

الجمهورية الجزائرية الديمقراطية الشعبية
People's Democratic Republic of Algeria
وزارة التعليم العالي والبحث العلمي



Ministry of Higher Education and Scientific Research

جامعة غرداية

University of Ghardaia

كلية العلوم والتكنولوجيا

Faculty of Science and Technology

قسم الآلية والكهروميكانيك

Department of Automatics and Electromechanics

Registration No

.../.../...

**Final Thesis Submitted in partial fulfillment of the requirements for
the degree of**

Master

Field of Study: Science and Technology

Major: Electromechanics

Specialization: Industrial Maintenance

Title

**Analysis of stress intensity factor in a
cracked structure**

Presented by:

Abderrahmane Reciou

Defended publicly on: 14/06/2026

Board of Examiners:

BELLAOUAR Abderrahmane

AKERMI Faouzi

AISSAOUI Faris

BOUSNANE Toufik

Pr

MAA

MCA

MCB

Univ. Ghardaïa

Univ. Ghardaïa

Univ. Ghardaïa

Univ. Ghardaïa

President

Examiner

Examiner

Supervisor

Academic Year: 2026 / 2025

Acknowledgments:

I would like to express my sincere thanks and deep appreciation to the Faculty of Science and Technology at Ghardaia University for its continuous support, valuable guidance, and dedicated efforts in providing an outstanding academic environment that encourages learning, research, and innovation.

I would also like to extend my heartfelt gratitude to my family and friends and my supervisor for their constant support, patience, and encouragement throughout my academic journey, and to everyone who supported me, stood by my side, and offered help in one way or another.

I am truly grateful for all the opportunities, knowledge, and support I have received, and I wish everyone continued success and excellence.

إهداء

أُتقدم بخالص الشكر وعظيم التقدير إلى كلية العلوم والتكنولوجيا بجامعة غرداية على دعمها المستمر، وتوجيهاتها القيّمة، وجهودها المخلصة في توفير بيئة أكاديمية متميزة تشجع على التعلم والبحث والابتكار.

كما أتوجه بجزيل الشكر والامتنان إلى أهلي الكرام و اصدقائي و المؤطر على دعمهم المتواصل، وصبرهم، وتشجيعهم الدائم طوال مسيرتي الدراسية، وإلى كل من ساندني ووقف إلى جانبي وقدم لي المساعدة من قريب أو بعيد.

أعبر عن امتناني العميق لكل ما تلقّيته من دعم ومعرفة، وأتمنى للجميع دوام النجاح والتوفيق.

Résumé:

Cette recherche évalue le facteur d'intensité de contrainte (SIF) dans les structures fissurées et étudie comment les variations des dimensions de fissure et des conditions de chargement influencent le comportement de la fracture. L'étude aborde les limites historiques de l'ingénierie classique en considérant explicitement la présence de défauts souvent décrits comme des « destructeurs silencieux » des matériaux afin d'assurer la sécurité et la fiabilité structurelle. Des simulations numériques ont été réalisées en utilisant la méthode des éléments finis (MEF) dans l'environnement ABAQUS. Une plaque d'acier inoxydable 2D a été modélisée avec un raffinement de maillage spécialisé à l'extrémité de la fissure afin de capturer le champ de contrainte singulier, et les valeurs de KI ont été extraites à l'aide de la méthode J-intégrale. La simulation a donné une valeur de KI de 19,98 pour le modèle de base. Des études paramétriques ont révélé que l'augmentation de la longueur de fissure ou de la pression externe élève considérablement le rapport KI/KIc, rapprochant ainsi le composant du seuil de propagation instable et catastrophique. Ce travail confirme que le prototypage virtuel est un atout stratégique pour l'évaluation de l'intégrité structurelle et l'analyse de la tolérance aux dommages. En quantifiant la singularité de contrainte mathématique à l'extrémité de la fissure, la méthodologie fournit un cadre robuste pour des conceptions d'ingénierie plus sûres. En fin de compte, la recherche prouve que la modélisation numérique prédit avec précision les limites de défaillance, garantissant la fiabilité des composants critiques dans les applications industrielles.

Mots-clés:

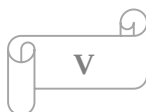
Mécanique de la rupture, facteur d'intensité de contrainte (FIC), ABAQUS, méthode des éléments finis (MEF), J-intégrale, mécanique linéaire élastique de la rupture (LEFM), tolérance aux dommages, singularité de contrainte.

Abstract:

This research evaluates the Stress Intensity Factor (SIF) in cracked structures and investigates how variations in crack dimensions and loading conditions influence fracture behavior. The study addresses the historical limitations of classical engineering by explicitly considering the presence of flaws often described as the "silent destroyers" of materials to ensure structural safety and reliability. Numerical simulations were performed using the Finite Element Method (FEM) within the ABAQUS environment. A 2D stainless steel plate was modeled with specialized mesh refinement at the crack tip to capture the singular stress field, and KI values were extracted using the J-integral method. The simulation yielded a KI value of 19.98 for the baseline model. Parametric studies revealed that increasing the crack length or external pressure significantly elevates the KI/K_{Ic} ratio moving the component closer to the threshold of unstable, catastrophic propagation. This work confirms that virtual prototyping is a strategic asset for structural integrity assessment and damage-tolerance analysis. By quantifying the mathematical stress singularity at the crack tip, the methodology provides a robust framework for safer engineering designs. Ultimately, the research proves that numerical modeling accurately predicts failure boundaries, ensuring the reliability of critical components in industrial applications.

Keywords:

Fracture Mechanics, Stress Intensity Factor (SIF), ABAQUS, Finite Element Method (FEM), J-Integral, Linear Elastic Fracture Mechanics (LEFM), Damage Tolerance, Stress Singularity.



الملخص:

يُقيم هذا البحث معامل شدة الإجهاد (SIF) في الهياكل المتشققة، ويتقصى كيفية تأثير التغيرات في أبعاد الشقوق وشروط التحميل على سلوك الكسر. وتعالج الدراسة الحدود التاريخية للهندسة الكلاسيكية من خلال الأخذ في الاعتبار بشكل صريح وجود العيوب، والتي غالبًا ما توصف بأنها "المدمرات الصامتة" للمواد، وذلك لضمان السلامة والموثوقية الهيكلية. أُجريت محاكاة عددية باستخدام طريقة العناصر المحدودة (FEM) ضمن بيئة برنامج أباكوس (ABAQUS)؛ حيث تم نمذجة صفيحة ثنائية الأبعاد (2D) من الفولاذ المقاوم للصدأ مع تكثيف مخصص للشبكة (Mesh refinement) عند قمة الشق لالتقاط حقل الإجهاد المفرد، واستُخلصت قيم KI باستخدام طريقة تكامل جي (J-integral). وقد أسفرت المحاكاة عن قيمة KI بلغت 19.98 للنموذج الأساسي. وأظهرت الدراسات البارامترية أن زيادة طول الشق أو الضغط الخارجي ترفع بشكل كبير من نسبة KI/KIc ، مما يقرب المكون الميكانيكي من عتبة الانتشار الكارثي غير المستقر للشق. يؤكد هذا العمل أن النمذجة الأولية الافتراضية تعد أداة استراتيجية لتقييم السلامة الهيكلية وتحليل تحمل الضرر. ومن خلال التحديد الكمي للتفرد الرياضي للإجهاد عند قمة الشق، توفر هذه المنهجية إطارًا قويًا لتصميمات هندسية أكثر أمانًا وتوازنًا. وفي النهاية، يثبت البحث أن النمذجة العددية تتنبأ بدقة بحدود الفشل، مما يضمن موثوقية المكونات الحساسة في التطبيقات الصناعية.

الكلمات المفتاحية:

ميكانيكا الكسر، معامل شدة الإجهاد (SIF)، أباكوس (ABAQUS)، طريقة العناصر المحدودة (FEM)، تكامل جي (J-Integral)، ميكانيكا الكسر المرن الخطي (LEFM)، تحمل الضرر، تفرد الإجهاد.

Table of contents

Acknowledgments.....	II
Abstract.....	V
List of tables.....	XI
List of figures.....	XII
List of Abbreviations.....	XIII
General introduction	1

Chapter 1: Overview of Material Failure

1.1) Introduction-Fracture Mechanics and Fatigue.....	4
1.1.1) Why Fracture Mechanics?	4
1.1.2) Stress Analysis near an elliptical hole.....	4
1.1.3) Stress Concentration: The Silent Destroyer of Materials.....	5
1.1.4) Mechanism: Flow Lines and Crowding.....	5
1.1.5) Elliptical Hole: A Critical Case Study.....	6
1.1.6) Fatigue makes it worse.....	6
1.2) stress intensity factor:	6
1.2.1) Mechanical Analysis:	6
1.3) Modes of Fracture:.....	8
1.4) Linear Elastic Fracture Mechanics (LEFM).....	9
1.4.1) SIF K:.....	9
1.5) Griffith's Energy Theory of Fracture	10
1.5.1) Thermodynamic Criterion:.....	11
1.5.2) Classical Griffith Equation:.....	11
1.5.3) Relationship between G and K:.....	12
1.6) Irwin's Modification of Griffith:.....	13
1.6.1) Plasticity at the Crack Tip R_p :.....	15
1.6.2) Irwin's Plastic Zone Correction:.....	15
1.7) Fatigue Curves and Fatigue Laws:.....	16
1.7.1) S-N curve (Stress-Number curve):.....	16

1.7.2) Fatigue Limit:.....	17
1.7.3) Endurance Limit:.....	17
1.7.4) Partitioning of the Wöhler Curve:.....	17
1.8) Goodman Diagram:.....	18
1.8.1) Goodman Criterion.....	18
Conclusion.....	20

Chapter 2: Numerical Modeling Using The Finite Element Method

2.1) Overview of numerical simulation software.....	22
2.2) Introduction of ABAQUS.....	22
2.3) Main characteristics.....	23
2.3.1) ABAQUS/Standard:.....	24
2.3.2) ABAQUS/Explicit:.....	24
2.3.3) ABAQUS/CAE:.....	24
2.4) Finite Element Modeling in ABAQUS.....	25
2.5) Unit systems in ABAQUS.....	26
2.6) Types of problems solved with ABAQUS.....	27
2.6.1) Linear and Nonlinear Problems:.....	27
2.6.2) Static and Dynamic Problems:.....	28
2.6.3) Structural Mechanics Problems:.....	28
2.6.4) Fracture Mechanics Problems:.....	28
2.7) ABAQUS/CAE Environment.....	28
2.7.1) Organization of the ABAQUS/CAE Interface.....	29
2.7.2) Model tree.....	39
2.7.3) Definition and Role:.....	30
2.7.4) Modular Structure:.....	31
2.8) Main Modules in ABAQUS.....	31
2.8.1) Part module.....	32

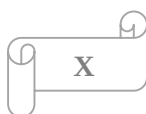
2.8.2) Property module.....	33
2.8.3) Assembly module.....	34
2.8.4) Step module.....	35
2.8.5) Interaction module.....	36
2.8.6) Load module.....	36
2.8.7) Mesh module.....	37
2.8.8) Job module.....	37
2.8.9) Visualization module.....	38
2.8.10) Sketch Module.....	38
2.9) Finite Elements Used in ABAQUS/.....	38
2.9.1) Element Characterization and Naming:.....	38
2.9.2) Continuum Elements for Fracture Analysis:.....	39
2.9.3) Singular Elements for the Crack Tip:.....	39
2.9.4) SIF Extraction via the J-Integral:.....	39
Conclusion.....	40

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

3.1) Modeling.....	42
3.1.1) Load, Boundary conditions and Mesh.....	43
3.2) Results.....	45
3.2.1) Stress intensity factor (Theory vs. Simulation).....	45
3.3) Stress distribution.....	45
3.4) Crack variation Effect.....	46
3.4.1) Comparing numerical results versus analytical results.....	46
3.4.2) Graph KI/ K _{Ic}	47
3.5) Load variation Effect.....	48
3.5.1) Comparing numerical results versus analytical results.....	48
3.5.2) Graph KI/ K _{Ic}	48
General conclusion:.....	50

references:.....	51
------------------	----

List of Tables:



List of Tables	Pages
Table 2.1: Unit systems in ABAQUS	27
Table 3.1: Stainless steel properties	42
Table 3.2: Numerical crack length results VS analytical results	47
Table 3.3: K_c / K_{Ic}	47
Table 3.4: Numerical load results VS analytical results	48
Table 3.5: K_c / K_{Ic}	48

List of Figures

Figures 1.1: Mode I, Mode II, and Mode III crack loading.	9
Figure 1.2 : Schematic representation of a Wöhler curve	16
Figure 1.3 : Wöhler curve	17
Figure 2.1: Some of the most commonly used elements	25
Figure 2.2: Model Tree	30
Figure 2.3: Implementation of the Simulation Model and Data Preparation	31
Figure 2.4: Part module	32
Figure 2.5: Geometry generation	33
Figure 2.6: Materials Property and sections	34
Figure 2.7: Instance and assembly	35
Figure 2.8: Creating steps	35
Figure 2.9: Creating load	36
Figure 2.10: Mesh generating	37
Figure 2.11: Computation run	37
Figure 2.12: Results Visualization	38
Figure 3.1: plate model	42
Figure 3.2: Simulation loading parameters	43
Figure 3.3: global size plate mesh	44
Figure 3.4: local size mesh around the crack tip	44
Figure 3.5 : stress distribution for mechanical loading $\sigma = 10\text{MPa}$	45-46
Figure 3.6: graph K_I / K_{Ic}	47
Figure 3.7: K_I / K_{Ic} load	48

List of Abbreviations

Abbreviation	Concept
A	is the half-width of the hole (perpendicular to load)
B	is the half-height of the hole (parallel to load)
Σ	the applied stress
a in K	the crack size/length
Y	geometry factor
I	refers to Mode I (opening mode)
C	denotes the critical value under specific (standardized) conditions
U	elastic energy
W	energy required for crack growth
E	Young's modulus
Σf	critical fracture stress
Y	specific surface energy
G_c	surface energy contribution
G_p	plastic energy contribution
σ_m	Mean stress
Σa	Alternating stress
σ_D	endurance limit
σ_R	ultimate tensile strength
Y	Poisson's Ratio
σ_y	Yield strength

General introduction

Until the beginning of the 20th century, it was generally assumed that material failure occurred when certain stress components reached critical values, such as tensile or compressive strength. Therefore, the engineering design was largely based on permissible pressures obtained from experiments on samples considered homogeneous and free of defects. However, several service failures have been observed at stress levels well below the theoretical resistance of the material, revealing the limitations of this approach.

A decisive step toward a better understanding of fracture was taken in 1920 by Griffith's pioneering work. He showed that real materials always contain small defects or microscopic cracks that generate large local stress concentrations. As a result, average pressures calculated assuming a continuous, defect-free medium cannot accurately represent the conditions that lead to rupture. The presence and development of cracks must be explicitly taken into account. This vision laid the foundation for the development of fracture mechanics.

Cracks can arise from manufacturing defects, intrinsic material heterogeneity, or progressive damage mechanisms such as fatigue and corrosion. Even when these defects are microscopic, they may significantly reduce the structure's ability to withstand loads. Crack growth can remain stable for long periods before suddenly becoming unstable, leading to sudden and sometimes catastrophic failure. For this reason, the study of cracked objects has become essential to ensure the safety, reliability and durability of engineering components across many applications.

The main challenge in analyzing such problems arises from the behavior of the stress field near the crack tip. In this region, stresses vary rapidly and exhibit a unique character, making classical descriptions of material strength inadequate. Fracture mechanics overcomes this difficulty by introducing parameters that determine the intensity of fields near the tip and relate them to the onset of crack propagation. Within the framework of linear elastic fracture mechanics (LEFM), the stress intensity factor K_I is one of the most important parameters. It characterizes the amplitude of single stress distribution and depends on the applied loads, crack dimensions and structural geometry. The crack is expected to extend when K_I reaches a critical value K_{Ic} , fracture toughness, which is an essential physical property.

While the theoretical foundations of LEFM are well established, modern structural integrity assessments rely heavily on advanced computational techniques. Recent literature emphasizes the use of the finite element method (FEM) to accurately simulate complex crack tip stress singularities

General introduction

and predict structural failure limits. In particular, virtual prototyping via software like Abaqus has become an indispensable engineering standard for conducting reliable damage tolerance analyses and improving material performance without the high costs of physical testing.

In this context, the primary objective of this thesis is to numerically evaluate the stress intensity factor in the open position KI in a stainless steel plate and to study the limits of structural stability under various mechanical constraints. Specifically, this paper addresses the following fundamental question: How do gradual variations in crack length (from 1 mm to 3 mm) and applied tensile loads (from 10 MPa to 100 MPa) affect the KI/KIc ratio and determine the unstable diffusion threshold? By improving understanding of these parameters, this work aims to provide a robust framework for engineers to improve component design, achieving the necessary balance between safety, cost and mechanical performance. [1]

To address these objectives, this thesis is organized as follows:

Chapter 1 provides a comprehensive review of the literature on fracture mechanics, detailing the theoretical principles of LEFM and the stress intensity factor.

Chapter 2 defines the methodology, describing the finite element method and the specific modeling procedures used within the Abaqus environment.

Chapter 3 presents numerical results and discussion, analyzing parametric simulations to determine critical thresholds for failure.

Finally, the general conclusion summarizes the main findings of the research and suggests potential avenues for future improvement studies.

Chapter 1: Overview of Material failure

1.1) Introduction-Fracture Mechanics and Fatigue:

Fracture mechanics and material fatigue are essential fields in mechanical engineering and materials science. They focus on the study of material failures under mechanical stresses, whether static or dynamic.[2]

1.1.1) Fracture Mechanics:

This discipline investigates how and why materials break. It involves analyzing cracks, internal defects, and the mechanisms of crack propagation. Griffith's laws and other theoretical models are often used to predict a material's fracture toughness.

a) Fatigue: Material fatigue refers to failure that occurs under repeated cyclic loads. Fatigue phenomena can lead to fractures even when the applied stresses are below the material's ultimate strength. S-N (stress-number) curves are common tools used to assess fatigue resistance.

b) Importance: Understanding these phenomena is crucial for the safe and sustainable design of structures and components. Applications range from bridges to aircraft, where the safety and longevity of materials are paramount. Analyzing fracture mechanics and fatigue helps minimize failure risks and optimize material performance

c) Why Fracture Mechanics?: Conventional design procedures based only on some maximum stress criterion are not adequate under all circumstances.

Fracture mechanics determine failure based on the interaction between the applied stress, the crack (or flaw) and material parameters (e.g., fracture toughness).

Instead of the magnitude of stress or strain, fracture mechanics is concerned primarily with the distribution of stresses and displacements in the vicinity of a crack tip.

1.1.2) Stress Analysis Near an Elliptical Hole:

Stress analysis near an elliptical hole is a critical area in materials mechanics and structural engineering. It aims to understand how external loads affect the stress distribution around elliptical voids in materials.

Understanding the stress distribution around elliptical holes is vital for the design and analysis of structural components in various applications, including aerospace, civil engineering, and mechanical systems. Proper assessment of stress concentrations can enhance safety, durability, and material performance. [3]

1.1.3) Stress Concentration: The Silent Destroyer of Materials:

In the fundamental study of mechanics often relies on a simplified assumption: that stress is distributed uniformly across a material. We calculate stress (σ) simply by dividing the applied force (F) by the cross-sectional area (A), giving us the nominal or average stress.

However, real-world components rarely behave with such uniformity. While the "average" stress in a part might appear perfectly safe and well within design limits, components often fail unexpectedly. And the real-world components rarely experience uniform stress distribution due to engineering disturbances. The culprit is usually a localized phenomenon known as Stress Concentration.

1.1.4) Mechanism: Flow Lines and Crowding:

To understand why this happens, it is helpful to visualize stress as "flow lines" moving through a material, much like water flowing through a pipe. In a uniform bar, these lines run parallel and evenly spaced.

However, any geometric disturbance acts as an obstacle. Common disturbances include:

- Holes
- Notches
- Sharp corners
- Threads
- Scratches
- Sudden changes in cross-section

When the stress flow lines encounter one of these discontinuities, they are forced to curve and detour around the obstacle. Just as a wide river moves faster and more violently when forced through a narrow gorge, stress lines "crowd" together tightly near the edges of the discontinuity.

Where these lines crowd, the local stress skyrockets, becoming significantly higher than the nominal average value. This amplification of stress at a specific point is the definition of stress concentration.

1.1.5) Elliptical Hole: A Critical Case Study:

A classic and essential example of this phenomenon is the presence of an elliptical hole in a material. The shape of the hole plays a massive role in how severe the stress concentration will be.

When an elliptical hole is present, the stress is not distributed evenly around the rim. Instead, it peaks at the tips of the ellipse that are perpendicular to the direction of the applied load. The sharper the ellipse (i.e., the narrower the tip), the more severe the crowding of the flow lines, and the higher the stress concentration.

Mathematically, this is often expressed using the Stress Concentration Factor (K_t), which is the ratio of the maximum stress ($\sigma_{\max}$) to the nominal stress (σ_{nom}):

$$\sigma_{\max} K_t = \sigma_{\text{nom}}$$

For an elliptical hole in a wide plate under tension, the stress at the tip of the hole is approximately:

$$\sigma_{\max} = \sigma_{\text{nom}} (1 + 2(a/b))$$

Where:

a : is the half-width of the hole (perpendicular to load).

b : is the half-height of the hole (parallel to load)

As the hole becomes flatter (smaller b), the stress concentration factor increases dramatically, mimicking the behavior of a sharp crack

1.1.6) Fatigue makes it worse:

Under cyclic loading, cracks can grow at stresses far below the static strength. Each cycle extends the crack slightly. Over thousands or millions of cycles, it becomes large enough for sudden fracture.

So a component can fail at "only" 40-50% of its yield strength if the geometry is unfavorable.

1.2) The stress intensity factor:

1.2.1) Mechanical Analysis:

Fracture mechanics examines the interaction between a geometric discontinuity (a crack) and the surrounding continuous medium, as well as the evolution of this discontinuity.

From a mechanical standpoint, three successive zones can be schematically distinguished in a cracked medium:

a) process zone:

This zone is located at the crack tip and in the wake left by the crack during its propagation. The study of this zone is highly complex due to the significant stresses that have severely damaged the material. It is discontinuous in the sense of solid mechanics. Classical fracture mechanics theory reduces this zone to a point in two-dimensional problems and to a curve in three-dimensional problems.

b) singular zone:

In this region, the displacement, strain, and stress fields are continuous and can be expressed through a formulation independent of the remote geometry of the structure. It can be demonstrated that, within this zone, the stress field components become infinite in the vicinity of the crack front ($r \rightarrow 0$). More precisely, the singularity follows a $1/\sqrt{r}$ behavior in linear elastic media.

Since materials possess a yield strength, there exists a radius r_p around the crack tip that defines the extent of the plastic zone. Depending on the magnitude of r_p , fracture is considered brittle when r_p is small and ductile when r_p is large.

This distinction, based on the parameter r_p , is crucial because it determines the validity of the theoretical framework employed:

Linear Elastic Fracture Mechanics (LEFM) for brittle fracture;

Nonlinear Fracture Mechanics for cases involving significant plastic zones (ductile fracture).

c) outer (Zone 3) region:

This region encompasses the far-field conditions, connecting on one side to the singular zone and on the other to the boundary conditions in terms of loads and displacements. In this area, the displacement, strain, and stress fields vary gradually and can be approximated by polynomials commonly used in the finite element method.

Various analytical methods are available to study displacement, strain, and stress fields in the vicinity of a crack. These methods can be grouped into two main categories:

Direct approaches, which are based on the use of Airy stress functions. These approaches solve planar problems and rely on the determination of analytical functions.

Energy-based approaches, which rely on the energetic analysis of a continuous medium containing a crack. This involves a global energy balance incorporating the energy release rate associated with a virtual crack extension.

1.3) Modes of Fracture:

Fracture mechanics was initially developed for planar cracks with a straight crack front.

For calculation purposes, the following convention is adopted:

The z-axis corresponds to the crack front line, running along the thickness of the material;

The x-axis lies in the plane of the crack and is oriented toward the uncracked (sound) material;

The y-axis is perpendicular to the crack plane.

In cylindrical coordinates, the crack faces therefore correspond to $\theta \pm 180^\circ$.

(Diagram to be included - elements to be provided to Mathieu.)

Three modes of fracture are distinguished according to the direction of the relative displacement of the crack faces:

- a) **Mode I (opening mode):** Displacements occur in the y-direction, perpendicular to the crack plane, causing the crack faces to separate.
- b) **Mode II (in-plane shear mode or sliding mode):** Displacements occur solely in the x-direction; the crack faces slide over one another in a direction perpendicular to the crack front.
- c) **Mode III (anti-plane shear mode or tearing mode):** Displacements occur parallel to the crack front, in the z-direction.

The general case corresponds to a superposition of two or three fracture modes; this is referred to as mixed-mode fracture.

It will be shown later that Mode I is the predominant fracture mode and is also the most critical.[4]

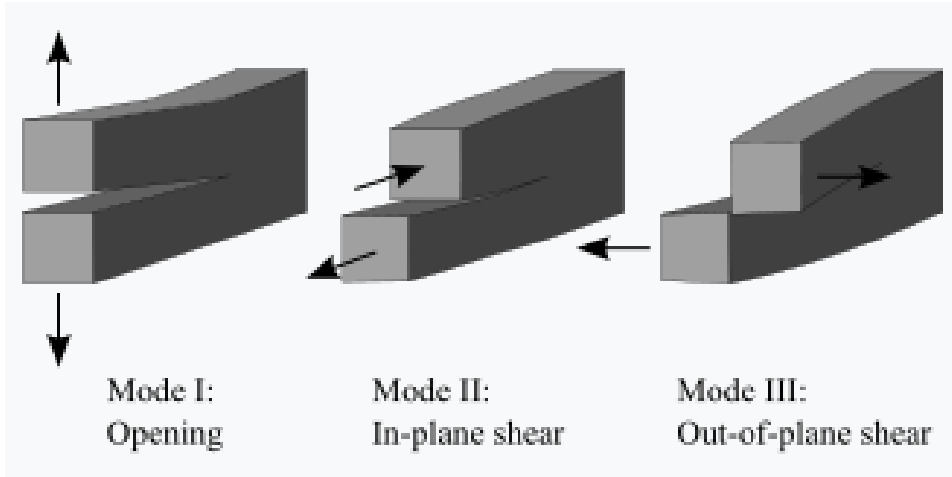


Figure 1.1: Mode I, Mode II, and Mode III crack loading.

1.4) Linear Elastic Fracture Mechanics (LEFM):

Definition:

LEFM (Linear Elastic Fracture Mechanics) studies the behavior of cracks in materials assuming linear-elastic behavior. It is valid when plastic deformation near the crack tip is negligible compared to the overall structure. The stress intensity factor K is the key parameter, describing the stress concentration near the crack tip. Rupture occurs when K reaches a critical value K_{IC} , which is a material property called fracture toughness.[5]

1.4.1) Stress Intensity Factor (K):

Stress Intensity Factor (**SIF**) K is a central concept in fracture mechanics, especially in the Linear Elastic Fracture Mechanics (LEFM) framework. It quantifies how severe the stress field is near the tip of a crack in a material.[6]

a) What is K?:

In fracture mechanics, K is a parameter that describes how intense the stresses are near the crack tip due to an external load. It consolidates the effects of:

- the applied stress (σ),
- the crack size/length (a)
- the geometry of the specimen (through a geometry factor often denoted Y or $f(a/W)$).

In its simplest form for a through crack in a wide plate under tensile loading (Mode I):

$$K = Y\sigma\sqrt{\pi a} \quad (1)$$

Chapter 1: Overview of Material failure

Here Y is a dimensionless correction factor depending on the specimen geometry and crack configuration.

b) Physical Meaning:

At the tip of a crack, classical elasticity predicts that stresses become very large mathematically tending toward infinity as the distance to the tip goes to zero.

However, in fracture mechanics, we don't need the full stress field everywhere; we only need this singular near-tip behavior, and K captures that.

All components of stress near the crack tip depend on K and scale with $\frac{1}{\sqrt{r}}$, where r is the distance from the crack tip.

This behavior makes K extremely useful for predicting crack propagation: as K increases with loading or crack length, it approaches a critical value at which unstable fracture occurs.

c) Fracture Criterion:

A crack will start to grow unstably when the stress intensity at its tip reaches a critical value characteristic of the material.

This critical value is called the fracture toughness, usually denoted:

$$K_{Ic}$$

K_{Ic} is expressed in [MPa-mv].

Where:

- I refers to Mode I (opening mode).
- c denotes the critical value under specific (standardized) conditions.

When:

- $K_I < K_{Ic} \rightarrow$ the crack remains stable.
- $K_I = K_{Ic} \rightarrow$ the crack begins to propagate.
- $K_I > K_{Ic} \rightarrow$ fracture occurs.

1.5) Griffith's Energy Theory of Fracture :

The Griffith energy criterion is a foundational theory in fracture mechanics that explains why microscopic cracks can cause material failure at much lower stresses than would be expected

Chapter 1: Overview of Material failure

from atomic bond strength alone. It was first proposed by A. A. Griffith in 1921 to describe brittle fracture phenomena.[7]

a) **Basic Idea:**

Griffith observed that:

- Real materials always contain microscopic flaws (cracks or voids).
- When a crack extends, there are two competing energy effects:
 1. Elastic strain energy is released from the material around the crack.
 2. Surface energy increases because new crack surfaces are created.
- A crack will grow spontaneously only if the decrease in elastic strain energy due to crack growth is greater than the increase in surface energy required to create the new crack surfaces.

1.5.1) Thermodynamic Criterion:

Griffith's original energy balance leads to a criterion for unstable crack propagation:

Energy Released by Extension > Energy Needed to Create New Surfaces

This can be written in terms of strain energy release rate and surface energy. In essence:

- The external work done on a cracked solid releases stored elastic strain energy.
- If enough strain energy is released per unit new crack area to offset the energy needed to create two new surfaces, the crack will grow.

1.5.2) Classical Griffith Equation:

For a through crack of length $2a$ in an ideally brittle material under uniaxial tension

Fracture occurs when: $G = G_c$.

Sufficient to provide all the energy that is required for crack growth. If the latter is not the case the stress has to be raised. The triangle ODE represents the amount of energy available if the crack would grow.

The condition for crack growth is:

$$\frac{d(U)}{da} = \frac{d(W)}{da} \quad (2)$$

Where U is the elastic energy and W the energy required for crack growth. Based upon stress field calculations for an elliptical flaw.

Griffith calculated dU/da as:

Chapter 1: Overview of Material failure

$$\frac{d(U)}{da} = \frac{2\pi\sigma^{(2)}a}{E} \quad (3)$$

Per unit plate thickness, where E is Young's modulus. Usually dU/da is replaced by:

$$G = \frac{\pi\sigma^{(2)}a}{E} \quad (4)$$

Which is the so called "elastic energy release rate" per crack tip. G is also called the crack driving force: its dimensions of energy per unit plate thickness and per unit crack extension are also the dimensions of force per unit crack extension.

The energy consumed in crack propagation is denoted by $R = \frac{d(W)}{da}$ which is called the crack resistance. To a first approximation it can be assumed that the energy required to produce a crack (the decohesion of atomic bonds) is the same for each increment da . This means that R is a constant.

The energy condition of equation (1) now states that G must be at least equal to R before crack propagation can occur. If R is a constant this means that G must exceed a certain critical value G_{Ic} . Hence crack growth occurs when:

$$\frac{(\pi\sigma^{(2)}a)}{E} = G_{Ic} \quad \text{OR} \quad \sigma_c = \sqrt{\frac{EG_{Ic}}{\pi a}} \quad (5)$$

Where:

σ_f = critical fracture stress

E = Young's modulus

γ = specific surface energy

a = half-crack length

Knowing that $G_c = 2\gamma$

This shows that fracture strength decreases with increasing crack size, which was a major advance over classical tensile-strength criteria that ignored cracks.

1.5.3) Relationship between G and K :

Griffith derived his equation for glass, which is a very brittle material. Therefore he assumed that R consisted of surface energy only. In ductile materials, such as metals, plastic deformation occurs at the crack tip. Much work is required in producing a new plastic zone at the tip of the advancing crack. Since this plastic zone has to be produced upon crack growth the energy for its

Chapter 1: Overview of Material failure

formation can be considered as energy required for crack propagation. This means that for metals R is mainly plastic energy: the surface energy is so small that it can be neglected.

The energy criterion is a necessary criterion for crack extension. It need not be a sufficient criterion. Even if sufficient energy for crack propagation can be provided, the crack will not propagate unless the material at the crack tip is ready to fail: the material should be at the end of its capacity to take load and to undergo further straining. However, the latter criterion is equivalent to the energy criterion, since it follows from equation (1) and (4) that: [8]

$$\frac{K^2}{E} = G \quad (6)$$

Apparently the stress criterion and the energy criterion are fulfilled simultaneously. Hence, equations (1) and (4) are equivalent. And it shows that equation (5) is valid for plane stress and that a term $(1 - \nu^2)$ has to be added in the case of plane strain, leading to

$$(1 - \nu^2) \left(\frac{K^2}{E} \right) = G$$

And

$$(1 - \nu^2) \left(\frac{K I c^2}{E} \right) = G I c$$

Why Griffith's Theory Needed to Be Modified:

Later research and experiments showed that:

- Many engineering materials (example: steels) undergo plastic deformation near the crack tip before actual crack propagation.
- Griffith's energy balance underestimates the energy needed for failure in ductile materials because it only accounts for surface energy.

Thus, a modification was needed to include plastic work in addition to surface energy.[9]

1.6) Irwin's Modification of Griffith:

In the 1950s, G. R. Irwin extended Griffith's theory by introducing the concept of stress intensity factor (K) and by including plastic energy dissipation at the crack tip.

a) Key Ideas of Irwin's Modification:

1. Plastic zone at the crack tip:

- Irwin recognized that materials like metals develop a plastic zone in front of the crack.
- Energy is consumed not just to create new surfaces but also to deform the material plastically.

2. Stress Intensity Factor K:

- Instead of only using energy balance, Irwin introduced K to quantify how "intense" stresses are around the crack tip under linear elastic conditions.
- He showed that this K directly relates to the energy release rate G.

3. Modified fracture criterion:

- Fracture occurs when:

$$K = K_c \text{ (critical stress intensity factor)}$$

- Or, in energy terms:

$$G = G_c + G_p$$

Where:

G_c = surface energy contribution

G_p = plastic energy contribution

1.6.1) Plasticity at the Crack Tip R_p :

In Linear Elastic Fracture Mechanics (LEFM), the material is assumed to behave elastically. However, very close to the crack tip, the stress field becomes extremely high:

$$\sigma = \frac{K}{\sqrt{2\pi r}}$$

r = radial distance from the crack tip

When the stress reaches the material yield strength σ_y , the material no longer behaves elastically. As a result, a plastic zone develops ahead of the crack tip. The size of this region is characterized by the plastic zone radius, denoted R_p . [10]

Expression for R_p :

The classical estimation proposed by **R. Irwin** is obtained by equating the maximum crack-tip stress to the yield strength.

a) Plane Stress Condition:

$$R_p = \left(\frac{1}{2\pi} \right) \left(\frac{KI}{\sigma_y} \right)^2$$

Characteristics:

- Relatively large plastic zone.
- More ductile behavior.
- Typically occurs in thin plates.

b) Plane Strain Condition:

$$R_p = \left(\frac{1}{6\pi} \right) \left(\frac{KI}{\sigma_y} \right)^2$$

Characteristics:

- Smaller plastic zone.
- Strong three-dimensional constraint.
- More brittle behavior.
- Typical of thick plates.

1.6.2) Irwin's Plastic Zone Correction:

Irwin proposed correcting the effective crack length to account for plasticity:

$$a_{eff} = a + R_p$$

This correction allows elastic solutions to remain applicable while partially accounting for crack-tip plastic deformation.

Why is it important? :

The size of the plastic zone is critical because it determines the validity of LEFM assumptions:

- If $R_p \ll a$, the linear elastic approach is valid.
- If R_p becomes significant relative to crack length or specimen thickness, elastic-plastic fracture mechanics, for example: J-integral methods must be used.

The plastic zone size directly influences:

- The measured fracture toughness **K_{Ic}**.
- The ductile-to-brittle transition.

1.7) Fatigue Curves and Fatigue Laws Definition:

In mechanics and materials science, fatigue refers to the progressive and localized structural damage that occurs in a material subjected to repeated or cyclic loading, even when the maximum stress in each load cycle is well below the material's yield strength or ultimate tensile strength. This process typically involves the initiation of microscopic cracks, their subsequent growth over many cycles, and ultimately the sudden failure of the component when the crack reaches a critical size.

1.7.1) S-N curve (Stress-Number curve):

Illustrates the relationship between the applied stress and the number of cycles required to cause failure in a material subjected to cyclic loading.

a) Principle:

Several identical specimens are subjected to different stress levels, and for each specimen, the number of cycles to failure (N_f) is recorded.

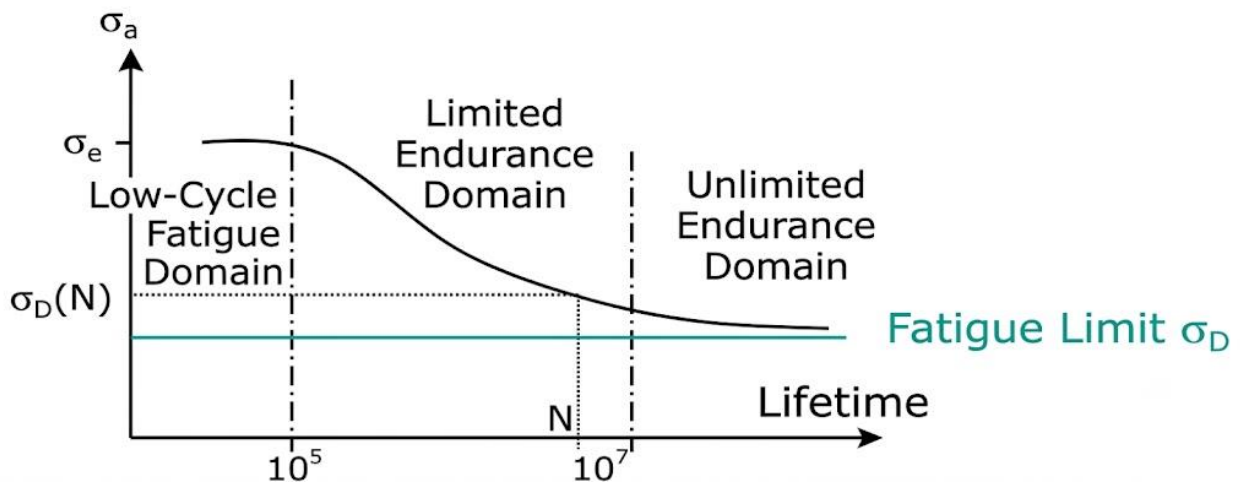


Figure 1.2: Schematic representation of a Wöhler curve

By plotting the maximum stress (S) on the vertical axis and the number of cycles to failure (N) on the horizontal axis-typically using a logarithmic scale for N -the Wöhler curve is obtained.

1.7.2) Fatigue Limit:

The fatigue limit is the stress level below which no failure is observed, even after an infinite number of cycles.

1.7.3) Endurance Limit:

The endurance limit corresponds to the asymptotic value toward which the stress amplitude tends as the number of cycles becomes very large.

In the case of steels, this endurance limit can be practically approached at approximately 10^7 cycles.

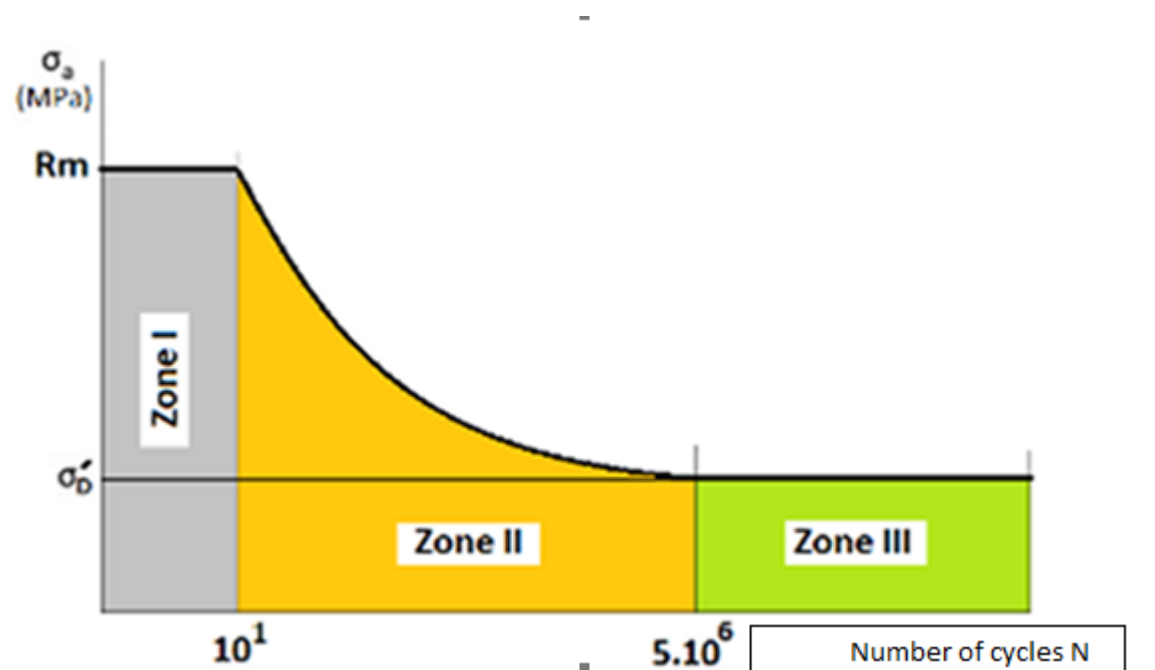


Figure 1.3: Wöhler Curve

1.7.4) Partitioning of the Wöhler Curve:

The Wöhler curve can be divided into three distinct regions:

a) Low-Cycle Fatigue (Zone 1) :

The stress amplitude σ_a is close to the ultimate tensile strength R_m . Specimens fail after only a small number of cycles (from 1 up to about 10 cycles).

b) Finit-Life Fatigue (Zone II) :

This region extends roughly from 10 cycles to $(5 \cdot 10^6)$ cycles and corresponds to an intermediate stress range.

Chapter 1: Overview of Material failure

In this region, all specimens fail after a certain number of cycles, for each specimen, this number represents its finite fatigue life.

c) Infinite-Life Region (Zone III):

Below the endurance limit, the material can theoretically withstand an infinite number of cycles without failure.[11]

1.8) Goodman Diagram:

In practice, the applied stress is not always fully reversed about zero; therefore, a mean stress (σ_m) may be present.

The Goodman diagram accounts for the combined effect of:

- Mean stress (σ_m)
- Alternating stress (σ_a)

It is widely used in fatigue design to evaluate whether a component subjected to fluctuating loading will fail.

1.8.1) Goodman Criterion:

The linear Goodman failure criterion is expressed as:

$$\left(\frac{\sigma_a}{\sigma_D}\right) + \left(\frac{\sigma_m}{\sigma_R}\right) < 1$$

where:

- σ_a = alternating stress
- σ_m = mean stress
- σ_D = endurance limit
- σ_R = ultimate tensile strength

If the left-hand side is less than or equal to 1, the component is considered safe.

Example:

A component is subjected to:

* Maximum stress: 120 MPa

* Minimum stress: 20 MPa

Material properties:

Chapter 1: Overview of Material failure

* Ultimate tensile strength: $\sigma_R = 400\text{MPa}$

* Endurance limit: $\sigma_D = 200\text{MPa}$

Step 1: Calculate Alternating and Mean Stresses

$$\sigma_a = \frac{120 - 20}{2} = 50\text{MPa}$$

$$\sigma_m = \frac{120 + 20}{2} = 70\text{MPa}$$

Step 2: Apply Goodman Criterion

$$(50/200) + (70/400) = 0.25 + 0.175 = 0.425$$

Since:

$$0.425 < 1$$

The component is safe according to the Goodman criterion.

Conclusion:

In conclusion, the first chapter established the basic theoretical bridge between classical engineering design and the modern damage-tolerant model. By going beyond the ideal assumption of defect-free materials, we identified stress concentration localized congestion of "flow lines" around geometric disturbances— as the primary "silent destroyer" responsible for unexpected structural failure. By studying elliptical holes and sharp cracks, it is clear that the integrity of materials is governed not by average pressures, but by the uniqueness of the intense stress present at the tip of the crack.

This chapter has detailed how linear elastic fracture mechanics (LEFM) provides the tools to measure this intensity through stress intensity factor (K) and energy-based parameters. We explored the fundamental contributions of Griffith, who linked crack propagation to energy release, and Irwin, whose crucial modification was responsible for the plastic region a vital consideration of the ductile behavior of engineering metals such as stainless steel. Furthermore, the inclusion of fatigue analysis and the Goodman criterion highlights the fact that even stresses well below the yield strength of a material can lead to catastrophic rupture under cyclic loading. Ultimately, the theoretical frameworks presented here ranging from first-mode opening mechanisms to Fuller curves provide the parameters necessary to define the boundaries between structural stability and failure.

However, since analytical solutions are often limited to simplified geometries, these principles serve as a prerequisite for the numerical modeling techniques that will be explored in the next chapter, where we will leverage ABAQUS to transform these abstract mathematical singularities into predictable geometric data

Chapter 2: Numerical Modeling Using The Finite Element Method

Chapter 2: Numerical Modeling Using The Finite Element Method

2.1) Overview of numerical simulation software:

Numerical simulation software has become an essential tool in modern engineering, enabling the analysis and prediction of complex physical phenomena that are difficult or impossible to solve analytically. These tools are based on advanced numerical methods such as the finite element method (FEM), finite volume method (FVM), and finite difference method (FDM), which allow engineers to discretize and solve governing equations over complex geometries.

A wide range of numerical simulation software has been developed to address different engineering applications, including structural analysis, fluid dynamics, heat transfer, and multiphysics problems. Among the most widely used tools are ABAQUS, ANSYS, and COMSOL Multiphysics. These software packages provide integrated environments that include pre-processing (model creation and meshing), solution algorithms, and post-processing tools for visualization and interpretation of results.

Numerical simulation software plays a critical role in improving product design, reducing development costs, and minimizing the need for experimental testing. By enabling virtual prototyping, engineers can evaluate multiple design configurations, optimize structural performance, and predict failure mechanisms under various loading conditions. This is particularly important in industries such as aerospace, automotive, and civil engineering, where safety, reliability, and efficiency are of paramount importance.

Despite their advantages, the accuracy of numerical simulations depends strongly on the quality of the mathematical models, including material properties, boundary conditions, and mesh refinement. Therefore, simulation results must often be validated through experimental testing to ensure their reliability. Nevertheless, numerical simulation software remains a powerful and indispensable tool in modern engineering analysis and design.[12][13]

2.2) Introduction of ABAQUS:

The rapid development of numerical simulation techniques has led to the emergence of advanced computational tools for solving complex engineering problems. Among these tools, ABAQUS has established itself as one of the most powerful and widely used software packages for finite element analysis in both academic research and industrial applications.

ABAQUS is a general-purpose simulation software based on the Finite Element Method (FEM), designed to model and analyze the physical behavior of engineering structures under various

Chapter 2: Numerical Modeling Using The Finite Element Method

conditions. It is capable of solving a wide range of problems, including structural, thermal, dynamic, and coupled multiphysics analyses.

The software allows engineers to simulate the response of materials and structures subjected to different types of loading, boundary conditions, and environmental effects.

One of the main strengths of ABAQUS lies in its ability to handle highly complex and nonlinear problems.

These include material nonlinearities such as plasticity and damage, geometric nonlinearities involving large deformations, and contact interactions between different bodies. Such capabilities make it particularly suitable for analyzing real-world engineering problems where ideal assumptions are no longer valid.

The software consists of several integrated components, including ABAQUS/Standard, which is used for implicit analysis of static and quasi-static problems, and ABAQUS/Explicit, which is designed for dynamic and highly nonlinear simulations such as impact and crash analysis. In addition, ABAQUS/CAE provides a user-friendly graphical interface for model creation, meshing, simulation control, and visualization of results.

In the field of fracture mechanics, ABAQUS offers advanced tools for modeling crack initiation and propagation, as well as for evaluating fracture parameters such as the Stress Intensity Factor (SIF). These capabilities make it an essential tool for the analysis of cracked structures and the prediction of failure mechanisms.

Due to its robustness, versatility, and accuracy, ABAQUS is widely used in various engineering domains, including mechanical, civil, aerospace, and materials engineering. It plays a crucial role in modern design and analysis processes by enabling virtual testing, reducing experimental costs, and improving the reliability and performance of engineering systems.[14]

2.3) Main characteristics:

ABAQUS was originally developed by Hibbitt, Karlsson & Sorensen (HKS), later known as ABAQUS, over more than three decades, alongside the development of the finite element method. The software has been continuously improved to incorporate advancements in numerical methods and to meet evolving industrial requirements. In October 2005, the company was acquired by Dassault Systèmes.

ABAQUS is a general-purpose finite element simulation software designed to solve a wide range of problems in mechanics and is widely recognized for its robust and efficient treatment of highly

Chapter 2: Numerical Modeling Using The Finite Element Method

nonlinear problems. The software supports multiple physical domains, including mechanical, thermal, and electrical analyses (such as piezoelectric and thermoelectric effects), as well as coupled multiphysics problems. It also provides an advanced material library, including conventional elastic and elastoplastic materials, in addition to specialized models for foams, concrete and soils materials.

The software is capable of solving various classes of engineering problems, including static and dynamic analyses, linear and nonlinear problems, and rigid body dynamics. The ABAQUS software package consists of three main components: ABAQUS/Standard, ABAQUS/Explicit, and ABAQUS/CAE.[15]

2.3.1) ABAQUS/Standard:

ABAQUS/Standard is a general-purpose solver based on a traditional implicit integration scheme.

- The solution procedure is based on the Newton-Raphson algorithm and the Riks method.
- It is capable of solving both linear and nonlinear problems and supports geometries in one-dimensional (1D), two-dimensional (2D), three-dimensional (3D), and axisymmetric forms.
- In addition, it provides numerous analysis procedures in both the time domain and frequency domain.

2.3.2) ABAQUS/Explicit:

ABAQUS/Explicit uses an explicit integration scheme to solve nonlinear dynamic or quasi-static problems.

- It is particularly suitable for transient and dynamic structural analyses.
- The solution is based on an explicit time integration method, which allows efficient handling of highly nonlinear problems involving complex contact and large deformations.

2.3.3) ABAQUS/CAE:

ABAQUS/CAE provides an integrated environment for modeling, simulation, and visualization associated with the ABAQUS solvers. It serves as a graphical interface that enables users to define models, run analyses, and interpret results. Each of these components can be complemented by additional and optional modules designed for specific applications.

ABAQUS offers a wide range of finite elements, which can be classified into several main categories.

Chapter 2: Numerical Modeling Using The Finite Element Method

One-dimensional (1D): elements include bar and beam elements, which may be straight or curved.

Two-dimensional (2D): elements are used for plane stress or plane strain problems, as well as for plate bending and shell structures with triangular or quadrilateral shapes.

Three-dimensional (3D): elements include tetrahedral and hexahedral solids. In addition, axisymmetric elements are used for structures exhibiting rotational symmetry in both geometry and loading conditions, while other elements such as springs, dampers, and rigid elements are also available.

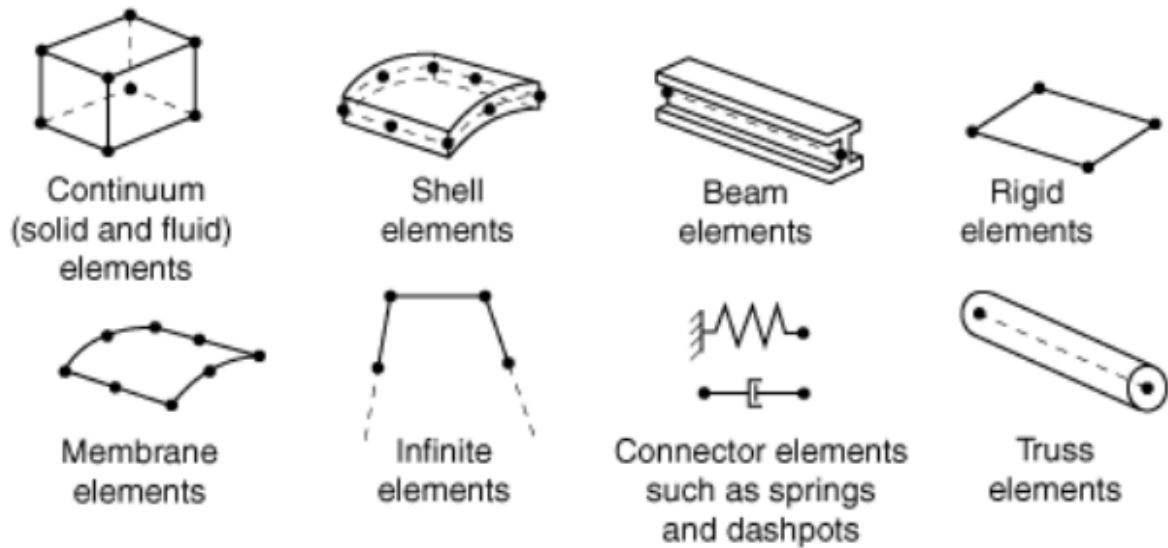


Figure 2.1: Some of the most commonly used elements

2.4) Finite Element Modeling in Abaqus:

Finite element modeling using ABAQUS follows a systematic procedure that enables the simulation of complex engineering problems. This process is generally divided into three main stages: pre-processing, solution, and post-processing.

The pre-processing stage involves the definition of the geometry, material properties, and boundary conditions of the model. The structure is first discretized into a finite number of elements through a meshing process, where the choice of element type and mesh density plays a crucial role in the accuracy of the results. Material behavior is then defined using appropriate constitutive models, and loading conditions are applied to represent real operating conditions.

Once the model is fully defined, the solution phase is carried out using the computational engine of ABAQUS. Depending on the nature of the problem, either ABAQUS/Standard or

Chapter 2: Numerical Modeling Using The Finite Element Method

ABAQUS/Explicit can be used. During this phase, the governing equations are solved numerically to obtain the response of the structure in terms of displacements, stresses, and strains.

The final stage is post-processing, which involves the visualization and interpretation of the results. ABAQUS provides powerful tools for analyzing the distribution of stresses, strains, and displacements within the structure. These results are essential for evaluating the mechanical behavior and identifying critical zones, such as regions of high stress concentration.

In the context of fracture mechanics, finite element modeling plays a crucial role in the analysis of cracked structures. Special attention must be given to mesh refinement near the crack tip in order to accurately capture the stress singularity. ABAQUS provides advanced techniques, such as contour integrals and specialized elements, for the evaluation of fracture parameters, including the Stress Intensity Factor (SIF).

The accuracy and reliability of the numerical results depend on several factors, including the quality of the mesh, the selection of element types, and the proper definition of material properties and boundary conditions. Therefore, careful modeling and validation are essential to ensure meaningful simulation outcomes.[16]

2.5) Unit Systems in ABAQUS:

Before defining a model, it is essential to select the system of units to be used. ABAQUS does not have a built-in unit system; therefore, all input data must be specified using a consistent set of units. The results obtained from the simulation will follow the same unit system. In most cases, a system based on the International System of Units (SI) is adopted, often using millimeters (mm) as the unit of length.[17]

Chapter 2: Numerical Modeling Using The Finite Element Method

Table 1.1: Unit Systems in ABAQUS

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	Mm	Ft	In
Force	N	N	Lbf	Lbf
Mass	Kg	tonne (10 ³ kg)	Slug	Lbf s ² / in
Time	s	S	S	S
Constraint	Pa (N/m ²)	MPa (N/mm ²)	Lbf / ft ²	Psi (lbf / in ²)
Energy	J	mJ (10 ⁻³)	ft lbf	in lbf
Density	Kg/m ³	tonne / mm ³	slug / ft ³	Lbf s ² / in ⁴

2.6) Types of problems solved with ABAQUS:

2.6.1) Linear and Nonlinear Problems:

ABAQUS can solve both:

a) Linear problems:

- Small deformations.
- Linear material behavior.
- No contact or geometric changes.

b) Nonlinear problems:

- Material nonlinearity (plasticity, damage).
- Geometric nonlinearity (large deformation).
- Contact nonlinearity.

Chapter 2: Numerical Modeling Using The Finite Element Method

2.6.2) Static and Dynamic Problems:

a) Static analysis:

- Structures under constant or slowly varying loads.
- Stress, strain, and deformation evaluation.

b) Dynamic analysis:

- Time-dependent problems.

Includes:

- Impact.
- Vibrations.
- Transient loading.

2.6.3) Structural Mechanics Problems:

ABAQUS is widely used for:

- Stress and strain analysis.
- Deformation of structures.

2.6.4) Fracture Mechanics Problems:

- Crack initiation and propagation.
- Stress Intensity Factor (SIF).
- J-integral evaluation.

2.7) ABAQUS/CAE environment:

The ABAQUS/CAE environment is an integrated graphical interface within ABAQUS that enables users to perform all stages of finite element modeling, including pre-processing, simulation execution, and post-processing. It is organized into several functional modules that correspond to different steps of the modeling process, such as geometry creation, material definition, meshing, and results visualization. This environment plays a crucial role in fracture mechanics applications, as it facilitates crack modeling, mesh refinement near the crack tip, and the extraction of parameters such as the Stress Intensity Factor.[18]

Chapter 2: Numerical Modeling Using The Finite Element Method

2.7.1) Organization of the ABAQUS/CAE Interface:

The ABAQUS/CAE interface is designed to provide a structured environment for finite element modeling within ABAQUS. It is organized into several main regions that facilitate interaction with the model.

Main components of the interface:

- Menu Bar and Toolbars:** Provide access to commands and modeling tools.
- Viewport:** The main graphical area where the model is displayed and manipulated.
- Toolbox Area:** Contains tools specific to the active module (e.g., meshing tools, sketch tools).
- Prompt Area:** Displays instructions and messages to guide the user during modeling.
- Module Selector:** Allows switching between different modules such as Part, Property, Assembly, Step, Interaction, Load, -Mesh, Job, and Visualization.

2.7.2) Model tree:

Items in the Model Tree are represented by small icons; for example, the Steps icon,. In addition, parentheses next to an item indicate that the item is a container, and the number in the parentheses indicates the number of items in the container. You can click on the “+” and “-” signs in the Model Tree to expand and collapse a container. The right and left arrow keys perform the same operation.

The arrangement of the containers and items in the Model Tree reflects the order in which you are expected to create your model. As noted earlier, a similar logic governs the order of modules in the module menu, you create parts before you create the assembly, and you create steps before you create loads. This arrangement is fixed, you cannot move items in the Model Tree.[18]

The Model Tree provides most of the functionality of the main menu bar and the module managers. For example, if you double-click on the Parts container, you can create a new part (the equivalent of selecting Part Create from the main menu bar).

The instructions for the examples discussed in this guide will focus on using the Model Tree to access the functionality of Abaqus/CAE. Menu bar actions will be considered only when necessary example: when creating a finite element mesh or post processing results.

The Results Tree uses the same space occupied by the Model Tree. Click the Results tab in the left side of the main window to switch from the Model Tree to the Results Tree. The Results Tree provides access to session-specific features (example: functionality available in only the

Chapter 2: Numerical Modeling Using The Finite Element Method

Visualization module). The Results Tree will be introduced during the course of the post processing exercises contained in this guide.

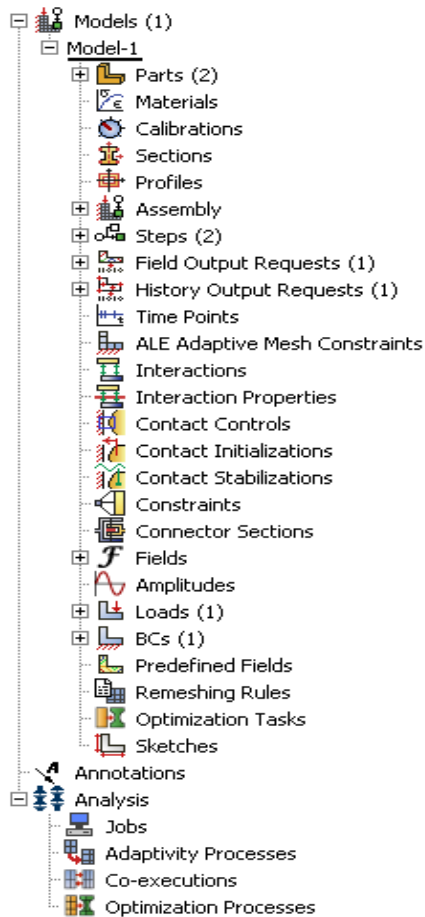


Figure 2.2: Model Tree

2.7.3) Definition and Role:

The ABAQUS/CAE environment provides tools for:

- Geometry creation and modification
- Material definition
- Assembly of components
- Step (analysis) definition
- Application of loads and boundary conditions
- Mesh generation
- Job submission and monitoring
- Visualization and interpretation of results

Chapter 2: Numerical Modeling Using The Finite Element Method

2.7.4) Modular Structure:

The ABAQUS/CAE environment is organized into several modules, each corresponding to a specific stage of the simulation:

- Part module.
- Property module.
- Assembly module.
- Step module.
- Interaction module.
- Load module.
- Mesh module.
- Job module simulation execution.
- Visualization module.

2.8) Main Modules in ABAQUS:

The complete definition of a simulation dataset is executed by systematically navigating through a succession of specialized modules.[19]

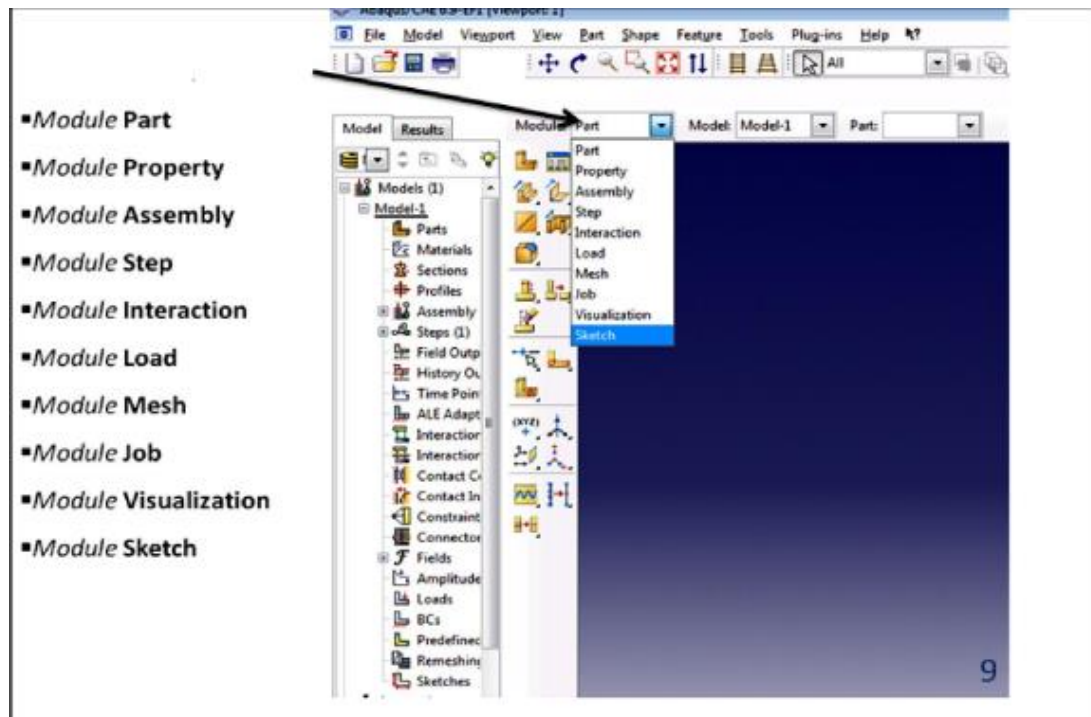


Figure 2.3: Implementation of the Simulation Model and Data Preparation

Chapter 2: Numerical Modeling Using The Finite Element Method

2.8.1) Part module:

The Part Module is used to define the geometry of the model entities (refer to Figure 6 and Figure 7), which can include both deformable and rigid components. New parts are generated by selecting the "Create Part" icon located in the upper-left toolbox of the interface.

To define a part correctly, the following parameters must be selected:

- Modeling Space: 3D, 2D Planar, or Axisymmetric.
- Type: Deformable, Discrete Rigid, or Analytical Rigid.
- Base Feature Shape: Solid, Shell, Wire, or Point.

Upon confirming these settings, the software transitions to an integrated CAD environment (the Sketcher), where the specific geometry of the components is drawn.

In the case of a rigid body, it is essential to establish a Reference Point (RP). This point represents the collective Degrees of Freedom (DOFs) for the entire rigid entity, facilitating the subsequent application of boundary conditions or concentrated loads. This is achieved via the Tools menu by selecting the Reference Point command.

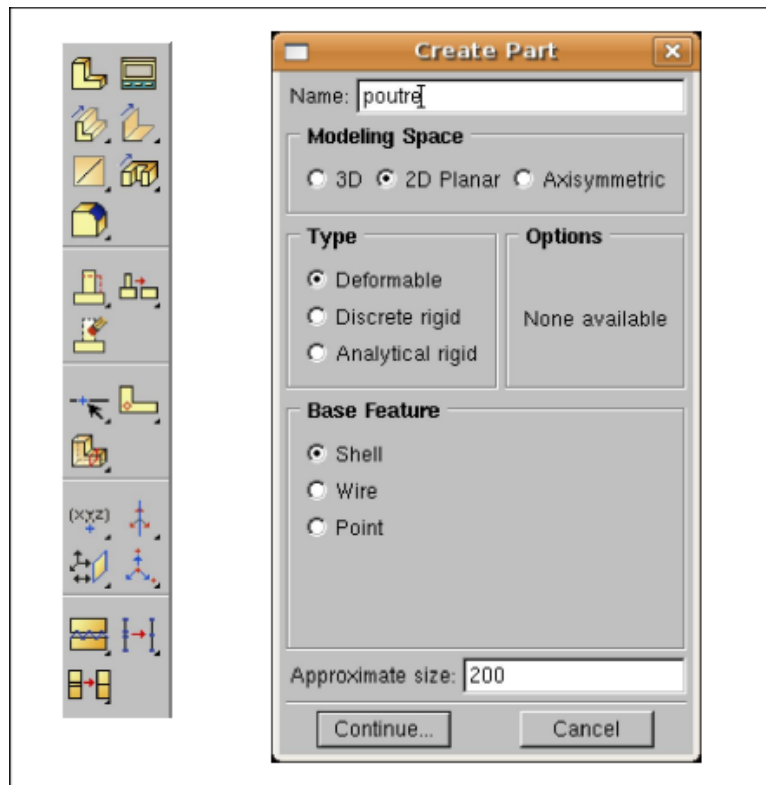


Figure 2.4: Part module

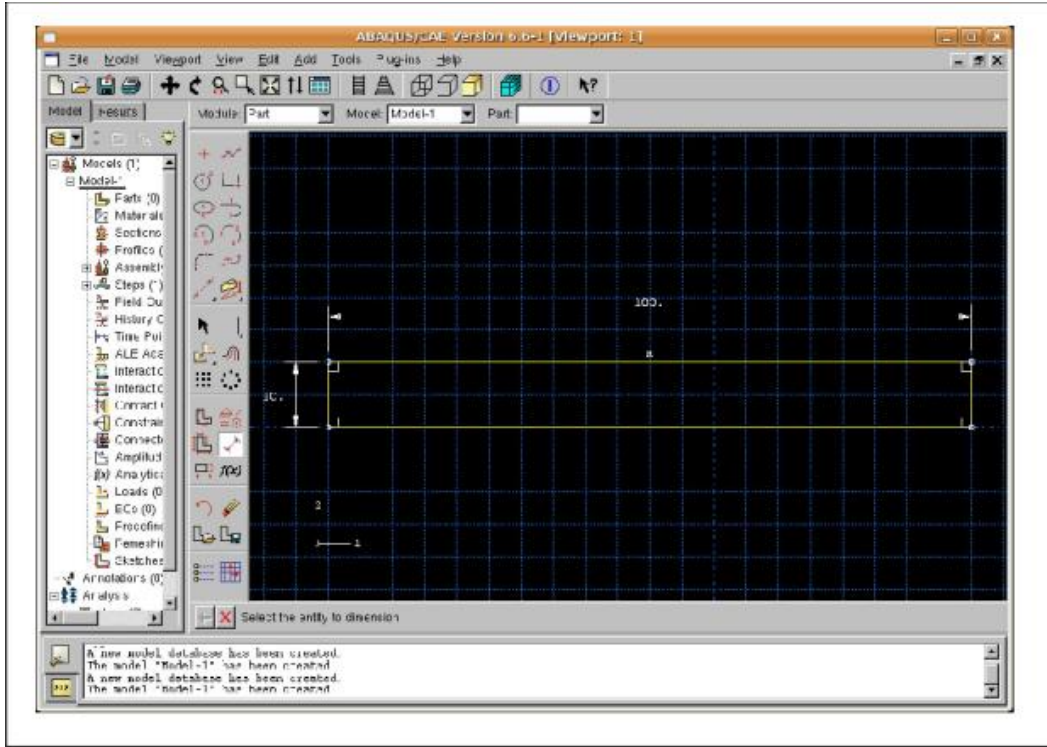


Figure 2.5: Geometry generation

2.8.2) Property module:

The Property Module is used to define the material characteristics of the deformable components within the model, as well as the inertial properties (mass) of rigid bodies when required (refer to Figure 8). For the current study, the material definition involves specifying elastic and/or plastic behavior.

It is considered best practice to assign unique identifiers to each material to facilitate the development of a reusable material database. The following properties are defined:

- Elastic Properties: Configured via Mechanical > Elasticity > Elastic. For an isotropic material, this requires the input of the Young's Modulus and Poisson's Ratio.

- Plastic Properties: Configured via Mechanical > Plasticity > Plastic. Under the assumption of isotropic hardening, the hardening curve is entered as a series of data points consisting of Yield Stress and Plastic Strain.

In accordance with Abaqus' requirement for finite strain analysis, these values must be input as True Stress (Cauchy stress, defined as $\sigma_{\text{true}} = F/A$) and Logarithmic Plastic Strain ($\ln(L/L_0)$).

Following the material definition, these properties must be assigned to the mesh entities. This process requires the creation of a Section (represented by the beam icon). The section references

Chapter 2: Numerical Modeling Using The Finite Element Method

the specific material and defines geometric constants, such as the Thickness for shell or membrane elements.

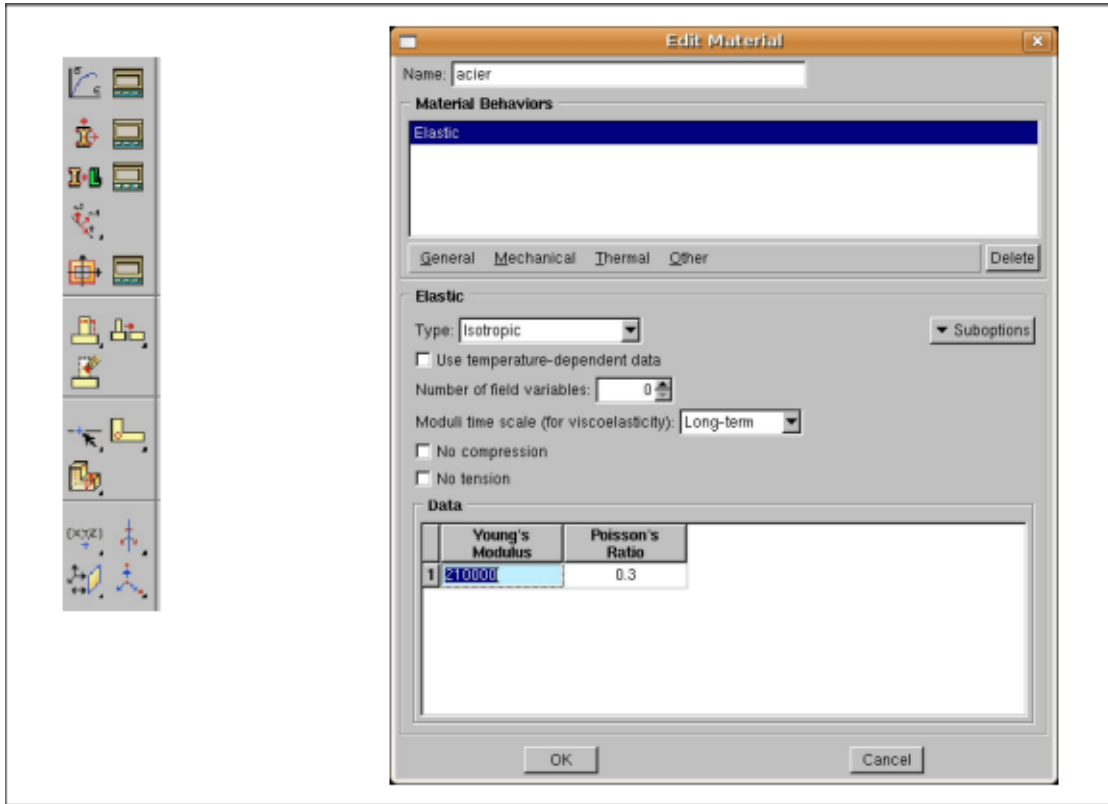


Figure 2.6: Materials Property and sections

2.8.3) Assembly module:

Assembly module: This module involves creating an instance (Figure 9) from the entities previously defined in the Part module. These instances are the elements that will subsequently be manipulated within the model.

In general, using the icon located at the upper left, you create an instance by selecting all components defined in the Part module. If necessary, global transformations can be applied to these instances in order to reposition them within the spatial domain.

Chapter 2: Numerical Modeling Using The Finite Element Method

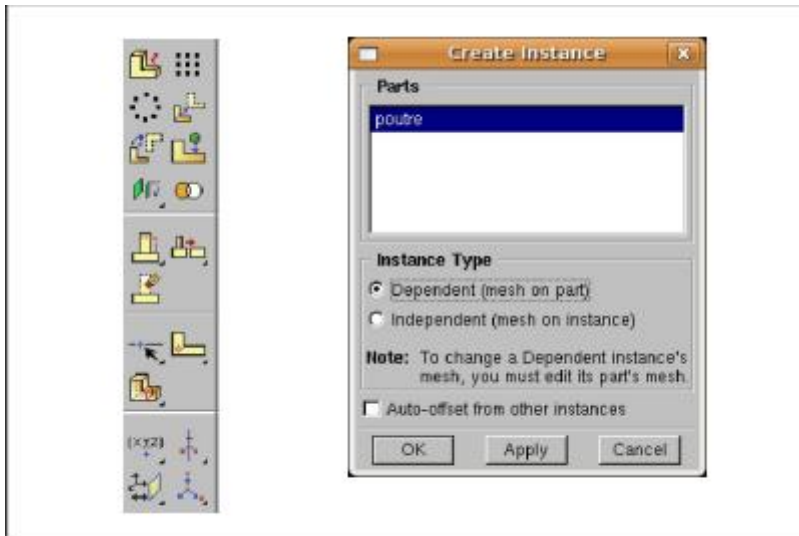


Figure 2.7: Instance and assembly

2.8.4) Step module:

At this stage, as illustrated in (Figure 10), you will select the Dynamic Explicit module for crash or impact analyses. It is worth noting the wide range of additional capabilities offered by this computational code.

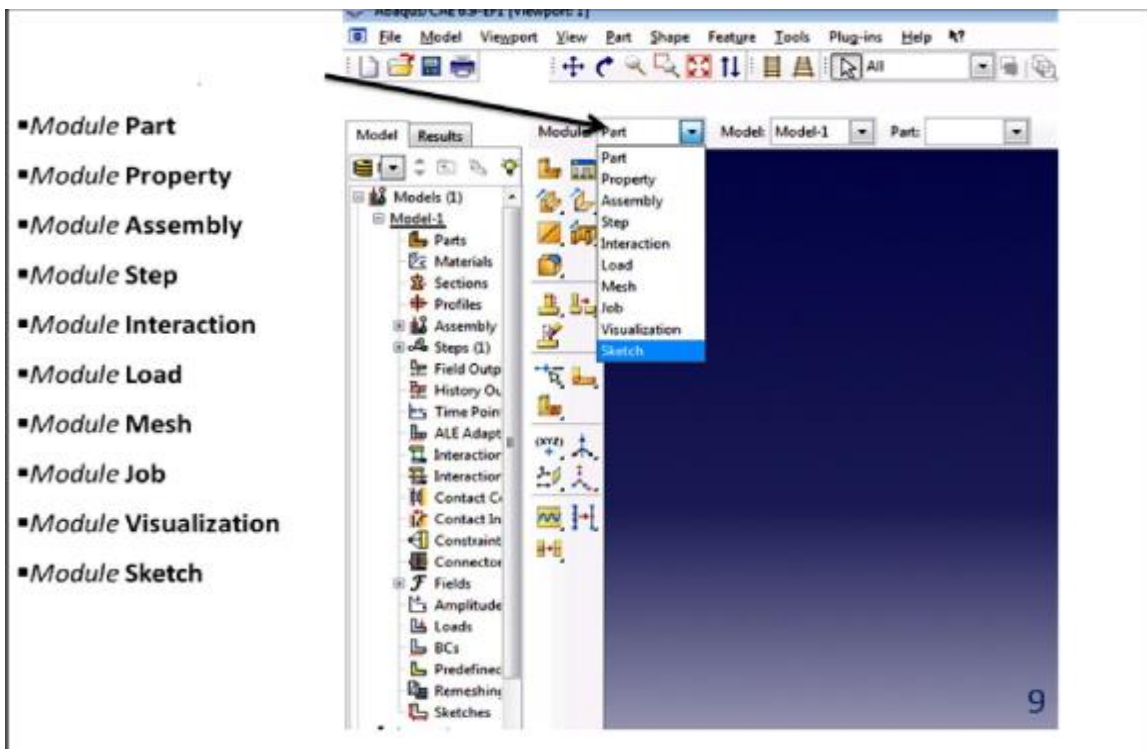


Figure 2.8: Creating steps

Chapter 2: Numerical Modeling Using The Finite Element Method

2.8.5) Interaction module:

This module enables the specification of all interactions between the various parts and regions of the model, whether mechanical, thermal, or otherwise. It is important to note that ABAQUS considers only explicitly defined interactions; mere geometric proximity is not sufficient.

2.8.6) Load module:

The Load module allows the definition of all loads, boundary conditions, and fields (Figure 11). It should be noted that loads and boundary conditions are step-dependent, for example, a force may be applied during Step 1 but remain inactive during Step 2.

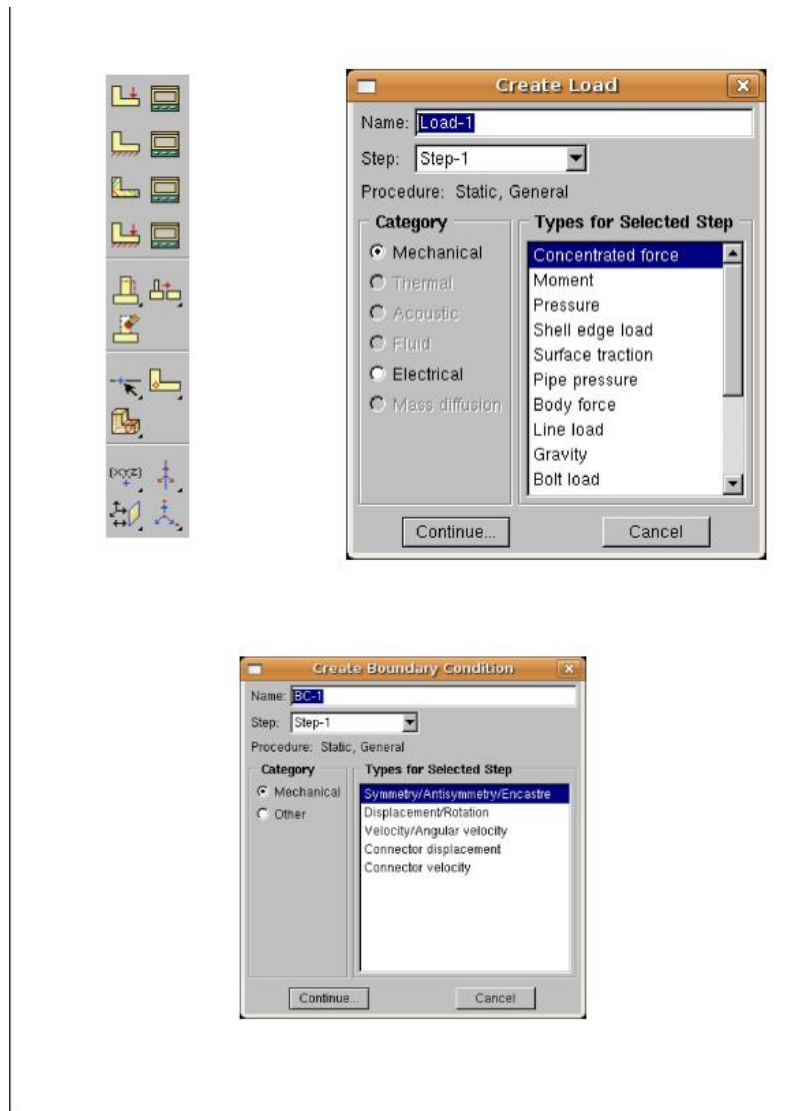


Figure 2.9: Creating load

Chapter 2: Numerical Modeling Using The Finite Element Method

2.8.7) Mesh module:

The Mesh module contains all the necessary tools for generating a finite element mesh on an assembly (Figure 12).



Figure 2.10: Mesh generating

2.8.8) Job module:

The initiation of the computation follows a standard procedure (Figure 13). Once the analysis has been launched, its progress can be monitored by clicking on Monitor, allowing real-time tracking of load increments and assessment of the convergence of the calculations.

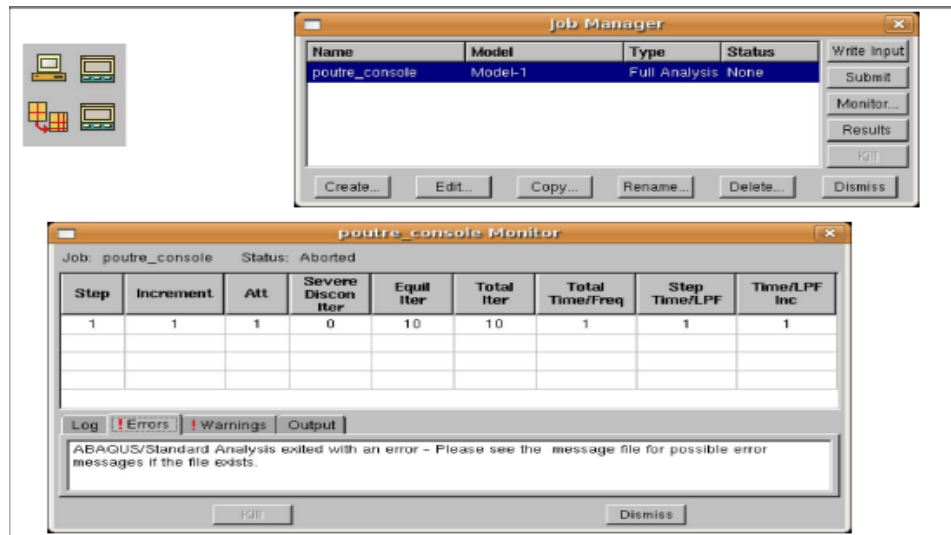


Figure 2.11: Computation run

Chapter 2: Numerical Modeling Using The Finite Element Method

2.8.9) Visualization module:

The Visualization module is used to display all results associated with the part, including stresses and deformations (Figure 14).

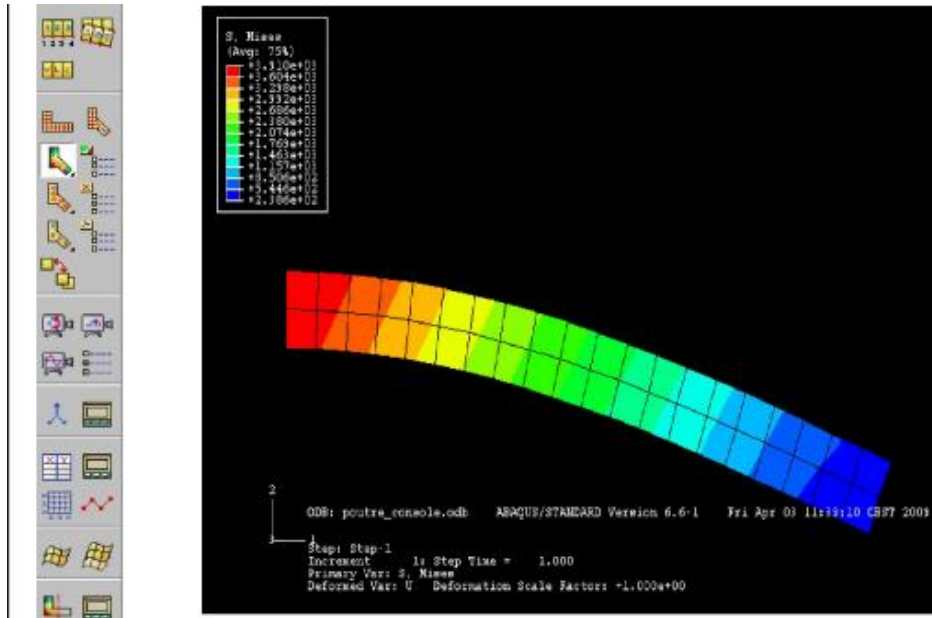


Figure 2.12: Results Visualization

2.8.10) sketch module:

The Sketch module enables the creation of two-dimensional geometries that are not yet associated with a specific object. It can, for example, be used as a basis for extrusion operations.

2.9) Finite Element Library in Abaqus:

Abaqus 6.14 provides an extensive library of finite elements designed to model a wide range of geometric and constitutive behaviors. For the analysis of stress intensity factors (SIF) in cracked structures, the selection is primarily concentrated on Continuum (Solid) Elements.[20]

2.9.1) Element Characterization and Naming:

Each element in Abaqus is identified by a unique alphanumeric code that defines its **family**, **degrees of freedom (DOF)**, and **numerical formulation**. For example, in the element **CPE8R**:

-**C**: Continuum (Solid) family.

-**PE**: Plane Strain formulation.

-**8**: Number of nodes (indicating a second-order/quadratic interpolation).

-**R**: Reduced integration scheme.

Chapter 2: Numerical Modeling Using The Finite Element Method

2.9.2) Continuum Elements for Fracture Analysis:

a) **2D Modeling:** If the structure can be simplified to a 2D plane, abaqus offers two critical variants

Plane Strain Elements (CPE): These are used for thick structures where the strain in the thickness direction is negligible ($\epsilon_z = 0$). In fracture mechanics, plane strain conditions are often preferred as they represent the most conservative "high-constraint" state at the crack tip.

The symbol ϵ (epsilon) represents Strain, which is the measure of how much a material deforms or stretches. When we say ($\epsilon_z = 0$), we are saying there is zero deformation in the thickness direction. This happens in very thick structures.

Plane Stress Elements (CPS): Utilized for thin plates where the out-of-plane stress is zero $\sigma_z = 0$.

b) **3D Solid Elements (C3D):** For complex three-dimensional geometries, hexahedral

Elements like the C3D20 (20-node quadratic) are favored for their superior convergence rates in high-stress gradient zones compared to tetrahedral elements.

2.9.3) Singular Elements for the Crack Tip:

Standard polynomial elements cannot naturally represent the $\frac{1}{\sqrt{r}}$ stress singularity predicted by Linear Elastic Fracture Mechanics (LEFM).

2.9.4) SIF Extraction via the J-Integral:

Because the stress exactly at the crack tip is technically infinite, measuring stress directly at a singular node will yield unstable and mathematically useless results. Instead, numerical modeling relies on energy methods.

The solver calculates the J-integral by evaluating the strain energy release rate along several concentric paths (contours) surrounding the crack tip. Once the J-integral is calculated across these contours, the software converts that energy value back into the Stress Intensity Factor (KI) using the fundamental relationship:

$$KI = \sqrt{J \cdot E'}$$

Where $E' = E$ for plane stress conditions, and $E' = \frac{E}{1 - \nu^2}$ for plane strain conditions (which provides a more conservative, worst-case scenario for fracture).

$\nu = \nu$ (Poisson's Ratio).

Chapter 2: Numerical Modeling Using The Finite Element Method

Conclusion:

Chapter Two created the technical framework necessary to move from abstract theoretical concepts to practical numerical application. We explored the ABAQUS environment as a powerful "virtual lab" capable of solving highly complex, nonlinear engineering problems that often challenge analytical solutions. Through its specialized solutions ABAQUS/Standard for Implicit Static Analysis and ABAQUS/Explicit for Transient Dynamic Events, the software provides a versatile engine for structural integrity assessment.

The ABAQUS/CAE modular architecture serves as a systematic roadmap, guiding simulation through the rigorous stages of preprocessing, resolution, and post processing. We have learned that the reliability of our results depends largely on the strategic selection of components from the finite element library, where choosing the correct family and numerical formula such as planar strain elements (CPE) or planar stress (CPS) is essential to capture precise stress gradients.

Most importantly, this chapter discusses in detail the specialized techniques required for fracture analysis. By using singular elements and the J-integration method, we overcome the challenge of mathematical stress singularity, allowing us to transform the "theoretical infinity" at the crack tip into measurable and predictable data. This technical basis now enables us to move to Chapter 3, where these methodologies will be applied to evaluate the stress intensity factor in a physical case study involving a stainless steel plate

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

3.1) Modeling:

To understand how our material behaves under stress, the first step was to build a reliable digital simulation using Abaqus. We started by creating a simple 2D rectangular panel, **75mm** long and **25mm** wide, we inserted a small **1 mm** crack right in the middle of the plate.

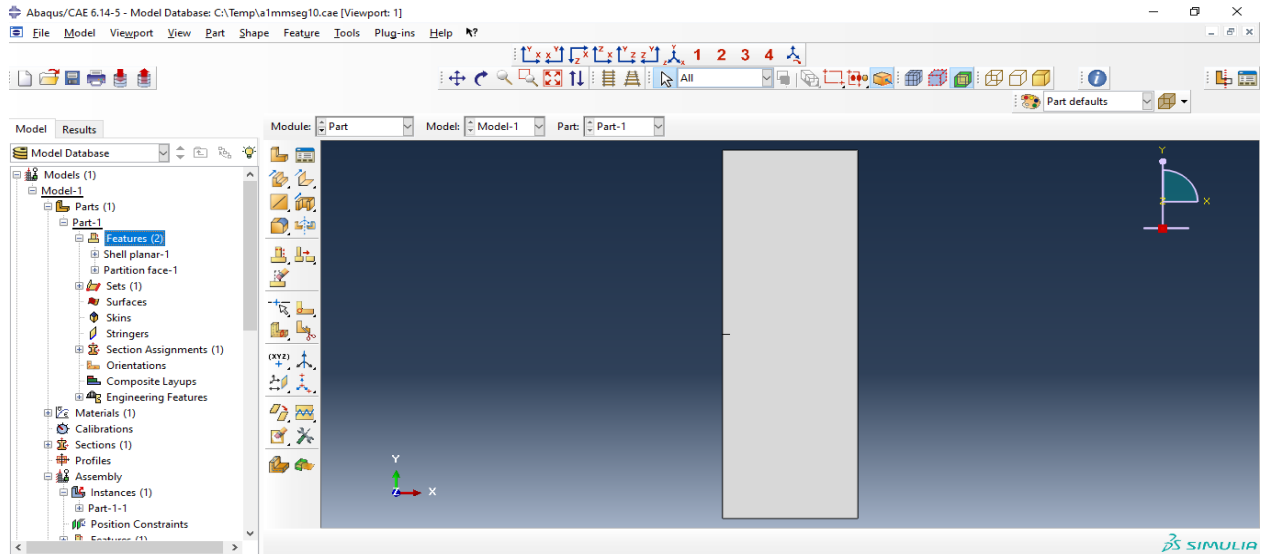


Figure 3.1: plate model

To give this numerical geometry the physical properties of a material in the real world, we defined it as a solid homogeneous section made of stainless steel. We used the standard linear elastic properties of this metal: Young's modulus (**E**) of **193 MPa** to determine its hardness, and Poisson's ratio (**ν**) of **0.29** to determine how it deforms.

Table 3.1: stainless steel properties

Variable	Symbol	SI Unit
Density	P	8000kg/m ³
Young's modulus	E	193*10 ³ MPa
Poisson's Ratio	Y	0.29
Yield strength	Σ_y	215 MPa
Fracture Toughness	K _{Ic}	4743 MPa $\sqrt{mm}$ 150a $\sqrt{m}$

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

3.1.1) Load, Boundary conditions and Mesh:

With material preparation and geometry, we needed to simulate real-world physical conditions. First, we locked the entire bottom edge of the panel in place. By completely stabilizing its motion and rotation (setting U1, U2, and UR3 to zero), the model base acts as if it were securely attached to a solid surface. Next, I applied force by adding a pressure of 10 MPa to the model.

We used quadratic elements with a global size of 0.5 and local size of 0.01 near the creak tip, the global size of 0.5 mm was sufficient because stress levels in those areas are relatively stable and do not change rapidly. However, near the crack tip, the pressure increases incredibly fast over a small distance due to the pressure singularity.

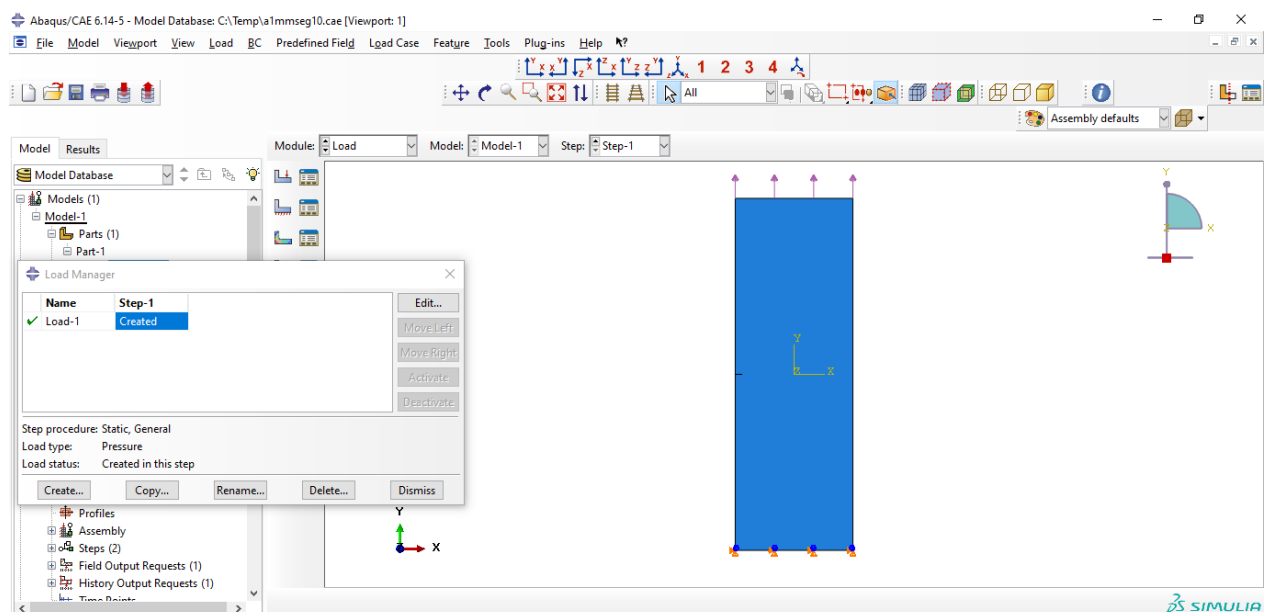


Figure 3.2: Simulation loading parameters

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

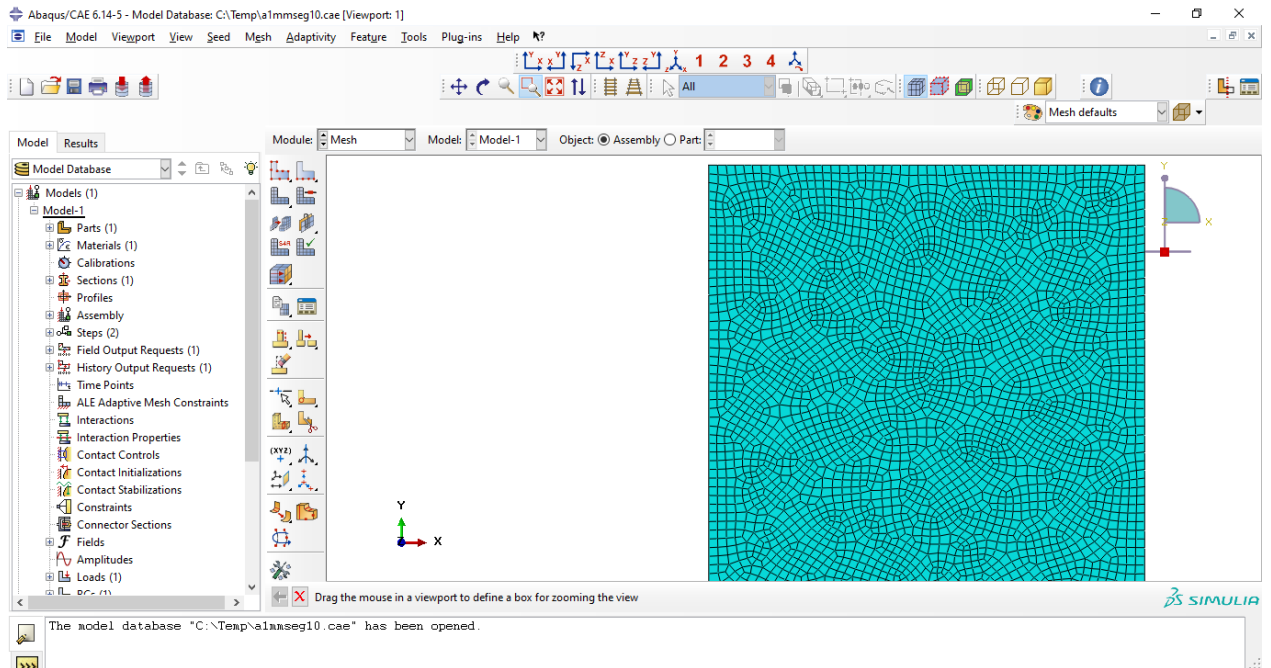


Figure 3.3: global size plate mesh

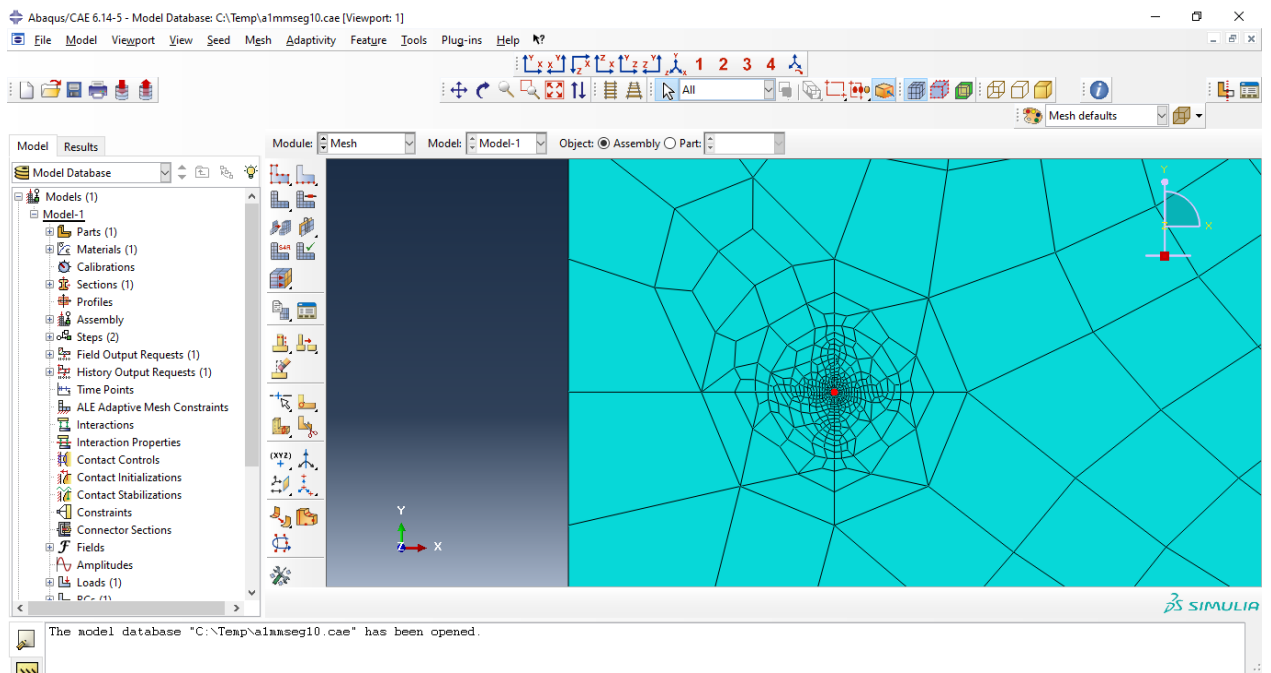


Figure 3.4: local size mesh around the crack tip

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

3.2) Results:

After running the simulation, the software accurately calculated how to focus the pressure around the crack. The most important number that comes out of this test is the stress intensity factor (KI), which measures the intensity of stress at the tip of the crack and helps us predict whether the material will fail. For this specific setting stainless steel plate, 1 mm notch, and 10 MPa drag force the simulation resulted in a KI value of 19.98 and for KII value of 0.1808

The calculated KII values are insignificant compared to KI because the tensile load of 10 MPa acts purely at a right angle to the crack. Since mode I (opening) is the absolute dominant failure mechanism here, KII is excluded from further analysis to focus strictly on the KI results.

3.2.1) Stress intensity factor (Theory vs. Simulation):

To prove that Abaqus's model is accurate and reliable, it is necessary to validate computer results against theoretical mathematics. You simply cannot blindly trust the software, the numerical results must conform to established physics. To do this, we manually calculated the theoretical stress intensity factor for a centrally cracked plate under tension using standard fracture mechanics formulas. Our theoretical manual calculations yielded a value of:

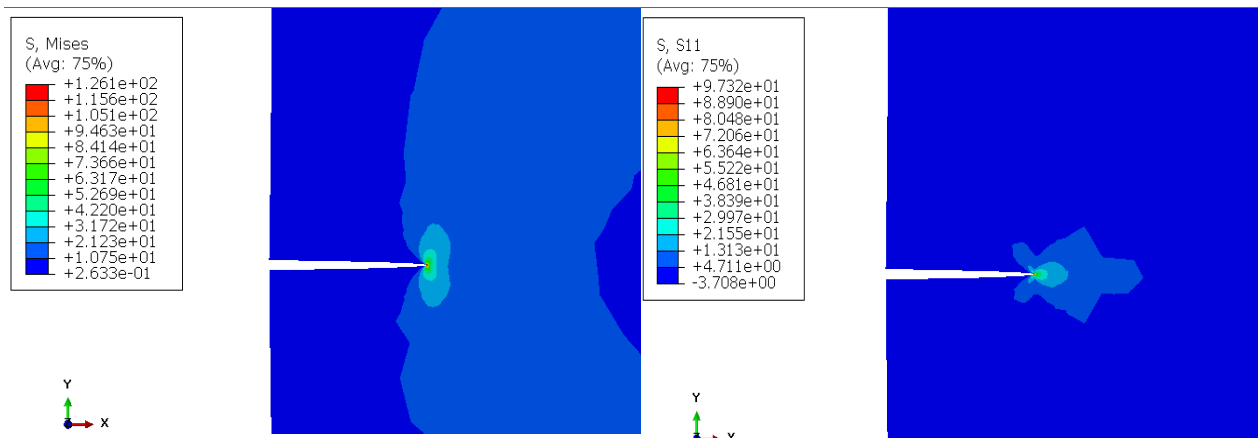
$$KI = Y\sqrt{\pi a}$$

$$KI = 1.2 * 10\sqrt{\pi} = 19.84$$

When comparing this theoretical baseline (19.84) with the numerical result generated by Abaqus (19.98), the values are remarkably close.

3.3) Stress distribution:

Figure 19 represents the equivalent von Mises stress distribution and normal stresses within the stainless steel plate. Stress field analysis clearly indicates that the stresses immediately take their highest level at the crack tip confirming the uniqueness of the expected stress and decrease to a low and stable level in the areas far from the crack under an applied tensile load of 10 MPa.



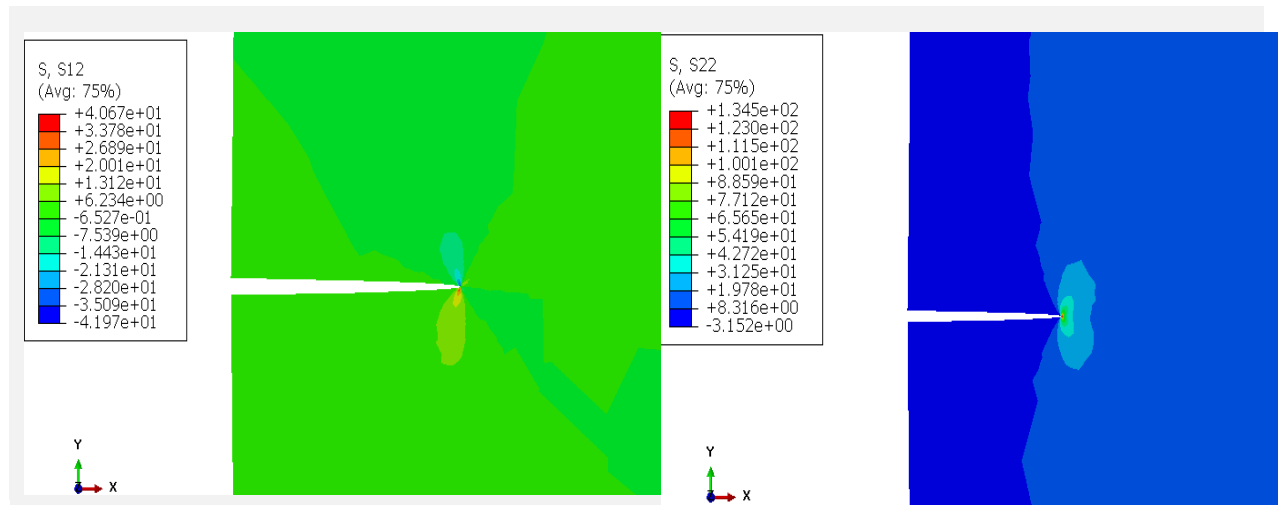


Figure 3.5 : stress distribution for mechanical loading $\sigma = 10\text{MPa}$.

3.4) Crack variation Effect:

After creating a basic model with a 1 mm initial incision, the investigation was expanded to evaluate the evolution of the stress intensity factor (SIF) during incision propagation. In industrial applications, structural defects may spread undetected over large operational periods. As a result, a comparative analysis was performed to determine the severity of crack lengths of 2 mm and 3 mm relative to the initial reference state of 1 mm. To do this, we kept the material properties and 10 MPa load exactly the same, but modified the plate geometry in Abaqus:

For the 2mm crack: we modified the sections and mesh to accommodate the longer crack. The simulation yielded a stress intensity factor K_I of $30.33 \text{ MPa}\sqrt{\text{mm}}$

For the 3mm crack: we increased the length further, again ensuring that the mesh remains polished at the tip. This time, the value of K_I reached $37.03 \text{ MPa}\sqrt{\text{mm}}$

3.4.1) Comparing numerical results versus analytical results:

To ensure the accuracy of the study in all scenarios, we performed a verification check for each fault length. We compared the stress intensity factor K_I generated by Abaqus (numerical) with the values we calculated using theoretical (analytical) LEFM formulas.

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

Table 3.2: numerical crack length results VS analytical results

Crack length (a)	Abaqus Results(KI)	Hand calculations(KI)
1mm (baseline)	19.98	19.84
2mm	30.33	28.07
3mm	37.03	34.38

3.4.2) Graph KI/ K_{Ic}:

Table 3.3: KI/ K_{Ic}:

Crack length	KI/ K _{Ic}
1	0.004213
2	0.006395
3	0.007807

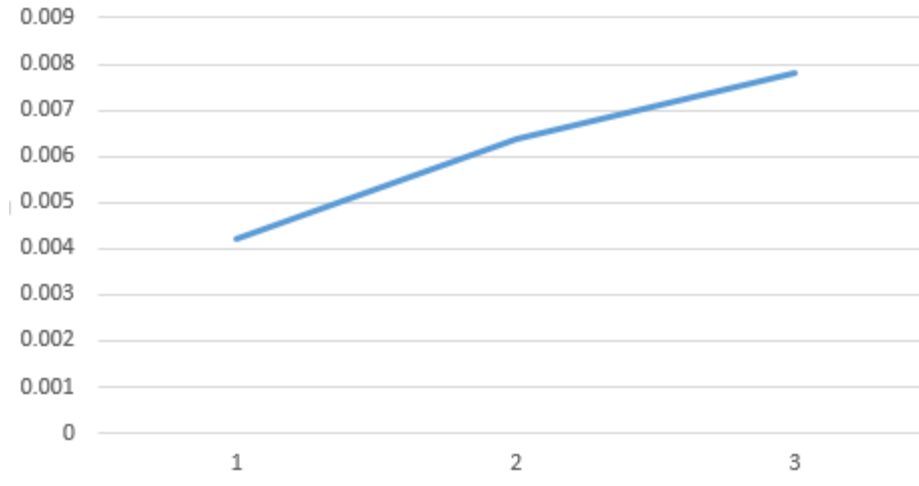


Figure 3.6: graph KI/ K_{Ic}

The ratio of the stress intensity factor **KI** to the fracture toughness **K_{Ic}** of stainless steel below 1 indicates that the crack is **stable** and will not propagate under the applied load of 10 MPa. As shown in the graph.

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

The value of 1 represents the theoretical limit of the resistance of the material. If the $K_I / K_{Ic} > 1$, At this point, the crack is no longer stable, it will undergo unstable propagation, which means that it will grow rapidly over the entire plate without any increase in the applied load.

3.5) Load variation Effect:

To better reflect the instability of industrial environments, the study was expanded to evaluate the effect of load change. Exceeding the initial reference of 10 MPa allows measuring the relationship between increased external force and crack tip condensation. By testing these differences against baseline, we can determine how sensitive a 1 mm diameter crack is to sudden pressure spikes common in service conditions.

To do this, we kept the material properties and geometry of the 1mm crack exactly the same, but adjusted the pressure applied in Abaqus:

For 50 MPa load: The simulation yielded a stress intensity factor K_I of **99.92 MPa $\sqrt{mm}$**

For 100 MPa load: The simulation yielded a stress intensity factor K_I of **199.8 MPa $\sqrt{mm}$**

3.5.1) Comparing numerical results versus analytical results:

To ensure the accuracy of the study in all scenarios, we performed a verification check for each fault load. We compared the stress intensity factor K_I generated by Abaqus (numerical) with the values we calculated using theoretical (analytical) LEFM formulas.

Table 3.4: numerical load results versus analytical results

load	Abaqus Results(K_I)	Hand calculations(K_I)
10	19.98	19.84
50	99.92	99.25
100	199.8	198.51

3.5.2) Graph K_I / K_{Ic} :

Table 3.5: K_I / K_{Ic}

load	K_I / K_{Ic}
10	0.0042
50	0.021
100	0.0421

Chapter 3: Calculation of the Stress Intensity Factor using Abaqus

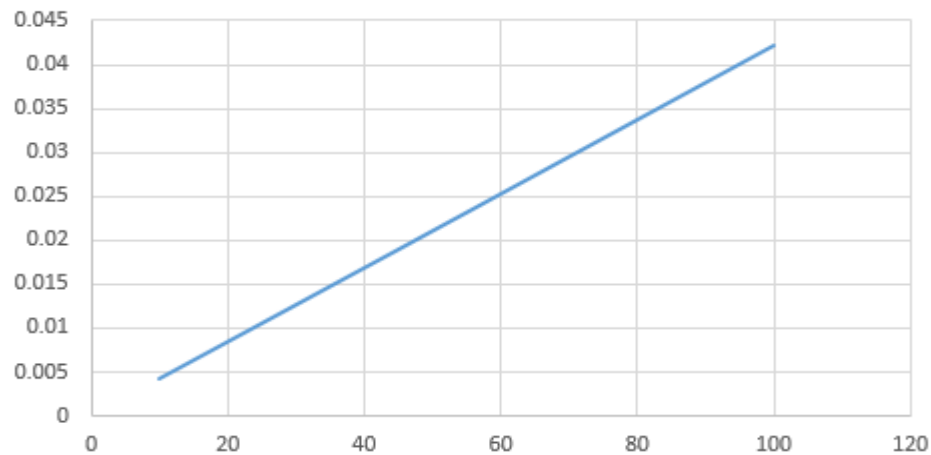


Figure 3.7: KI/ KIc load

The ratio rises, indicating that the plate is approaching its critical limit. While the 10 MPa and 50 MPa cases show a high margin of safety, the 100 MPa case represents the most significant risk as the stress intensity at the 1 mm crack tip is closest to reaching the material failure threshold.

General conclusion:

Engineering is not about making things that last forever or finding perfect, indestructible materials. Instead, it is the art of solving problems within constraints finding the "perfect point" where the design is safe enough to work, but effective enough to be practical. This thesis focused on this balance. By studying how cracks grow and how loads vary, we moved away from "ideal" theoretical models and closer to the true behavior of stainless steel components. Using computational simulation, we explored the exact point at which a controllable defect becomes a serious danger.

The structural integrity is a dynamic relationship between geometry and strength. By starting with a baseline of a 1 mm crack under a load of 10 MPa, we created a reference for normal operation. When the crack extended to 2 mm and 3 mm, and the loads increased to 50 MPa and 100 MPa, the stress intensity factor not only grew but escalated. We found that the interaction between increased cleft and sudden load surge is much more serious than any factor alone. A component that appears stable under normal conditions can reach a breaking point almost immediately when these variables fluctuate.

Beyond digital data, the most important thing to learn is the power of predictability. In modern engineering, we cannot avoid all defects, but we can manage them. Using elastic linear fracture mechanics (LEFM) and software like Abaqus, we can transform invisible pressure into clear, actionable data. This research has shown that maintaining structural integrity requires us to treat the physical defect and external load as a single connected system.

In conclusion, this work bridges the gap between academic theory and practical application. The results provide a framework for better damage tolerance and more accurate monitoring of structural health. As engineers, our goal is to improve the world around us. By accurately modeling, we can design structures that are not only efficient and cost-effective, but more importantly, reliable under unexpected stresses, we enable engineers to look beyond "average" stresses and move toward a future of predictive structural integrity, ensuring that tomorrow's components are safer, stronger, and profoundly more reliable.

References

References:

- [1] Han, Sungkon, Fatigue & Fracture Mechanics Analysis of Components Containing Residual Stress Fields. Released in 2003
- [2] Griffith A.A., 1920. ; Phil. Trans. Roy. So. London; A 221, pp 163-198 The phenomena of rupture and flow in solids
- [3] Mansoura Engineering Journal, vol. 49, article 3.
- [4] Mécanique de la rupture (Course presentation).
- [5] Linear Elastic Fracture Mechanics: Understanding Crack Behavior in Materials," Journal of Powder Metallurgy & Mining. In March 3, 2025
- [6] David Roylance of MIT. LibreTexts, Introduction to Fracture Mechanics. March 28, 2025
- [7] A. A. Griffith, Communicated by G. I. Taylor. "The phenomena of rupture and flow in solids," 1921, vol. 221.
- [8] D. Broek, Elementary Engineering Fracture Mechanics. Springer, 1986.
- [9] T. L. Anderson, Fracture Mechanics: Fundamentals and Applications. March 3, 2017
- [10] N. E. Dowling, Mechanical Behavior of Materials. CVM, Essai de fatigue.
- [11] A. Bouleftour and K. Meftah, Evaluation d'éléments finis volumiques dans Abaqus sur des cas tests non linéaires géométriques.
- [12] Dassault Systèmes, Abaqus Analysis User's Manual, vol. 1.
- [13] Tarek Maifi (2018) ; Chap 3 - Logiciel de simulation, Université Badji Mokhtar Annaba. Évolution des paramètres de rupture en fonction de la charge et la taille de fissure en flexion 3 et 4 points, Université Badji Mokhtar Annaba.

References

- [14] ABAQUS for Engineers.
- [15] M. Boumenkar, Évolution des paramètres de rupture en fonction de la charge et la taille de fissure en flexion 3 et 4 points.
- [16] Dassault Systèmes Simulia Corp, "What is the model tree?" in Abaqus 2017 online documentation.
- [17] Dassault Systèmes, Getting Started with Abaqus: Interactive Edition.
- [18] Dassault Systèmes, Abaqus Analysis User's Guide.

الجمهورية الجزائرية الديمقراطية الشعبية
وزارة التعليم العالي والبحث العلمي

Université de Ghardaia
Faculté des Sciences
et de la technologie



جامعة غرداية
كلية العلوم والتكنولوجيا
قسم الآلية والكهروميكانيك

غرداية في: 2026/07/15

شعبة/ الكهروميكانيك
مخصص: الهندسة الميكانيكية

شهادة ترخيص بالتصحيح والاياداع:

انا الاستاذ : بلعور عبد الرحمان
بصفتي المشرف المسؤول عن تصحيح مذكرة تخرج (ليسانس/ ماستر/ دكتورا) المعنونة بـ

Analysis of stress intensity factor in a cracked structure

من انجاز الطالب :

عبد الرحمان رسيوي

التي نوقشت بتاريخ : 2026/06/14

اشهد ان الطالب/الطالبة قد قام /قاموا بالتعديلات والتصحيحات المطلوبة من طرف لجنة المناقشة وقد تم التحقق
ذلك من طرفنا
وقد استوفت جميع الشروط المطلوبة.

مصادقة رئيس القسم

امضاء المسؤول عن التصحيح

