Self Project

SMART Based Crack Growth Simulation of a Center-Cracked Plate in ANSYS

Submitted by:

24M0045 - Jayshree Chauhan

Aerospace Engineering Department

Indian Institute of Technology, Bombay

Contents

List of Figures

1 Introduction

The presence of cracks in engineering structures can significantly reduce their load-carrying capacity and lead to catastrophic failure if not properly assessed. Predicting the growth of cracks under service conditions is therefore a key aspect of structural integrity analysis. Traditional fracture mechanics methods provide analytical solutions for simple geometries and loading conditions; however, real-world components often exhibit complex geometries and loading patterns that require numerical simulation.

Finite Element Analysis (FEA) has become a standard approach for evaluating crack propagation. In particular, the Separating Morphing Adaptive Remeshing Technology (SMART) in ANSYS Mechanical provides a powerful tool for simulating crack growth with high accuracy. SMART automatically updates the mesh as the crack advances, eliminating the need for manual re-meshing and reducing computational effort while maintaining accuracy.

In this project, a static crack growth analysis is performed on a steel plate containing an initial edge crack subjected to monotonic tensile loading. The objective is to determine the crack path, evaluate the evolution of the stress intensity factor (SIF), and assess the conditions under which crack propagation occurs. The material is modeled using linear elastic fracture mechanics (LEFM), with crack growth governed by the critical fracture toughness criterion (K_{IC}). The SMART method allows for continuous crack advancement based on the computed K_I values, with adaptive meshing ensuring precise SIF calculations at each growth increment.

The outcomes of this study have direct relevance for engineering applications where fracture resistance is critical, such as in aerospace, automotive, and pressure vessel components. By accurately predicting crack growth behavior under static loads, engineers can design safer and more reliable structures while optimizing material usage.

2 Model Geometry, Material Properties, and Setup

2.1 Geometry Definition

The simulation was carried out using a plate model with an center crack, designed to study crack propagation under monotonic loading conditions. The geometry was constructed in ANSYS Workbench using precise dimensions to match a standard fracture mechanics test configuration.

The main features of the geometry include:

- A rectangular plate with defined length, width, and thickness to represent a structural component under tensile loading.
- An initial straight crack inserted at the center of the plate, perpendicular to the applied load, serving as the crack initiation point.

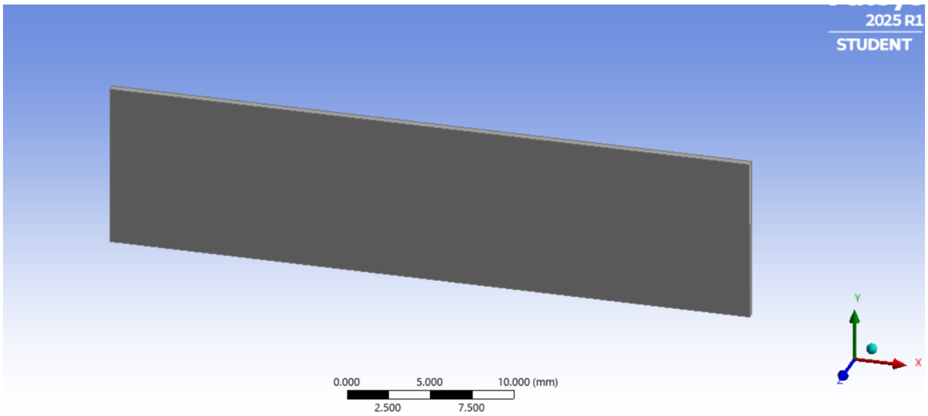

Figure 1: Plate Model for Analysis

The simulated component is a rectangular plate with a center crack, designed to represent a simplified structural element under tensile loading. The dimensions of the plate are as follows:

- **Length:** 40 mm
- **Width:** 10 mm
- **Thickness:** 0.5 mm

2.2 Material Properties

The component was assumed to be made from structural steel, which is commonly used in engineering due to its high strength and fracture resistance. Linear elastic material properties were defined in ANSYS Engineering Data, including:

- Elastic modulus (E)
- Poisson's ratio (ν)

Properties of Outline Row 3: Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Isotropic Elasticity		
4	Derive from	Young's Modulus an...	
5	Young's Modulus	2E+11	Pa
6	Poisson's Ratio	0.3	
7	Bulk Modulus	1.6667E+11	Pa
8	Shear Modulus	7.6923E+10	Pa

Figure 2: Steel Properties

The fracture toughness value was used by ANSYS to determine whether the crack would propagate under the applied stress intensity factor.

2.3 Simulation Setup

The geometry was imported into ANSYS Mechanical, and the SMART Crack Growth tool was activated to simulate crack extension under static loading. The following steps were taken in the setup:

- **Meshing:** A refined mesh was generated in the vicinity of the crack tip to ensure accurate calculation of the stress intensity factor (SIF). The SMART adaptive remeshing zone was set around the crack tip to maintain mesh quality as the crack propagated.

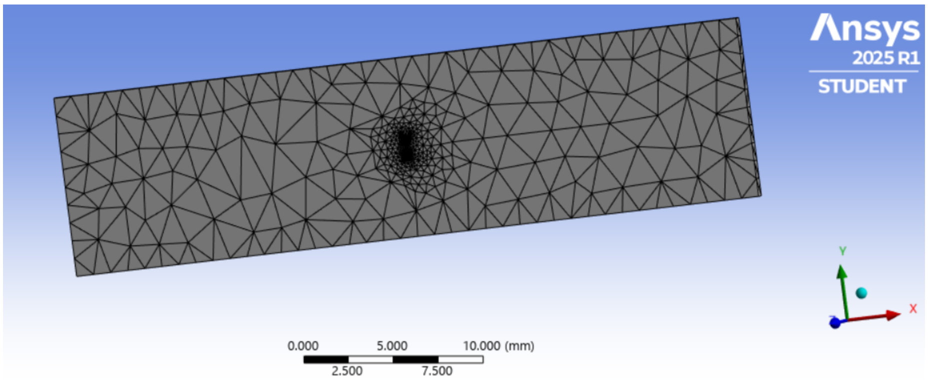

Figure 3: Meshing

- **Boundary Conditions:** Fixed supports were applied to one side of the plate to restrict displacement, and a uniform tensile displacement or force was applied on the opposite side to induce crack opening (Mode I loading).

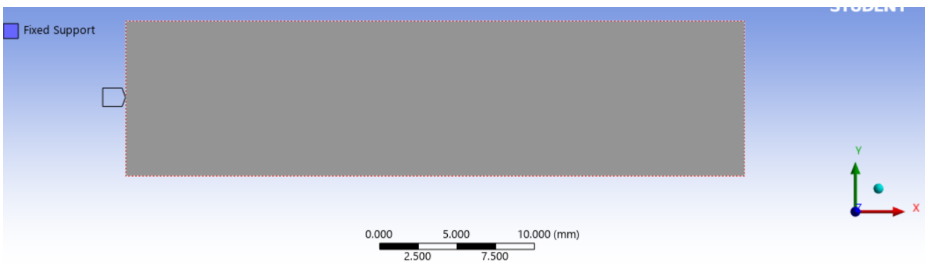

Figure 4: Fixed Support

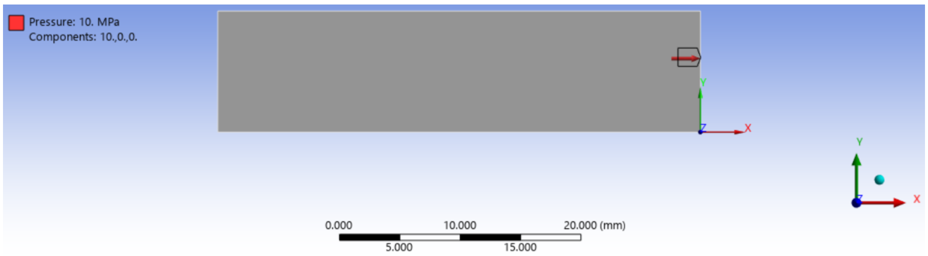

Figure 5: Pressure

- **Crack Definition:** The initial crack front was defined at the edge notch, with the crack path set to propagate perpendicular to the loading direction.

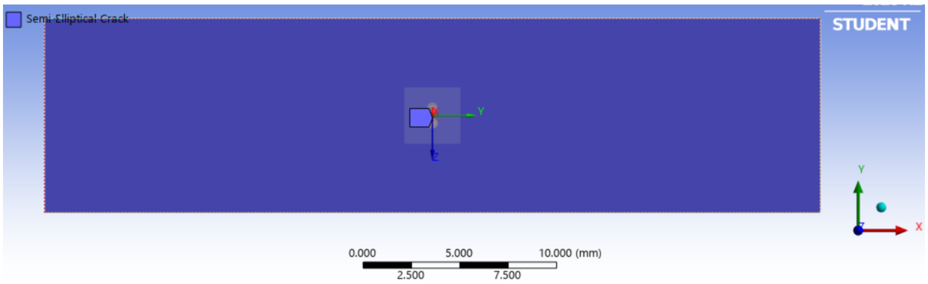

Figure 6: Elliptical Crack Definition

Initial Crack Length: 0.4 mm (total length across the plate centerline)

The crack is positioned at the geometric center of the plate, oriented perpendicular to the loading direction, creating a pure Mode I (opening mode) fracture scenario. This configuration allows symmetrical crack growth in both directions from the center under the applied tensile load.

- **Fracture Criterion:** Crack growth was controlled by comparing the computed Mode I SIF (K_I) to the material's fracture toughness (K_{IC}). If K_I exceeded K_{IC}, the crack advanced incrementally.

- **Solution Controls:** Convergence criteria were tightened to ensure accurate results, especially during automatic remeshing operations.

This approach allowed for an accurate representation of static crack propagation, with the SMART tool ensuring that mesh refinement and crack growth path updates occurred automatically as the simulation progressed.

3 Results and Discussion

The crack growth analysis was carried out using the SMART (Seamless Mesh Adaptation for Realistic Tearing) technique in ANSYS Workbench. The simulation provided insights into stress distribution, Stress Intensity Factors (SIFs), and crack propagation behavior for a center-elliptical crack in a thin plate.

3.1 Crack Growth

The simulated crack exhibited symmetric propagation from the initial elliptical geometry, consistent with Mode I opening conditions under uniform tensile loading. The adaptive meshing maintained high element quality near the crack front throughout the simulation. Figure 7 shows the crack propagation.

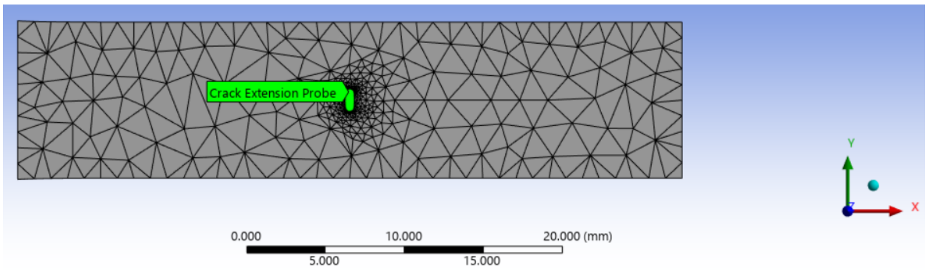

Figure 7: Crack growth from the initial elliptical crack under tensile loading.

3.2 Stress Distribution

The von Mises stress contour revealed high stress concentration at the crack tips. As the crack length increased, these concentration zones intensified, and redistribution of stress occurred in the surrounding regions. Figure 8 presents the stress profile for a representative step.

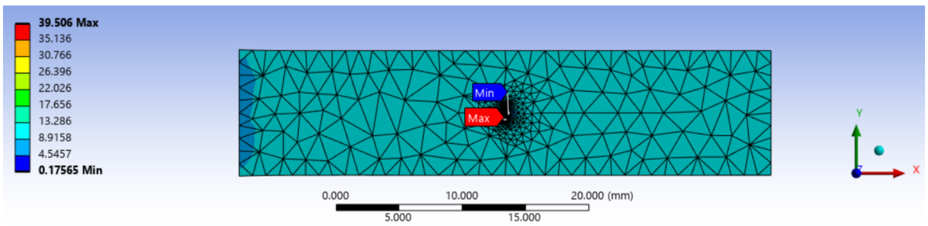

Figure 8: Von Mises stress distribution near the crack tip during loading.

3.3 Stress Intensity Factor (SIF) Analysis

The Mode I Stress Intensity Factor, K_I, for a center crack in a finite-width plate is given by:

$$K_I = \sigma\sqrt{\pi a} \cdot F\left(\frac{a}{W}\right)$$

where:

- σ = applied stress (MPa)
- a = half crack length (m)
- W = plate width (m)
- $F(a/W)$ = geometry correction factor

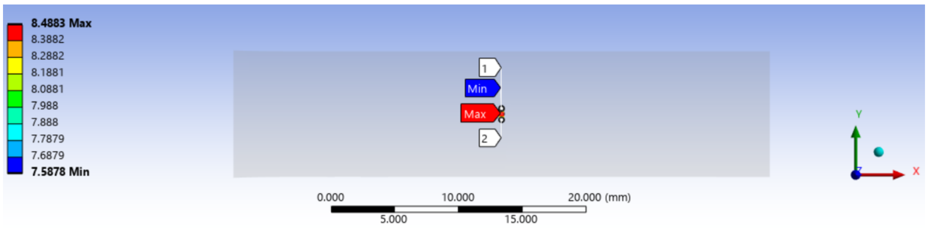

Figure 9: Stress Intensity Factor for mode 1

3.4 Discussion

In this study, a static structural analysis was performed in ANSYS to evaluate crack growth behavior. The Stress Intensity Factor (SIF, K1) was extracted along the crack front, with values ranging from approximately 6.62 to 8.49 MPa$\sqrt{\text{mm}}$. The maximum SIF occurred near the mid-region of the crack front, indicating a higher driving force for crack propagation in that area.

The stress distribution plot showed a maximum stress of 39.506 MPa concentrated around the crack tip, confirming it as the critical location for failure initiation. Crack front length measurements helped relate the SIF variation to crack geometry. These results highlight that both SIF and stress reach peak values at the crack tip region, which governs the crack growth behavior.