FINITE ELEMENT ANALYSIS OF A FRACTURED SPECIMEN

Final Project report

Ishaan Karode
ishaan.karode@stonybrook.edu
SBU ID: 110457152

Objective

The objective of the project is to do finite element analysis of standard fractured specimen loaded under tension. Due to symmetry of the model only top half of the model is used in FEM. Von Mises and Stress contours of the model were obtained followed by the computation of the stress intensity factor (K_1) from the model using the required formula and compare with the exact solution and find the error between the exact solution and computed ones.

Description of model

The model is an ASTM compact tension specimen. The thickness of the specimen is 1m as it is relatively thick. Since the model is symmetrical, only the top half is modelled with required boundary condition. The material is structural steel and with young's modulus of 200 GPa and Poisson's ratio of 0.3. The width of the specimen is 50 mm. The crack length is found using the following equation

a= (0.45+2/100)*0.05=0.0235m.

Then the model is made according to the crack length with the dimensions as given in the figure below.

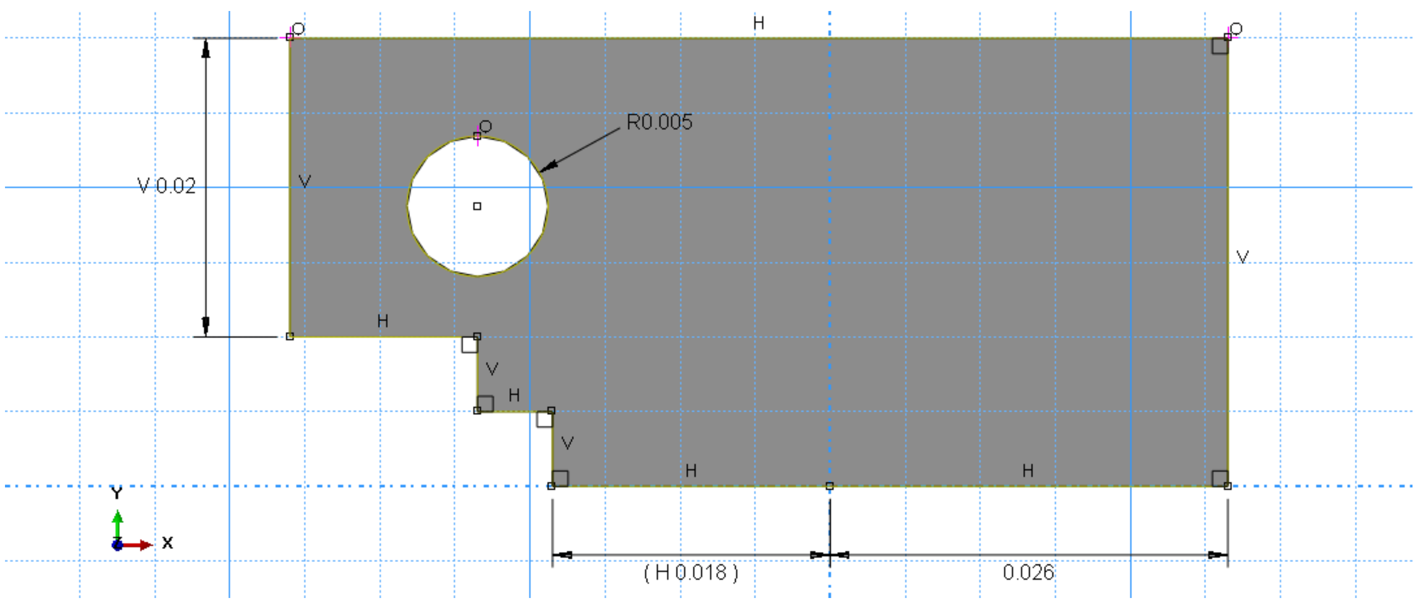


Figure 1 Top half of the model with dimensions in meters

Procedure

The model is created with 2D planar with approximate size of 1. Appropriate design and material properties are given. The section is created as solid and homogenous with plane strain of thickness 1. The assembly is created as independent. Then the model is given step size of 1. Then load is applied at the top part of the circle as 100kN.

The model is then partitioned to get good mesh for getting optimum results. A very small rectangle of 0.002m is made near the crack tip so as to get fine elements near the crack tip. The region near the circle is structured mesh with 9 elements in the side and 5 elements in the top. The region above the crack is also structured mesh with 6 elements on each side. The small rectangle near the crack tip is modelled with 7 elements on all side. After seeding, the mesh is done.

Next to create the crack tip, two tri noded element is created near the crack tip and they are combined to 4 node. After two four noded element is created, a set is made selecting the nodes right to them for the purpose of boundary condition. Then 5 nodes between the four noded elements is combined to get the crack tip. Next the boundary conditions are applied. First the U2 is constrained for the created set. Next U1 is constrained for the bottom left node. The partitioning and boundary condition is showed in figure 2.

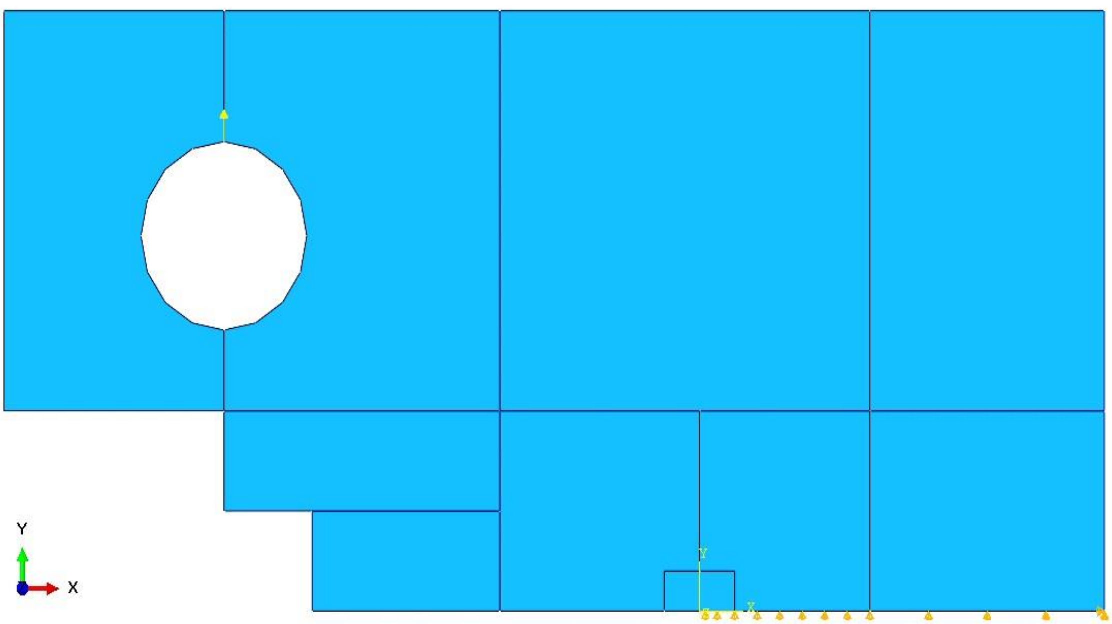


Figure 2 Partition, loading and boundary condition of the model

After applying loads and boundary condition for the model, the model is submitted for analysis. Deformed and un-deformed contours are obtained along with von Mises and Stress contour plots. **The total number of nodes is 692 and elements is 617**. Plain Strain condition CPE4 is considered for meshing

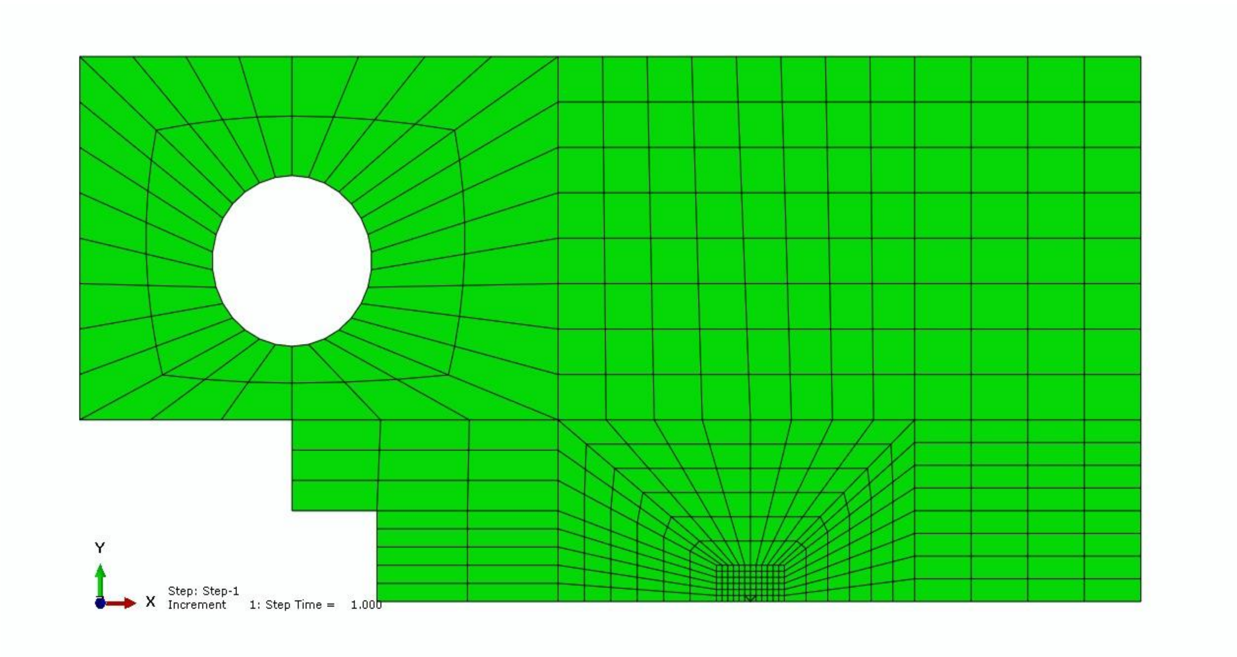


Figure 3 Mesh plot of the model

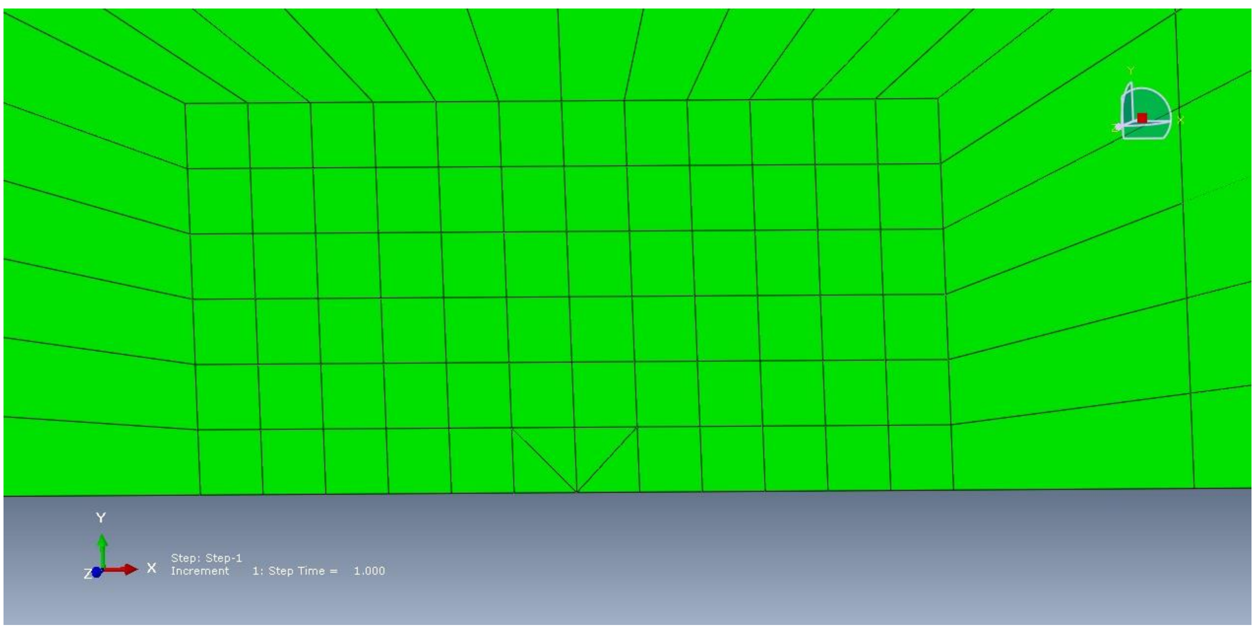


Figure 4 Mesh near crack tip

The von Mises contour plot of the model is obtained with maximum stress as $2.941*10^8$ Pa. The stress is concentrated on the tip as expected and decreasing away from the crack. Figure 5 shows the Von Mises contour plot of the whole model. Figure 6 gives a better understanding as it gives the Von Mises stress distribution.

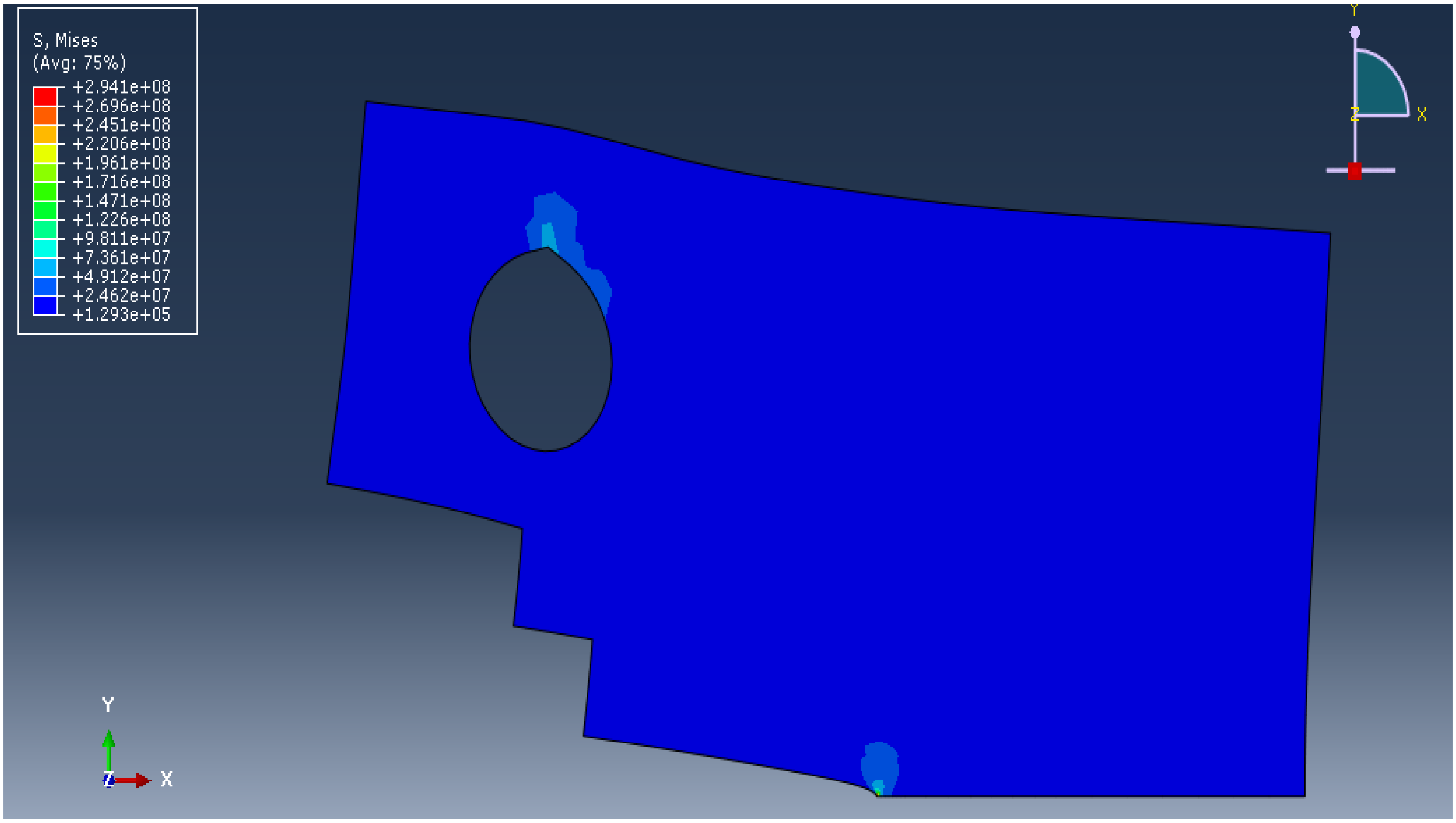


Figure 5 Von Mises contour plot of the model with units of Pa

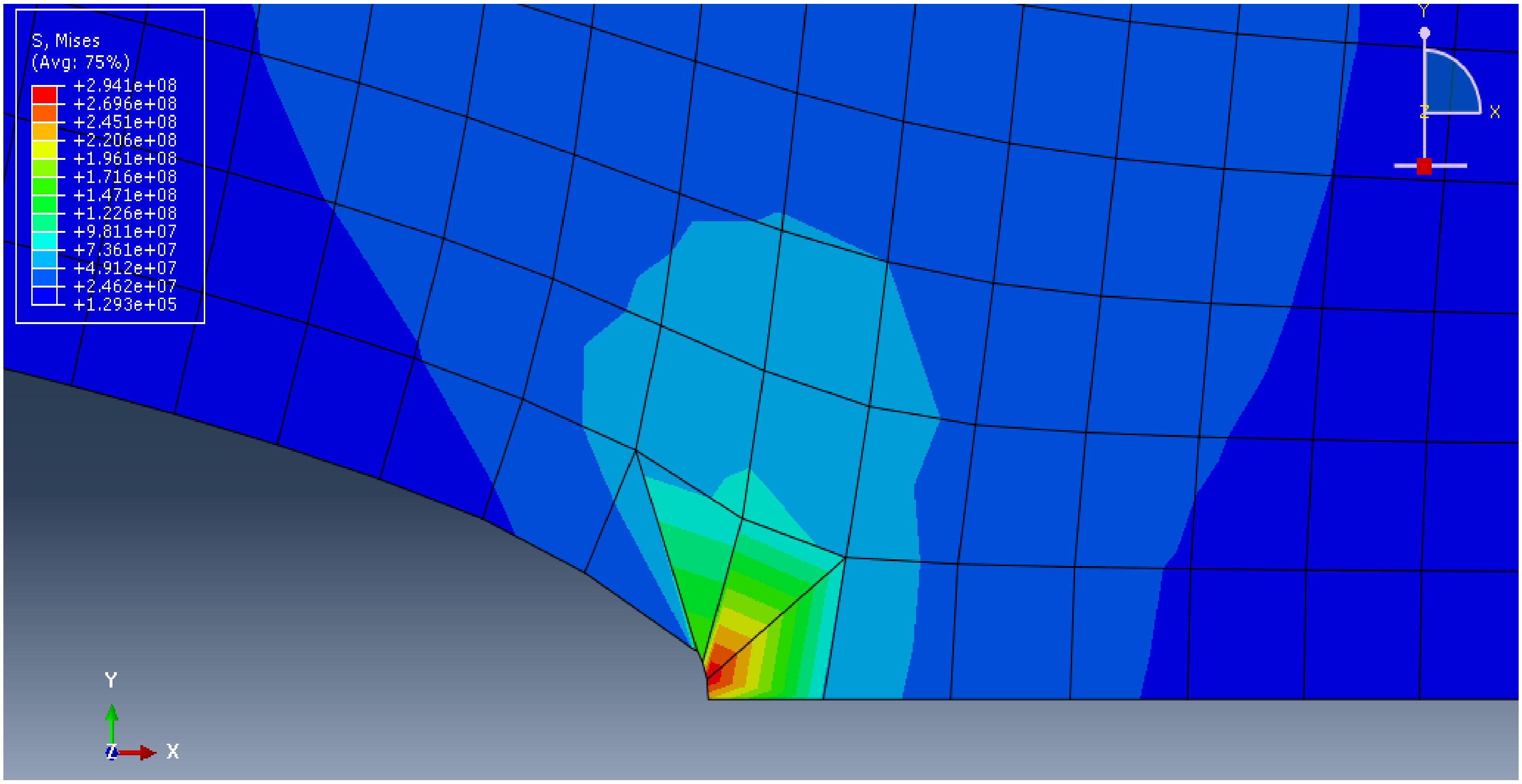


Figure 6 Von Mises contour plot near the crack tip with units of Pa

The S22 plot of the model is shown in figure 7.

The stress value was found to be 2.615*10^8 Pa. The stress is low at the boundary and its high near the crack tip. Figure 8 give clear details about the stress near the crack tip.

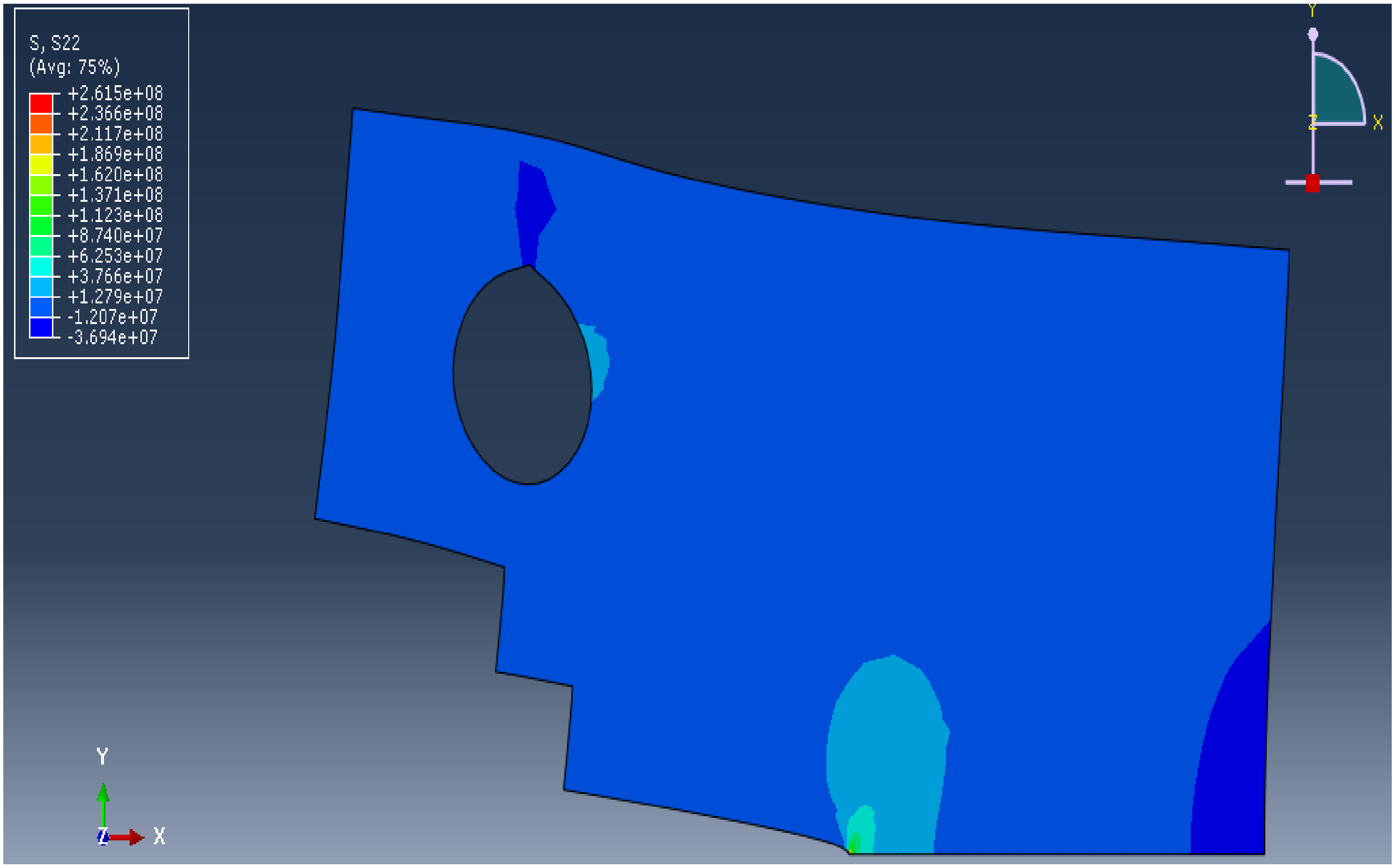


Figure 7 Stress contour of the model with units of Pa

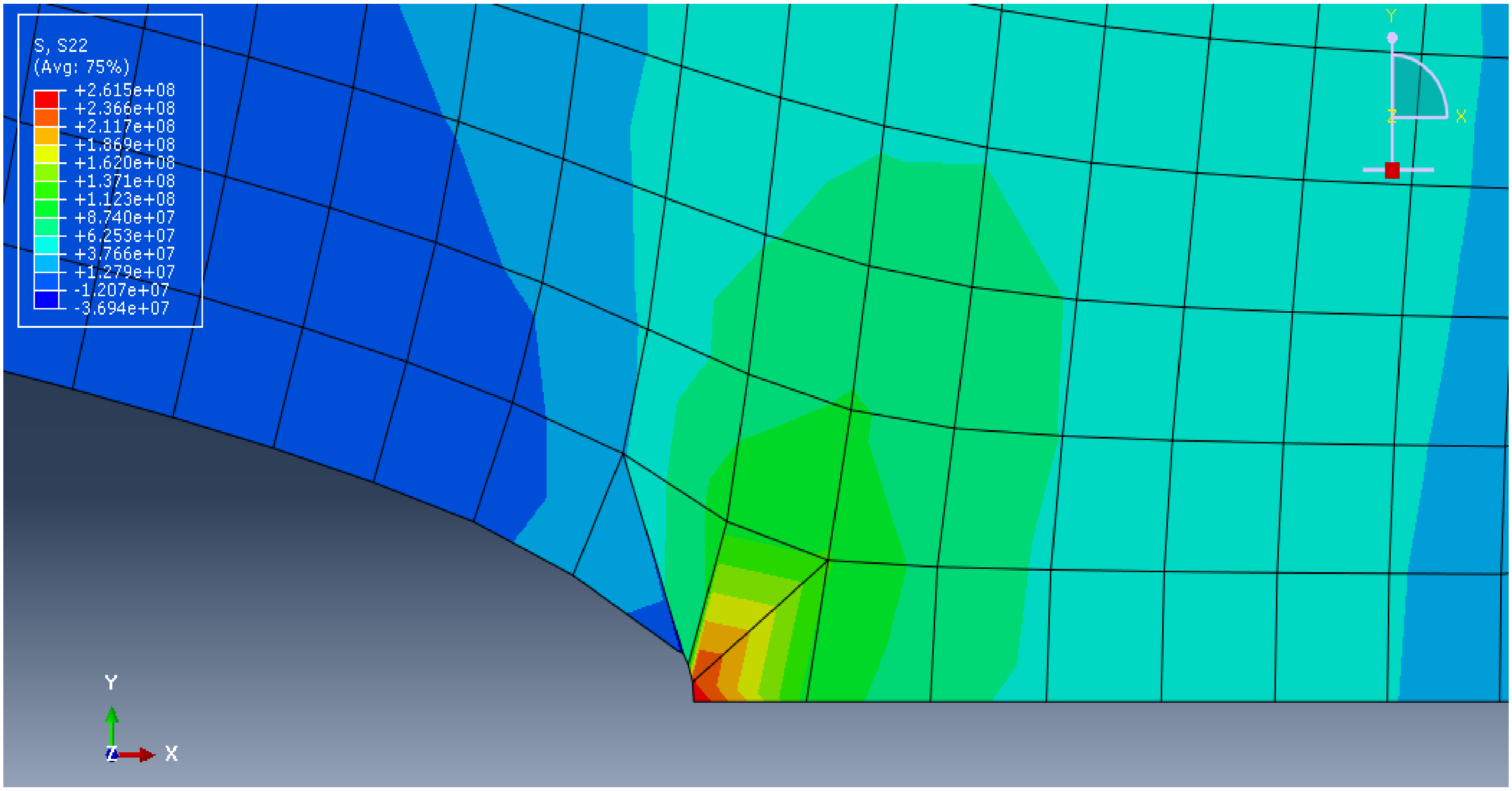


Figure 8 Stress Contour near the crack tip with units of Pa

Calculations

(a) The Stress Intensity is calculated using the following hand book formula

$$K_I = \frac{P}{\sqrt{W}} \frac{2 + a/W}{(1 - a/W)^{3/2}} \left[0.866 + 4.64\left(\frac{a}{W}\right) - 13.22\left(\frac{a}{W}\right)^2 + 14.72\left(\frac{a}{W}\right)^3 - 5.60\left(\frac{a}{W}\right)^4 \right]$$

Substituting, a=27mm; w=50mm; P=100kN, we get

K_I =3.95506 MPa.

(b) From the opening displacement behind the crack tip,

U=4.3361*10^{-7} m

l=2.16636*10^{-4} m

Then value of K is found using the formula

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

Here E= 200GPa; υ=0.3

After substituting all the values, it was found that K_I^{disp} =4.0564 MPa $m^{0.5}$

The error is found to be 2.5%.

From the opening stress ahead of the crack tip 6 different nodes are selected whose coordinates lie within r < (a/10) and 0 < θ < (π/4), where a is the crack length.

a/10 = 0.00235 m

Table 1 Values of stress and x and y coordinates

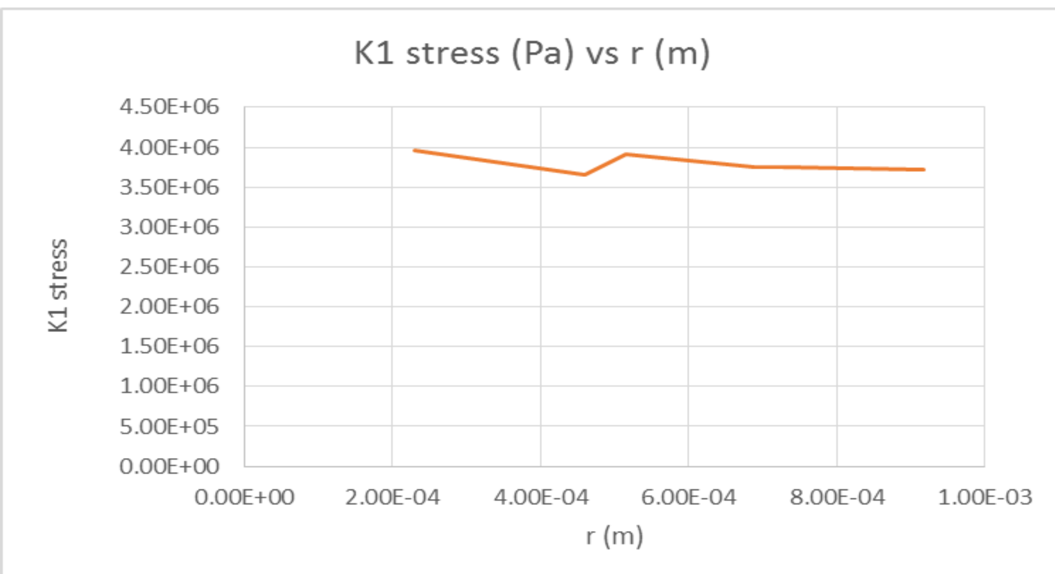
S22(Pa)	X(m)	Y(m)	R	θ
1.04E+08	2.30E-04	0	2.30E-04	0
5.90E+07	6.89E-04	2.34E-04	7.28E-04	18.8
7.70E+07	4.59E-04	2.34E-04	5.15E-04	27.012
6.8E+07	4.59E-04	0	4.59E-04	0
5.70E+07	6.89E-04	0	6.89E-04	0
4.90E+07	9.18-04	0	9.18E-04	0

The K_I stress is estimated from K field solution at each node

Table 2 Values of r and K

r (m)	$K1^{stress}$ (MPa)	K	Error(%)
2.30E-04	3.95E+06	3.955	0.064
4.59E-04	3.65E+06	3.955	7.711
5.15E-04	3.91E+06	3.955	1.10
6.89E-04	3.75E+06	3.955	5.1833
7.28E-04	3.76E+06	3.955	4.93
9.18E-04	3.72E+06	3.955	5.941

Graph 1 Plot of K1 Vs r



(3)

Behavior of opening stress around the crack

A set of nodes is chosen around the crack tip and their values are noted.
R and theta are calculated from those values

Table 3 Values of Stress and coordinates from the nodes around the crack

S22 (MPa)	x(m)	y(m)	r	θ (degrees)
6.80E+07	4.59E-04	0	0.000459	0
7.70E+07	4.59E-04	2.34E-04	0.000515206	27.01
7.70E+07	4.59E-04	4.68E-04	0.000655519	45.556
9.00E+07	2.30E-04	4.68E-04	0.000521463	63.82
7.60E+07	-1.00E-06	4.68E-04	0.000468001	90.122
4.00E+07	-2.17E-04	4.68E-04	0.000515861	114.84
1.60E+07	-4.33E-04	4.68E-04	0.000637584	132.797
3.00E+06	-4.33E-04	2.34E-04	0.000492184	151.63
8.00E+06	-4.33E-04	6.18E-07	0.000433	179.918

(c)Then the normalized stress is calculated using the equation

$$\tilde{\sigma}_v^{\text{f.e.}} = \sigma_v^{\text{f.e.}} \sqrt{2\pi r} \;/ K_{\mathrm{I}}$$

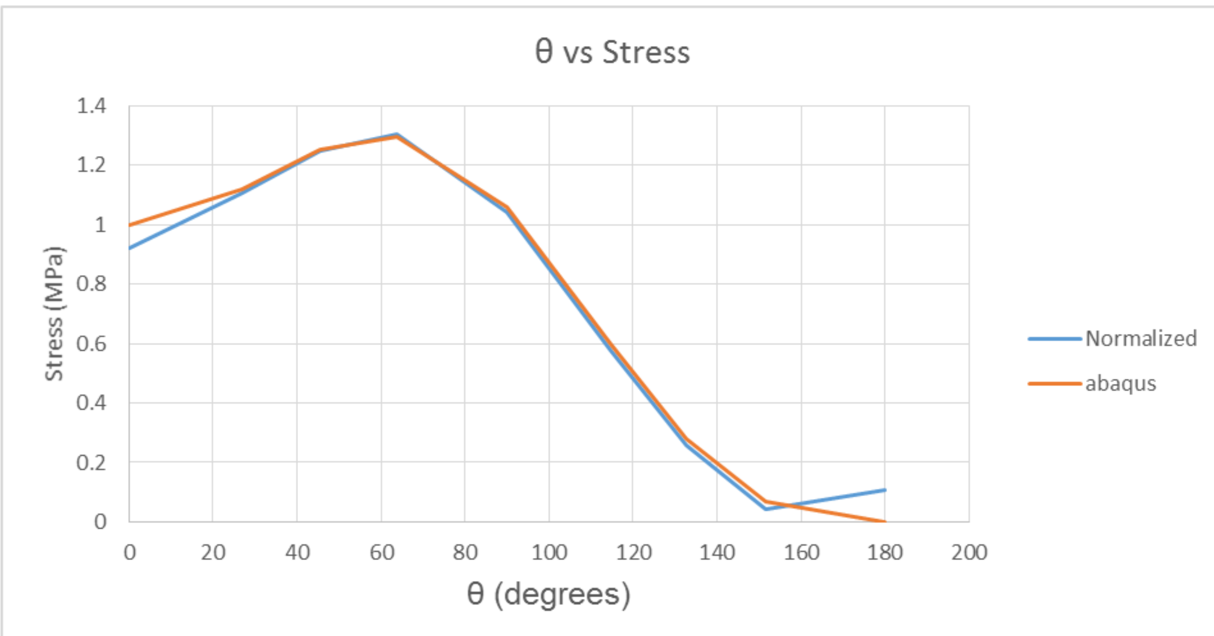
(d) The angular variation of stress is found using the equation

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

Table 4 Angular Variation of Stress

θ(degrees)	S22 Normalized	S22 Abaqus (MPa)
0	0.9230	1
27.01	1.1074	1.1198

45.556	1.2491	1.2537
63.82	1.3022	1.2953
90.122	1.0417	1.0587
114.84	0.5756	0.5995
132.797	0.2559	0.2797
151.63	0.0421	0.0700
179.918	0.1054	0



Discussion

As expected, the stress was high at the crack tip. The stress was seen decreasing away from the crack tip. The error between the calculated and theoretical Stress intensity was 2.5%. From the first plot it was found that the stress intensity factor decreases away from the crack tip. The angular variation of normalized stress and normal stress were plotted and it was found that at the beginning the normalized stress was low compared to model one but as the time progresses the stresses were found to be same.

The accuracy of the results obtained can be further increased by doing spider mesh near the crack tip that would render better angular variation of stress near the crack tip and by making many elements near the crack tip may also prove useful to get accurate results.