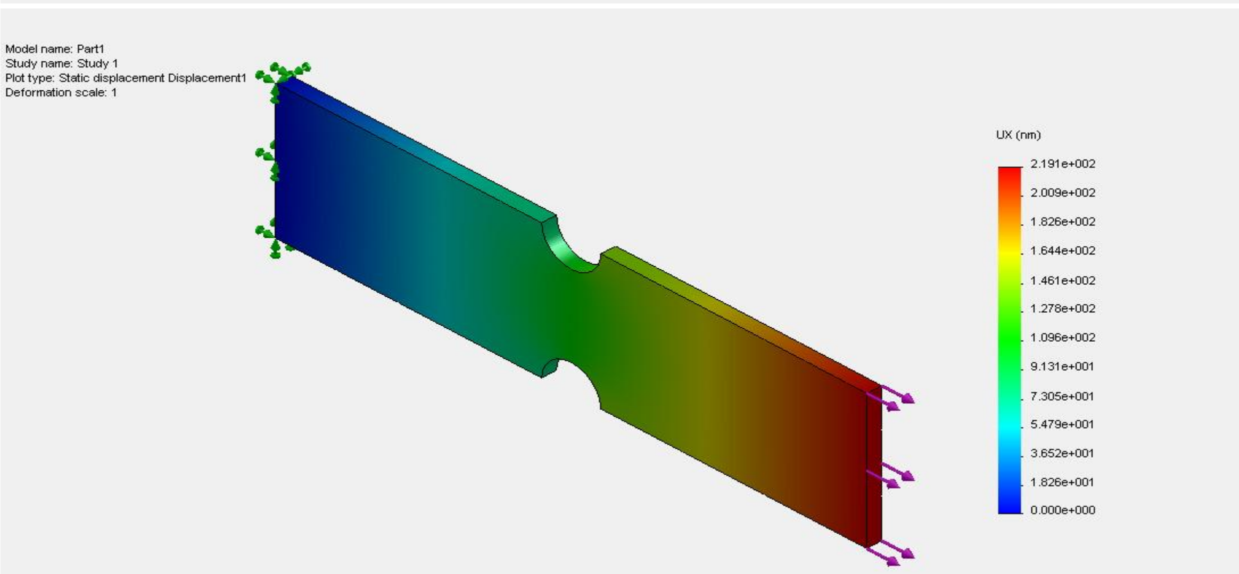

Model name: Part1
Study name: Study 1
Plot type: Static displacement Displacement1
Deformation scale: 1
UX (nm)
2.189e+002
2.007e+002
1.824e+002
1.642e+002
1.459e+002
1.277e+002
1.095e+002
9.121e+001
7.297e+001
5.473e+001
3.648e+001
1.824e+001
0.000e+000

7.2. High Quality

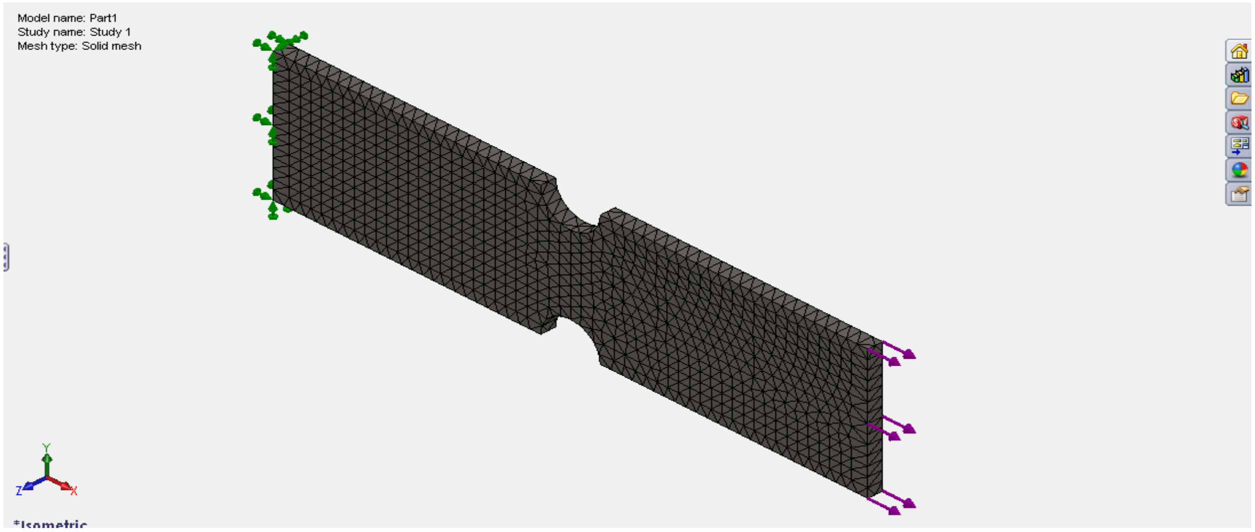
- Coarse

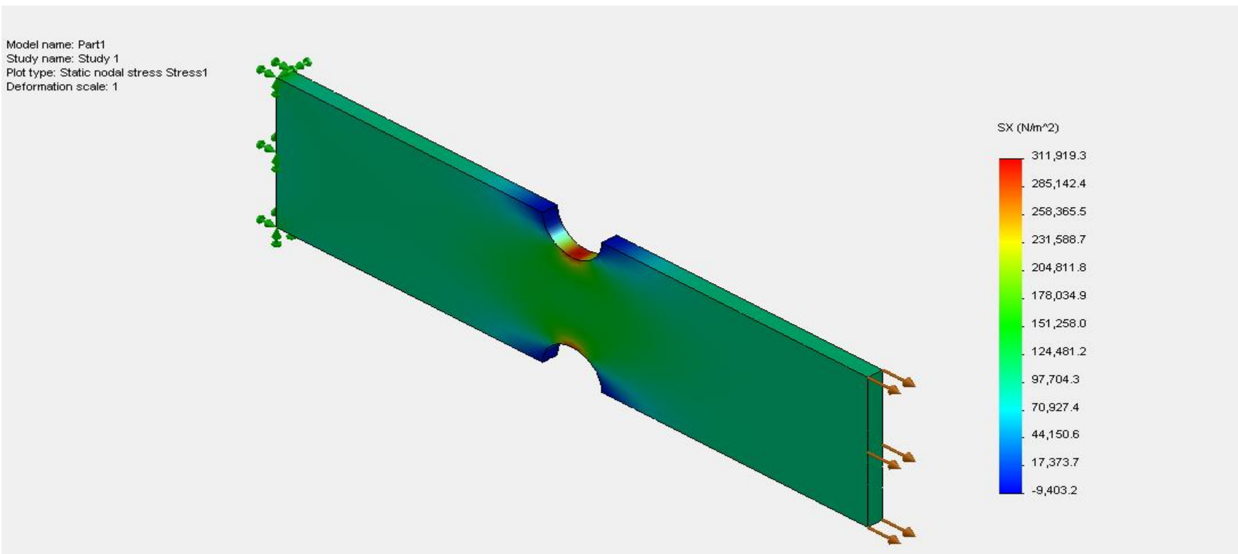


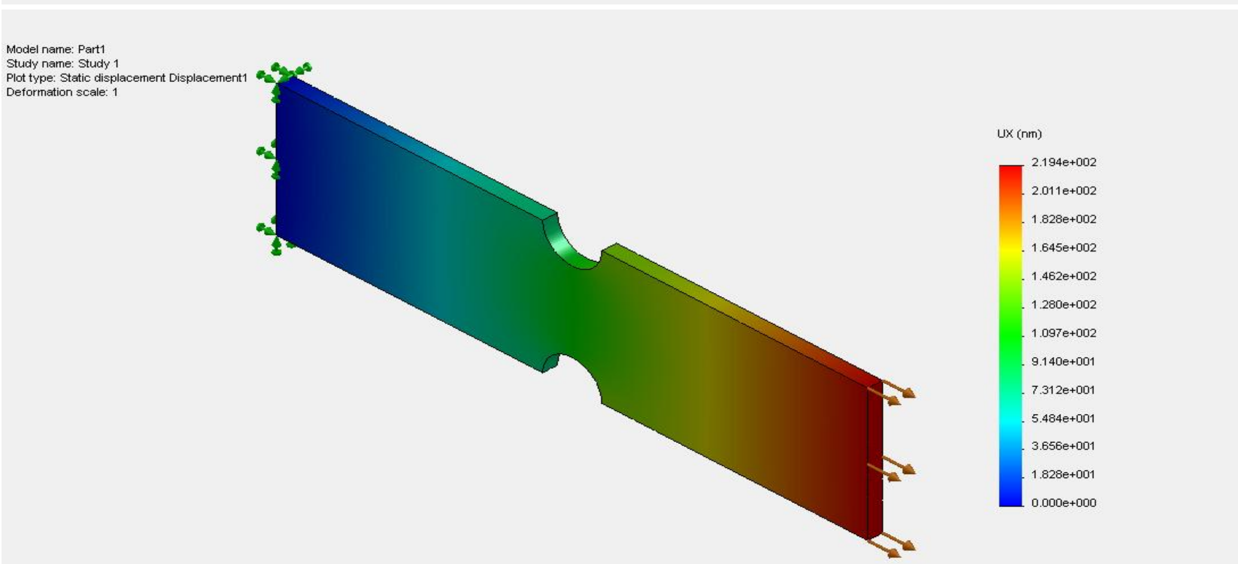




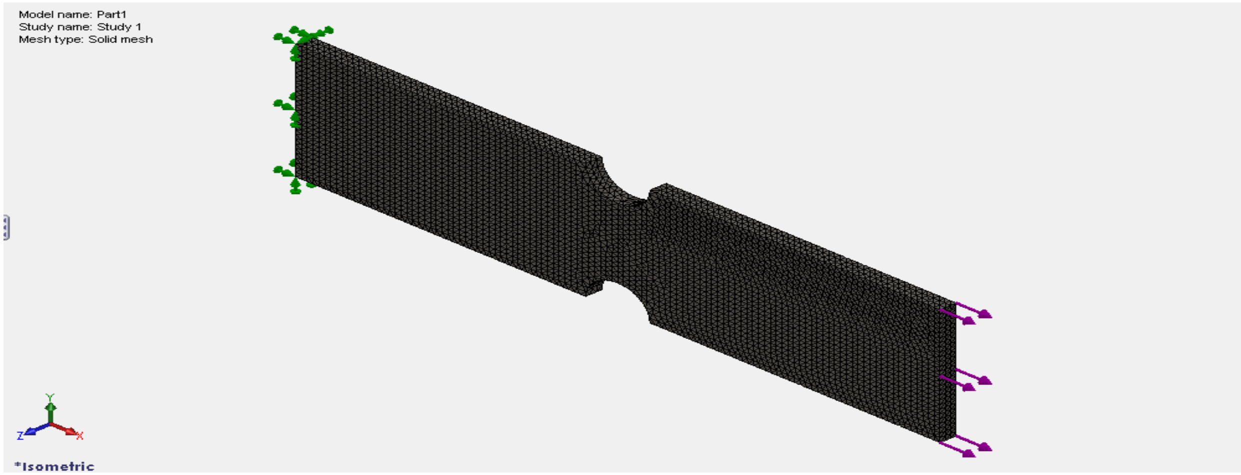
- Medium

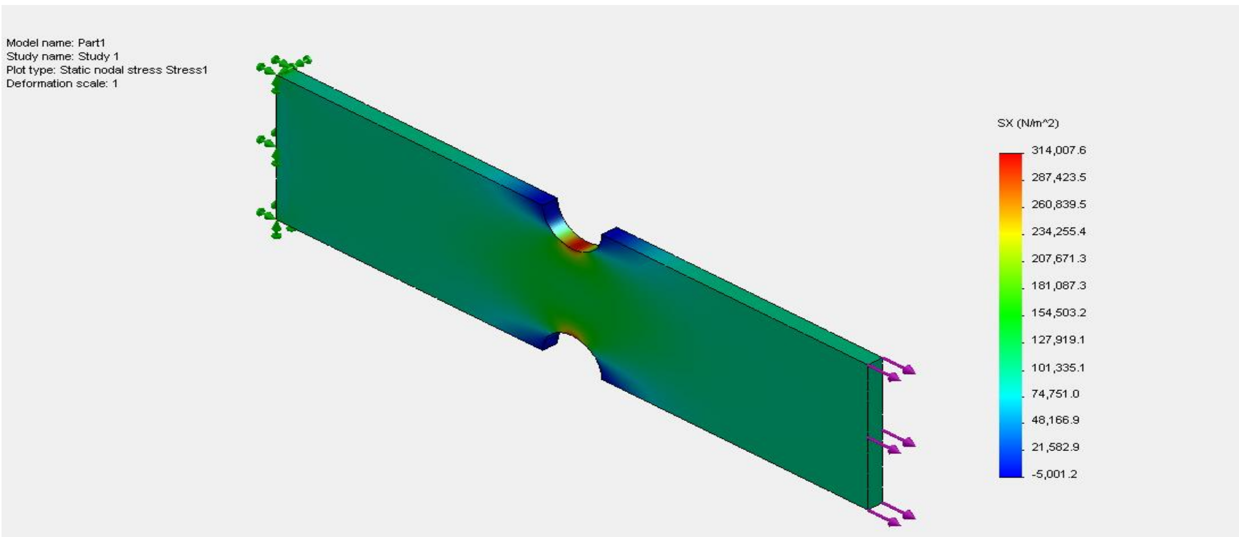

Model name: Part1
Study name: Study 1
Mesh type: Solid mesh
*Isometric

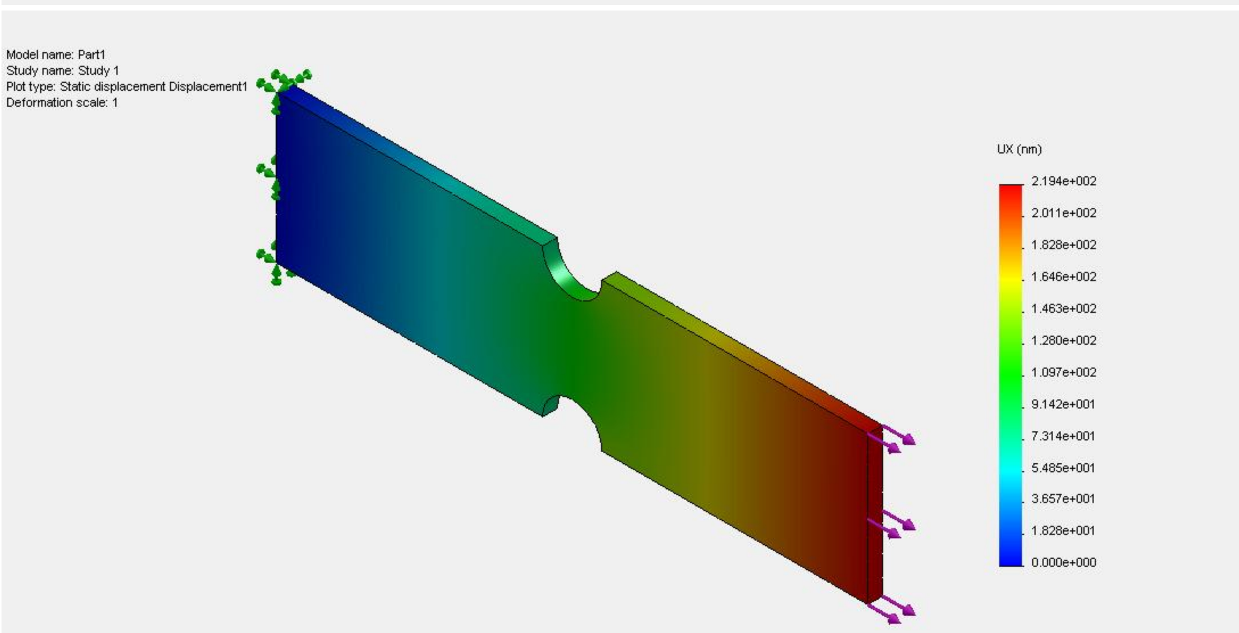

Model name: Part1
Study name: Study 1
Plot type: Static nodal stress Stress1
Deformation scale: 1
SX (N/m^2)
311,919.3
285,142.4
258,365.5
231,588.7
204,811.8
178,034.9
151,258.0
124,481.2
97,704.3
70,927.4
44,150.6
17,373.7
-9,403.2


Model name: Part1
Study name: Study 1
Plot type: Static displacement Displacement1
Deformation scale: 1
UX (nm)
2.194e+002
2.011e+002
1.828e+002
1.645e+002
1.462e+002
1.280e+002
1.097e+002
9.140e+001
7.312e+001
5.484e+001
3.656e+001
1.828e+001
0.000e+000

- Fine

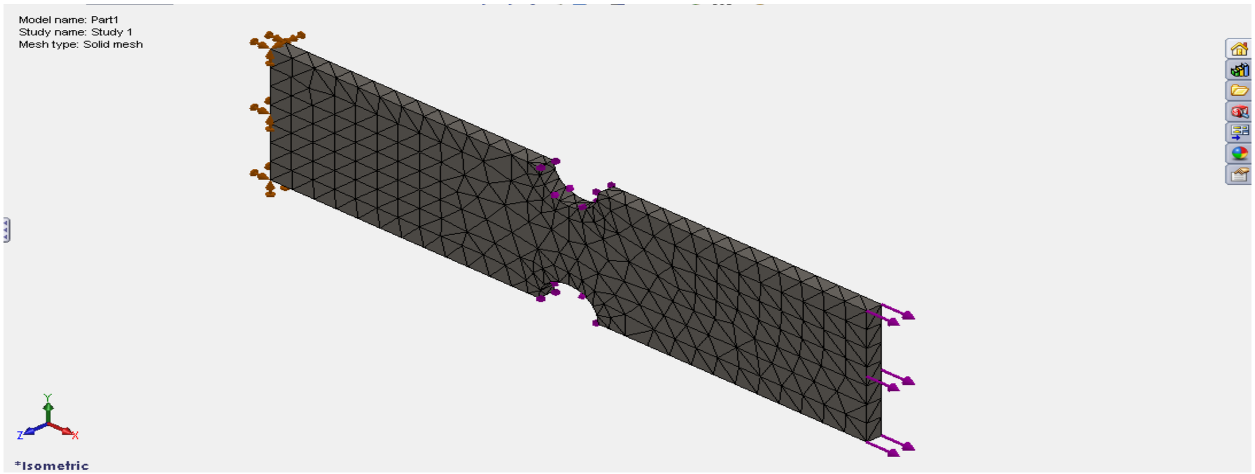

Model name: Part1
Study name: Study 1
Mesh type: Solid mesh
Y
Z
X
*Isometric


Model name: Part1
Study name: Study 1
Plot type: Static nodal stress Stress1
Deformation scale: 1
SX (N/m^2)
314,007.6
287,423.5
260,839.5
234,255.4
207,671.3
181,087.3
154,503.2
127,919.1
101,335.1
74,751.0
48,166.9
21,582.9
-5,001.2

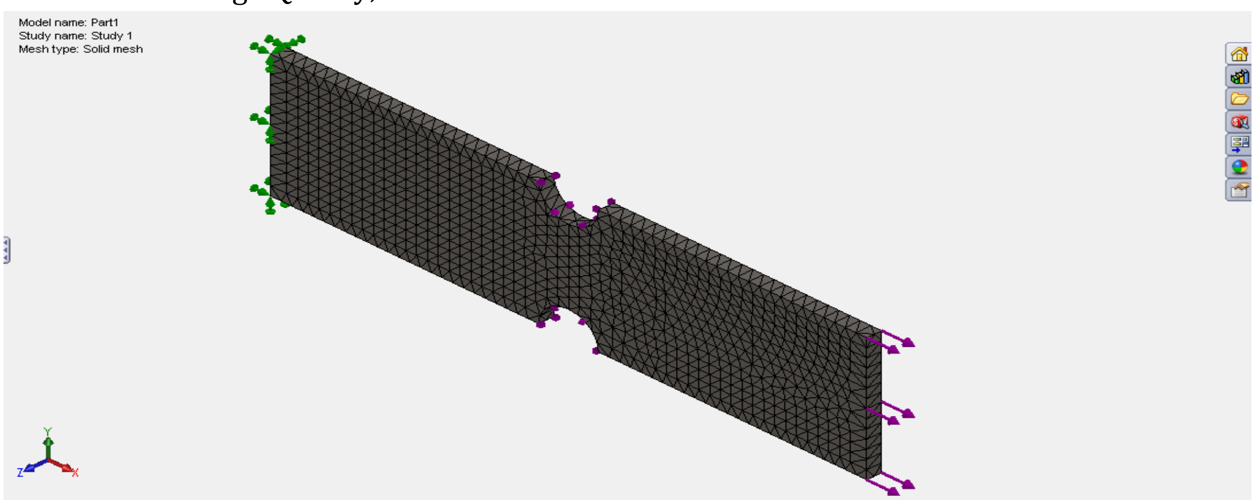

Model name: Part1
Study name: Study 1
Plot type: Static displacement Displacement1
Deformation scale: 1
UX (nm)
2.194e+002
2.011e+002
1.828e+002
1.646e+002
1.463e+002
1.280e+002
1.097e+002
9.142e+001
7.314e+001
5.485e+001
3.657e+001
1.828e+001
0.000e+000

7.3. Apply Control

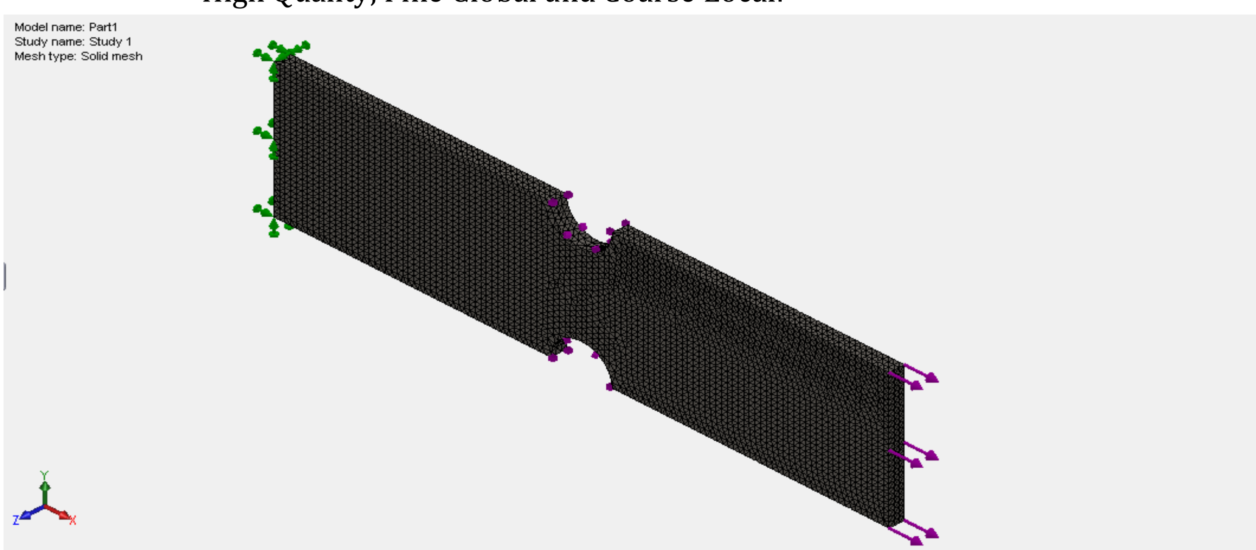
- High Quality, Coarse Global and Coarse Local:



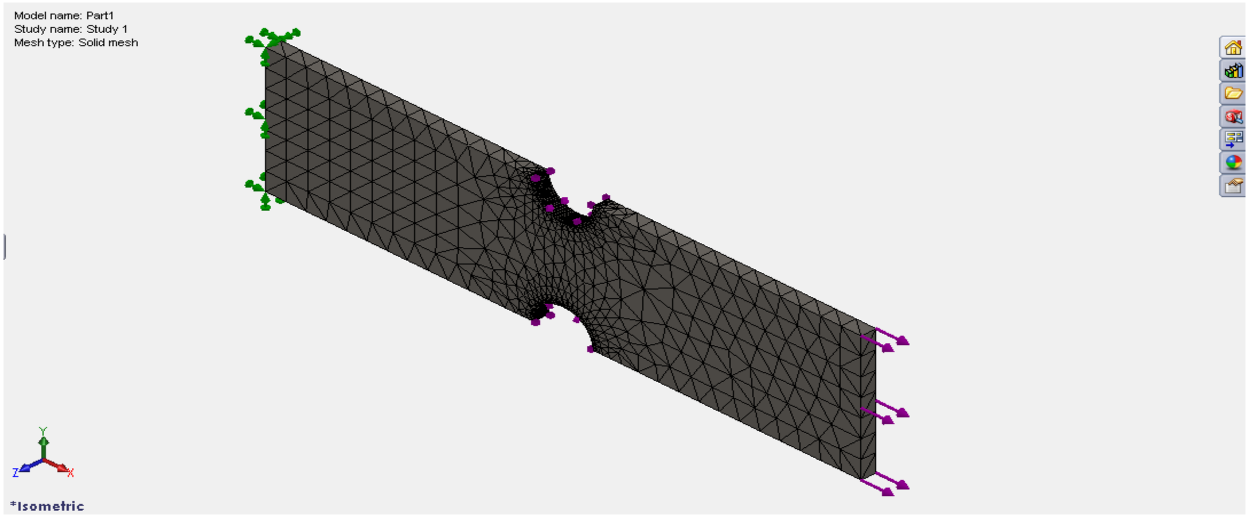
- High Quality, Medium Global and Coarse Local:



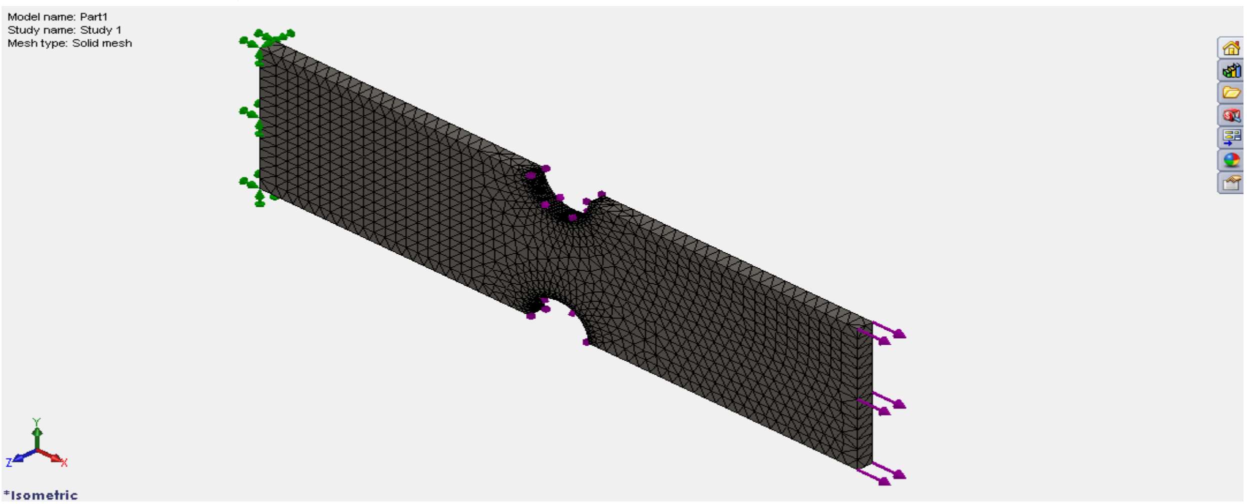
- High Quality, Fine Global and Coarse Local:



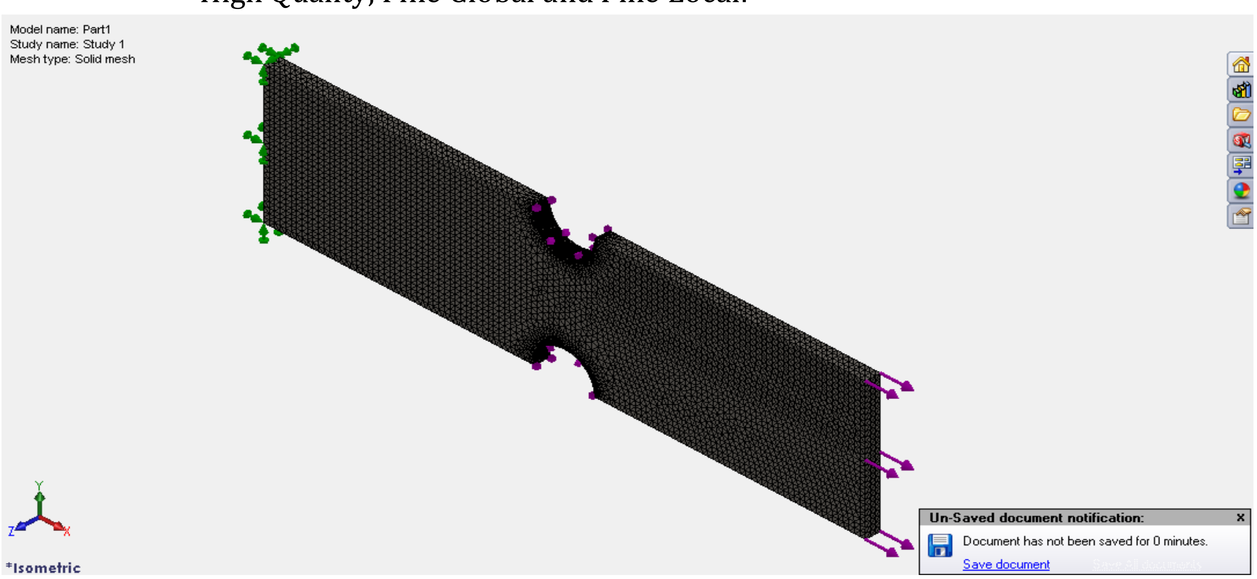
- High Quality, Coarse Global and Fine Local:



- High Quality, Medium Global and Fine Local:

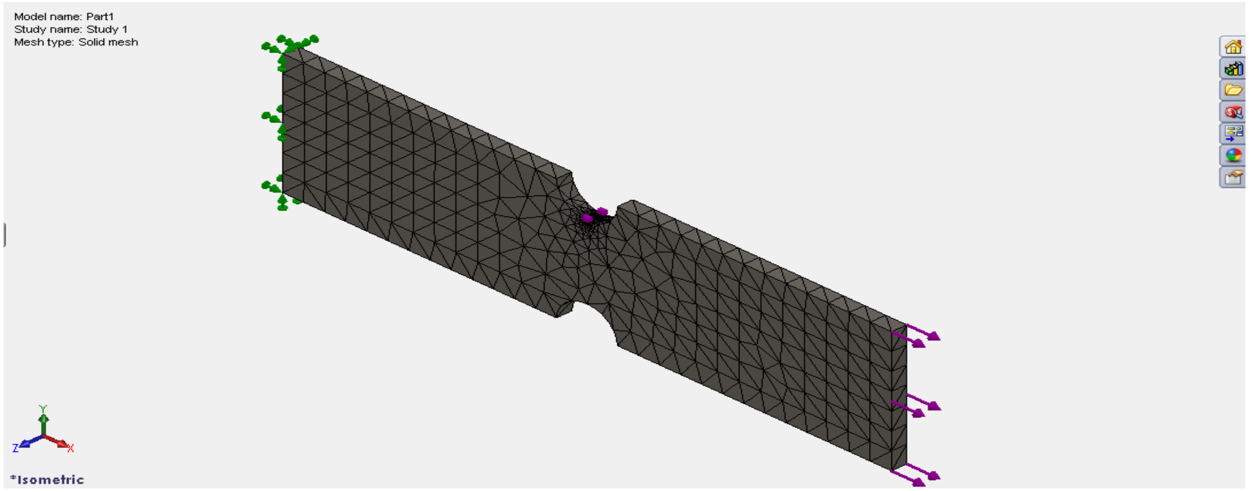


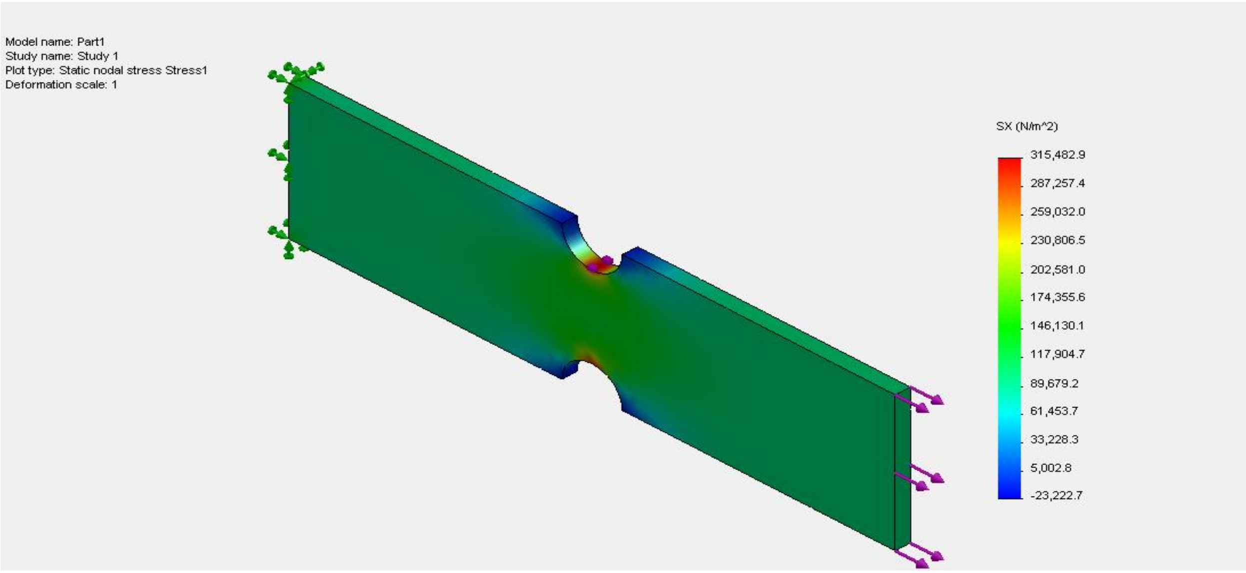
- High Quality, Fine Global and Fine Local:

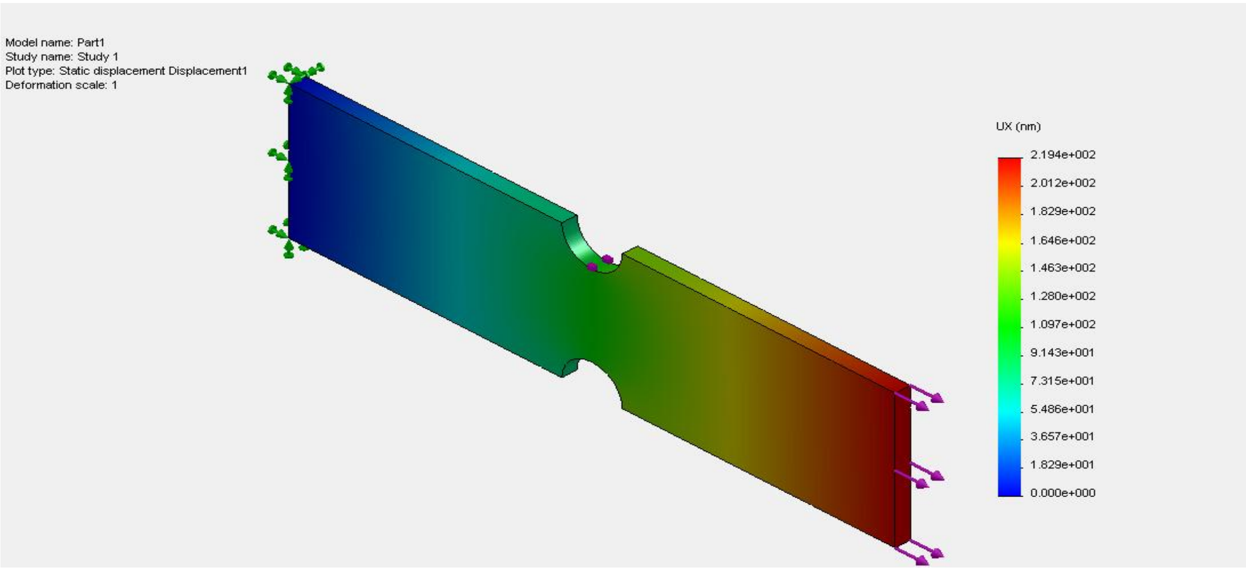


7.4. Apply Control by Split line

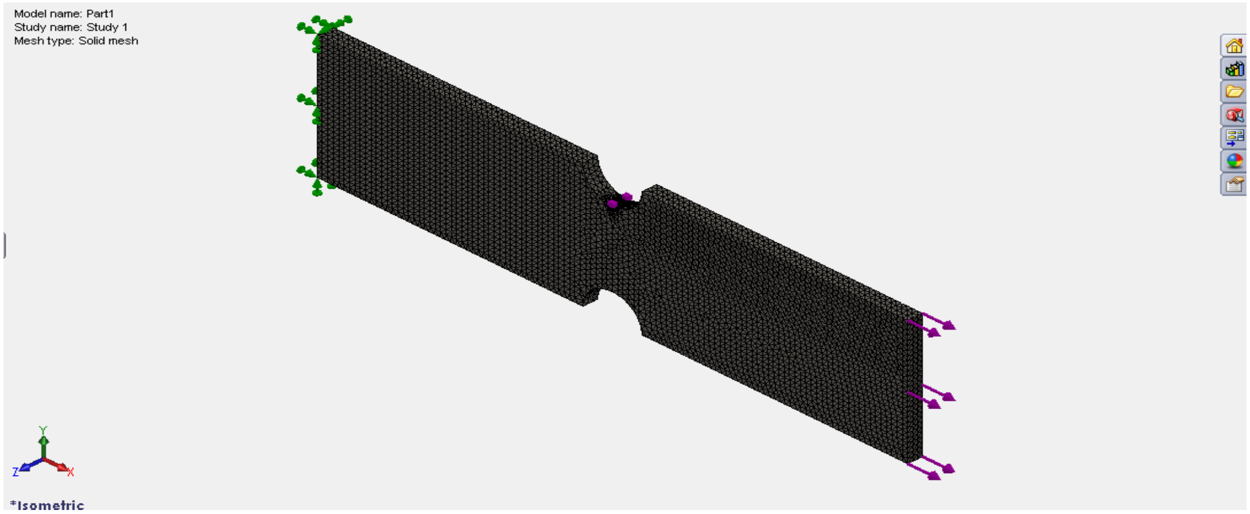
- High Quality, Coarse Global and Coarse Local:

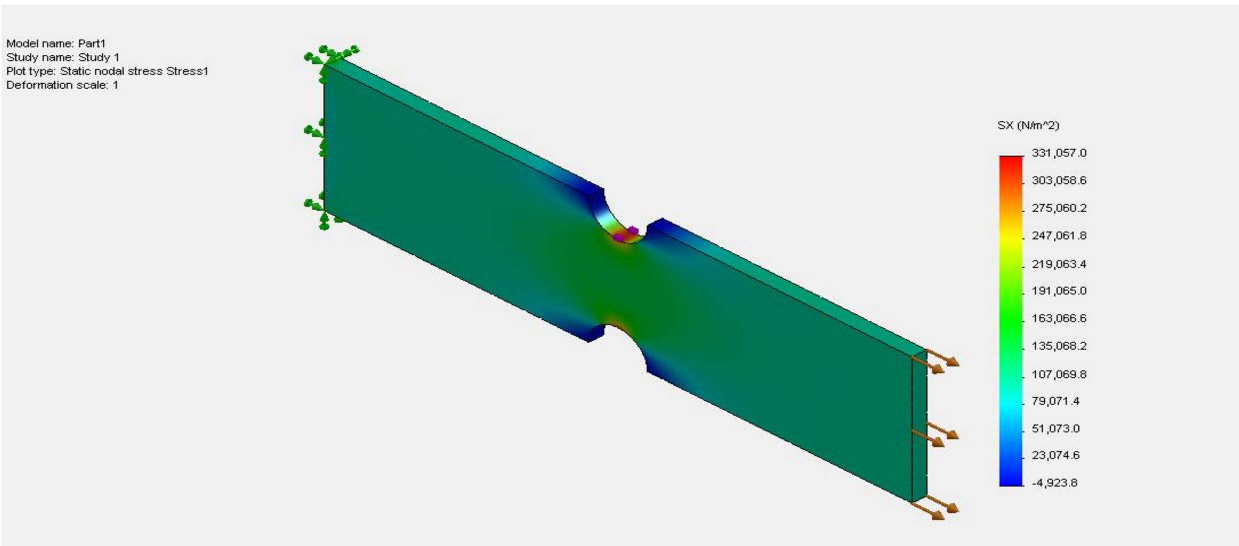


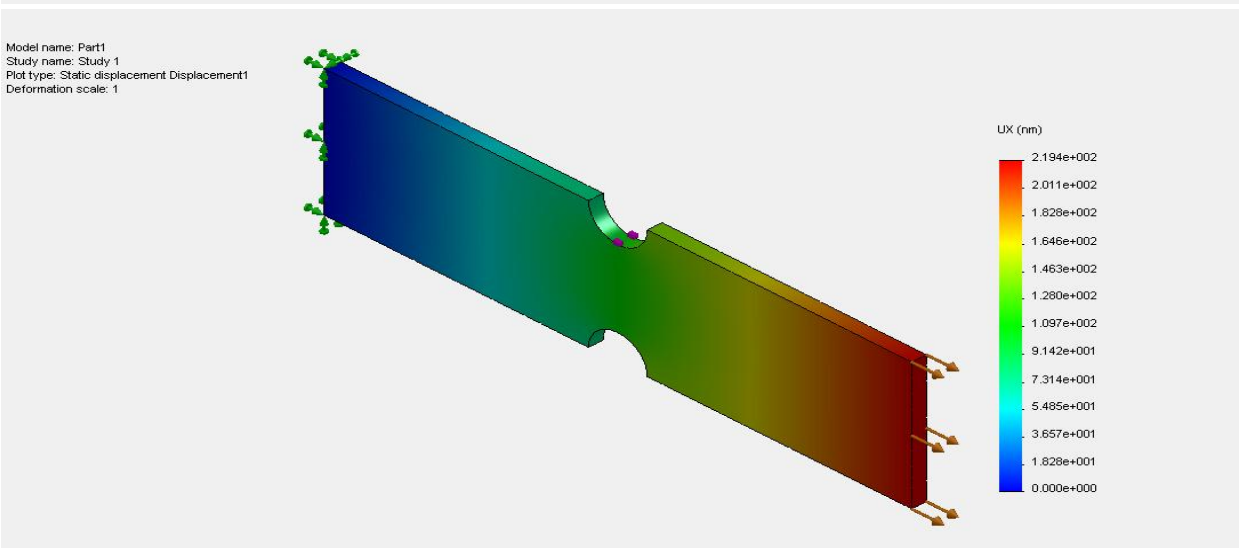




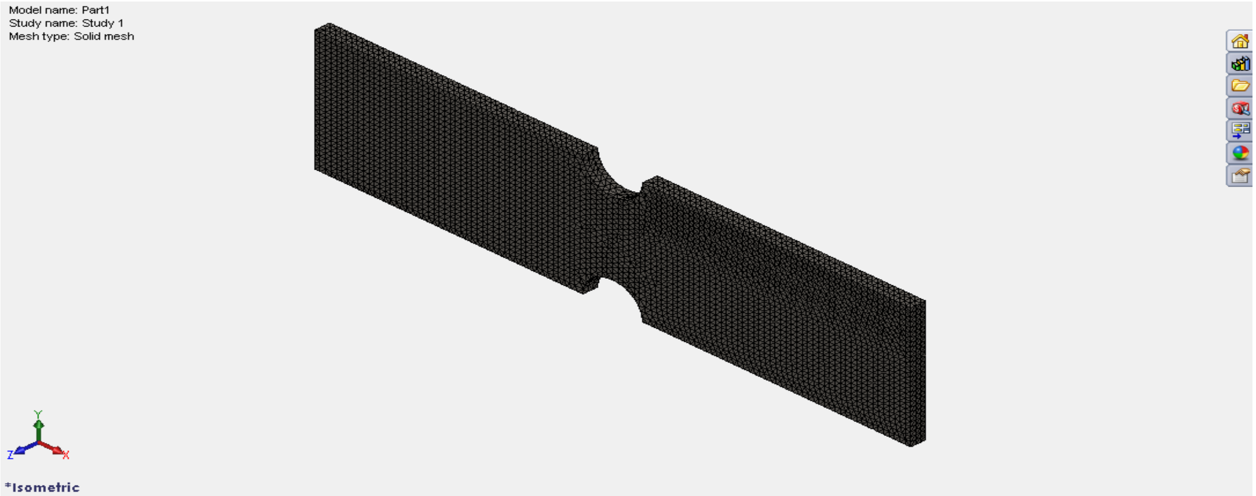
- High Quality, Fine Global and Fine Local:

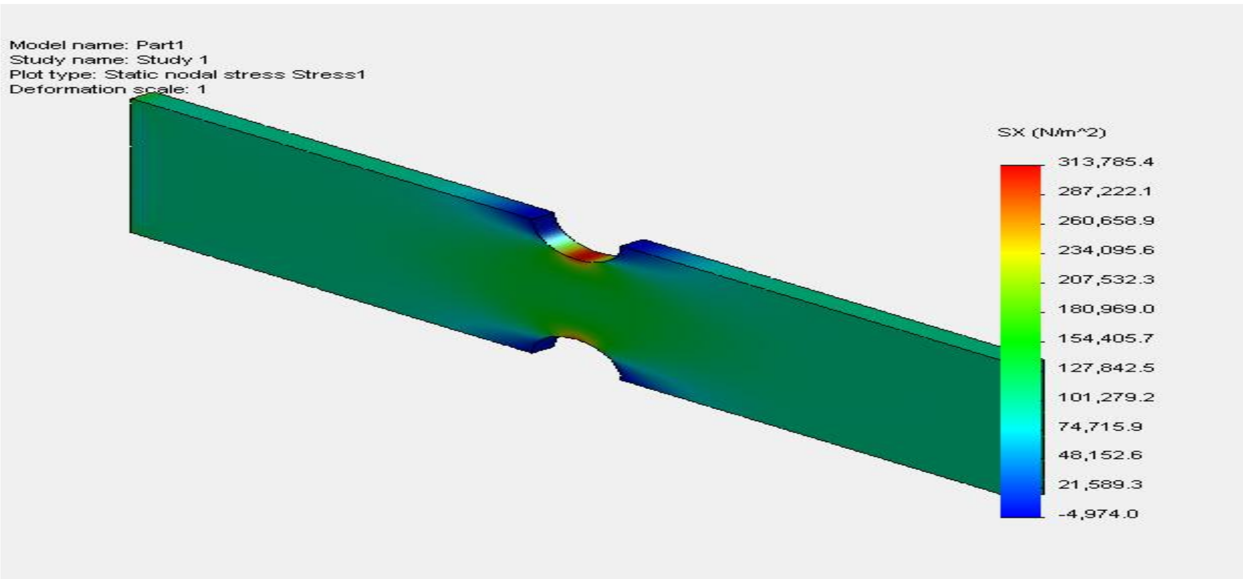


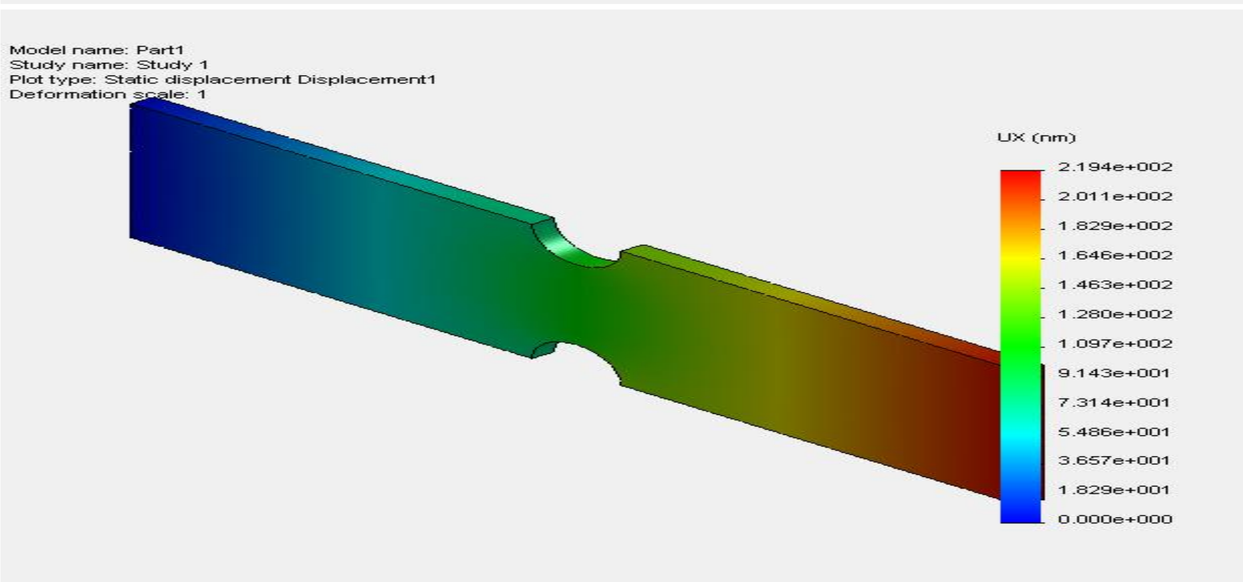




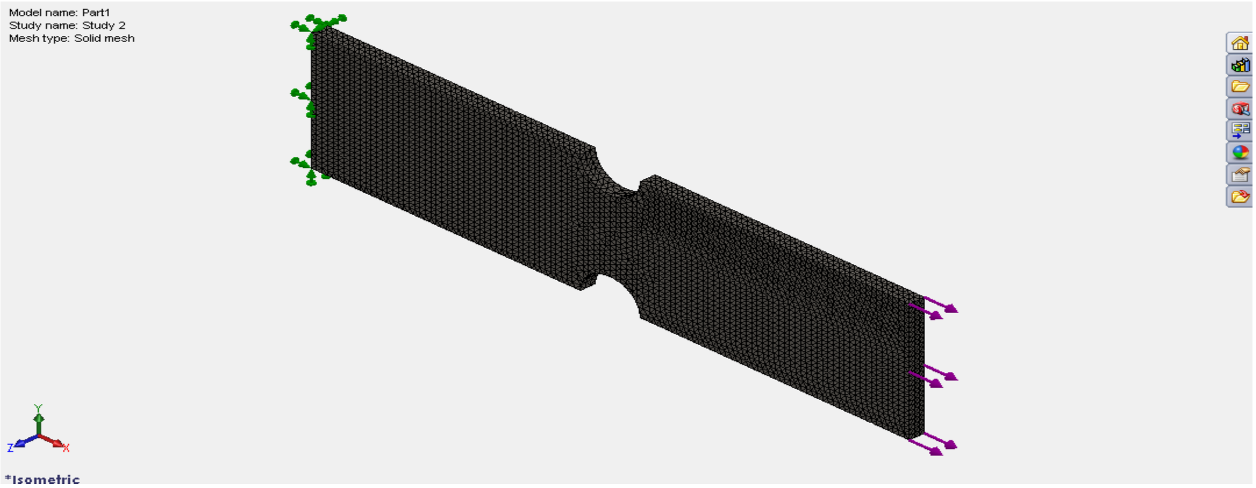
7.5. P-adaptive

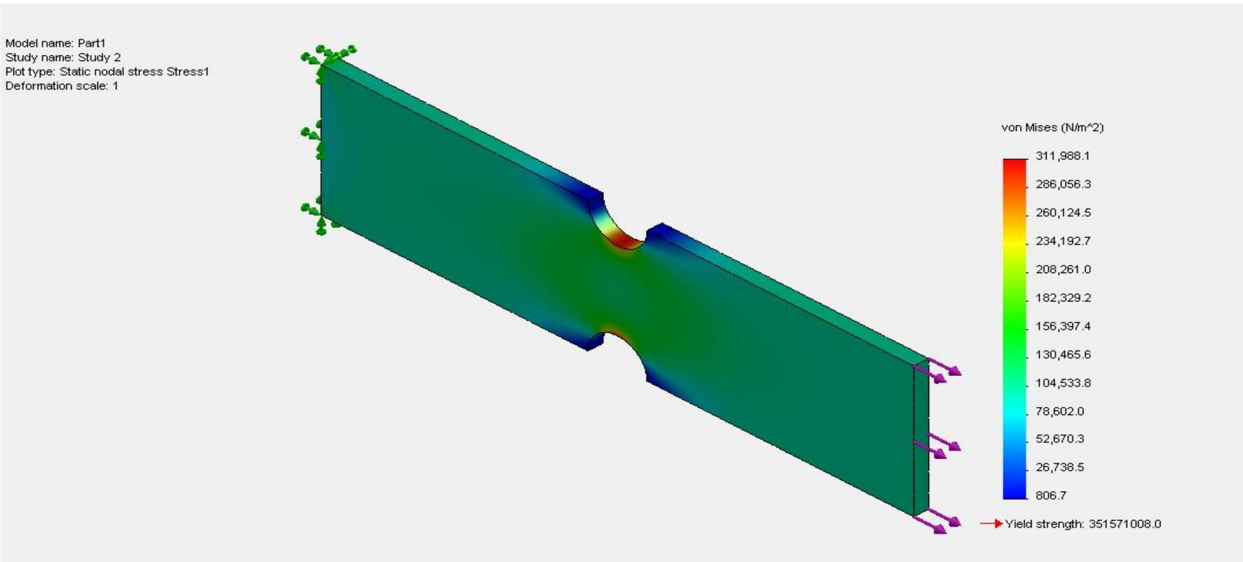


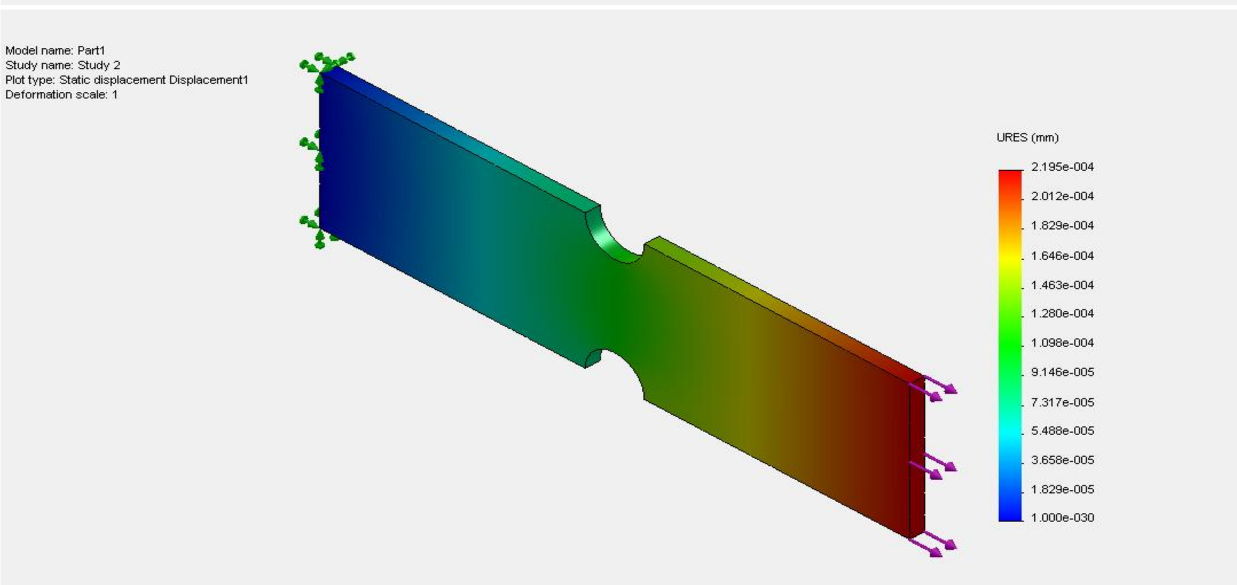




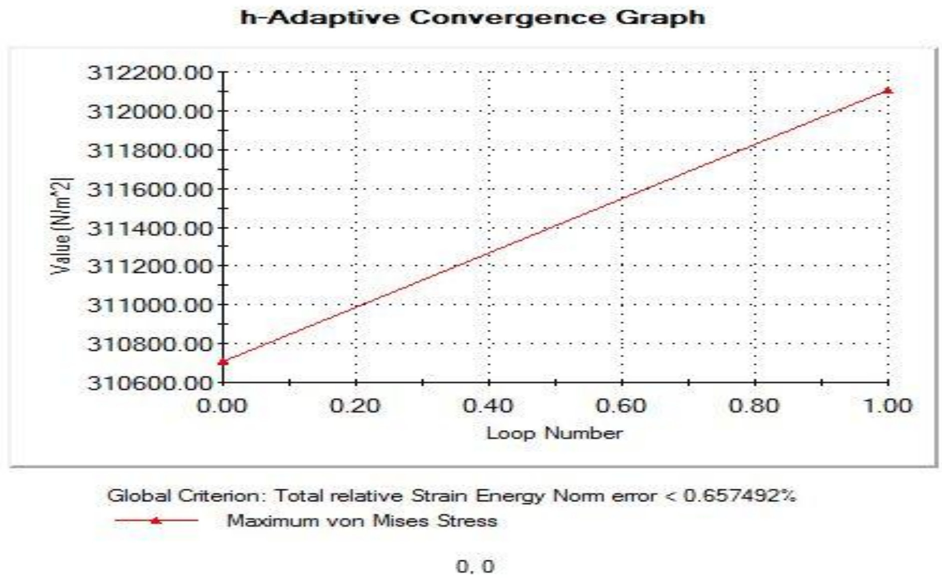
7.6. H-adaptive

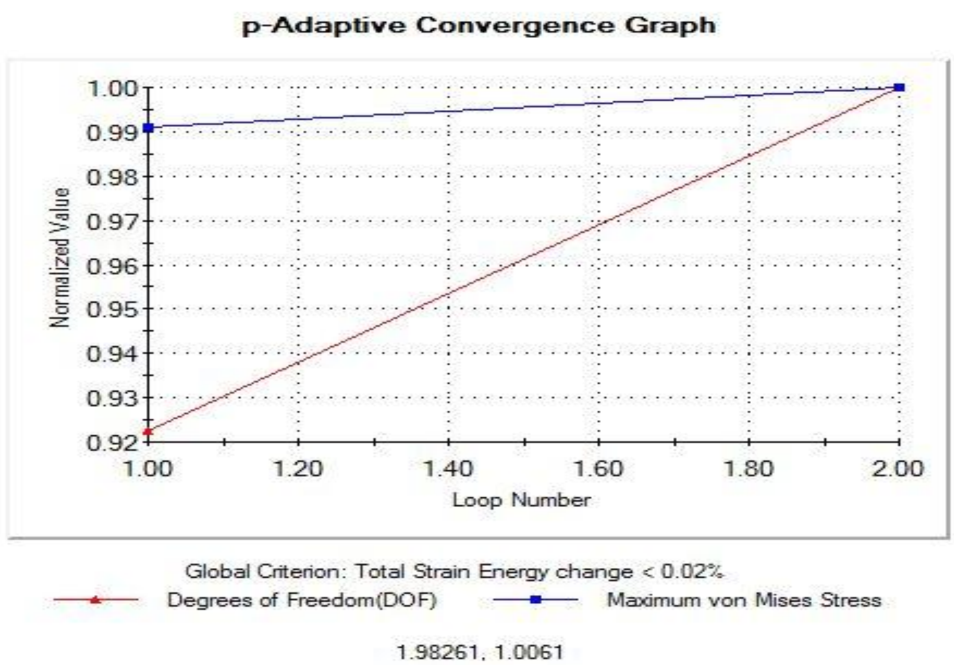




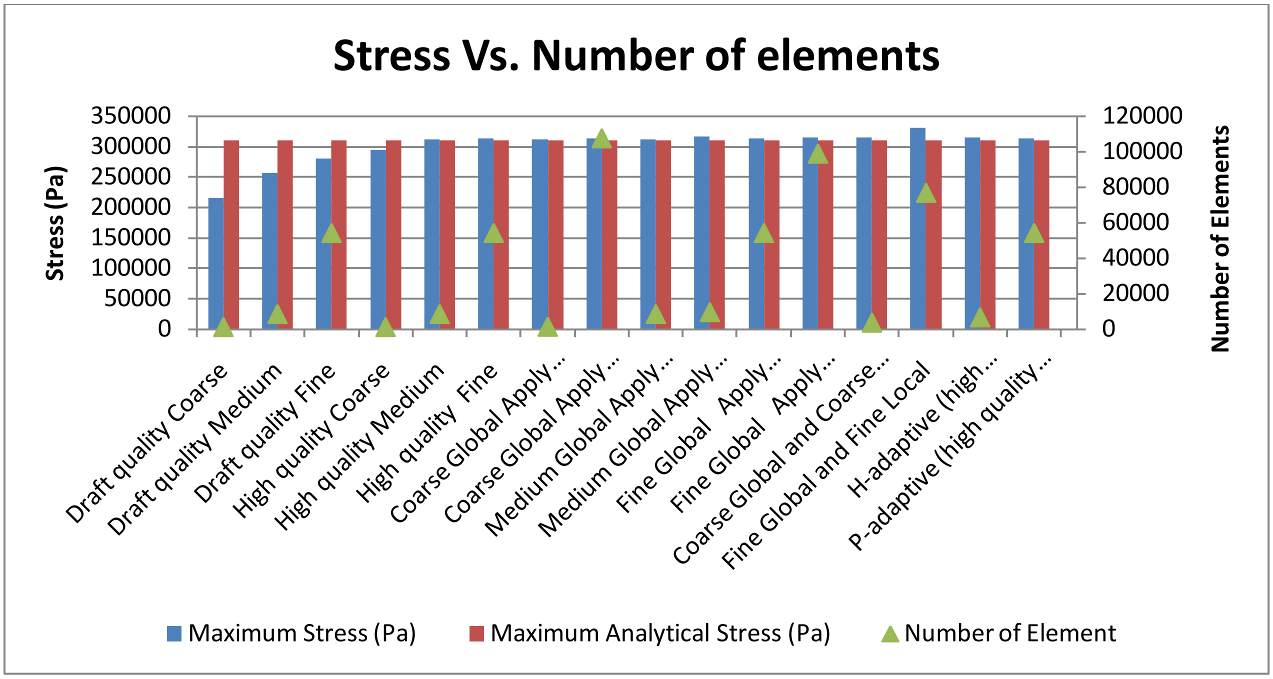


8. Comparison:

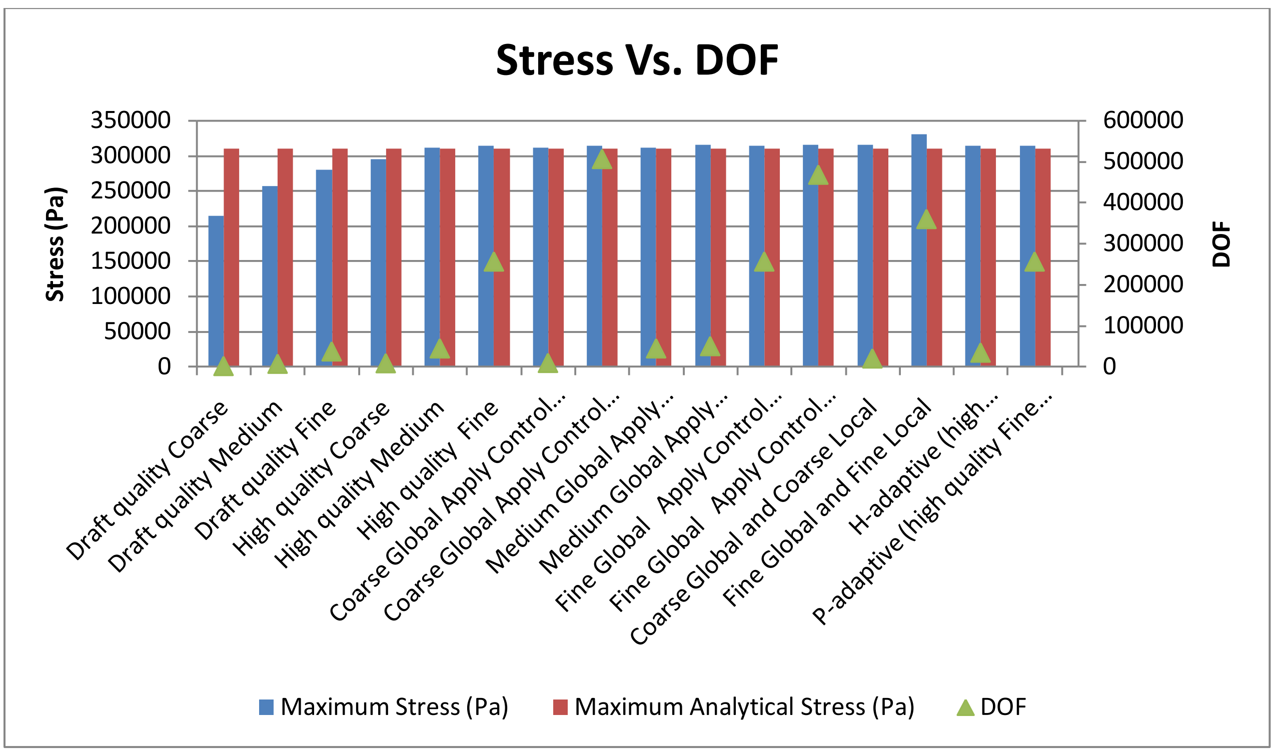




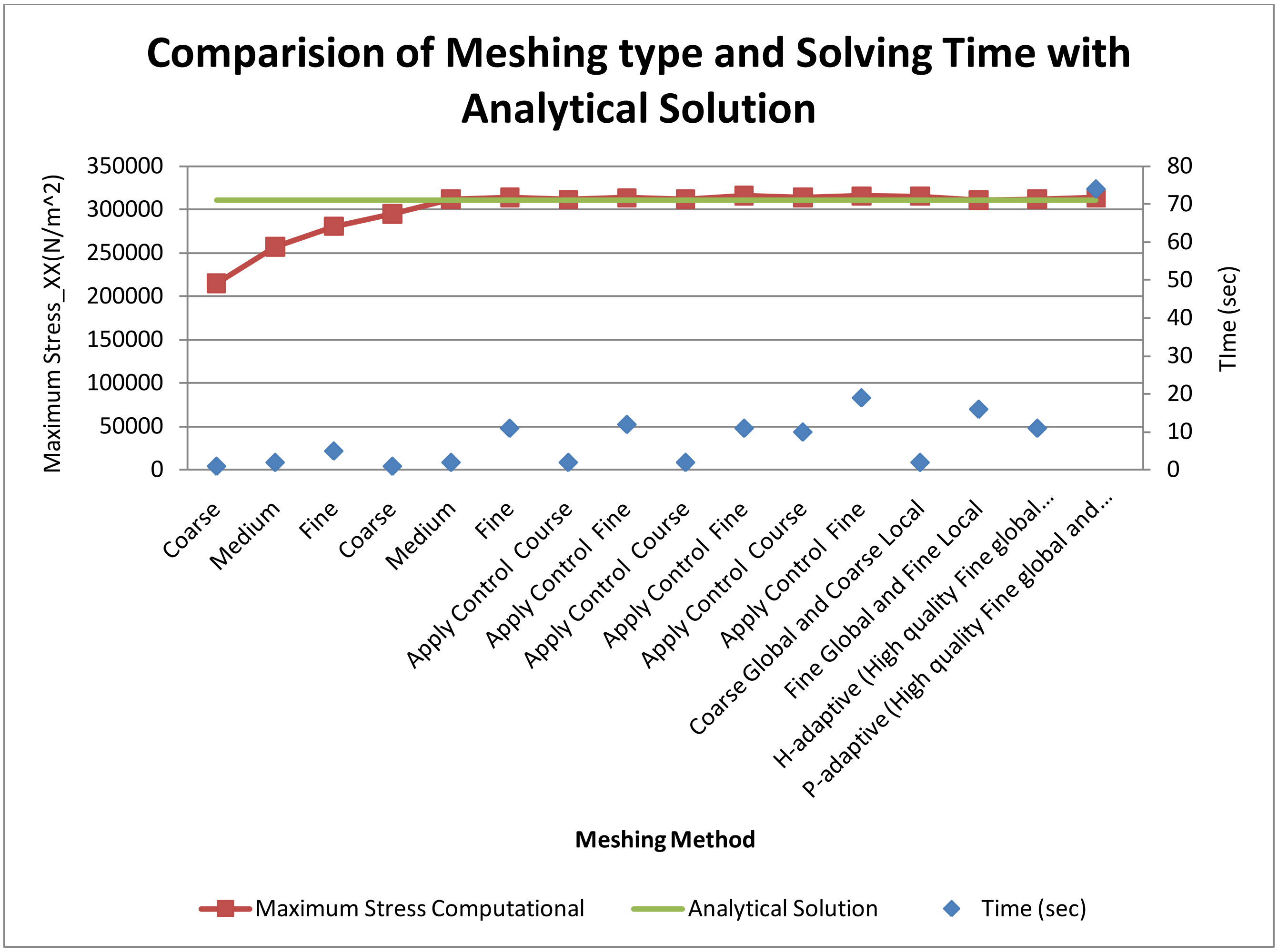
The validity of your model is true when the computer generates model result or computational solution from the solver comes closer to the true or analytical value. Adaptive test is a very effective tool while comparing the result if the true value is not known, that is by convergence plot. By constraining the solver by number of iteration or percentage error in the value, the solver obtains the value with least percentage error in iterative value.



Graph 1.0

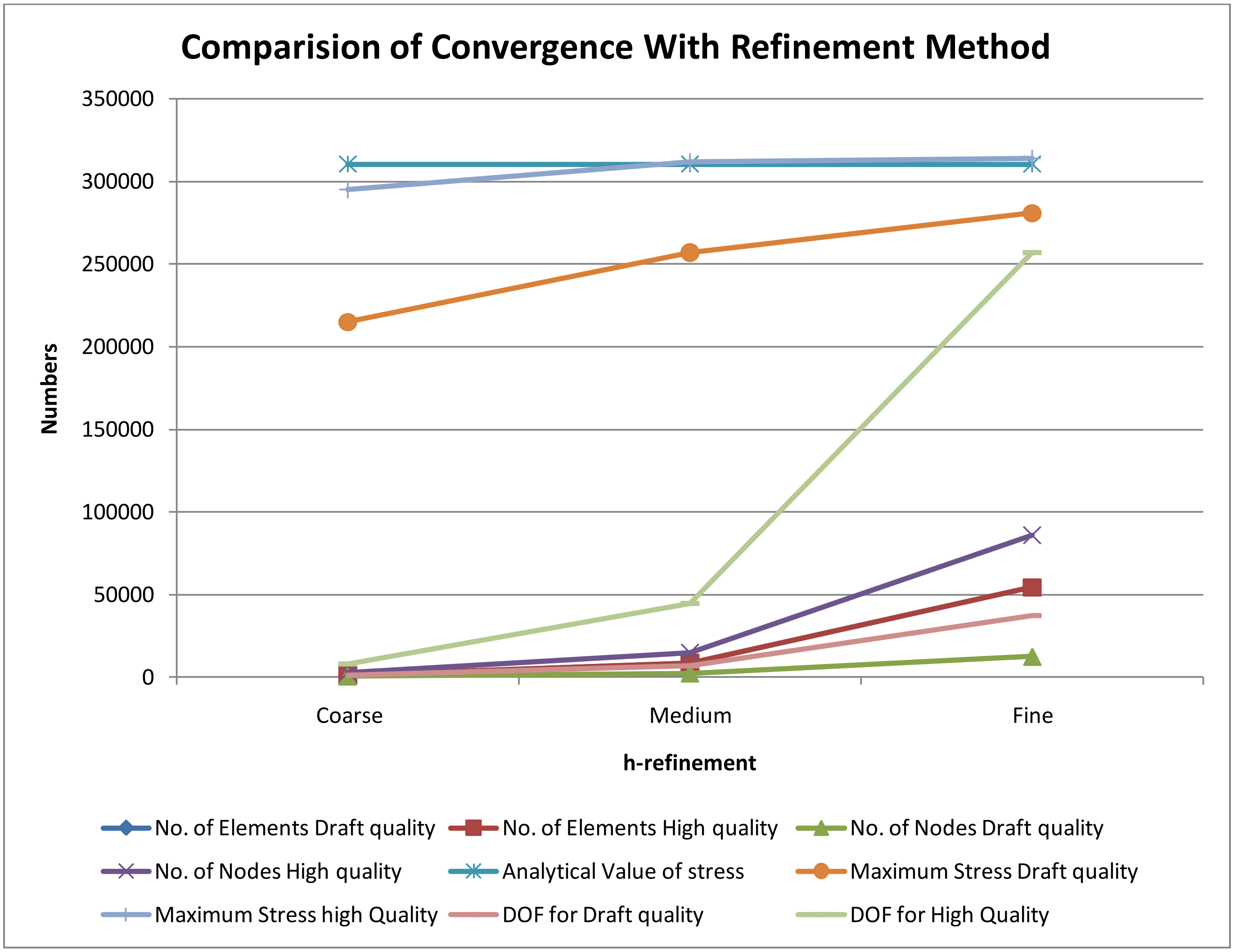


Graph 2.0



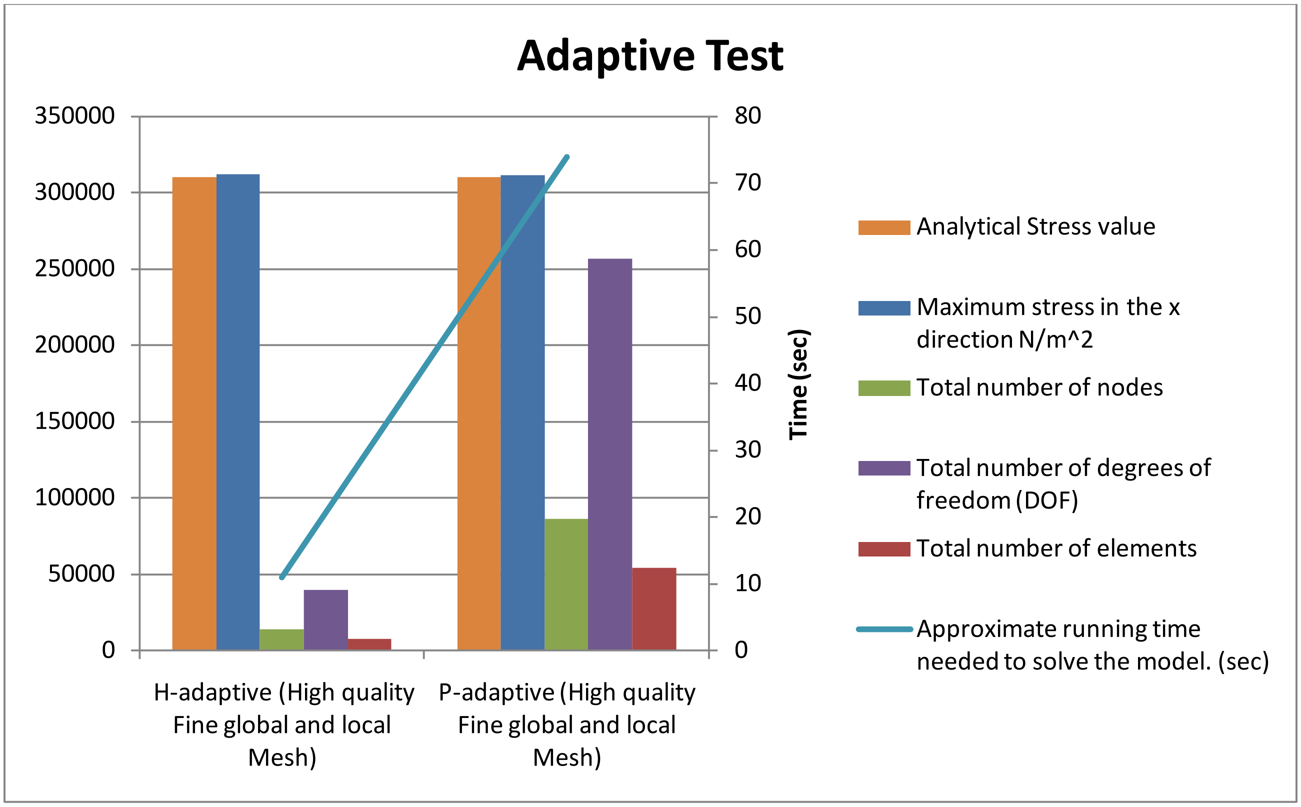
Graph 3.0

From the graph above we can visualize that computational solution made by FEM solver converge as the mesh refinement i.e. local as well as global, is done. As we learn from the lecture, as the mesh size are refined and order of polynomial is increased the approximation error is reduce and result become closer to the analytical or true value if know. From the graph above we can see the value generated by the high quality fine global and local mesh, h-adaptive and p-adaptive are the closest value to the one predicated by the analytical solution which validate the solver approximation. However as we look for time scale, i.e. time taken by the solver to provide the result also increases with respect to the increase in meshing size and in the graph the highest time was taken by p-adaptive method for P=4, this approximation is done by 4^{th} order polynomial which increase stiffness matrix size and eventually leads to maximum accuracy however cost more time to solve the model. This was a simple model with few constrain where 74 seconds was compromise, however for complicated model this time cold be huge. Nevertheless, depending on the sensitivity of the analysis time and accuracy are to be compromise as required.



Graph 4.0

From the graph above we can see how h-refinement and p-refinement has resulted into the convergence and the accuracy of the result. For the h-refinement i.e. coarse, medium and fine, the solution converges as the element size decrease and for p-refinement i.e. from draft quality (P=1) to high quality (P=2), along with h-refinement the solution converges faster and is more accurate method of solving for more accurate results. We can also as number of nodes increase as h-refinement and for p-refinement also, however the number of element remains constant for p-refinement however it increase with respect to h-refinement. Degree of freedom is as approximately 3 times the number of nodes, that include three dimension motion excluding rotation. As we found that number of nodes increases with both p and h refinement the degree of freedom also increase in as meshing element are more refined.



Graph 5.0

From the graph above, we can say that for our model p-adaptive method was more effective than p-adaptive as the constrain in h-adaptive was to meet the % error in total strain was the boundary for calculation and as the limit was reached in first iteration and the solver was stopped however in p-adaptive for P=4, the Von Mises Stress % change was the constrain which increase the solver iteration process and more accurate and precise result was obtain that was close to analytical solution. However we can also see that the p-adaptive method took 74 seconds in compare to 11 sec for h-adaptive method which is also an important factor to consider when one has to model complicated model in quick time. The number of nodes, DOF, and elements are higher for p-adaptive however the element size are same for both local and global control.

9. Conclusion:

The stress calculated from analytical solution and numerically from simulation in COSMOS by FEM solver for p-adaptive control is 310.44 and 311.78 KPa respectively. The validity of your model can be confirmed as we can see the percentage error in the approximation is about 0.4%. The local and global refinement also plays an important role and result can be confirmed by the Graph 1.0, Graph 2.0 and Graph 3.0 as refining your model which respect to both local and global control the solution converges to the true value.

Moreover, displacement on contrary was calculated by the FEM solver was approximately 219.4 nm which is higher than the displacement calculated analytically solution which was because solution did not the consideration the notches and calculated to be 200 nm. This concludes that FEM solution is valid and the approximation is more real world solution of engineering problem rather than evaluating via expensive experiment in some case. However, it does not FEM are always correct in prediction, rather the result are not be taken as primary source in optimizing model to its best proximate before prototyping and validating the product specification and degree of safety.

SolidWorks COSMOS help validate a CAD generated model and ensures the quality and performance of your design before its production. Not only static study COSMOS allows a comprehensive analysis tools for thermal, fatigue, frequency, optimization and fluid flow analysis. Though we used COSMOS 2006 for the analysis, SolidWorks has realized its latest edition which might have definitely modified the short comes and solver efficiency than its previous edition. COSMOS not only help study physical interaction that is kinematics and kinetics study but also help determining the methods to reduce weight and material costs, improve durability and manufacturability, optimize margins, and compare design alternatives.