Stony Brook University
Mechanical Engineering

MEC 530 (APPLIED STRESS ANALYSIS)

Project Report

FINITE ELEMENT ANALYSIS OF FRACTURE SPECIMEN

Submitted by

SRAVANA SUMANTH KONERU

(ID: 112526331)

Date: May 13, 2020

1. INTRODUCTION:

In this project, Finite element analysis is carried out on a standard ASTM (American Society for Testing and Materials) **Compact Fracture specimen** using **ABAQUS** to study the fracture. Investigating a fracture is important to prevent the catastrophic failure of the engineering structures. These failures are majorly caused by poor design, materials, manufacturing, stress concentration, environmental conditions, and degradation.

The **finite element method** is one of the most common solution techniques used in the fracture analyses. Stress intensity factors for many configurations have been obtained using the finite element analyses. In majorities of cases, stresses and displacements computed from the finite element analyses are used to calculate necessary fracture parameters or relevant fracture variables via various computational procedures.

Figure 1 Standard Fracture test specimen

To solve a problem, the FEM subdivides a large system into smaller, simpler parts that are called finite elements. This is achieved by a particular space discretization in the space dimensions, which is implemented by the construction of a mesh of the object. Studying or analyzing a phenomenon with FEM is often referred to as finite element analysis (FEA). In all finite element analyses (whether fracture or not), success to obtaining good solutions depends on the mesh design. A poorly designed mesh can yield erroneous results. Naturally, other factors, such as precise representation of material behavior, can also influence the accuracy of the solutions.

In this project we conduct a **finite element analysis on a compact Fracture test specimen** (Figure.1).

2. PROCEDURE:

For the finite element analysis, a fracture test specimen made of **Structural Steel (Young's modulus = 200GPa and Poisson's ratio = 0.3)** is used. Also, while carrying out the finite element analysis, we assume that the specimen is relatively thick (**unit length**), so that **plane strain condition** exists.

2.1 MODEL DESCRIPTION

As mentioned above, we use a standard fracture specimen. From figure. 1 we can observe that the specimen is symmetric about x – axis. So, we consider only half of the specimen (upper half) and impose **symmetry conditions** on that. The dimensions of the model used are given below in **Figure 2**. with **crack length = 0.0276m.** (**All Dimensions in the figure are in meters**)

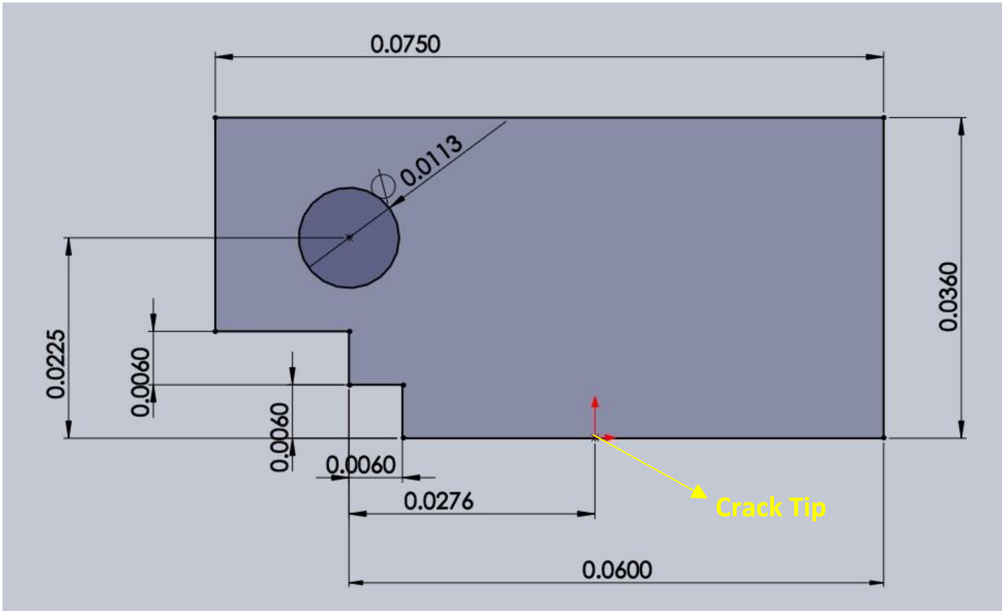


Figure 2 Sketch of the specimen, all dimensions are in meters

2.2 FINITE ELEMENT ANALYSIS

After the above model is modelled in ABAQUS, material properties i.e**. E = 200Gpa and v = 0.3** are imposed to the model. The model is assigned a **unit thickness** to assume a plain strain model. Then the loading is applied. A **concentrated load** of **100 KN/m** was applied at the top node of the pin hole.

For meshing, the model is also partitioned into various regions for obtaining an accurate mesh. The whole part is assigned a "**Plain Strain**" condition with reduced integration. Then each of the surface is assigned individual seeds to obtain the best mesh. Trial and error method is used to obtain an optimized mesh. The obtained mesh is shown below. Care is taken that the number of nodes does not exceed 1000. A **"Focusing / Spider-Web mesh"** is used at the crack tip to obtain a more accurate stress field. After the mesh is obtained, the nodes to the right of the tip are selected and are given a boundary condition of **Uy = 0**. Also, the right most node is given a boundary condition of **Ux = 0.** The load and the boundary conditions are shown in **figure**.**3** , the undeformed mesh is shown in **Figure 4** and deformed mesh is shown in **Figure 5.**

Mesh Properties:

- **Number of nodes:** 900
- **Number of elements:** 843
- **Type of element:** CPE4 (Four Node Plain Strain element without reduced integration)

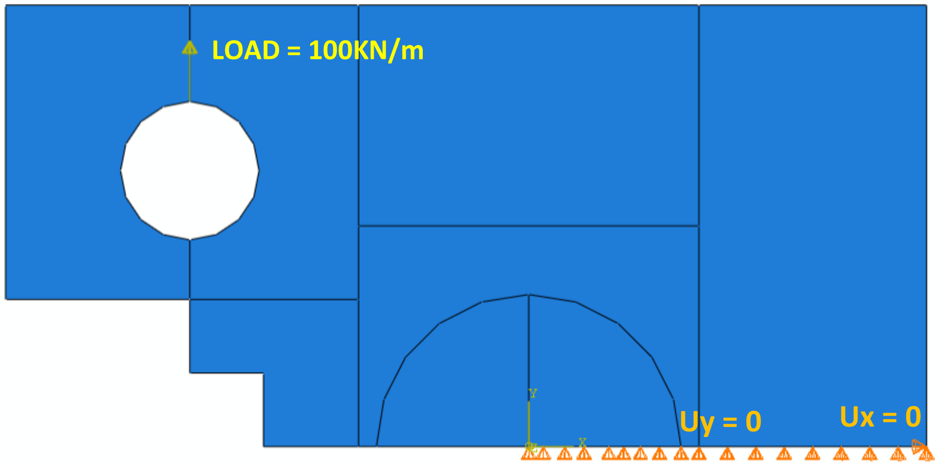


Figure 3 Partitions, Load and boundary conditions on the model

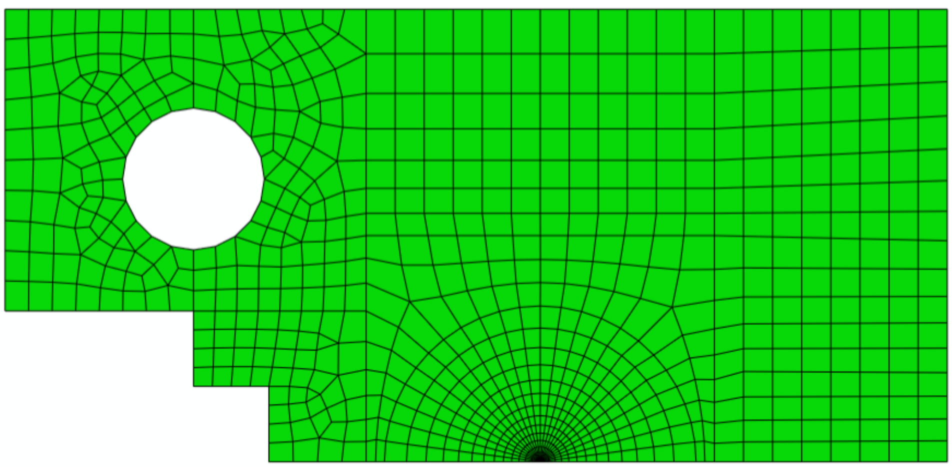


Figure 4. Obtained undeformed Mesh

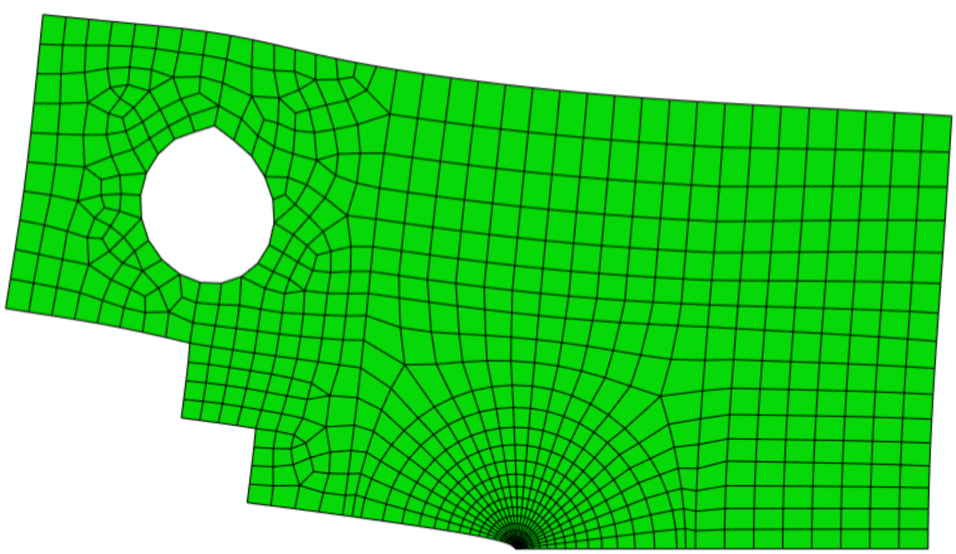


Figure 5. Deformed Mesh

The stress contours are then obtained. The obtained stress contours are shown below:

Von Mises Stress contour on deformed plot (w/o edges) (Figure 6)

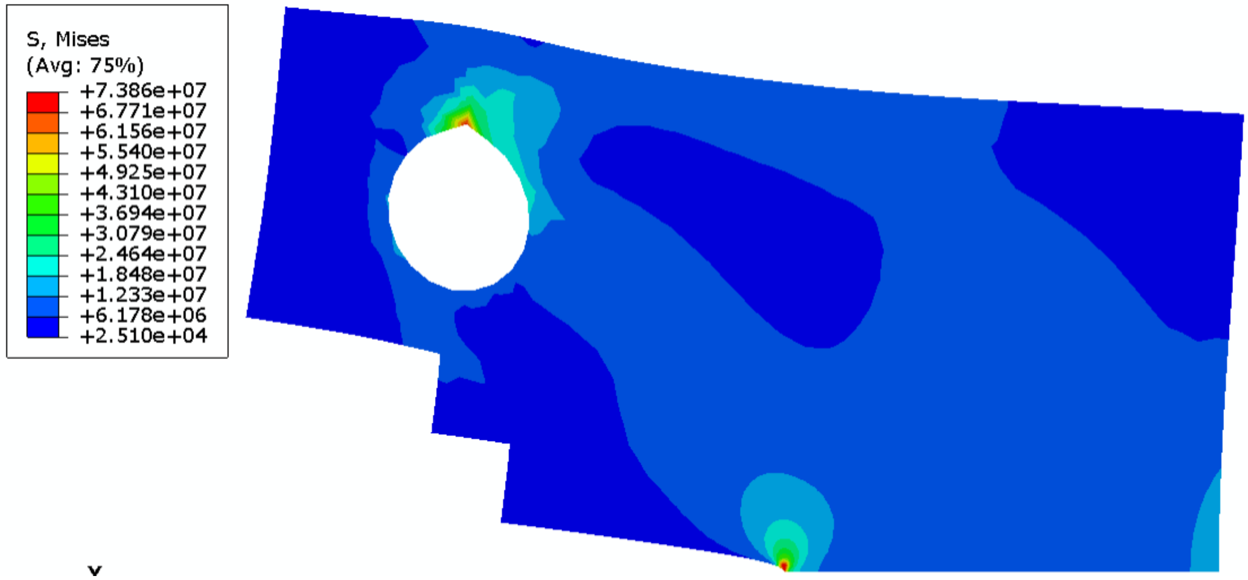


Figure 6. Von Mises Contour Plot on deformed mesh

We take a closer look at the Von Mises contour plot at the crack tip (Figure 7)

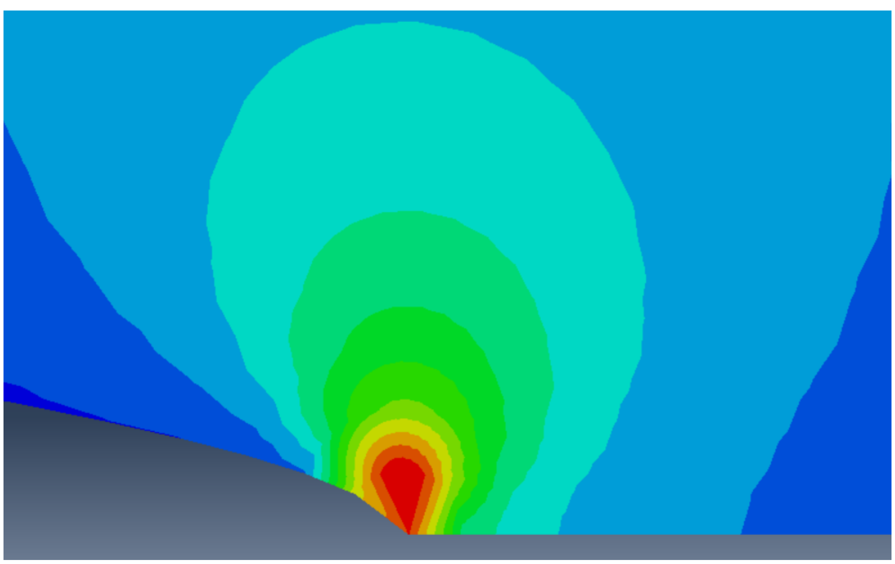


Figure 7. A closer look at Von Mises contour at crack tip

S22 Stress contour on deformed plot (w/o edges) (Figure 8)

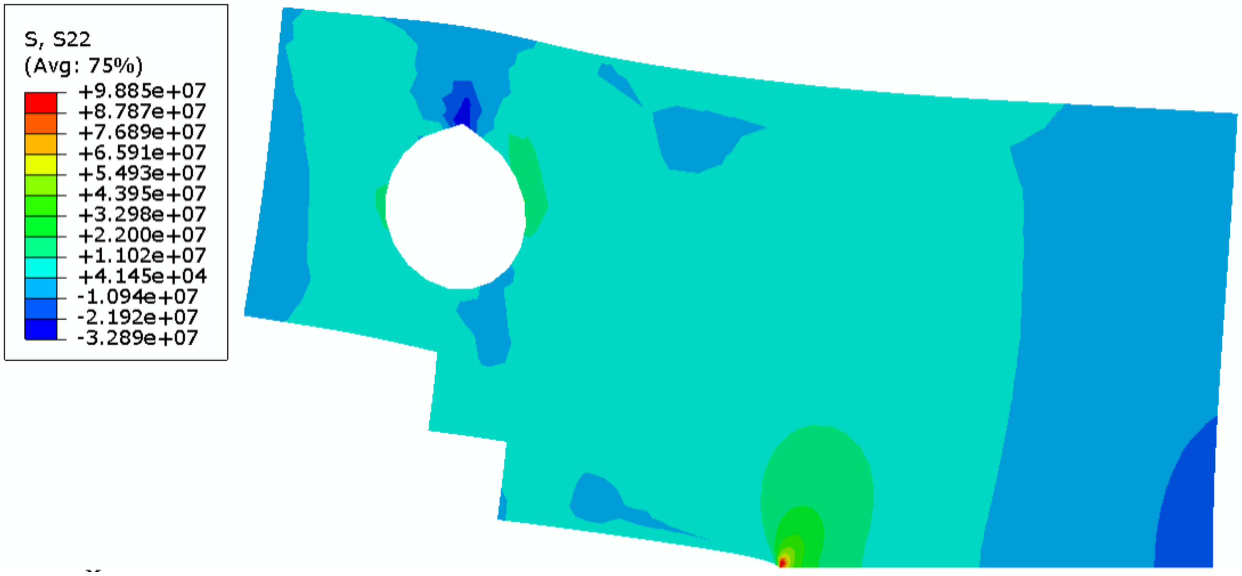


Figure 8. S22 Contour plot on deformed mesh

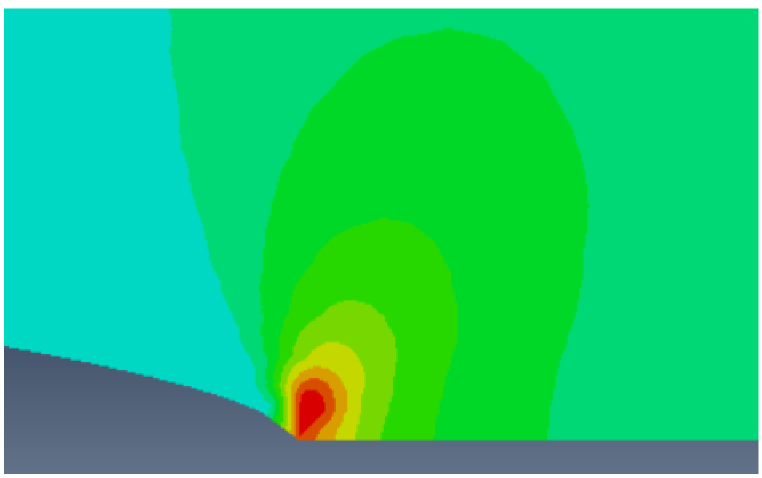


Figure 9. Closer look at S22 Contour plot at crack tip

3. <u>ANALYSIS and RESULTS</u>

3.1 Stress Intensity Factor (K_I)

Since the stress field near the crack tip (i.e., at small r) is proportional to $1/\sqrt{r}$, one can define a proportional constant. This constant is called **"stress intensity factor"** and denoted by $\mathbf{K_I}$**.** It is important to determine $\mathbf{K_I}$ for a given geometry and under various boundary conditions in fracture analysis. This information is used to predict whether the crack propagates or not.

Firstly, exact stress intensity factor for the compact specimen is found out from the handbook formula.

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

$$K_I^{Exact} = \frac{100 * 10^3}{\sqrt{0.06}} \frac{2 + 0.46}{(1 - 0.46)^{\frac{3}{2}}} \begin{bmatrix} 0.866 + 4.64(0.46) - 13.22(0.46)^2 \\ +14.72(0.46)^3 - 5.60(0.46)^4 \end{bmatrix}$$

$$therefore, \quad \boxed{\boldsymbol{K_I^{Exact} = 3.505\ Mpa/\sqrt{m}}}$$

Now, the stress intensity factor is calculated from the Finite element analysis is done and is compared to the exact stress intensity factor.

(a) From the opening displacement behind the crack tip and the force at the crack tip. First, $\mathbf{K_I^{FU}}$ is calculated from the formula given:

$$\boxed{K_{\mathrm{I}}^{\mathrm{Fu}} = \sqrt{\frac{F_y u_y E}{l(1-\upsilon^2)}}}$$

From the FE Analysis, we obtain:

Crack tip nodal force, $\mathbf{F_y^{tip}}$ = 43877.1 N ,

Opening nodal displacement, $\mathbf{u_y}$ = 4.6167×10^{-7}m,

and **l** = 3.508×10^{-4}m

By substituting these values in the above formula,

$$K_I^{Fu} = \frac{43877.1 * 4.6167 * 10^{-7} * 200 * 10^9}{3.508 * 10^{-4}(1 - 0.3^2)}$$

Therefore, $\boxed{\boldsymbol{K_I^{FU} = 3.5018\ Mpa/\sqrt{m}}}$

The error percent between K_I^{Exact} and K_I^{Fu} is:

$$\boxed{\%\ \mathrm{Error} = \frac{|\mathrm{K}_1^{\mathrm{Fu}} - \mathrm{K}_1^{\mathrm{exact}}|}{\mathrm{K}_1^{\mathrm{exact}}} * 100 = \frac{|3.5018 - 3.5055|}{3.5055} * 100 = 0.105\ \%}$$

(b) From the opening displacements behind the crack tip, the K_I^{disp} was calculated using given formula and compared with the K_I^{exact}.

$$K_I^{disp} = \frac{E}{4(1-v^2)}\sqrt{\frac{2\pi}{r}}u_y$$

The values of crack tip nodal force F_y^{tip} and the opening nodal displacement u_y immediately behind the crack tip obtained from FEA are shown in the table below.

R (meters)	Uy (meters)	K_I^{disp}(Mpa/m$^{1/2}$)	Error (%)
7.48 x10^{-4}	6.74 x10^{-7}	3.40	3.14
1.20 x10^{-3}	8.75 x10^{-7}	3.48	0.61
1.71 x10^{-3}	1.06 x10^{-6}	3.54	1.07
2.28 x10^{-3}	1.25 x10^{-6}	3.60	2.80
2.93 x10^{-3}	1.44 x10^{-6}	3.66	4.49
3.67 x10^{-3}	1.64 x10^{-6}	3.73	6.31
4.51 x10^{-3}	1.85 x10^{-6}	3.80	8.27

Sample calculation:

$$\mathrm{K_I^{disp}} = \frac{(200*10^9)}{4(1-0.3^2)}\sqrt{\frac{2\pi}{0.000120}} * 6.74 * 10^{-7} = 3.40 \text{ MPa m}^{1/2}$$

K_I^{disp} versus r is plotted as shown below (Figure 10) and is compared with K_I^{exact}:

Figure 10. KI Disp vs R

(c) From the opening stress (σy) near the crack tip, we choose at least 6 nodes whose coordinates are within r < a/10 and ~10° < θ < ~50°, where a is the crack length. We select nodes with different **r** and obtain the opening stress σ_y, r and θ (via x& y) of each node and using the formula given below, we estimate K_I^{stress} using the K-field solution below at each node.

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

The values of r, Θ, K_I^{Stress} obtained from the FEA are given below:

σ_y^{Fe}(Mpa)	R (meters)	Theta(degrees)	KI stress (Mpa/m$^{1/2}$)	Error(%)
68.940	3.51x10^{-4}	15	3.11	11.304
50.243	7.48 x10^{-4}	22.5	3.17	9.628
41.483	1.20 x10^{-4}	30	3.15	10.183
34.080	1.71 x10^{-3}	30	3.09	11.923
30.499	2.28 x10^{-3}	37.5	3.04	13.198
27.657	2.93 x10^{-3}	45	3.00	14.358
23.883	3.67 x10^{-3}	45	2.90	17.276
21.824	4.50 x10^{-3}	52.5	2.85	18.598

Sample calculation:

K_I^{Stress} = 68.94*sqrt(2*pi*2.51*10^{-4})/cos(7.5)(1+sin(7.5)*sin(3*7.5)) = **3.11Mpa/m$^{1/2}$**

K_I^{stress} versus r is plotted as shown below (Figure 11) and is compared with K_I^{exact}:

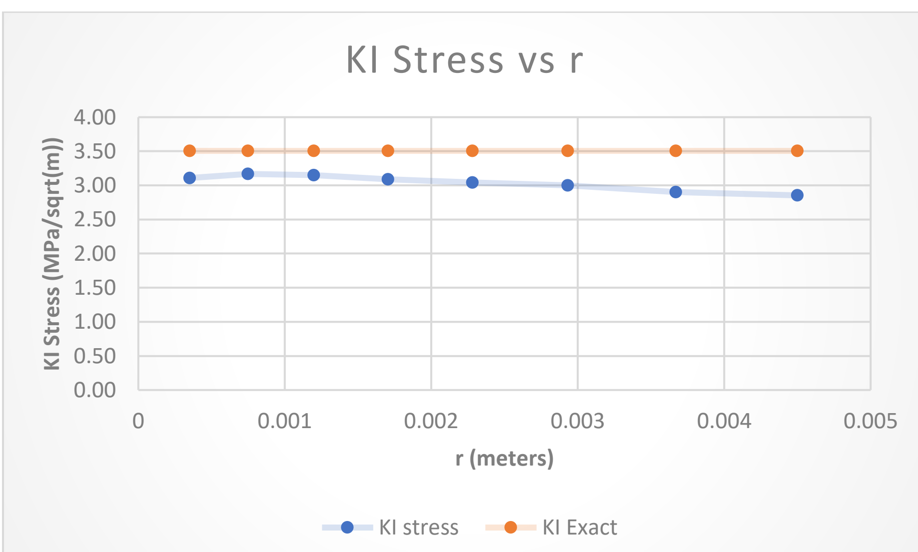


Figure 11. KI stress vs r

3.2 Behavior of opening stress (σy) around crack tip (0 < Θ < π):

Another method to check whether the mesh yields accurate stress field is to compare the angular variation of the computed stress with the angular variation of actual K-field stress shown earlier. For this, we choose a set of nodes that surrounds the crack tip, with similar radial coordinate from the tip. For each node, we record the opening stress $\sigma_y^{f.e.}$ and the coordinate locations in r and θ.

We normalize each stress using the following equation:

$$\boxed{\tilde{\sigma}_y{}^{f.e.} = \sigma_y{}^{f.e.}\sqrt{2\pi r}\ /\ K_1^{exact}}$$

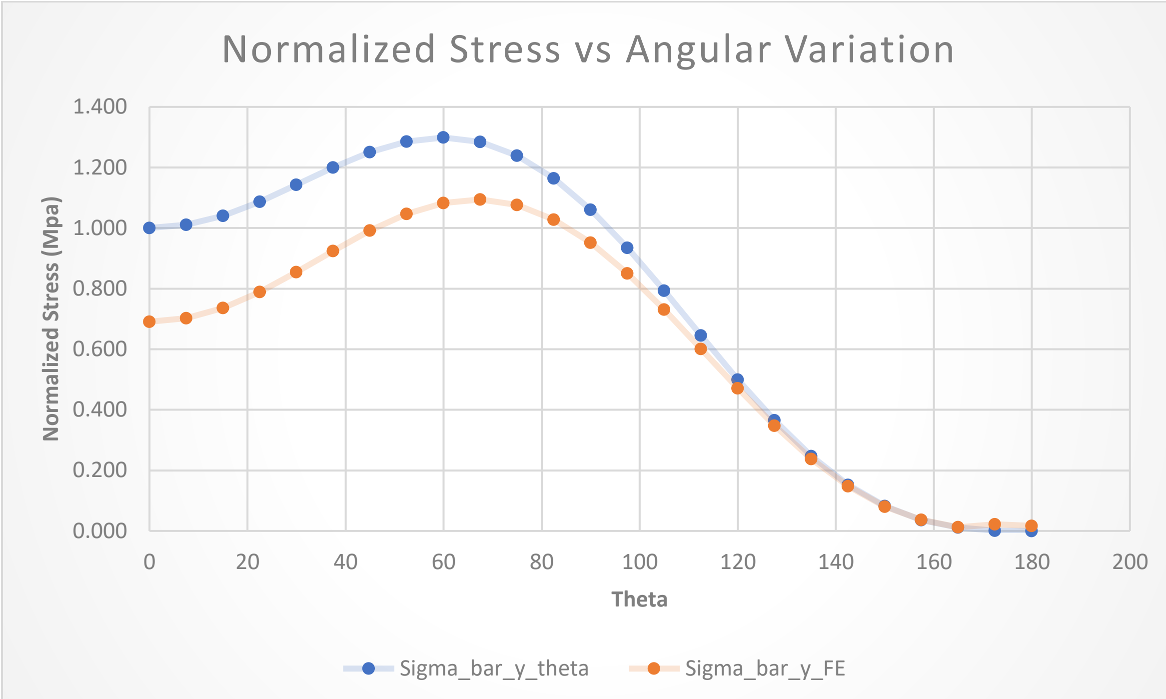
We also plot both the normalized stress found and the angular variation (from 0° to 180°) of stress according to the K-field solution given below:

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

The $\sigma_y^{f.e}$, r ,Theta, $\tilde{\sigma}_y{}^{f.e.}$, and $\tilde{\sigma}_y(\theta)$ obtained are shown below:

R (meters)	Θ (degrees)	$\sigma_y^{f.e}$(MPa)	$\tilde{\sigma}_y{}^{f.e.}$(MPa)	$\tilde{\sigma}_y(\theta)$ (MPa)
4.51×10^{-3}	180	0.354	0.017	0.000
4.51×10^{-3}	172.5	0.467	0.022	0.001
4.51×10^{-3}	165	0.251	0.012	0.011
4.51×10^{-3}	157.5	0.762	0.037	0.036
4.51×10^{-3}	150	1.680	0.081	0.082
4.51×10^{-3}	142.5	3.082	0.148	0.152
4.51×10^{-3}	135	4.950	0.238	0.247
4.51×10^{-3}	127.5	7.231	0.347	0.365
4.51×10^{-3}	120	9.811	0.471	0.500
4.51×10^{-3}	112.5	12.530	0.601	0.646
4.51×10^{-3}	105	15.230	0.731	0.794
4.51×10^{-3}	97.5	17.720	0.850	0.935
4.51×10^{-3}	90	19.830	0.952	1.061
4.51×10^{-3}	82.5	21.430	1.028	1.164
4.51×10^{-3}	75	22.433	1.076	1.240
4.51×10^{-3}	67.5	22.808	1.094	1.285
4.51×10^{-3}	60	22.570	1.083	1.299
4.51×10^{-3}	52.5	21.824	1.047	1.286
4.51×10^{-3}	45	20.671	0.992	1.251
4.51×10^{-3}	37.5	19.270	0.925	1.200
4.51×10^{-3}	30	17.809	0.855	1.143
4.51×10^{-3}	22.5	16.450	0.789	1.087

The normalized Stress vs angular variation is plotted and is given below:



4. CONCLUSION

The main purpose of this project was to carry out a finite element analysis of a standard fracture specimen to study the fracture. ABAQUS was used to obtain relevant stress values and displacement components. For this a specimen of **Structural Steel (Young's modulus = 200GPa and Poisson's ratio = 0.3)** is used. Plain strain condition is taken for the specimen. A good mesh is made with a "Spider-Web" at the crack tip for more accurate results. A concentrated load of 100KN/m is applied at the topmost node of the pin hole. Von Mises and S22 Stress contours are also shown. The calculation of stress intensity factor was also performed by using various methods. The calculated value of **exact stress intensity factor** was **3.5055 MPa/m$^{1/2}$**. The calculated value of **K_I^{FU}** is **3.5018 MPa/m$^{1/2}$** with an **error of 0.1%.** Later, K_I^{Disp} and K_I^{Stress} were also calculated and compared against exact stress intensity factor. K-field stress, a set of nodes that surrounds the crack tip was obtained. The calculated values of θ, $\tilde{\sigma}_y^{\text{f.e.}}$, $\tilde{\sigma}_y(\theta)$were recorded and compared.