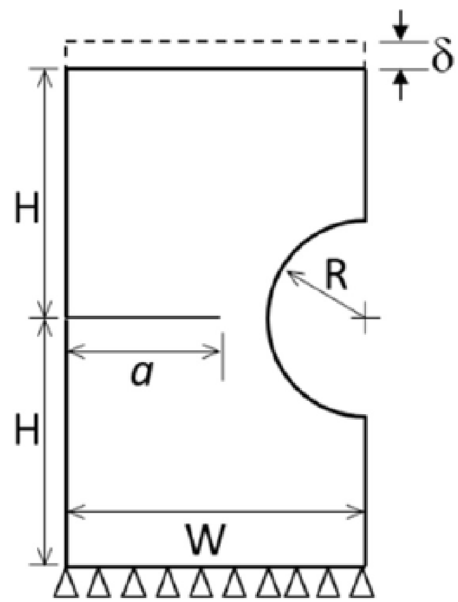
**NAME: DHAVAL PANWALA**

**ROLL NO. 231010022**

# Project Report

## Problem Statement




The objective of the project is to analysis of cracked geometries using Finite Element Method. The geometry of the problem is shown in figure. The loading is uniform displacement of the top edge by $\delta = 5 \text{ x } 10^{-5}$ m, while the bottom edge is restricted from moving vertically. Assume plane stress conditions with unit thickness. The elastic modulus is $70\ GPa$ and Poisson's ratio is $0.3$. Use only SI system, i.e. force in N, stress in $N/m^2$and all lengths in meter. Dimensions are $W = 0.06m, H = 0.084, a = 0.024\ and\ R = 0.021m$

## Software Used

The software used for this project is **Abaqus**. Abaqus is a comprehensive finite element analysis software suite that is widely used in both academia and industry for its robust and versatile capabilities.

In this project, Abaqus was utilized for several key tasks:

1. **Creating the Plate**: The geometry of the plate, including the crack, was modelled in Abaqus. The dimensions are $\mathrm{W} = 0.06\mathrm{m}, \mathrm{H} = 0.084, \mathrm{a} = 0.024$ and $\mathrm{R} = 0.021\mathrm{m}$.

2. **Defining Material Properties**: The material properties, including the elastic modulus of $70\ GPa$ and Poisson's ratio of $0.3$, were defined in Abaqus. These properties were crucial in accurately simulating the behavior of the plate under load.
3. **Meshing**: A finite element mesh was created for the geometry using Abaqus. Special attention was paid to the mesh around the crack tip, as this region is critical in the analysis of cracked geometries.
4. **Analysis**: The displacement loading, and boundary conditions were applied, and the stress analysis was carried out in Abaqus. The software's powerful solver was used to compute the stress intensity factor (SIF) and other key results.

The use of Abaqus in this project not only facilitated a detailed and accurate analysis of the cracked geometry but also provided valuable insights into the behaviour of the plate under the given loading conditions.

## Finite Element Mesh

The Problem with Boundary condition is shown in figure 1, where the topmost edge of the plate is subjected to uniform displacement of $5 \times 10^{-5}\mathrm{m}$ and the bottommost edge is fixed that is restricted from moving vertically.

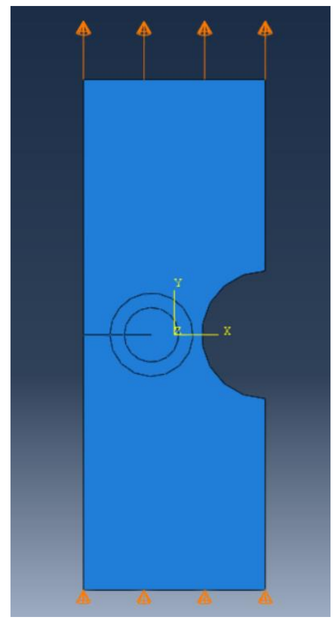


**Figure 1. Boundary condition applied on Plate with Hole**

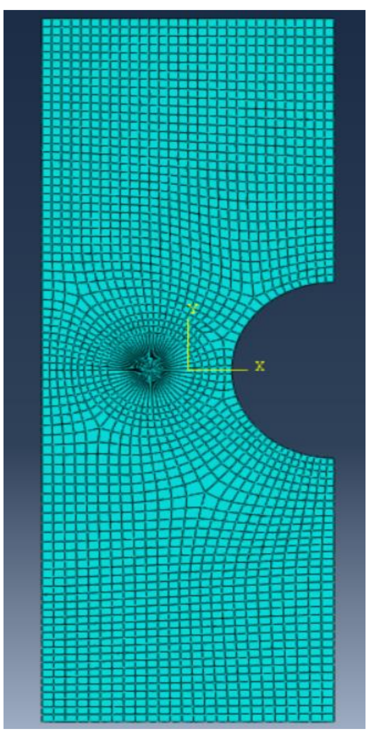


Figure 2(a)

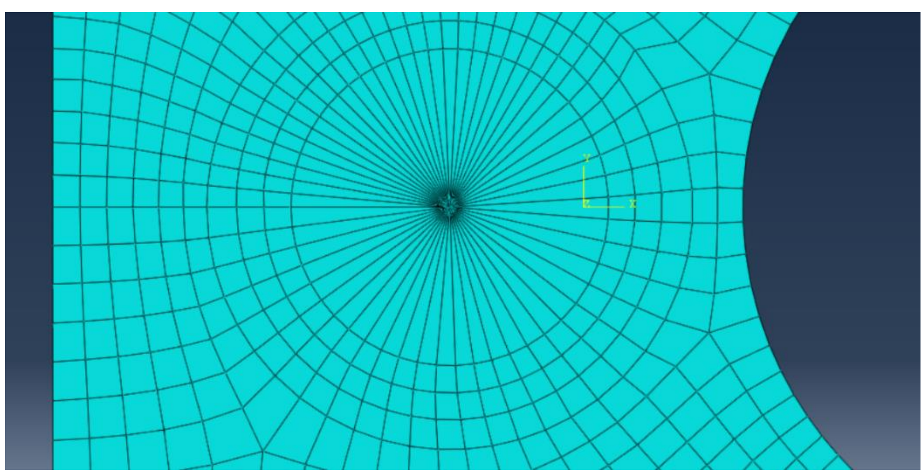


Figure 2(b)

**Figure 2. (a) Meshing of the Plate and (b) Meshing near the plate**

Now, figure 2(a) and figure 2(b) shows the meshing of plate and meshing near the crack tip respectively.

## Element Details

A triangular mesh, ideal for capturing stress variations, is used near the crack tip. As we move away from the crack tip, the mesh transitions from fine to coarse to optimize computational efficiency. In regions far from the crack tip, a quadrilateral mesh is employed. This approach ensures a balance between accuracy and computational power.

# Results

1. **Estimation of SIF using Stress Extrapolation Method**:

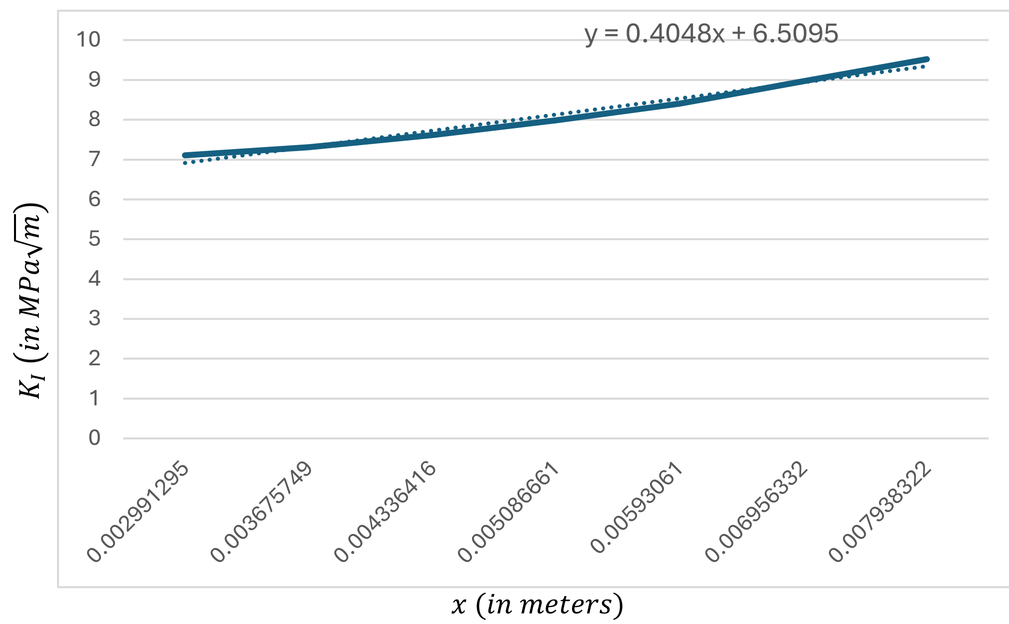



**Figure 3. Variation of $K_I$ with x using stress extrapolation method.**

In the Stress Extrapolation Method, we plot the stress values against the distance from the crack tip on a graph. The stress intensity factor (SIF), denoted as $K_I$, is then extrapolated from this graph.

In this project, the stress extrapolation method yielded a linear curve represented by the equation $y = 0.4048x + 6.5095$ which gives apparent stress intensity factor as $6.5095\ MPa\sqrt{m}$.

2. **Estimation of SIF using Displacement Extrapolation Method**:

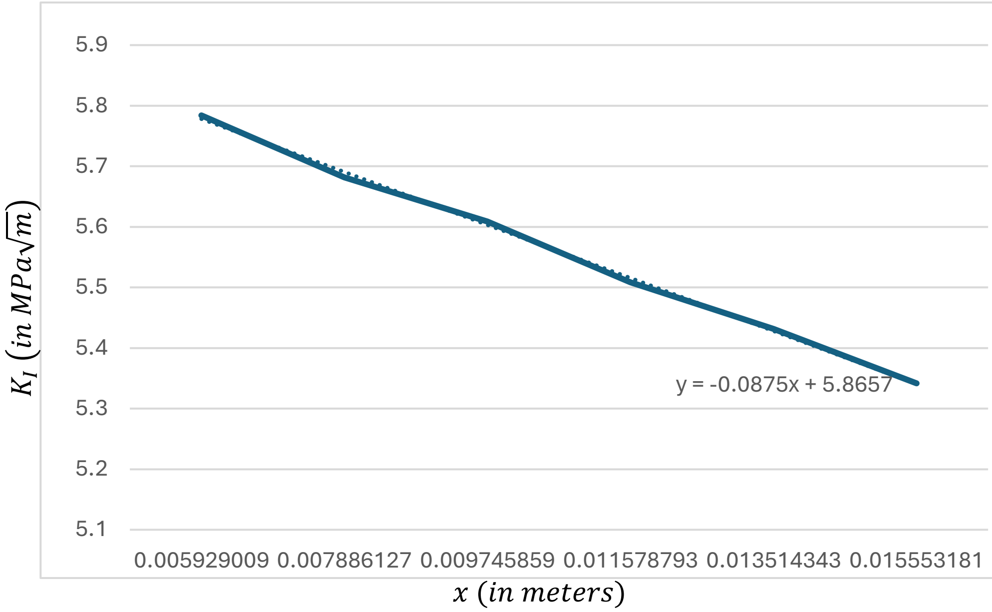



**Figure 4. Variation of $K_I$ with x using displacement extrapolation method.**

Figure 4 shows the linear variation of $K_I$ with respect to $x$ having equation $y = -0.0875x + 5.8657$ where $k_{app} = 5.8657\ MPa\sqrt{m}$

3. **Estimation of SIF using MVCCI**:

For MVCCI,

$$G = \frac{F_{yi}(V_k - V_k')}{2\Delta B} = \frac{K^2}{E}$$

Here, $F_{yi} = 110.6961\ N$, $V_k = V_k' = 7.2465 \times 10^{-6} m$;

Therefore,

$$K_I = 7.4934\ MPa\sqrt{m}$$

4. **SIF obtained directly from the software using J integral**:

J integral value for 3 different paths are 0.6178, 0.6182 and 0.6181

5. **Comparison with SENT case**:

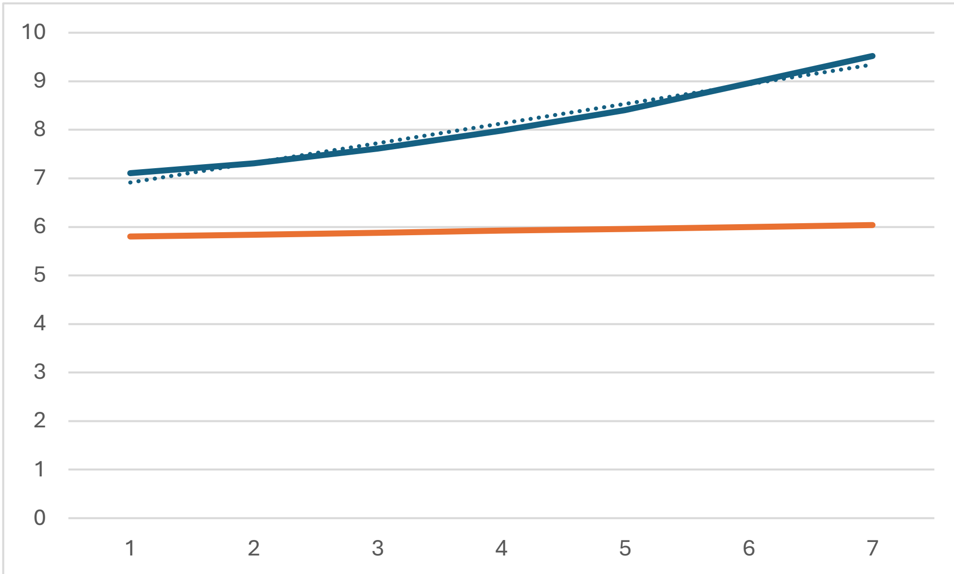



**Figure 5. Comparison of $K_I$ variation for plate with hole (represent by Blue) and plate without hole (represented by Orange) using Stress extrapolation method.**

Figure 5 shows how the stress intensity factor calculated using stress extrapolation method will variates differently for straight boundary and for semi-circular boundary.

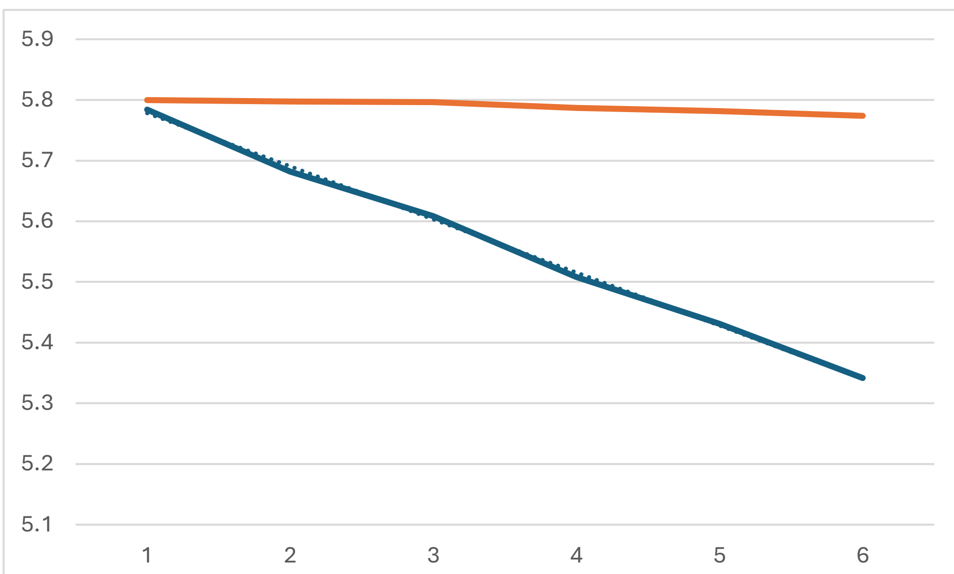



**Figure 6. Comparison of $K_I$ variation for plate with hole (represent by Blue) and plate without hole (represented by Orange) using displacement extrapolation method.**

Figure 6 shows how the stress intensity factor calculated using displacement extrapolation method will variates differently for straight boundary and for semi-circular boundary.

For MVCCI,

Stress intensity factor for straight boundary is $5.5725\ MPa\sqrt{m}$ whereas Stress intensity factor for semi-circular boundary is $7.4934\ MPa\sqrt{m}$.

J integral for Straight boundary for 3 different paths are 0.5339, 0.5339 and 0.5339 whereas for semi-circular boundary J integrals are 0.6178, 0.6182 and 0.6181 respectively.

6. **Opening Stress and Displacement Plots**:

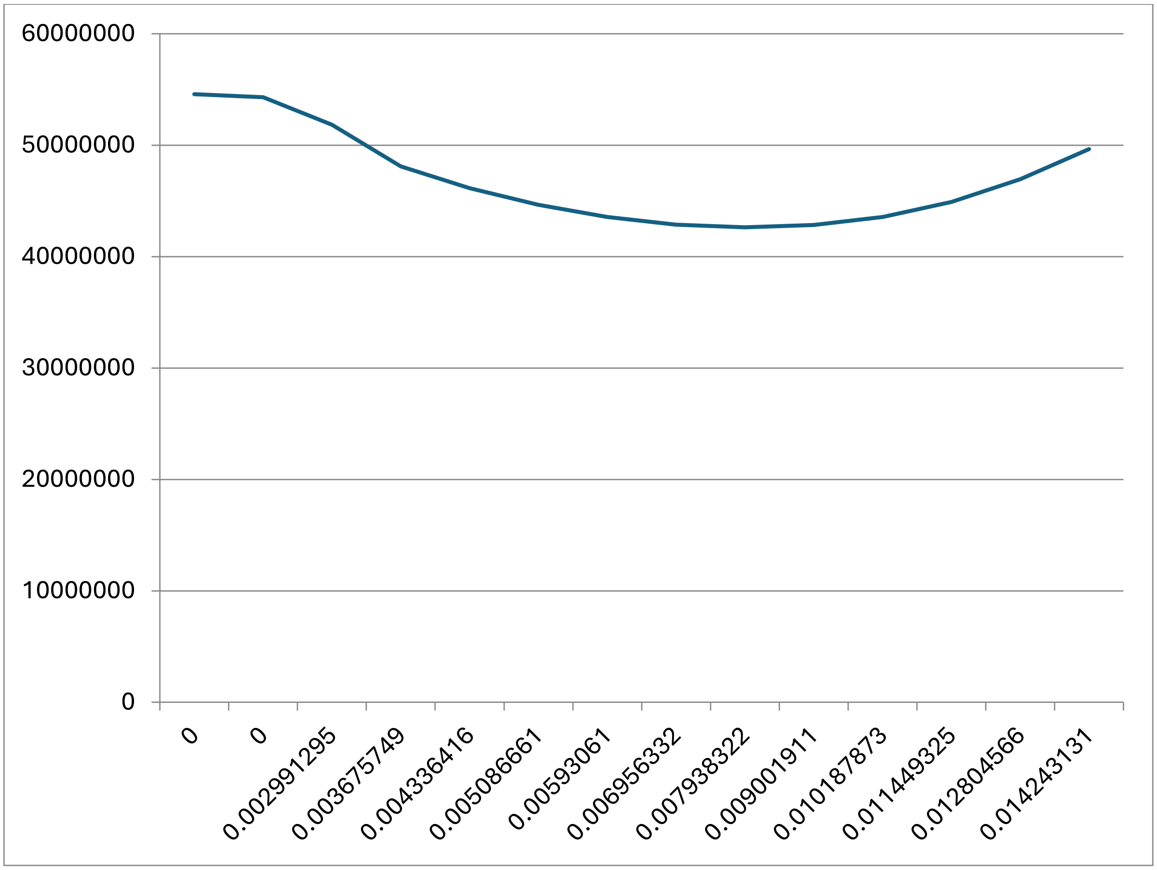



**Figure 7. Opening Stress variation along $\theta = 0$**

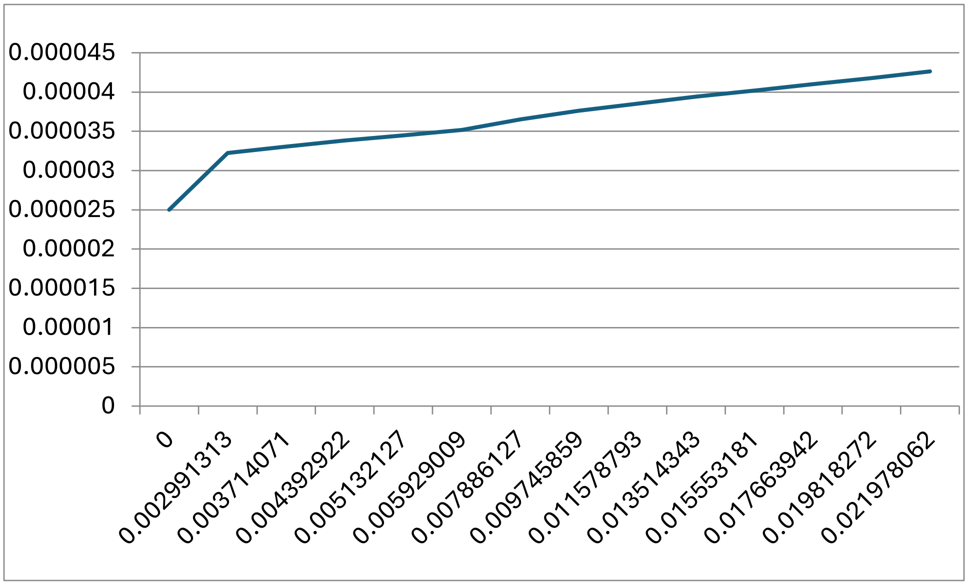



**Figure 8. Opening displacement as a function of r for $\theta = \pi$**

## 7. Accuracy of SIF:

Value of Stress intensity factor of a semi-circular boundary obtained from simulation is 6.5775 $MPa\sqrt{m}$.

| Method | SIF obtained from method | SIF obtained from simulation | Difference |
|---|---|---|---|
| Stress Extrapolation method | 6.5095 | 6.5775 | 1.033% |
| Displacement extrapolation method | 5.8675 | 6.5775 | 10.79% |
| MVCCI | 7.4934 | 6.5775 | 13.92% |

From above table, we can conclude that stress extrapolation method gives accurate results compare to other methods.