MEC 530: APPLIED STRESS ANALYSIS

FINITE ELEMENT PROJECT

Submitted by: Shashank Sharma, 110533045

Introduction

The problem being solved in this report is to determine the behavior of a standard fracture test component known as "Compact Tension Specimen" resembling a rectangular plate with a fracture pattern when a certain amount of load is applied to it. First we model the part with the standardized dimensions given by ASTM. Then we go on to analyze the effect of given loading and find out the stress concentration factor for the given fracture specimen using our FEM analysis. We also compute the stress concentration factor analytically using formulas from engineering handbook and compare it to experimental value. This gives an insight into how the specimen will fracture and the critical loading conditions for fracture.

Geometrical Model

The part geometry of fracture specimen being analyzed is displayed in the Figure 1 below as modelled on Autodesk Inventor. The first step is to model this geometry on Abacus accurately. The value of a was calculated to be .025m as my SBU ID ends with number 5.

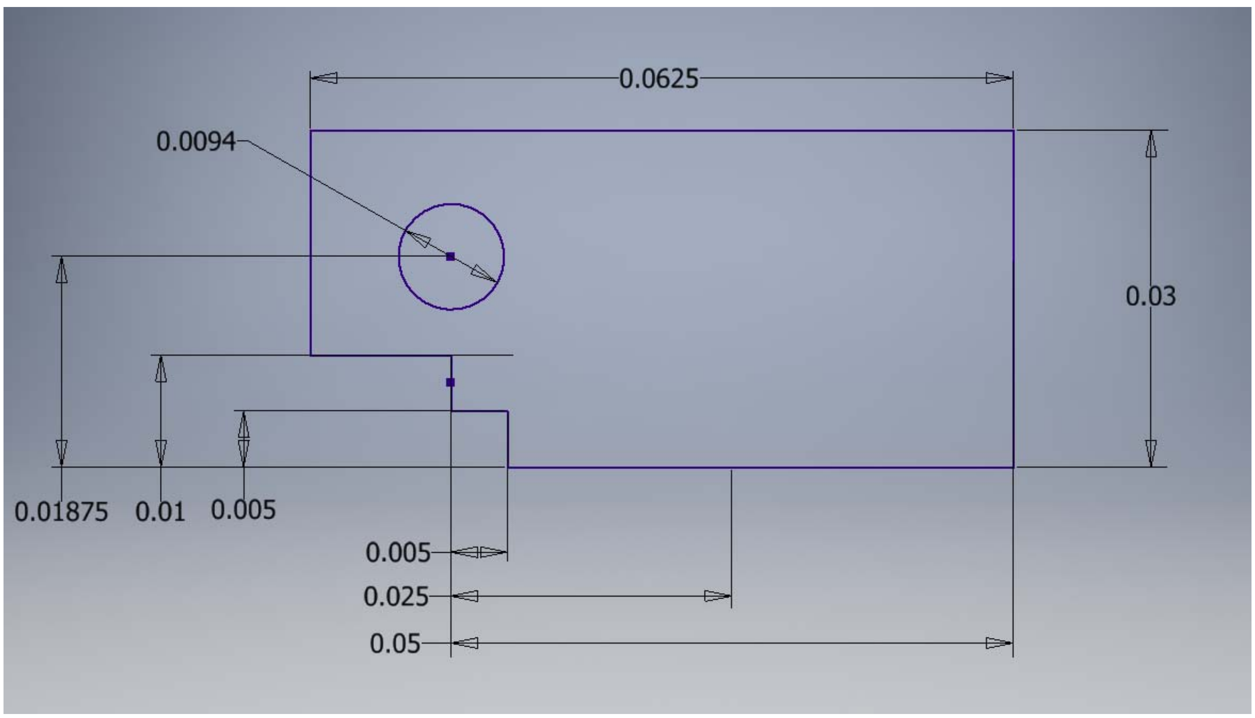


Fig 1. Geometry

Material Properties

For the analysis of our models, we have taken the material to be steel having E=200GPa and ν =.3. The thickness of the plate has been taken as 1m.

Meshing

Plane strain elements (CPE4) have been used to compute the analysis. A Structured and Quad type mesh has been created to represent the model using mesh nodes. The model is split into parts to gain control over the mesh. To maximize the accuracy, denser elements and web-type mesh around the crack-tip is generated. Mesh of Model is shown in Figure 2 below. Mesh at crack-tip is shown in Figure 3 below. There is a total of 990 nodes and 902 elements in the mesh.

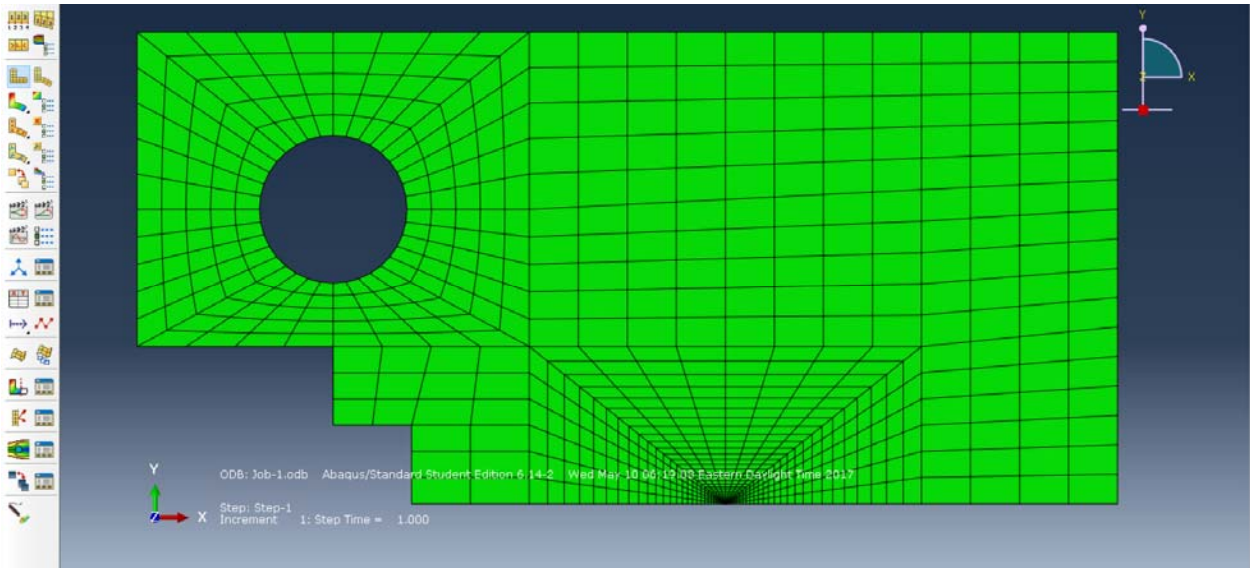


Fig 2. Structured meshing of the Geometry

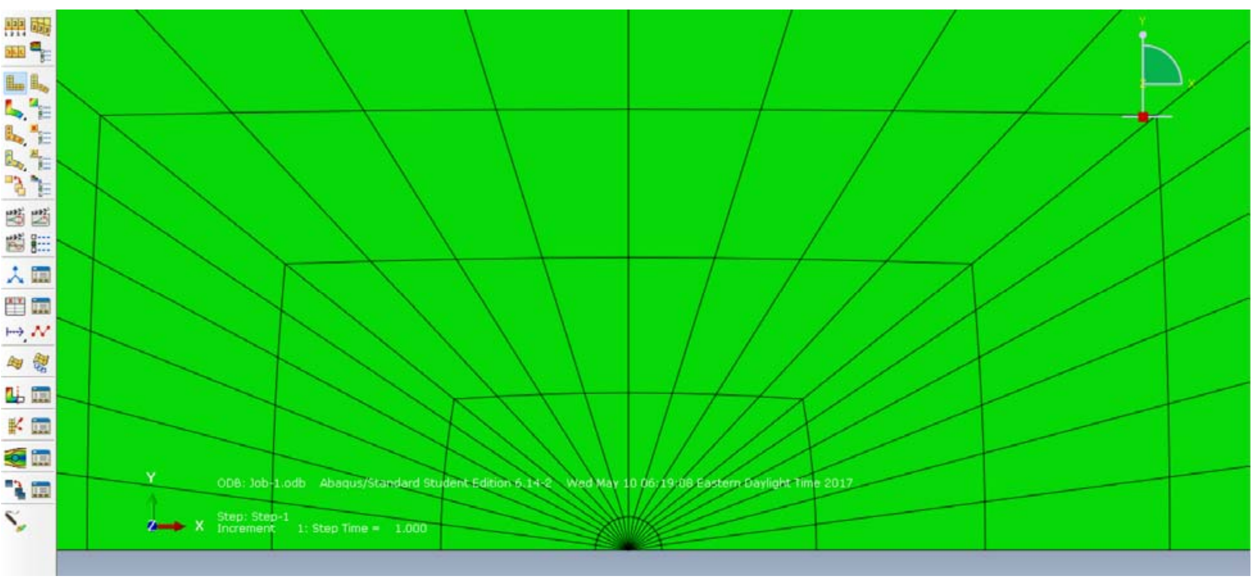


Fig 3. Web-type meshing at crack-tip

Loading and Boundary Conditions

We have modelled only a half of the test specimen in our analysis. The top node in the hole carries a concentrated force of 100 kN (Figure 4). The bottom edge till crack-tip is constrained in Y direction using symmetric condition. The right bottom corner is taken to be fixed to keep the model in static equilibrium. No conditions are imposed on the other edges or the edge representing the holes.

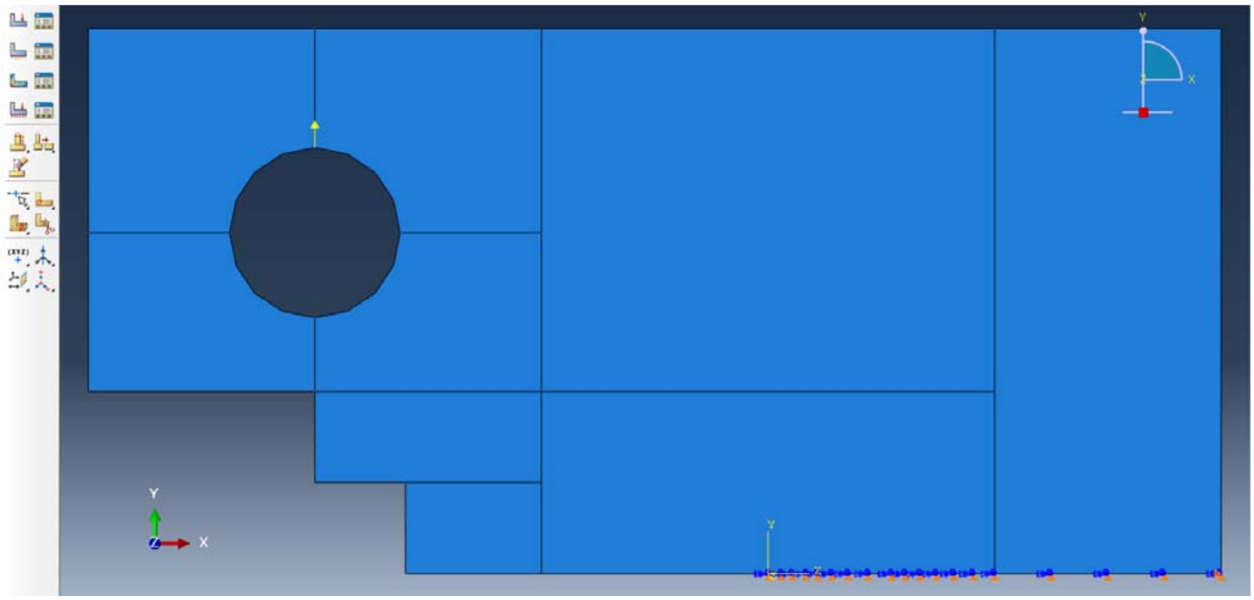


Fig 4. Loading and Boundary Conditions on Model

Results

Below in Figure 5,6,7,8 we display the Stress contour plots of Von mises and σY obtained for the model after Analysis with a zoomed in view at Crack-tip.

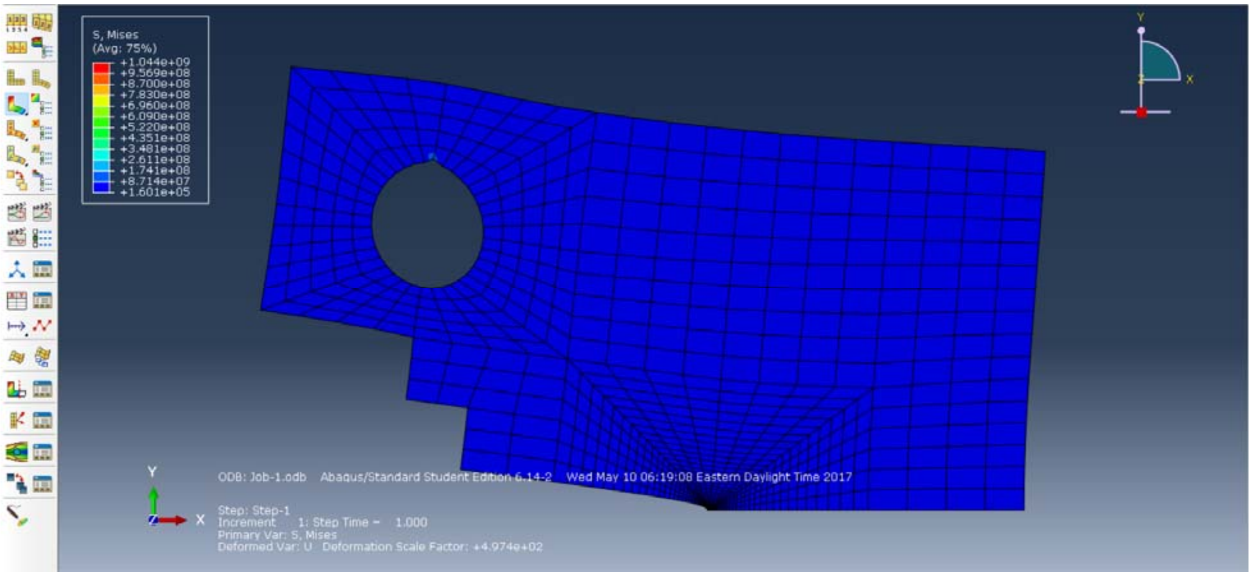


Fig 5. Von-Misses Stress Contour

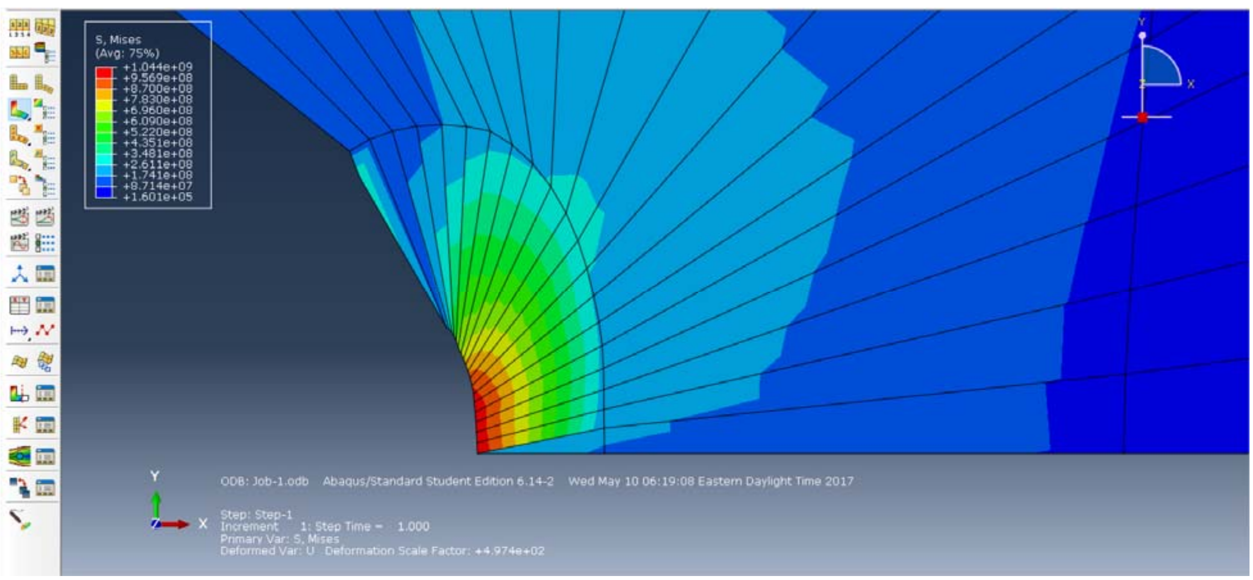


Fig 6. Von-Misses Stress Contour at Crack-tip

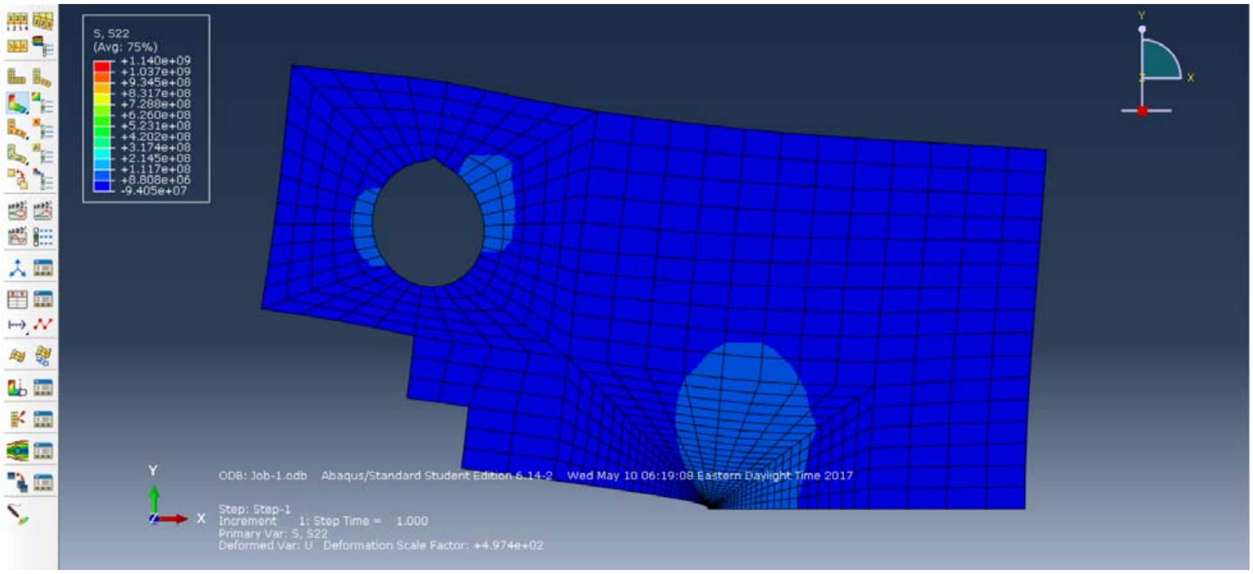


Fig 7. σ_Y Stress Contour

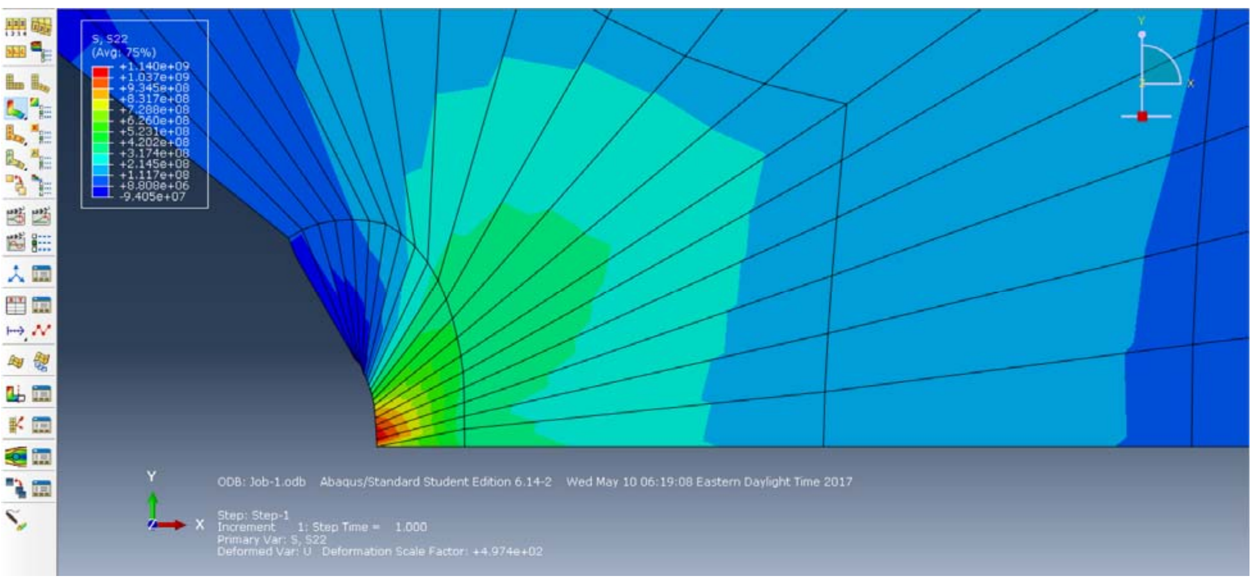


Fig 8. σ_Y Stress Contour at Crack-tip

Below in Figure 9, we display the deformation contour plot obtained for the model

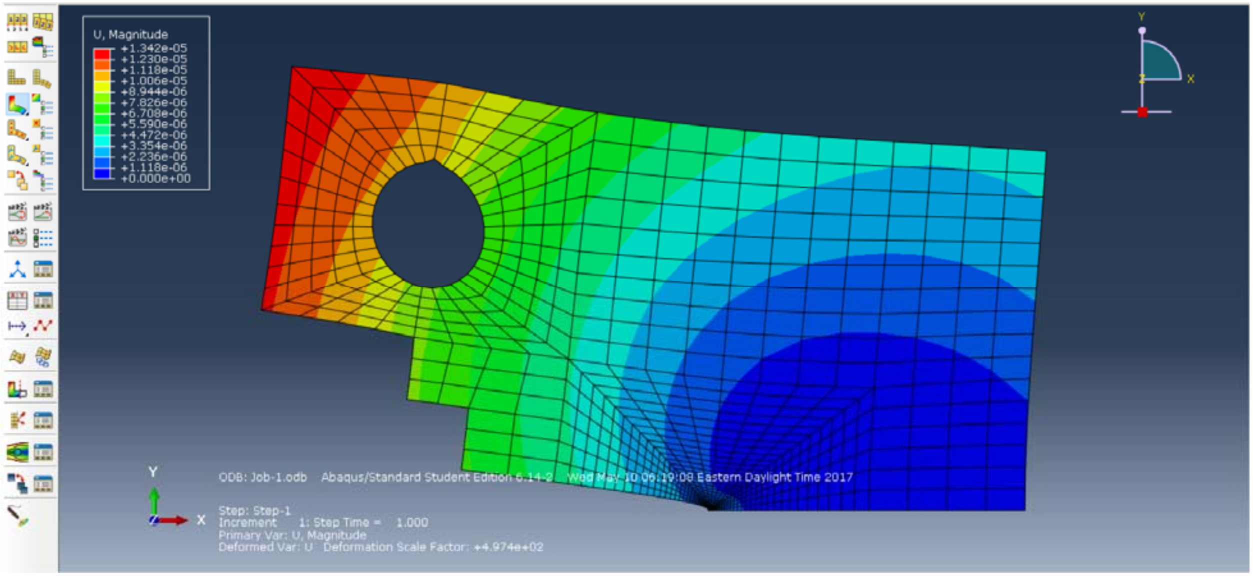


Fig 9. Deformation Contour

Stress Intensity Factor K_I

1. Analytical Stress Intensity Factor

To compute the stress intensity factor analytically for the given specimen, we use the following formula.

$$K_I = \frac{P}{\sqrt{W}} \frac{2 + \left(\frac{a}{W}\right)}{\left(1 - \left(\frac{a}{W}\right)\right)^{\frac{3}{2}}} \left[0.866 + 4.64\left(\frac{a}{W}\right) - 13.22\left(\frac{a}{W}\right)^2 + 14.72\left(\frac{a}{W}\right)^3 - 5.6\left(\frac{a}{W}\right)^4\right]$$

Substituting the value: $P = 100\ kN, \quad a = .025\ m, \quad W = .05\ m$

we calculate

$$K_I = 4335482.67\ Pa\sqrt{m}$$

2. Experimental Stress Concentration Factor using opening displacement behind the crack tip

To compute the stress intensity factor from the opening displacement behind crack tip, we use the following formula.

$$K_I^{disp} = \frac{E}{4(1-\upsilon^2)}\sqrt{\frac{2\pi}{l}}\,u_y^{f.e}$$

Substituting the value: $Ey = 200\ GPa,\quad \upsilon = .3,\quad l = 2.5 * 10^{-5}m,\quad u_y^{f.e} = 1.7 * 10^{-7}m$

We calculate

$$K_I^{disp} = 4682712.16\ Pa\sqrt{m}$$

$$Error = 8\%$$

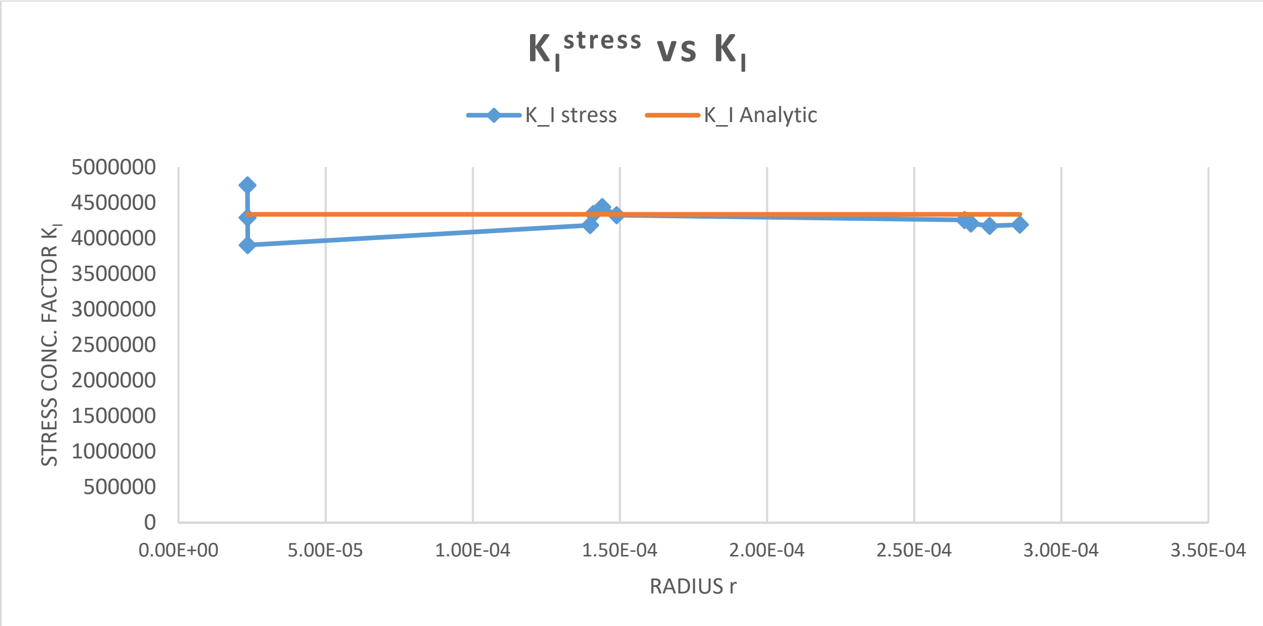
3. Experimental Stress Concentration Factor using opening stress ahead of the crack tip

To compute the stress intensity factor from the opening stress ahead of the crack tip, we choose 11 nodes which lie very near crack tip and have angles between 0 and $\pi/4$. We find the $\sigma_y^{f.e}$ at these points using the analysis results. The data for our analysis is given in Table 1. We then use the following formula to find K_I^{stress} and plot the results in Graph 1.

$$K_I^{stress} = \sigma_y^{f.e}\frac{\sqrt{2\pi r}}{\cos\left(\frac{\theta}{2}\right)\left(1+\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right)}$$

X-Coord	Y-Coord	Radius (r)	Angle ($\boldsymbol{\theta}$, deg)	$\boldsymbol{\sigma_y^{f.e}}$ (Pa)	$\boldsymbol{K_I^{stress}}$	$\boldsymbol{K_I}$	Error (%)
2.34E-05	1.38E-32	2.34E-05	3.37E-26	3.912E+08	4745138.8	4335480	9.45
2.31E-05	3.92E-06	2.34E-05	9.619348	3.596E+08	4290547.8	4335480	1.04
2.22E-05	7.73E-06	2.35E-05	19.21045	3.422E+08	3903843.6	4335480	9.96
0.00014	1.84E-32	1.40E-04	7.52E-27	1.411E+08	4183953.1	4335480	3.50
0.00014	1.94E-05	1.41E-04	7.925153	1.476E+08	4341105.3	4335480	0.13
0.000139	3.88E-05	1.44E-04	15.62002	1.541E+08	4437438.1	4335480	2.35
0.000137	5.79E-05	1.49E-04	22.88905	1.541E+08	4324129.7	4335480	0.26
0.000267	1.36E-32	2.67E-04	2.93E-27	1.039E+08	4257310.1	4335480	1.80
0.000267	3.64E-05	2.69E-04	7.766144	1.035E+08	4207493.6	4335480	2.95
0.000266	7.27E-05	2.76E-04	15.28661	1.045E+08	4173147.6	4335480	3.74
0.000264	0.000109	2.86E-04	22.36149	1.074E+08	4190623.5	4335480	3.34

Table 1. Node data for K_I^{stress} calculation using opening stress



Graph 1. Plot comparing K_I^{stress} with analytically calculated K_I

Behavior of the opening stress around the crack tip

We want to check our stress field accuracy by comparing angular variation of the computed stress with the angular variation of actual K-field stress. To do this we select 20 nodes at reasonable distance from crack tip with similar radial coordinates. The data for these nodes is displayed in Table 1 below. We again find the $\sigma_y^{f.e}$ at these points using the analysis results. Next we compute the normalized stresses for our model and analytic values using the formulas below. We then plot the results in Graph 2 to compare gradual change of σ'_y with Angle.

Normalized Stress :

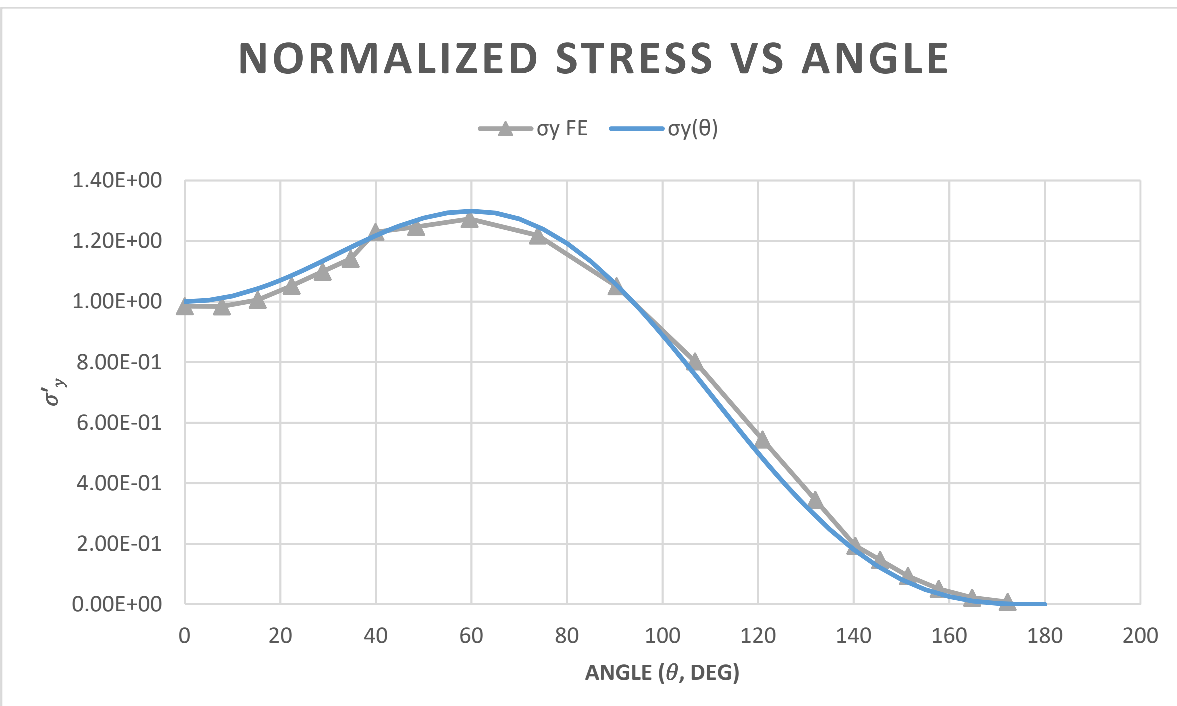
$$\sigma'^{f.e}_y = \sigma_y^{f.e}\frac{\sqrt{2\pi r}}{K_I}$$

$$\sigma'_y(\theta) = \cos\left(\frac{\theta}{2}\right)\left(1+\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right)$$

X-Coord	Y-Coord	Radius (r)	Angle (θ, deg)	$\sigma_y^{f.e}$ (Pa)	$\sigma'^{f.e}_y$	$\sigma'_y(\theta)$
0.000267	1.36E-32	0.000267	2.93E-27	1.04E+08	0.984687	1
0.000267	3.64E-05	0.000269	7.766144	1.03E+08	0.984158	1.011347
0.000266	7.27E-05	0.000276	15.28661	1.05E+08	1.006011	1.042474
0.000264	0.000109	0.000286	22.36149	1.07E+08	1.052313	1.086129
0.000262	0.000145	0.0003	28.85679	1.10E+08	1.099222	1.13391
0.00026	0.00018	0.000316	34.71383	1.11E+08	1.141421	1.179059

0.000257	0.000215	0.000335	39.92968	1.16E+08	1.230053	1.217534
0.000193	0.000217	0.00029	48.40973	1.26E+08	1.246831	1.268957
0.000128	0.000219	0.000253	59.62249	1.38E+08	1.273047	1.299003
6.34E-05	0.00022	0.000229	73.89073	1.39E+08	1.218438	1.248161
-1.4E-06	0.00022	0.00022	90.37002	1.23E+08	1.050722	1.054925
-6.6E-05	0.00022	0.000229	106.7829	9.18E+07	0.802453	0.7587
-0.00013	0.000219	0.000255	120.9033	5.91E+07	0.544874	0.483014
-0.0002	0.000217	0.000293	131.9652	3.50E+07	0.345752	0.292451
-0.00026	0.000216	0.000338	140.3294	1.83E+07	0.193917	0.177347
-0.00026	0.000181	0.000319	145.52	1.43E+07	0.146906	0.121018
-0.00027	0.000145	0.000302	151.3375	9.35E+06	0.093773	0.072113
-0.00027	0.000109	0.000289	157.7776	5.27E+06	0.051653	0.034722
-0.00027	7.32E-05	0.000279	164.7808	2.32E+06	0.022364	0.011448
-0.00027	3.69E-05	0.000272	172.2169	8.60E+05	0.008178	0.001557

Table 2. Node data for $\sigma'^{f.e}_y$ calculation using opening stress



Graph 2. Plot comparing $\sigma^{f.e}_y$ with analytically calculated $\sigma'_y(\theta)$

Discussions

As seen in the above Stress Contour Plots, the maximum stress concentration is in the areas around the loading point and crack tip shown by the red color pattern which is obvious.

The stress intensity factors were calculated for nodes ahead of the crack tip both analytical and experimental. These were then plotted against the radii as seen from the Graph 1. It is seen that the experimental values obtained via simulation are in most cases very near the analytical value.

We also observe that we get a more accurate result by finding Stress Intensity using opening stress rather than opening displacement.

In Graph 2, the stress values were normalized to obtain a graph plot against the angle. There were 20 nodes that were selected for tabulation which were located around the crack tip thus covering a semi circle of 180 degrees. The values when plotted in the graph shows an initial increasing trend. The values for both Normalization Stress initially increase to about a maximum of 1.3 after which it decreases uniformly to its minimum value. Both the analytic and experimental curves are very similar which substantiates our claim that the analysis is accurately performed.

Thus, we have a good simulated approximation of a real-life fracture specimen.

Improvements

Since the analysis done in the report was limited to a maximum of 1000 nodes, the mesh was not particularly very refined. A finer mesh could achieve a better accuracy to the analytical solutions.