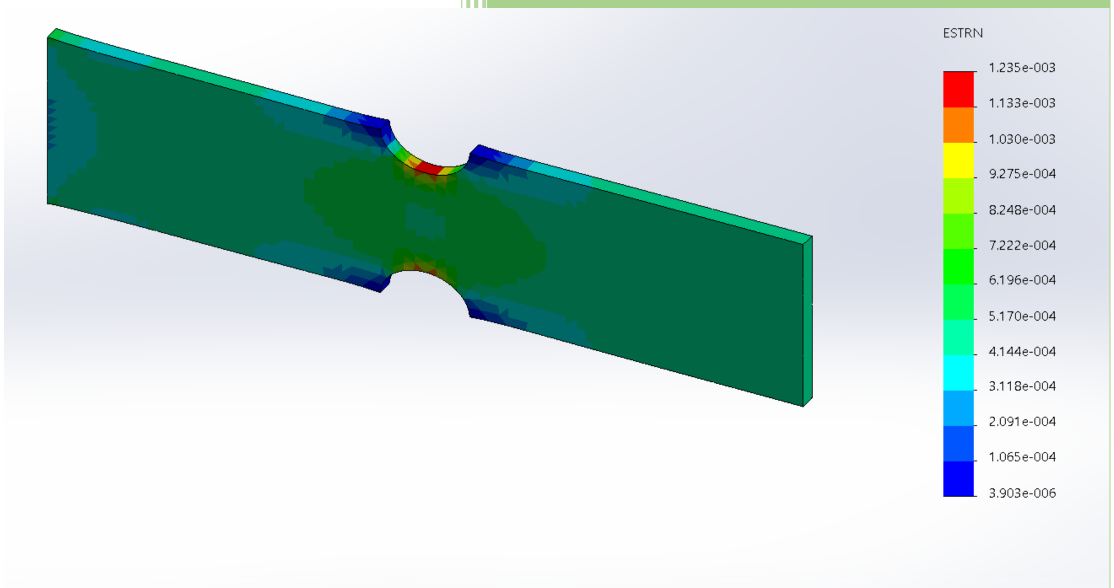
2015



Disha Barua
Instructor: Gary Benenson
11/4/2015

Overview

The finite element method is a dominant strategy to conduct structural engineering. The displacement formulation of the finite element method is used in solid work simulations in order to calculate displacement, stress and strain. The purpose of this study is to check the validity of a FEM solution as well as explore the effects of element size and order on the stress and displacement results.

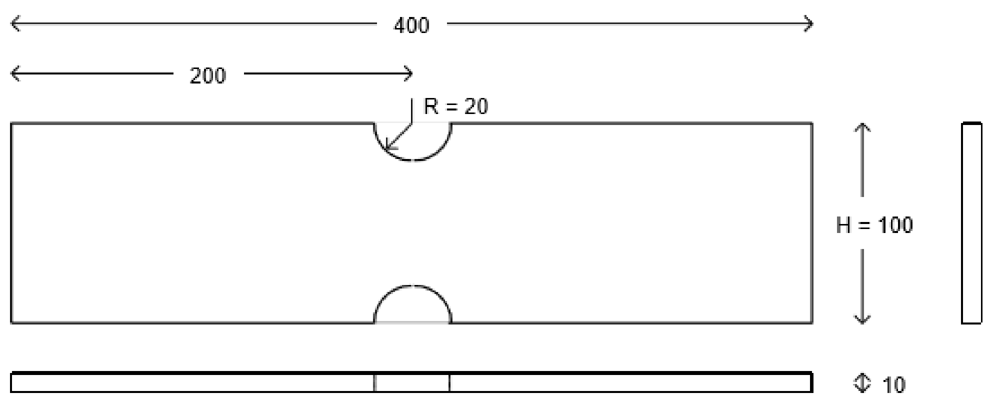


Figure 1: Plate with semicircular notches

In this case, final element method was used to investigate the validity of stress and displacement along a plate with two semicircular notches for an external load. Analytical evaluation was performed to obtain nominal stress, maximum stress, displacement, and strain using the Stress Concentration Theorem. Stress and displacement were approximated using the FE analysis in order to minimize the computational cost. The report demonstrates all the steps that were taken for this analysis: meshing, result, comparison in various studies, and convergence plots. Furthermore, I would like to end my discussion with some suggestions to improve SolidWorks.

Procedure

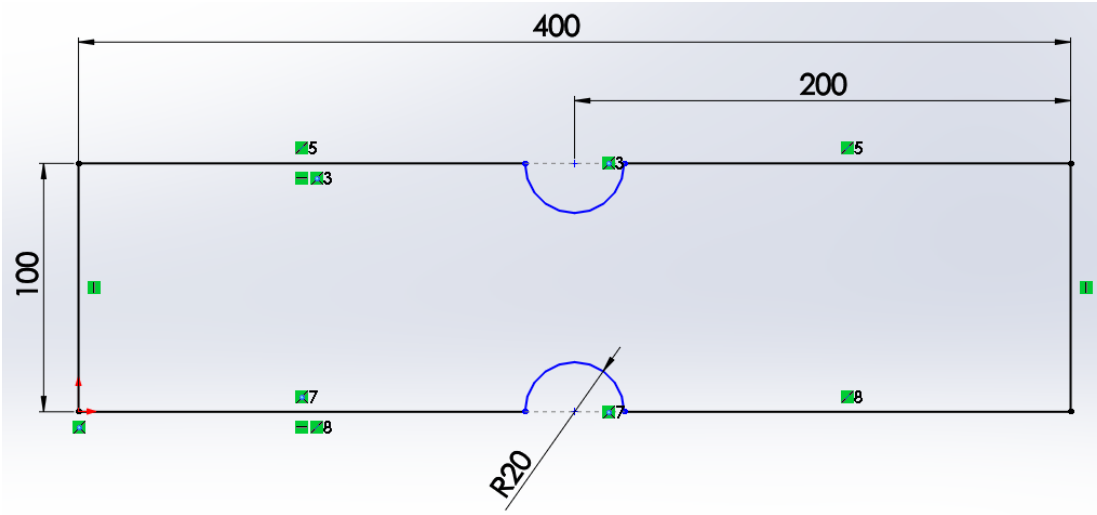


Figure 2: Solid model of the plate with semicircular notches

I performed FEM studies after addressing the geometry, material and boundary conditions of the plate. First, I created a solid model of the plate (Figure 2). The sketch above was created and extruded to the required thickness, 10mm. Steel 1020 was selected as material of the plate. Then, I applied two boundary conditions on the plate. The fixed geometry constraint was applied on the left end of the model, and the external force was applied to the right end of the model (Figure 3).

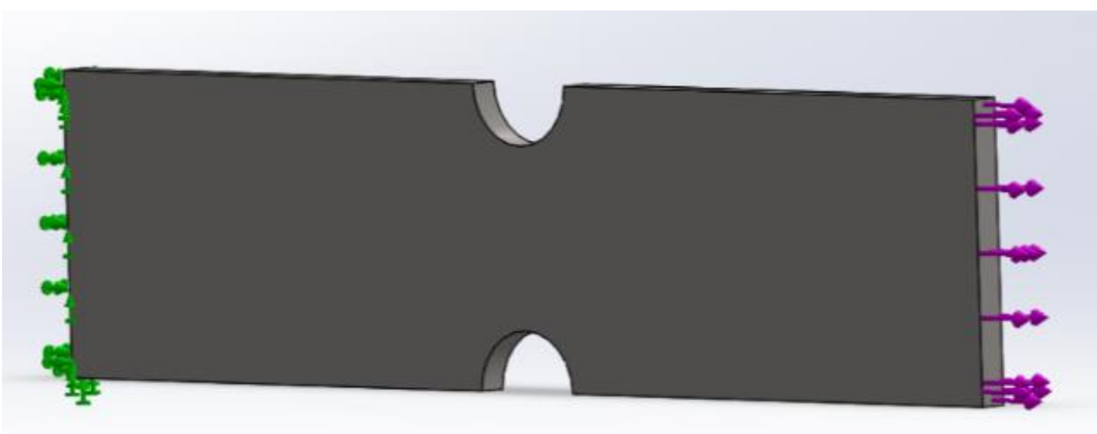


Figure 3: boundary conditions (Fixture at the left end and External load in

Several predictions were made based on the possible outcome of these constraints. Looking at the object, I predicted that the patterns of the stress should be uniform except in the notch areas.

The load path would become concentrated because of the change in geometry in those areas. The stress pattern should be more congested at the notch. The patterns of displacement are predicted to be the same along the plate except at the notch area. In this case, the pattern should be less congested around the notch area.

Out of curiosity, I conducted a small experiment to test the validities of my prediction. I took two identical pieces of polybag. I drew parallel lines on it and cut two semi-circular shape to represent the notches of our object. The left end was held in place by my left hand, and external force was applied on the right end by using my right hand to pull it. The experiment allowed me to formulate a better prediction for a displacement pattern.

According to my prediction, the maximum stress should be in the notch. When the load is applied, there will be stress concentration around that region. The minimal stress will be in the region in between the fixed end and notches that are far from the loading. Maximum deflection should occur where the load applied, which means at the right end of the plate while minimum deflection should be at the base of the fixed end. Noticing that the two notches are in the middle, the cross section of that area will be lower than the other cross section along the plate. Thus the maximum stress should be located at the smallest cross section of the notch.

After predicting the pattern and position of the stress and displacement in the plate, my next step was to find converged maximum stress and displacement. Different studies were conducted using draft quality mesh and high quality mesh to find how the result converges in each case. Draft quality mesh generates 1st order elements while high quality generates 2nd order. I controlled the mesh element using h-refinement and p-refinement. p-refinement changes the element types without changing the element size. Nodes can be added to existing element, or degrees of freedom can be added in existing nodes in p-refinement. h-refinement changes the element sizes without changing the element types. h-refinement can be applied as global refinement or local refinement. Global refinement is when the mesh is created with the same element size while local refinement is when specific refinements are made to some selected areas.

In order to conduct all those studies and organize them, first I wrote down the process using a notebook to record all the data. I started conducting studies with the draft quality element and changed the mesh type using different element sizes. Data was obtained for five different element sizes for 1st order elements. Similar steps were taken to conduct the next study with 2nd order elements. More simulations were run for the 2nd order elements in order to find singularity plot. The elements varied from 11.584 mm to 1.5mm.

Local refinement was processed at the notch surface and edges to check the validity of stress concentration in that area. Processing local refinement was an interesting challenge. I used “Apply Mesh Control” in order to change the element size in the notch surface and notch edges (Figure 4). At first, I made an error from not changing the local mesh parameter, resulting in an unwanted mesh. Hence, I recorded my desired element size first in order to avoid repeating the mistake.

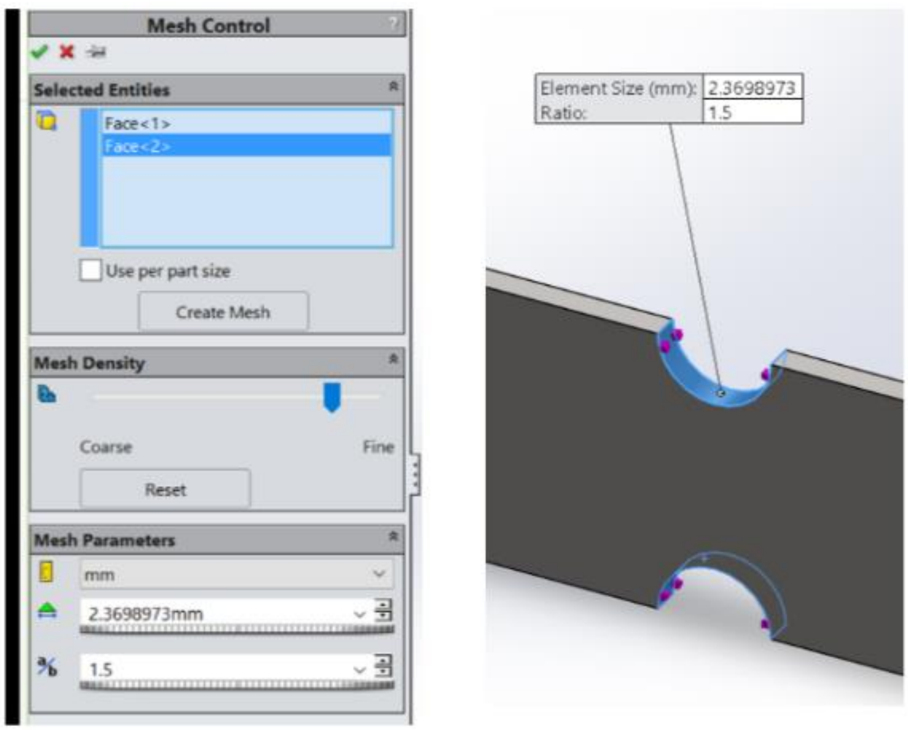


Figure 4: Local refinement

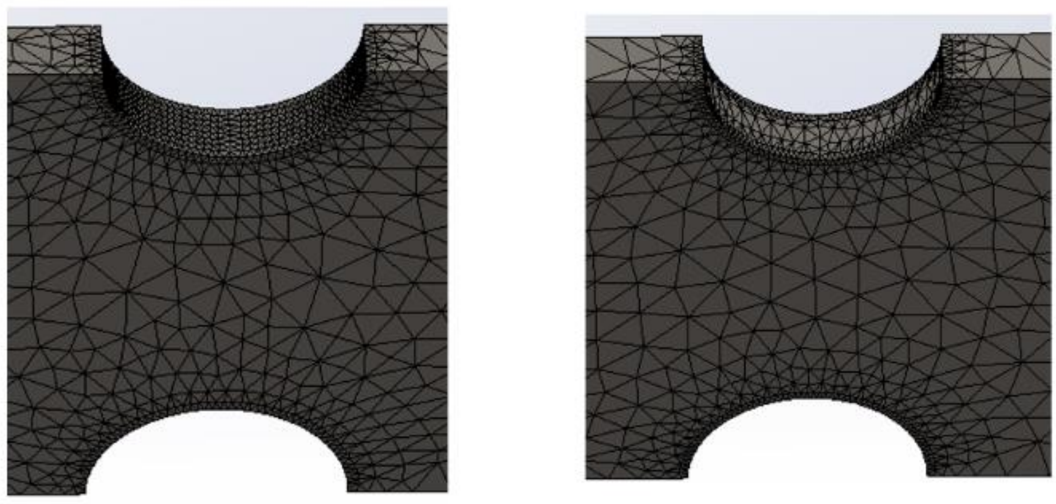


Figure 5: Local refinement at notch surface (left) and notch edge (right)

The global element size was kept constant at 7.29 mm, but the local element size varied from 2.36 mm to 0.6 mm.

Besides other mesh types, curvature-based mesh and auto transition were used to run more simulation studies. I defined the element size to create curvature-based mesh. The SolidWorks simulation allows us define the range of element size, element per circle, and element size growth. Auto transition is a mesh type where a global element size is defined and it applies a local element size to fillet, holes. Five simulation studies were run each case.

I noted down all the data needed and tabulated them in the result section. Then, nodes vs stress graphs were plotted to check for the convergence and the accuracy of the results.

<u>Analytical solution</u>

The maximum stress was determined by using the stress concentration theory and given formulas.

$$\sigma_{\text{nom}} = \frac{F}{A_{\text{reduced}}} \quad [1]$$

$$\sigma_{\max} = K_{\text{e}}\sigma_{\text{nom}} \quad [2]$$

$$K_{\text{e}} = 3.065 - 3.370\left(\frac{2R}{H}\right) + 0.647\left(\frac{2R}{H}\right)^2 + 0.658\left(\frac{2R}{H}\right)^3 \quad [3]$$

$$\sigma = E\varepsilon \quad [4]$$

$$\sigma = \frac{F}{A} \quad [5]$$

$$\delta = \frac{FL}{EA} \quad [6]$$

$$\varepsilon = \frac{\delta}{L} \quad [7]$$

$$A_{\text{reduced}} = (10\text{mm})(H - 2R) \quad [8]$$

Where,

External load, F = 100 kN

Radius of notch, R = 20 mm = .02m

Total width across the section, H =100 mm = 0.1 mm

Length of the plate, L = 400mm = 0.4 m

Young's modulus for steel 1020, E =200 GPA

K_e = stress concentration factor

A_{reduced}= reduced cross-sectional area, mm^2

σ_{nom} = nominal stress between the notches, MPa

σ_{max} = maximum stress, MPa

δ = displacement, mm

From equation (8) we get,

$$A_{\text{reduced}} = (10\text{mm})\big(100\text{mm} - 2(20\text{mm})\big) = 600\text{mm}^2$$

$$K_e = 3.065 - 3.370\left(\frac{2(20\text{mm})}{(100\text{mm})}\right) + 0.647\left(\frac{2(20\text{mm})}{(100\text{mm})}\right)^2 + 0.658\left(\frac{2(20\text{mm})}{(100\text{mm})}\right)^3 = 1.8626$$

$$\sigma_{\text{nom}} = \frac{100000\text{N}}{600\text{mm}^2} = \frac{100000\text{N}}{600 \times 10^{-6}\text{m}^2} = 166.7\text{MPa}$$

Substituting values of σ_{nom} and K_e in equation [2] we get,

$$\sigma_{max} = (1.8626)(166.7\text{MPa}) = \boxed{310.50\text{MPa}}$$

From equation (6) we get,

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

$$= \frac{100000\text{N}.0.4\text{m}}{(200.10^9).0.1\text{mm}.0.01\text{mm}}$$

$$= \boxed{0.2\text{ mm}}$$

Result

Table 1: Standard global mesh with 1st order elements

Element size, mm	Element no	Nodes	DOF	Mesh time, s	run time, s	Sx_{max} (MPa)	δmax (mm)	ε_{xmax}
14.584	1308	488	1416	0	0	211.9	0.216	1.221*10^-3
10.932	2657	860	2514	0	0	218.5	0.217	1.302*10^-3
7.29	8668	2334	6906	1	1	257.1	0.218	1.371*10^-3
5.196	18106	4727	13992	1	1	268	0.219	1.5*10^-3
3.646	54777	12553	37311	1	2	280.4	0.219	1.5*10^-3

Table 2: Standard global mesh with 2nd order elements

Element size, mm	Element no	Nodes	DOF	Mesh time, s	run time, s	Sx_{max} (MPa)	δmax (mm)	ε_{xmax}
11.485	2384	4643	13758	1	0	296.7	0.217	1.03*10^-3
7.292	8668	14961	44604	1	1	311.9	0.2195	1.308*10^-3
5.287	18441	31166	92925	1	2	310.5	0.2195	1.43*10^-3
3.646	54777	86516	258333	4	8	312.5	0.2195	1.43*10^-3
3	101745	155143	463800	6	11	312.6	.2195	1.47*10^-3
2.5	158668	239885	717165	11	17	313.6	0.2195	1.483*10^-3
2	309340	456051	1364712	27	34	313.5	.2195	1.47*10^-3

Table 3: p-refinement with global element size 7.29 mm and local refinement at notch surface

Element size, mm	Element no	Nodes	DOF	Mesh time, s	run time, s	Sx_{max} (MPa)	δmax (mm)	ε_{xmax}
2.36	10650	18419	54978	1	1	316	0.2195	1.499*10^-3
1.82	12206	21043	62850	1	1	314	0.2195	1.507*10^-3
1.5	14224	24331	72714	1	2	314.4	0.2195	1.504*10^-3
1.2	17802	30175	90246	1	2	314.6	0.2195	1.522*10^-3
0.6	48132	77740	232968	3	5	313.9	0.2196	1.539*10^-3

Table 4: p-refinement with global element size 7.29 mm and local refinement at notch edge

Element size, mm	Element no	Nodes	DOF	Mesh time, s	run time, s	Sx_{max} (MPa)	δmax (mm)	ε_{xmax}
2.36	1064	18435	55026	1	2	312.4	0.2195	1.481*10^-3
1.82	11455	19748	58965	1	2	312.8	0.2195	1.485*10^-3
1.5	12442	21401	63924	1	2	314.8	0.2195	1.494*10^-3
1	16044	27375	81846	1	2	313.8	0.2196	1.511*10^-3
0.5	27441	46236	138429	2	3	314.8	0.2196	1.525*10^-3

Table 5: Curvature based mesh, 2nd order, global refinement

Element size, mm	Element no	Nodes	DOF	Mesh time, s	run time, s	Sx_{max} (MPa)	δmax (mm)	ε_{xmax}
14.584	1590	3398	10041	1	1	300.2	0.2194	1.239*10^-3
7.292	12317	21210	63237	1	2	314.1	0.2195	1.364*10^-3
5.56	16231	27713	82308	1	2	312.3	0.2195	1.425*10^-3
4	77209	119363	356778	3	8	313.4	0.2195	1.459*10^-3
3.646	93022	146370	426771	3	11	313.6	0.2196	1.492*10^-3

Table 6: Automatic Transition mesh, 2nd order, global refinement

Element size, mm	Element number	nodes	DOF	Mesh time, s	runtime, s	SX, Mpa	δmax, mm	εmax
7.292	8769	15168	45177	1	2	314.7	0.2194	1.355*10^-3
5.65	15460	26421	78708	2	1	310.3	0.2196	1.419*10^-3
4.467	31852	51251	153042	3	4	316.7	0.2195	1.425*10^-3
3.646	54777	86510	258333	3	7	312.5	0.2195	1.427*10^-3
3	101848	155250	464283	5	11	314.2	0.2195	1.472*10^-3

Table 7: global refinement, 2nd order elements (no local refinements)

Nodes	Mesh runtime	run time	sx
4643	1	0	296.7
14961	1	1	311.9
31166	1	2	310.5
86516	4	8	312.5
155143	6	11	313.8
239885	11	17	313.6
456051	27	34	313.5
1035609	104	79	343

Convergence Plots

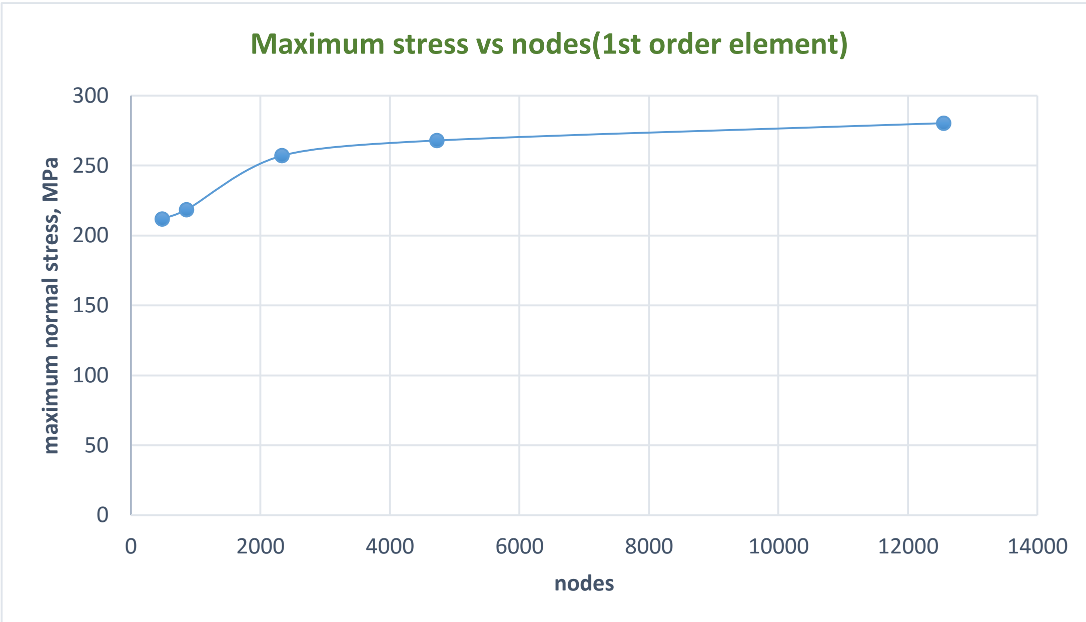


Figure 6: Standard global mesh with 1st order elements (Stress vs Nodes)

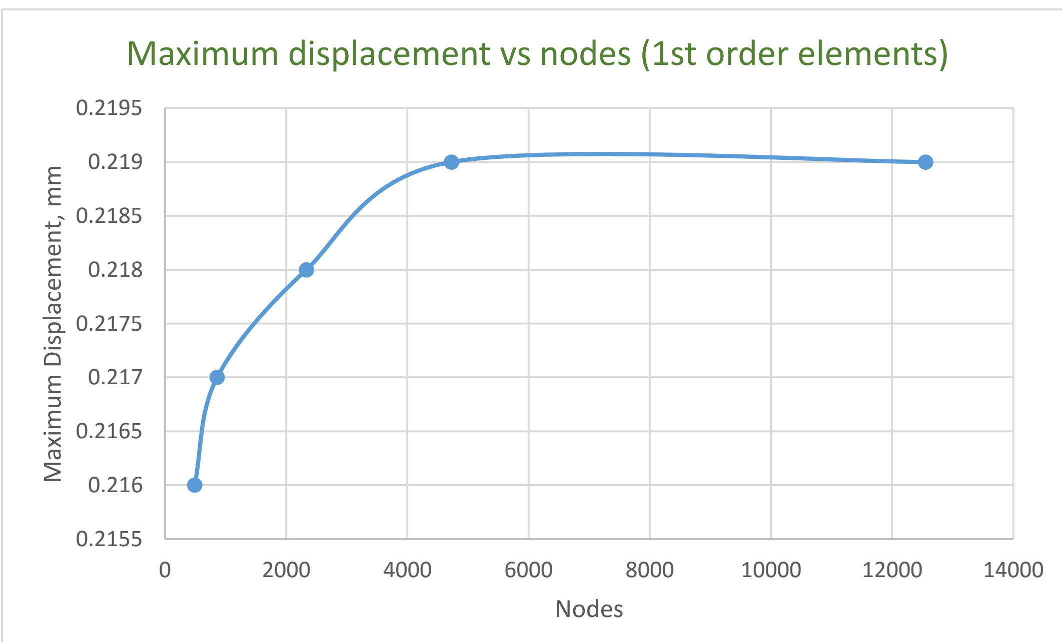


Figure 7: *Standard global mesh with 1st order elements (displacement vs Nodes)*

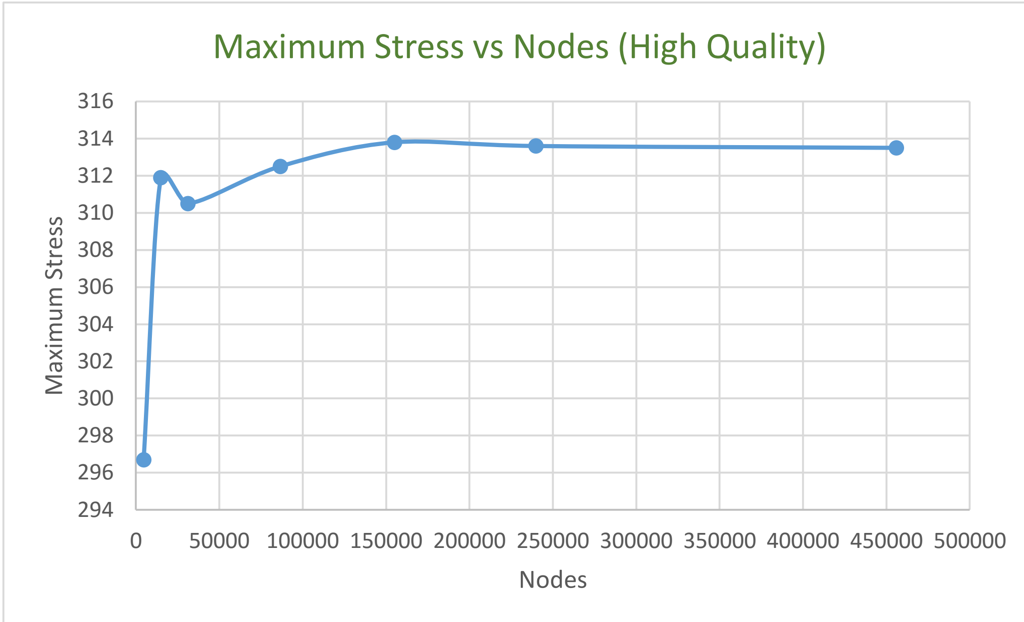


Figure 8: *Standard global mesh with 2nd order elements (Stress vs Nodes)*

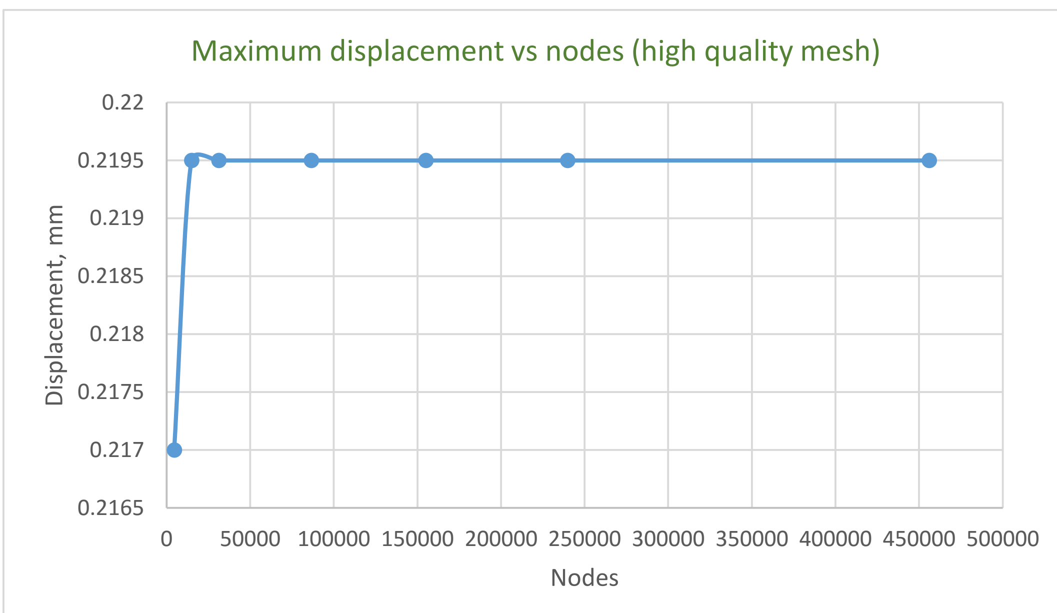


Figure 9: *Standard global mesh with 2nd order elements (Displacement vs Nodes)*

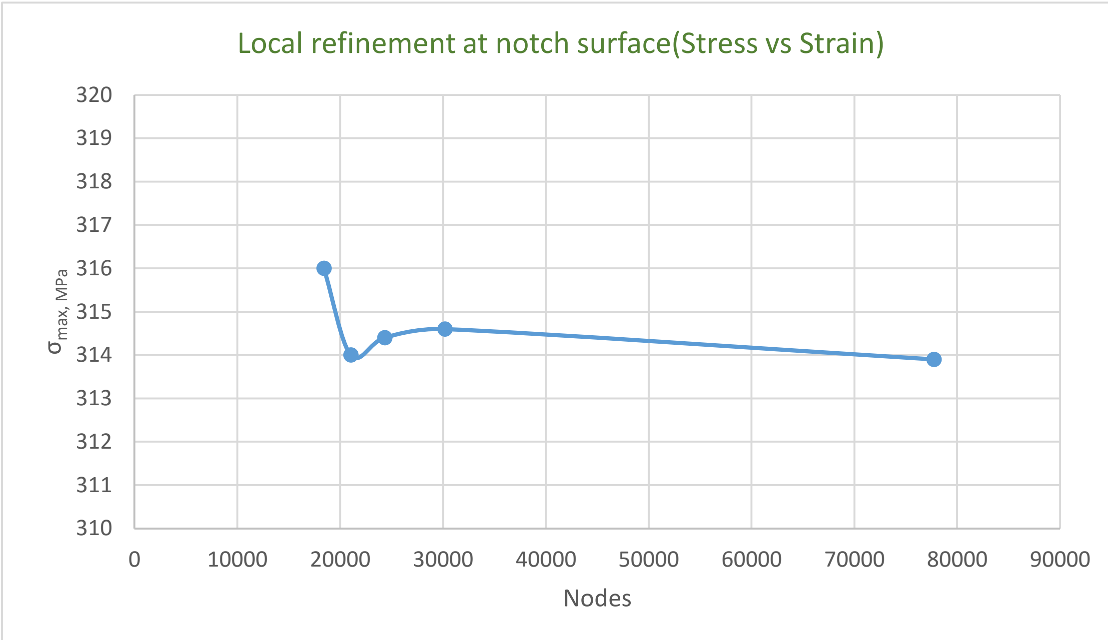


Figure 10: p-refinement and local refinement at notch surface

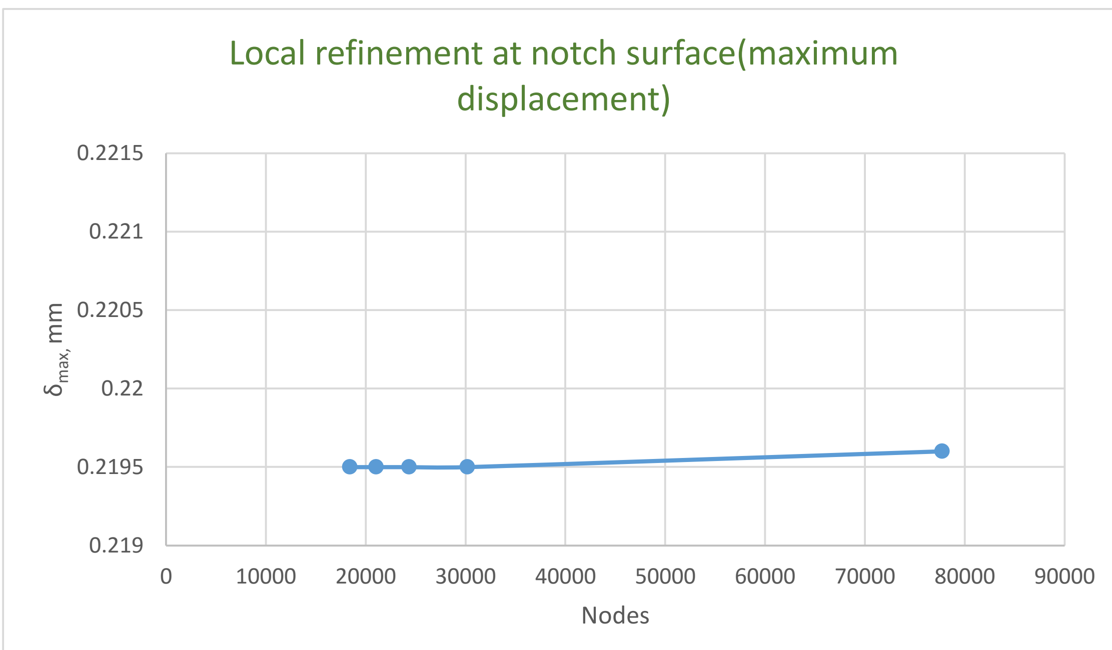


Figure 11: *p-refinement and local refinement at notch surface (Displacement vs Nodes)*

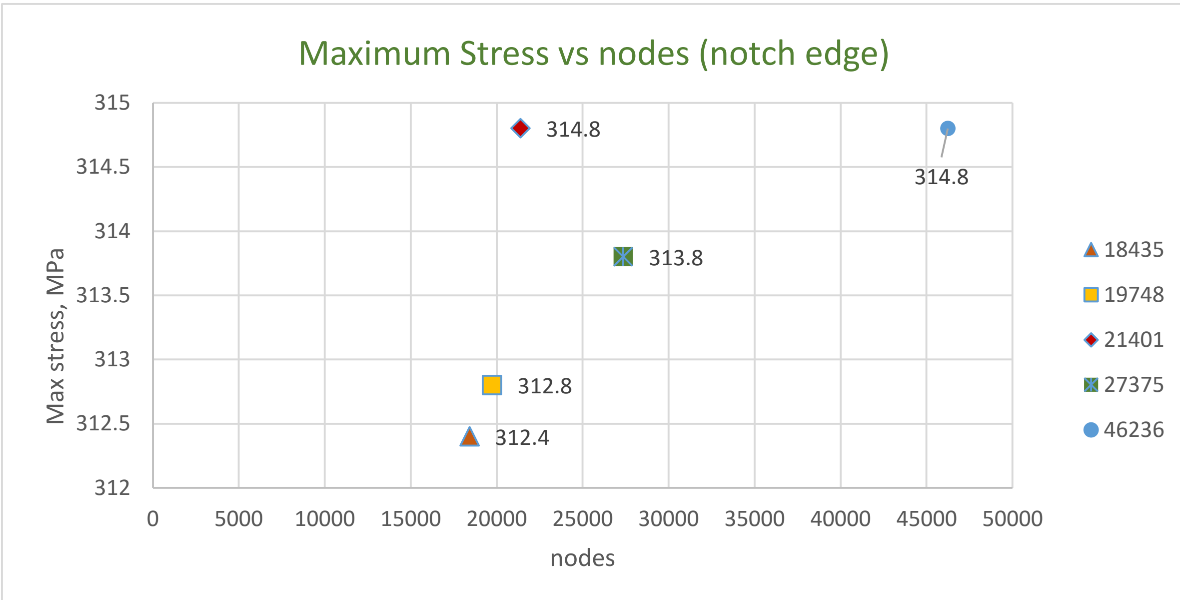


Figure 12: *p-refinement and local refinement at notch edge* (Stress vs Nodes)

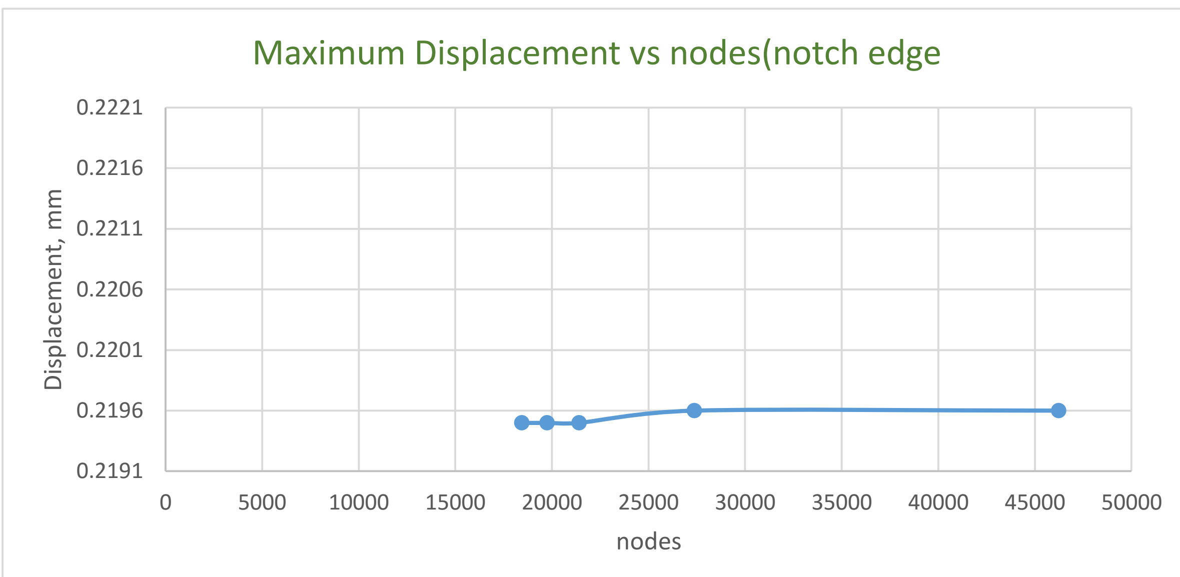


Figure 13: *p-refinement and local refinement at notch edge (Displacement vs Nodes)*

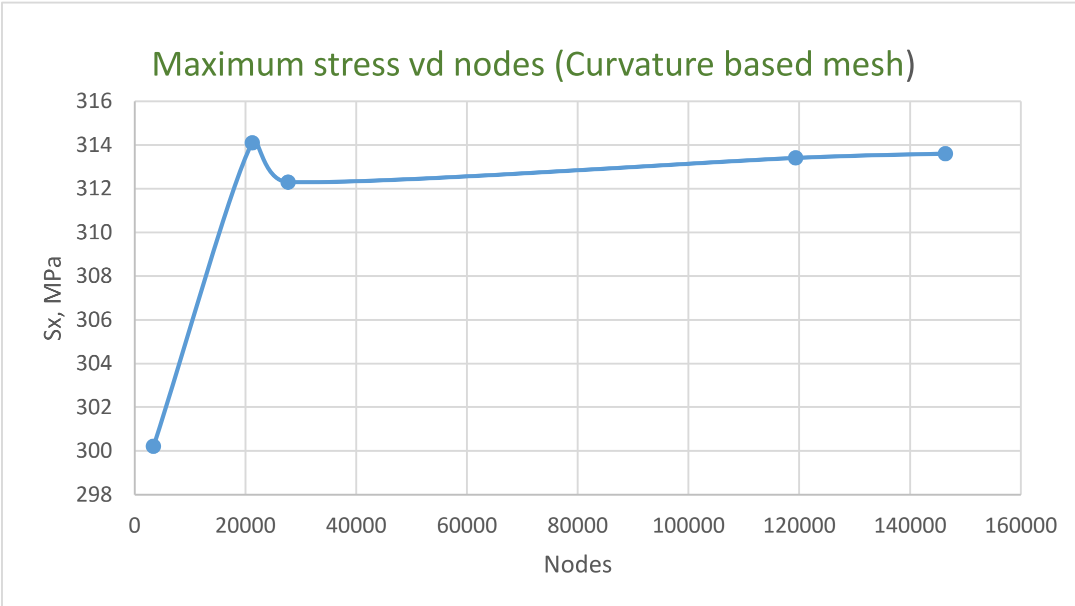


Figure 14: Curvature based mesh, second order, global refinement (Stress vs Nodes)

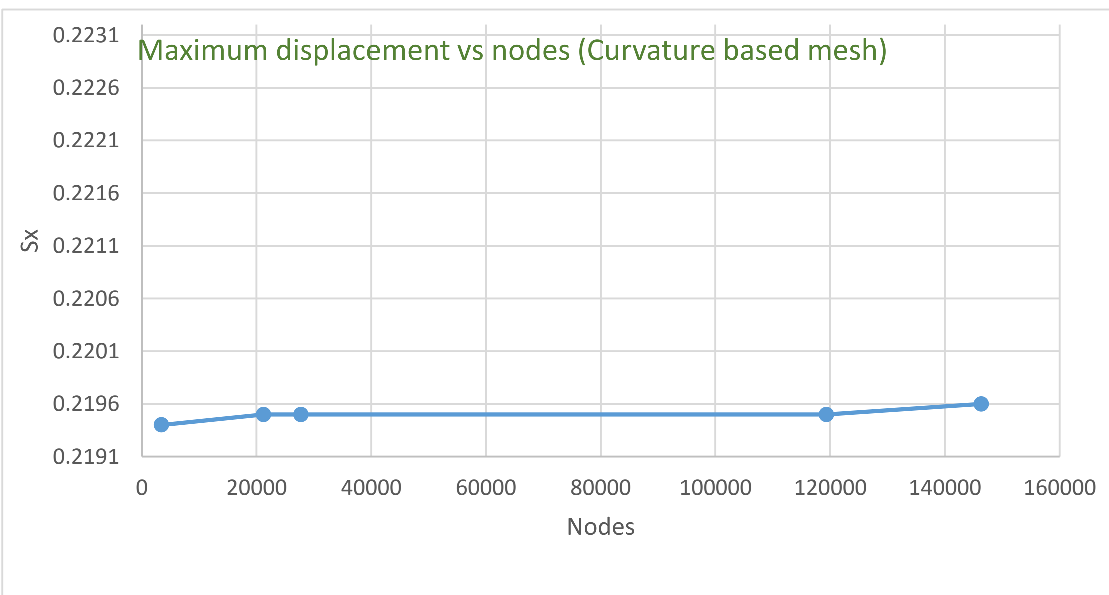


Figure 15: Curvature based mesh, second order, global refinement (displacement vs Nodes)

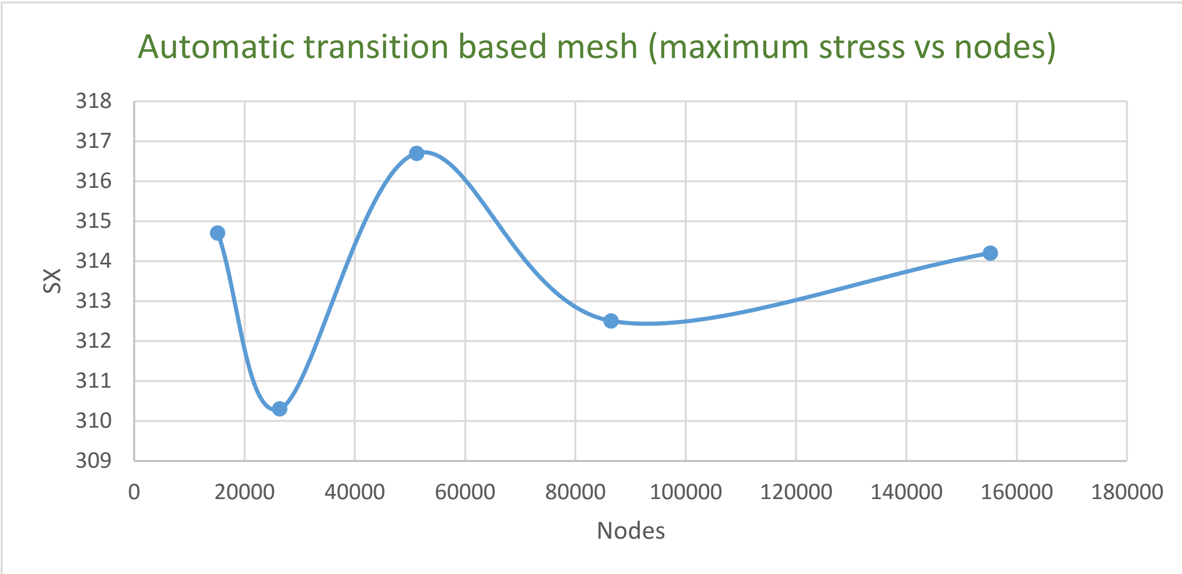


Figure 16: *Auto transition based mesh, second order, global refinement (stress vs Nodes)*

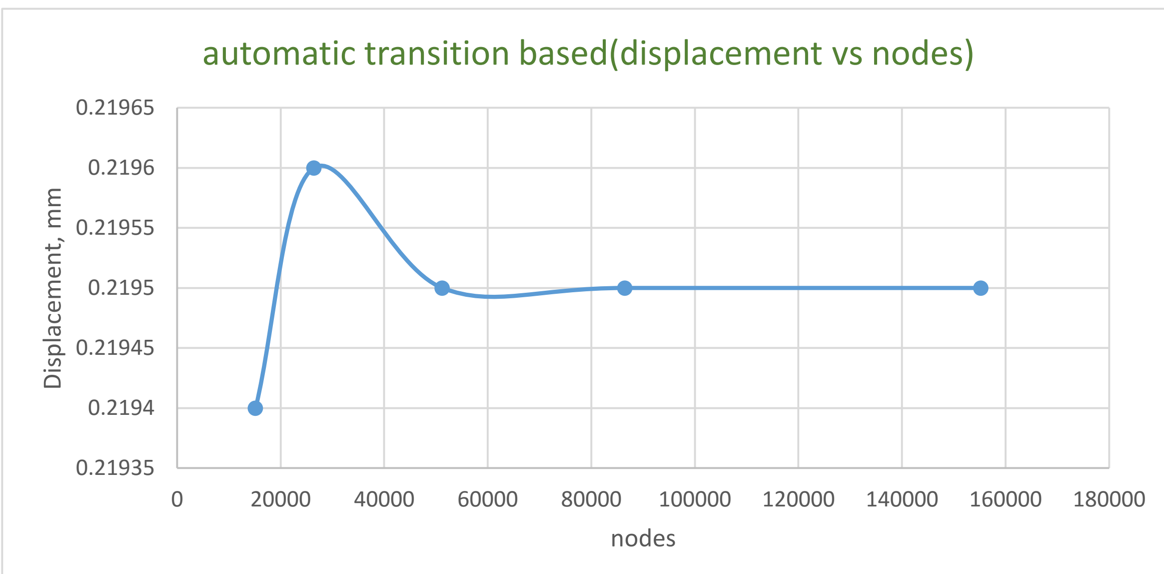


Figure 17: *Auto transition based mesh, second order, global refinement* (Displacement vs Nodes)

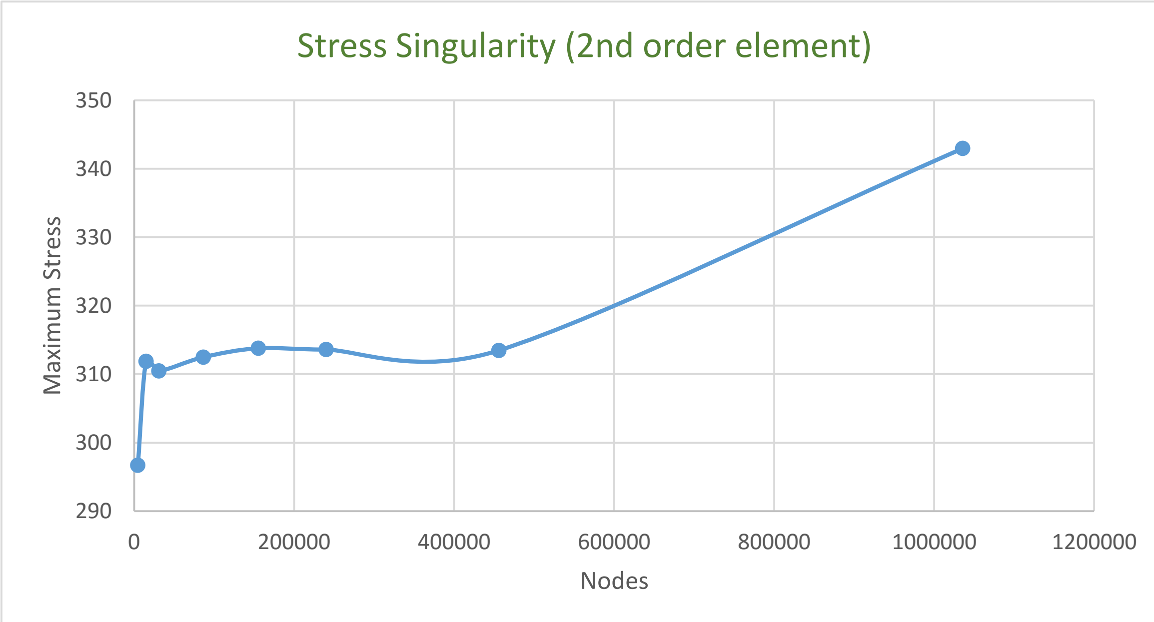


Figure 18: 2nd order stress vs nodes, singularity

Discussion:

The maximum stress and displacement of the plate was calculated analytically using the given formulas. The maximum stress was 310.5 MPa and the maximum displacement found was 0.2mm. To check the validity of the FEM solution, I conducted numerous experiments changing elements, element size, and orders. The steps were repeated to obtain convergence, so that I will be able to check the FEM solution and compare it with the analytical solution.

By analyzing Figures 6, 8, 10, 12 and 14, we see that maximum stress plot somewhat converges near 310 MPa. For the 1st order elements, the convergence happens at 280MPA, which is barely close to the analytical value. On the other hand, for 2ndorder elements the plot converged at 312 MPa which was very close to the analytical solution. The plot didn't converge for local refinement at the edges of notches and, instead, the stress plot was more irregular. This could have been caused by the high stress concentration in that area. However, according to Figures 7, 9, 11, 13, and 15, the maximum displacement plot converged at 0.2195mm, which is very close to the analytical value 0.2 mm. Therefore, the FE solution gives validity to our model.

While it may not be possible to obtain a 100% accurate result, we can achieve better accuracy by refining the model. In the tables and graphs, we see the plot converges with the increasing number of nodes. The result gets closer to the analytical result with the increasing number of nodes. As shown in the data tables, the 2nd order elements give more accurate results than the 1st order elements.

In order to get a hands on experience, and to obtain the convergence plot, I performed a lot of refinements. By using different refinements, I was able to control the mesh in the model and even mesh in a specific region (local refinement). Obtaining the convergence plots and valid FEM solutions would be impossible without refinements.

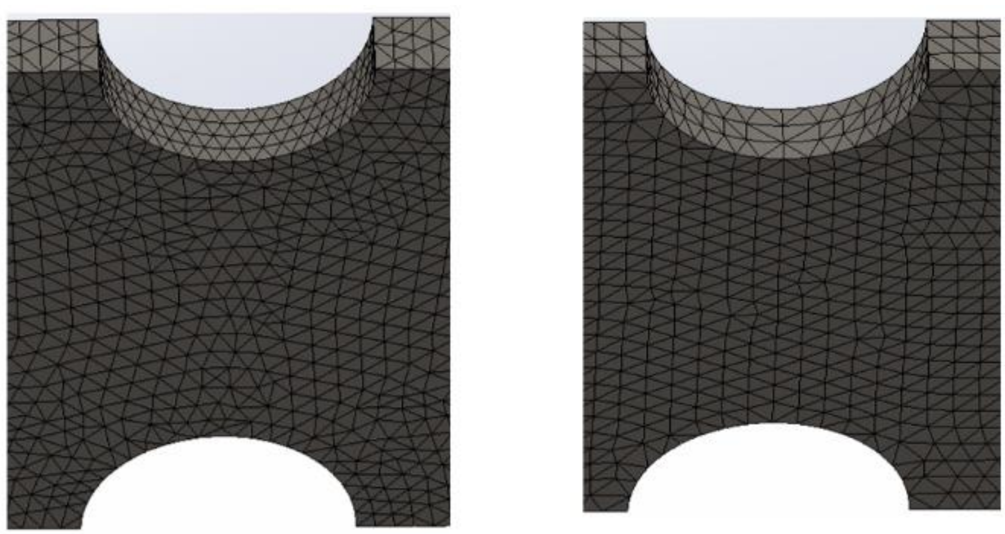
The error was calculated based on analytical result.

Global element size, mm	Sx (1st order element) MPa	Error %	Sx (2nd order element) Mpa	Error %
11.4	218	29.7%	280.4	9.7%
3.646	296.7	4%	312.4	0.5%

h-refinement is a process of controlling mesh by increasing element number by decreasing element size. I used this process to obtain a more valid result. Since element size decreases, it would give us a better result, but it takes longer time to create the mesh and obtain the result. For instance, I obtained 218 MPa for normal stress after conducting a study in FEA using 1st order element with global element size 11.4 mm. It had 29.7% error relative to the analytical solution. Then, I conducted another experiment using the smaller element size 3.646 mm which resulted in a more accurate approximation. This time the maximum stress was 280.4 with a 9.7% error relative to the analytical solution. This proves that it is possible to get a better approximation of result by performing h-refinement. However, if we keep decreasing the element size, stress singularity may occur, which I will discuss later in this report.

p-refinement is another process that affects the number of nodes present on each element. For this refinement, mesh quality was increased to 2nd order and experiments were performed using global and local element sizes. A tetrahedral element has 4 nodes in 1st order mesh, while it has 10 nodes in the 2nd order mesh. Eventually, the meshing around the curve or fillet is more consistent compared to 1st order meshing. Hence, it is more convenient to obtain a convergence plot using p-refinement. The stress of using 2nd order meshing is 296.7 MPa with an error 4% relative to analytical result, but for the same global element size 1st order element meshing gives us a result with a huge error.

Local refinement was used in this experiment to find stress in notches and edges. It is an effective process to obtain better stress approximation, but it has a higher computational cost. We will get more nodes for the decreasing element size in specific areas (Figure 5). From Figures 11 and 13, we can see that the displacement plot converges. In Figure 10, the plot is irregular at first, but eventually it tends to be convergent with the increasing nodes, even though the stress plot for the notch edge doesn't converge. Instead, we obtain different maximum stress with increasing nodes. It might be due to stress concentration on sharp edges, or due to stress singularity.



Curvature based mesh (high quality) Standard based mesh (High quality)

Figure 19: 2nd order element meshing

I also performed some experiments using Curvature based meshing. It was faster than standard based meshing. It created more elements compared to standard meshing. Besides it reaches to convergence fast. Figure (14) and (15) are respectively maximum stress and displacement convergence graph. The maximum stress is 313.4 MPa for the highest number of nodes.

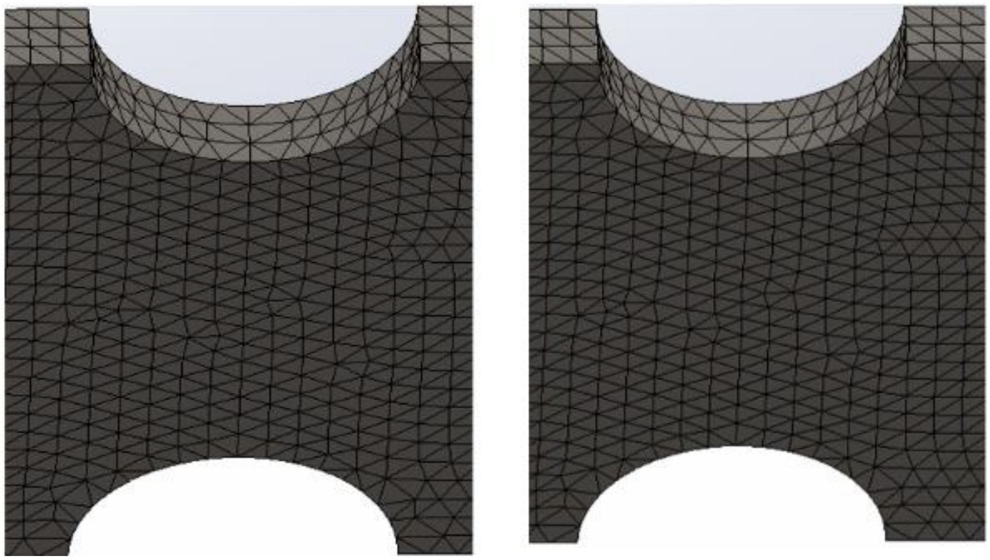


Figure 20: Automatic Transition mesh (left), standard meshing (right)

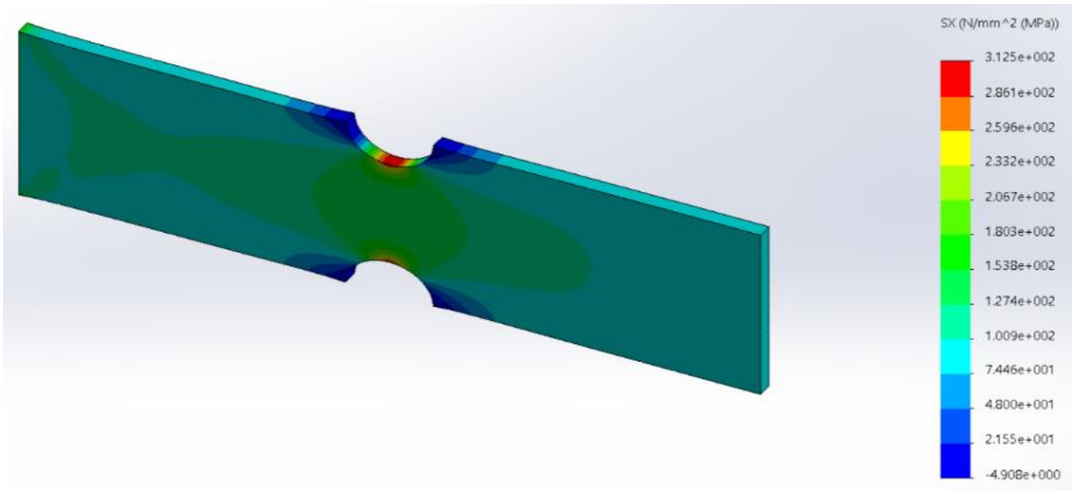
Automatic transition applies mesh control automatically to small features, holes, and fillets. I was curious how it would generate the meshing, but the results were lackluster. I was expecting to see smaller elements at the notch surface, but it wasn't different than high quality meshing with global element sizes (Figure 20). As seen in table 6, the stress increases and decreases again with the increasing number of nodes. This means it doesn't give us a stress convergence plot (Figure16), but it does give us a displacement convergence plot (Figure 17).

Lastly, I wanted to perform an experiment on stress singularity. If the stress becomes infinity, it is called stress singularity. The stress should be infinity in perfectly shaped edges, but there are no perfectly shaped edges in real world. For instance, if we keep decreasing the element size and perform mesh analysis in a given object, the plot will converge first. However, it can become non-convergence and stress becomes infinity if we keep decreasing element size (class notes, G. Benenson). In Figure 18, the stress converges first, then becomes infinity with increasing nodes.

It is not possible to get accurate result using Solidworks, but it does give us a good approximation of the solution. we can easily estimate the failure load of a building due to earthquake, or measure the strength of a cyclone using finite element analysis. Sometimes its possible to get wrong solution, but it does give us a estimation of posible outcome. Thus, it can be a very useful tool for obtaining estimation for stress and fatigue load for complicated real world problem.

I learnt a lot while performing the simulation studies. I became familiarized with all the refinements, mesh, Aspect ratio, jacobian points while conducting the experiments. Solidworks is an amazing tool to estimate stress, displacement and strain, eventhough it could still use some improvements. I would suggest to update automatic transition so that it create meshes in a proper format. SolidWorks would also be more effective if it would create a simple text file with all the data for each simulation study that was run so I wouldn't have to re-write the data in my notebook. That way we could easily have a back up for the data, and it would be more time efficient.

Appendix



Stress along the plate

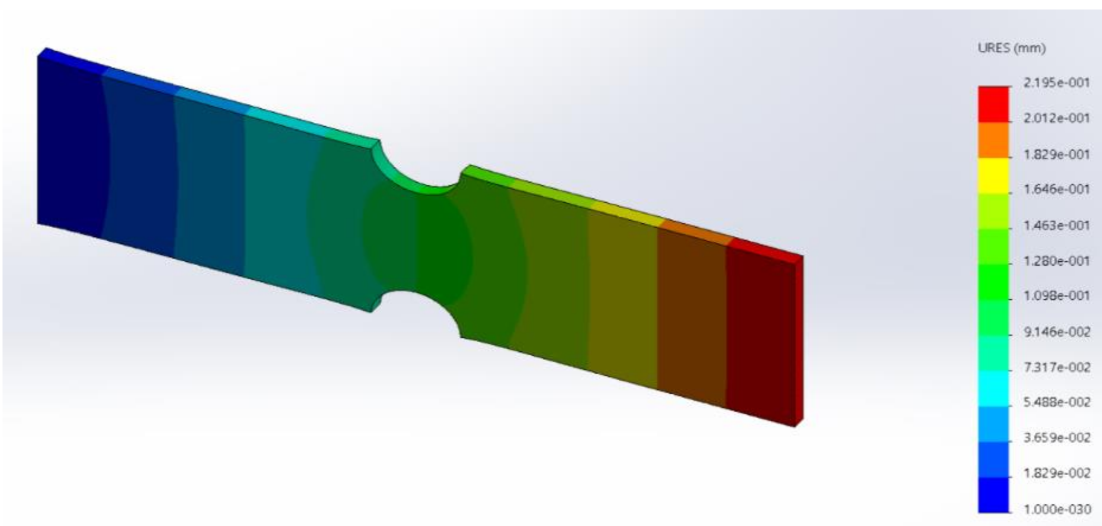


Figure 19: Displacement along the plate

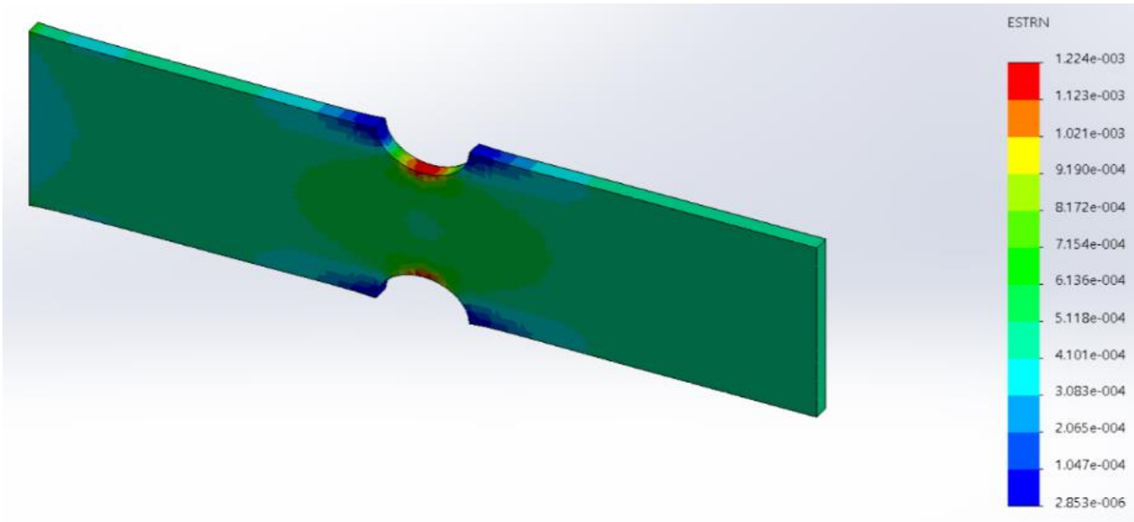


Figure 19: Strain along the plate