University of Stavanger

FACULTY OF SCIENCE AND TECHNOLOGY

MASTER'S THESIS

Study programme/specialisation: Engineering, construction and materials	Spring semester, 2021 Open
Author: Espen S. Næss	
Programme coordinator: Knut Erik Giljarhus Supervisor(s): Ove Mikkelsen Co-supervisor: Mostafa Atteya	
Title of master's thesis: Crack propagation in steel plates using FEM and XFEM	
Credits: 10	
Keywords: FEM XFEM Stress intensity factor Crack propagation	Number of pages: 31 Stavanger, 15.06/2021 date/year

Summary

Fracture mechanics is a very useful tool to analyse components with cracks. One useful parameter used in fracture mechanics is the stress intensity factor. This factor can be obtained in different ways, either using fracture mechanics theory or using some numerical method.

Since the technological advances of computers, methods like finite element method and extended finite element method have been more and more popularized. These methods are numerical and are used from everything between fluid dynamics to fatigue analysis. In this thesis the applicability and accuracy will be looked at regarding fracture mechanics.

FEM and XFEM are both useful tools for estimating the stress intensity factor but have some deviation from the empirical formula. Crack propagation for FEM is a bit tedious but accurate, while for XFEM is much more practical but difficult.

Preface

I would first like to thank my main supervisor Ove Mikkelsen and co-advisor Mostafa Atteya for helping me with my master thesis. I would also like to thank Jørgen Løining for helping me with all technical issues. All my friends and family that has supported me during this time also have my regards.

This project has given me a deeper understanding and greater appreciation for these subjects. I have learned many new things and refreshed some old, and hope to learn even more about this in the future.

Table of contents

1. Introduction

Structures and components that are exposed to fluctuating loads can be found almost everywhere. From bridges that experience traffic and wind loads, a car`s suspension system, materials with temperature changes, ships and oil platforms exposed to wind and wave loads. If these loads reach a certain magnitude and are repeated a certain number of times, the component will experience fracture. This is because when these fluctuating loads are applied to a component, it creates small cracks. This process is called fatigue.

Finite element method (FEM), developed in the 1940`s, is a numerical tool to solve partial differential equation (PDE). It is a widely used tool both in engineering and mathematic. Some application of FEM is analysis of stress fields, fluid flow and heat transfer. The main idea in FEM analysis is to subdivide components (2D or 3D) into smaller parts, these parts are called elements. These elements come in many shapes, such as triangles and rectangles in 2D, brick and tetrahedra in 3D. The same elements may differ from one another, since the elements are constructed of nodes, and there can be a different number of nodes in each element.

Extended finite element method (XFEM), developed in the late 1999`s. It extends the classical FEM approach by enriching the solution space for solution to diff. equation with discontinuous functions [10]. XFEM is very useful for discontinuity, for instance cracks.

1.1 Problem statement and objective

The problem statement is crack propagation in steel plates using FEM and XFEM.

The objectives are:

- Comparison of estimated stress intensity factor for selected steel plate problem using FEM and XFEM
- Comparison in methodology for crack propagation using FEM and XFEM
- Comparison between the numerical analysis FEM and XFEM vs empirical data for the stress intensity factor

2. Theory

2.1 Fracture mechanics

2.1.1 Linear elastic fracture mechanics

Linear elastic fracture mechanics (LEFM) is fracture mechanics based on analysis of cracks in linear elastic materials. It is used to solve many practical engineering problems such as life expectancy and safety in cracked components. one key difference in LEFM and elastic-plastic fracture mechanics is that LEFM assumes sharp cracks, whereas elastic-plastic fracture mechanics have blunted cracks.

Fractures occur when the atomic bond breaks, this happens when the applied stress and work on a component is high enough. The atomic bond strength is dependent on the attractive forces between the atoms. The bond energy is given by (all the following theory is based on [1])

$$E_b = \int_{x_0}^{\infty} P dx \tag{2.1.1.1}$$

Where P is applied tensile force and x_0 is the equilibrium spacing. Thru some simplification and calculation, it can be shown that this formula can be rewritten to

$$\sigma_c = \sqrt{\frac{E\gamma_s}{x_0}} \tag{2.1.1.2}$$

Where σ_c is stress at fracture, γ_s is the length of half the sinus curve and E is the Young`s modulus. These formulas show the theoretical cohesive strength in a brittle material is approximately E/π. This value is way too high and through experiments it is shown that the strength of brittle materials are typically 3 or 4 orders of magnitude below.

To fix this problem, scientists came up with a new concept, called the stress intensity factor. The stress intensity factor K is a concept for characterizing the stress field near a crack tip. This is a theoretical construct often applied to homogeneous, linear elastic materials to provide a failure criterion in brittle materials.

There are 3 different modes for types of modes for cracks. Mode I, II and III. Mode I is opening of the crack, mode II is in-plane shear and mode III is out-of-plane shear, see figure 1

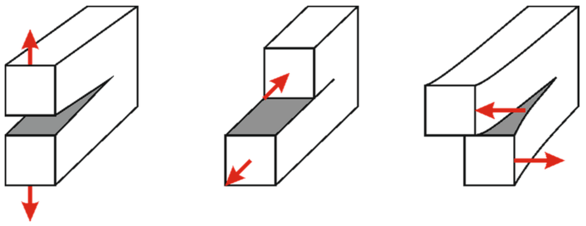

Figure 1: illustrates the different modes of cracks [1]

These modes have their specific stress intensity factor, K_I, K_{II} and K_{III} respectively. Then the stress field ahead of a crack tip in an isotropic linear elastic material is given as

$$\lim_{r\to 0} \sigma_{ij}^{(I)} = \frac{K_I}{\sqrt{2\pi r}} f_{ij}^{(I)}(\theta) \tag{2.1.1.3}$$

$$\lim_{r\to 0} \sigma_{ij}^{(II)} = \frac{K_I}{\sqrt{2\pi r}} f_{ij}^{(II)}(\theta) \tag{2.1.1.4}$$

$$\lim_{r\to 0} \sigma_{ij}^{(III)} = \frac{K_I}{\sqrt{2\pi r}} f_{ij}^{(III)}(\theta) \tag{2.1.1.5}$$

for mode I, II and III respectively. The stress components are additive, so the total stress in a mixed mode problem is given as

$$\sigma_{ij}^{(total)} = \sigma_{ij}^{(I)} + \sigma_{ij}^{(II)} + \sigma_{ij}^{(III)} \tag{2.1.1.6}$$

2.1.2 Fatigue crack propagation

Most failures in structures are due to fatigue crack propagation. In the early 1960`s theories were developed in this area and found that fracture mechanics is a useful tool for characterizing crack growth by fatigue (all of the following theory is based on [1]). However, there are some uncertainties remaining. Large-scale plasticity, variable-amplitude loading, and short cracks are not yet fully understood.

The theoretical basis for fracture mechanics comes from the concept of similitude. Similitude, when it applies, implies that crack tip conditions are uniquely defined by a single parameter such as the stress intensity factor. This means that two systems with a stationary crack will fail at the same critical K-value, if there exists an elastic singularity near the crack tip.

Consider a growing crack with constant amplitude loading. This will form a cyclic plastic zone at the crack tip and leave behind a plastic wake, see figure 2. If the plastic zone is small enough, and embedded in the elastic singularity, then the conditions at the crack tip are uniquely defined by the current K-value.

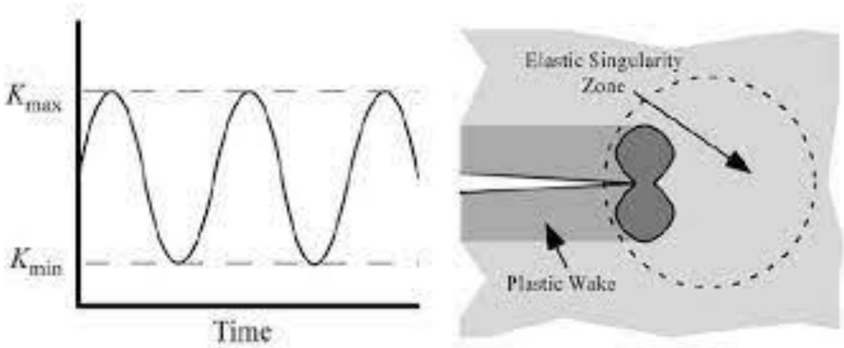

Figure 2: The graph on the left is the constant amplitude loading, and the picture on the right is an illustration of the crack tip with an elastic singularity and plastic wake [1]

$$\frac{da}{dN} = f_1(\Delta K, R) \qquad (2.1.2.1)$$

Where ΔK=(K_{max} - K_{min}), the stress ratio R=K_{min}/K_{max} and da/dN is the crack growth per cycle. If there is any variation in K_{max} or K_{min} during the cyclic loading, the crack growth may depend on the history as well as the current values of K_{max} and K_{min}.

$$\frac{da}{dN} = f_2(\Delta k, R, \mathcal{H}) \qquad (2.1.2.2)$$

Where $\mathcal{H}$ indicates the history dependence that comes from previous plastic deformation. Note that the similitude concept doesn't work here unless the two cases have the same prior history as well.

These formulas above are based on theory and understanding of fatigue crack growth and can be a bit difficult to use. There are some empirical formulas developed that are much more convenient to use. In figure 3 is a log-log plot of a da/dN versus ΔK, which describes a typical fatigue crack growth behaviour in metals.

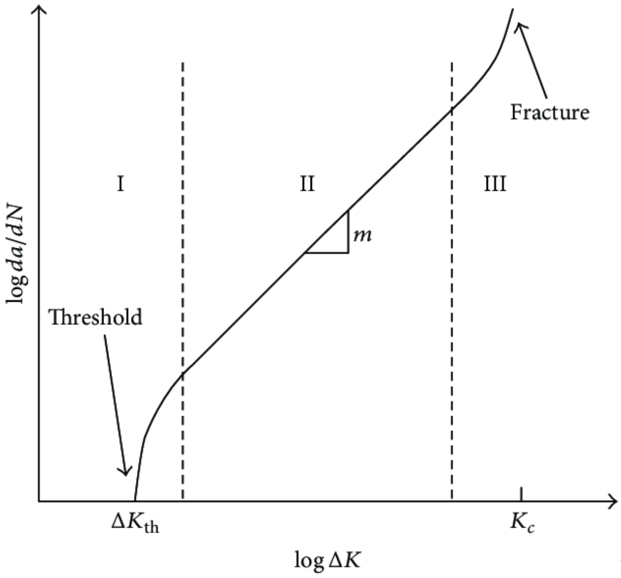

Figure 3: this is an illustration of fatigue crack growth, note there are 3 distinct regions [1]

In figure 3 there are 3 distinct regions, regions I, II and III. Region I describes the start of the crack growth, note that if ΔK is equal or less than ΔK_{th} there is no crack growth. This is due to something called crack closure. Region III describes when fracture will occur, this happens when ΔK is equal or higher than K_c. There are two promising hypotheses for why this happens. One is that "microscopic fracture events (e.g., pop-ins) contribute to crack growth". The other is that the acceleration in da/dN is not real but is due to the influence of crack tip plasticity. Region II, where the focus is, has a linear growth. This can be described by a power law. Paris and Erdogan, were the first to suggest this and came up with the following formula

$$\frac{da}{dN} = C\Delta K_{eff}^{m} \qquad (2.1.2.3)$$

Where C and m are material constants that are experimentally determined. This is probably the most important formula for fatigue crack growth analysis.

2.1.3 Crack propagation direction

Crack propagation happens when a cyclic load is present and reaches a certain threshold. For completely symmetrical cases, it is expected that the crack propagation is a straight line. However, most cases aren't completely symmetrical, this comes from various factors such as load, geometry, crack angle and flaw in material.

For homogeneous, isotropic elastic materials there are three different criteria to use when calculating the crack propagation direction. The maximum tangential stress criterion, the maximum energy release rate criterion or K_{II}=0 criterion [4].

2.1.3.1 The maximum tangential stress criterion

The stress field near the crack tip for a homogenous, isotropic linear elastic material is given by [7]

$$\sigma_{\theta\theta} = \frac{1}{\sqrt{2\pi r}} cos\frac{1}{2}\theta \left(K_I cos^2\frac{1}{2}\theta - \frac{3}{2}K_{II} sin\theta\right) \quad (2.1.3.1.1)$$

$$\tau_{r\theta} = \frac{1}{2\sqrt{2\pi r}} cos\frac{1}{2}\theta [K_I sin\theta + K_{II}(3cos - 1)] \quad (2.1.3.1.2)$$

the direction of the propagating crack can be obtained by using the following criterion

$$\frac{\partial \sigma_{\theta\theta}}{\partial\theta} = 0 \; and \; \frac{\partial^2 \sigma_{\theta\theta}}{\partial\theta^2} < 0 \quad (2.1.3.1.3)$$

And then end up with

$$\hat{\theta} = cos^{-1}\frac{3K_{II}^2 + \sqrt{K_I^4 + 8K_I^2K_{II}^2}}{K_I^2 + 9K_{II}^2} \quad (2.1.3.1.4)$$

" criterion states that a crack grows when the maximum average tangential stress in the fracture process zone ahead of the crack tip reaches its critical value and the crack growth direction coincides with the direction of the maximum average tangential stress along a constant radius around the crack tip." [5]

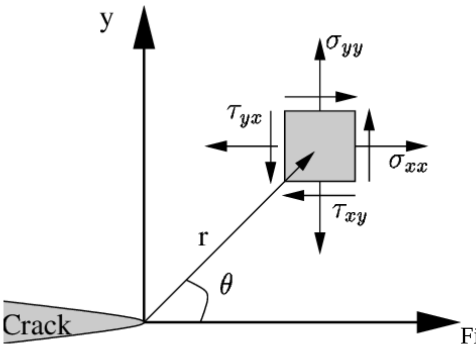

Figure 4: illustrates the forces acting on an element. Note, these are in polar coordinates [1]

2.1.3.2 The maximum energy release rate criterion

The main idea behind this approach is to use Griffiths energy release rate concept. This is done by determining the energy release rate as a function of crack propagation direction, and then determine the critical direction by maximizing the energy release rate [9]. It starts by analysing a crack with a kink in it. The energy release rate at the tip kink is

$$G_{kink} = \frac{k+1}{8\mu}(K_I^2 + K_{II}^2) \tag{2.1.3.2.1}$$

2.1.3.3 K_{II}=0 criterion

When a crack grows, it can be a 1 mode problem or a mixed mode problem, the K_{II}=0 criterion states that the crack will grow where K_{II}=0, so it becomes a single mode problem.

-

2.2 Finite element method

The main idea in FEM is to subdivide a component into smaller parts called finite elements. When this is done you will get a set of algebraic equations. These simple equations for the elements are then assembled back into a larger system that models the whole problem.

2.2.1 The principle of virtual work

The principle of virtual work can be used to obtain formulas for the element stiffness matrix, load vectors associated with initial strains, body forces, and surface traction (the following theory is based on [2]). "The virtual work approach provides essential formulas without requiring much mathematics".

"A virtual displacement is an imaginary and a very small change in the configuration of a system". For the purpose of the analysis, imagine that the virtual displacement takes place relative to the equilibrium configuration and the displacement is admissible. Neither the loads nor the stresses are changed by the virtual work. "The *principle of virtual work,* also known as the *principle of virtual displacement,* can be stated in the form"

$$\int \{\delta\varepsilon\}^T\{\sigma\}\, dV = \int \{\delta u\}^T\{F\}dV + \int \{\delta u\}^T\{\Phi\}dS \qquad (2.2.1.1)$$

$\{\delta\varepsilon\}$ is the vector of strains produced by the following equation

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} \qquad (2.2.1.2)$$

And $\{\delta u\} = \lfloor \delta u, \delta v, \delta w \rfloor^T$. Note that the symbol δ means the same as d for differential, but δ is used instead due to convention when displacements are virtual. Equation (2.2.1.1) says that for any admissible and quasistatic virtual displacement $\{\delta u\}$ from an equilibrium configuration, " the increment of strain energy stored is equal to the increment work done by body forces {F} in volume V and surface tractions $\{\Phi\}$ on surface S".

Let the displacement {u} be interpolated over an element

$$\{u\} = [N]\{d\} \qquad (2.2.1.3)$$

Where {u}=$\lfloor u, v, w \rfloor^T$ and {d} list the nodal displacement degrees of freedom of an element. Strains are determined from displacement $\{\varepsilon\}$=$[\partial]$\{u\}. Then we get

$$\{\varepsilon\} = [B]\{d\} \qquad (2.2.1.4)$$

Where [B]=$[\partial]$[N]. The matrix [B] is the so-called strain-displacement matrix. From equations (2.2.1.3) and (2.2.1.4) the following can be expressed

$$\{\delta u\}^T = \{\delta d\}^T [N]^T \; and \; \{\delta\varepsilon\}^T = \{\delta d\}^T [B]^T \qquad (2.2.1.5)$$

From all the equation above we can obtain

$$\{\delta d\}^T \left(\begin{array}{c} \int [B]^T [E][B] dV \{d\} - \int [B]^T [E]\{\varepsilon_0\} dV \; + \int [B]^T \{\sigma_0\} dV \\ - \int [N]^T \{F\} dV - \int [N]^T \{\Phi\} dS \end{array} \right) = 0 \qquad (2.2.1.6)$$

Note that the vectors $\{\delta d\}$ and {d} do not appear in any of the integrals, this is because they are not functions of the coordinates, thus they can be treated as constants. The equation above "must be true for *any* admissible virtual displacement $\{\delta d\}$ from the equilibrium configuration". This yields

$$[k]\{d\} = \{r_e\} \qquad (2.2.1.7)$$

Where the element stiffness matrix is

$$[k] = \int [B]^T [E][B] dV \qquad (2.2.1.8)$$

"and the vector of loads applied to structures nodes by elements, due to all sources but element deformation, is "

$$\{r_e\} = \int [N]^T \{F\} dV + \int [N]^T \{\Phi\} dS + \int [B]^T [E]\{\varepsilon_0\} dV - \int [B]^T \{\sigma_0\} dV \qquad (2.2.1.9)$$

Note that $\{r_e\}$ and the element stiffness matrix are determined by the same shape functions, this is due to this equation defining "consistent" nodal loads. In some particular problems, some of the integrals in the equation above may vanish. Even if the integral is there, it may vanish for most elements. For example, "$\{\Phi\}$ is nonzero only for elements surfaces that lie on the boundary of the structure and are loaded by surface traction".

2.2.2 FEM in 2D and 3D

There are few elements that can be formulated using the direct method, as applied to bars and beams (the following theory is based on [2]. For problems where the direct method is not suitable other tools must be applied such as stress-strain relations, strain-displacement relations and energy consideration. All the following theory and formulas are based on

Derived from Hook`s law is the stress-strain relationship σ=E*ε and in matrix form

$$\{\sigma\} = [E]\{\varepsilon\} + \{\sigma_0\} \qquad (2.2.2.1)$$

Note that σ_0 and ε_0 can both be present at the same time and may be caused by temperature change or swelling due to radiation or moisture. This relation applies to one, two and three dimensions. For two dimensions this extends to

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} \\ E_{21} & E_{22} & E_{23} \\ E_{31} & E_{32} & E_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} + \begin{Bmatrix} \sigma_{x0} \\ \sigma_{y0} \\ \tau_{xy0} \end{Bmatrix} \qquad (2.2.2.2)$$

Note that the matrix [E] is symmetric, this means $E_{ij} = E_{ji}$. [E] the material can be both isotropic or anisotropic. "For isotropy and plane stress conditions (σ_z=τ_{yz}=τ_{zx}=0)." [E] can be expressed as

$$[E] = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \qquad (2.2.2.3)$$

Where ν is Poisson`s ratio.

"A displacement field describes how a body deforms as well as how it displaces". Engineering definitions of strains are used. Strain in x-direction uses u and y-direction uses v, but since they are function of the coordinates it must be written as u=u(x,y) and v=v(x,y). by taking the partial derivative of these and let Δx and Δy approach 0, the strain-displacement ratio can be written as

$$\varepsilon_x = \frac{\partial u}{\partial x} \qquad \varepsilon_y = \frac{\partial v}{\partial y} \qquad \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \tag{2.2.2.4}$$

2.3 Extended finite element method

The extended finite element method or XFEM for short is an extension of the conventional FEM and is especially designed for treating discontinuity [4]. The discontinuity is divided into strong and weak discontinuity. "Strong discontinuity are discontinuities in the solution variable of a problem" [4].

"The basic idea is that the conventional finite element basis can be enriched to represent a given function ψ(x) on a given domain Ω in the following form:" (All the following theory and formula is based on [10])

$$\psi(\mathrm{x}) = \sum_I N_I(x)\, \phi(x) \tag{2.3.1}$$

N_I(x) satisfies the partition of unity, that is $\Sigma_I N_I(x)$ =1. This satisfies the reproducing condition of function. In XFEM, by adding the enriched terms based on the standard displacement field the discontinuous displacement field can be described more accurately. The displacement field for FEM can be expressed

$$u^h = \sum_I N_I(x)\, u_I + \psi(x) \tag{2.3.2}$$

Where N_I(x) is the shape function of a standard finite element and u_I is the degree of freedom (DOF) at the finite element node. $\psi(x)$ is an enriched term used to approximate the features of the unknown displacement field. Combining the two formulas above we get.

$$u^h = \sum_I N_I(x)\, u_I + \sum_J N_J(x)\, \phi(x) q_J \tag{2.3.3}$$

“The first term on the right-hand side is a standard finite element approximation, and the second term is an enriched term approximation based upon partition of unity”. The extra term q_J describes the newly added degrees of freedom at the original element node. q_J has no physical meaning and is only used to adjusting the amplitude of the enriched function $\phi(x)$ to get the best possible approximation of the displacement field. Equation (2.2.3) is the basic formulation. “Compared with conventional finite element, the principal difference is that additional DOFs are introduced at the element node”. If the definition of enrichment shape function $\psi_J(x) = N_J(x)\phi(x)$ is introduced, equation (2.2.3) simplifies to

$$u^h = \sum_I N_I(x)u_I + \sum_J \psi_J(x)q_J \tag{2.3.4}$$

It is worth noting that it is necessary to choose different enriched functions for different discontinuous problems in equation (2.2.3). To increase the convergence rate, the established enriched function $\phi(x)$ is required to have some features of the actual displacement solution.

It is also worth noting that the description of XFEM above is similar to p-order refinement from conventional FEM. P-refinement does not increase the number of elements, which is bad when it comes to computational time, but rather improves the features of the shape function to approximate the real solution better. The difference is that in p-refinement the added nodal degree of freedom is normally located in the interior of the element domain, while the nodal degree of freedom in XFEM is located at the original element node.

2.3.1 Level set method

XFEM in contrast to FEM allows discontinuous surfaces across the element, so the element mesh is independent of the fracture surface. Thus, it is necessary to provide a geometrical description of the surface. The level set method in XFEM are often used to describe the interface (all the following theory is based on [10])

The level set method is one of the numerical methods used to describe the different discontinuous such as cracks. It is indicated that the level set method is not necessary to use when running XFEM simulations, but it has its advantages. These advantages are: 1) “the geometrical features of a discontinuity can be depicted completely by the level set function", 2) "the geometrical features of a discontinuity can be depicted completely by the level set function", 3) "it is easily extended to the multidimensional situation".

Level set method introduces a zero level set function f(x(t),t) that are both spatial and time dependent that are used to describe the discontinuity, it is also dependent on the element mesh. "Because of the presence of a time variable, the level set function f is increased one more dimension than the discontinuity".

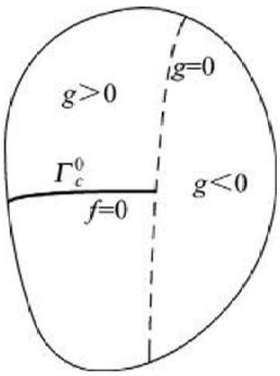

Figure 5:" Level set functions f and g representing a two-dimensional crack Γ_c^0"[10]

The function at the interrupted point at any time during the computation duration will always satisfy

$$f(x(t), t) = 0 \qquad (2.3.1.1)$$

All the points that satisfy this condition form a set called $\gamma(t)$. The signs for the level set function are opposite on both sides of the discontinuity. The location of the crack Γ_c^0 in figure 5 can be expressed by the zero level set function f(x(t),t)=0

One frequently used method for creating the level set function is the signed distance function

$$f(x, t) = \pm min_{x_\gamma \in \gamma(t)} \|x - x_\gamma\| \qquad (2.3.1.2)$$

The physical meaning of this equation is that the signed distance function at any point in the calculation domain is equal to the shortest distance from this point to the discontinuity. The *physical meaning of this equation is that the signed distance function at any point in the calculation domain is equal to the shortest distance from this point to the discontinuity*.

Since only one signed distance function cannot decide the position of the crack tip, another level set function g(x(t),t)=0 has to be introduced. Assume that the moving velocity v_i and position x_i for the crack tip is known, then the new level set function g_i can be expressed as follows

$$g_i = (x - x_i)\frac{v_i}{\|v_i\|} \qquad (2.3.1.3)$$

If $g_i = 0$ this represents a straight line that is passing the crack position x_i and is perpendicular to the velocity v_i. Note that on both sides of the line, the signs of the function g_i are opposite, this is a feature of the level set function.

The position of the crack surface Γ_c^0 in figure 5 can be expressed by the level set function

$$\Gamma_c^0 = \{x \in \Omega_o | f(x) = 0 \; and \; g(x,t) > 0\} \qquad (2.3.1.4)$$

2.3.2 strong discontinuity

Strong discontinuity is related to displacement, in contrast to weak discontinuity, that describes strains and acceleration (all the following theory is based on [10]). Consider figure 6

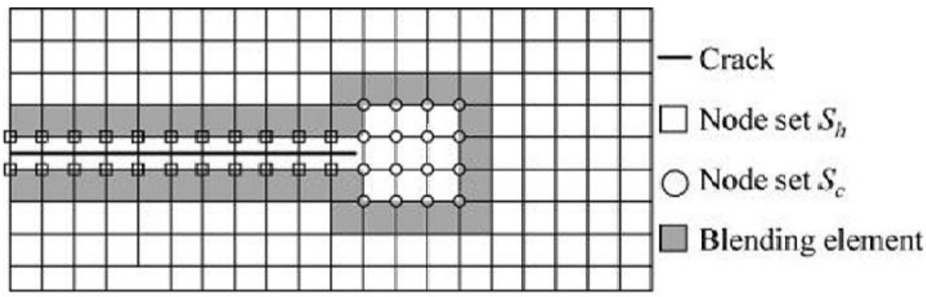

Figure 6: illustrates a crack in 2D with enriched schemes of nodes. [10]

"A crack Γ_c^o passes through the element mesh and terminates in the interior of the element". The set S consists of all the nodes in the element mesh. The node set S_h are the nodes for crack-crossed elements, marked as squares. The node set S_c are the nodes of crack-embedded elements, marked as circular.

The displacement is discontinuous on both sides for crack-crossed elements. The shape function $\psi_J(x)$ may take the following form

$$\psi_J(x) = N_J(x)H\big(f(x)\big) \tag{2.3.1.5}$$

Where H(x) is the Heaviside step function

$$H(x) = \begin{cases} 1 & x \geq 0 \\ -1 & x < 0 \end{cases} \tag{2.3.1.6}$$

And f(x) is the signed distance function

$$f(x) = min_{\bar{x}\epsilon\Gamma_c^0}\|x - \bar{x}\|sign\big(n^+(x - \bar{x})\big) \tag{2.3.1.7}$$

"Where n^+ is the unit normal vector to the discontinuity line Γ_c^0 ". The function f(x) describes the shortest distance for any point x, outside the crack Γ_c^0, to the crack Γ_c^0. Note that the positive and negative distance are dependent on n^+, if x lies on the same side n^+ is pointing, then the distance is positive and negative when x lies on the opposite side.

The node set S_c are describing the crack-embedded elements, and the shape function $\psi_J(x)$ may take the following from

$$\psi_J(x) = N_J(x)\phi(x) \tag{2.3.1.8}$$

"Where $\phi(x)$ could be a linear combination of the function base"

$$\phi(x) = \left[\sqrt{r}sin\frac{\theta}{2}, \sqrt{r}sin\frac{\theta}{2}sin\theta, \sqrt{r}cos\frac{\theta}{2}, \sqrt{r}cos\frac{\theta}{2}sin\theta\right] \tag{2.3.1.9}$$

Note that r and θ are in polar coordinates defined in the crack tip. Note that the enriched function equation above is just terms of the closed form solution in a crack tip displacement field for a plane crack.

Combining the two enriched shape functions, the displacement field of a 2D cracked plate can be expressed as

$$u^h(x) = \sum_{I \in S} N_I(x)u_1 + \sum_{J \in S_h} N_J(x)H\big(f(x)\big)a_J(t) + \sum_{K \in S_c} N_K(x)\phi(x)b_K(t) \qquad (2.3.1.10)$$

Where a_J and b_K are the enriched degrees of freedom at the element node. Note that "the selection of the node set S_c has some flexibility". The convergence speed may increase if one or more elements ahead of the crack tip are chosen. This is due to the extended area being increased.

In figure 6 there are something called blending elements as well, these are marked with grey colour. The blending elements are elements that only have an extended degree of freedom in some of the nodes. This can affect the convergence rate and accuracy. Note that the partition of unity is no longer satisfied.

"For the Heaviside enrichment function, Belytschko et al. (2003b) proposed a modified method, which makes the shape function zero in the blending element"

$$\psi_J(x) = N_J(x)\Big(H\big(f(x)\big) - H\left(f(x_J)\right)\Big) \qquad (2.3.1.11)$$

The extended degrees of freedom by either embedded or crossed crack has no longer any influence on the neighbouring element. This shape function is called shifted enrichment function and is very applicable in fracture problems.

The crack tip enrichment function, there is an idea of a weight function to modify the enriched shape function for blending elements

$$\psi_J(x) = N_J(x)\phi(x)R(x) \qquad (2.3.1.12)$$

"where R(x) is a ramp function that decreases progressively in the blending elements". This method reserves the feature of partition of unity in the blending elements.

3. FE Software

The software used to analyse the plates is Abaqus. Abaqus is a general-purpose software that can also handle XFEM analysis.

3.1 Finite element types

The element used in the 2D analysis for both stationary and growing crack, is the so-called S4R element. The S4R elements has 4 nodes and is a general-purpose shell, with finite membrane shell. Note that when these S4R element are collapsed, like at the crack tip, the calculation of the membrane strains follows the same procedure as if they were not collapsed [3]. This element has reduced integration, this means that the program can calculate the stiffness matrix and then invert it to calculate the displacement directly. In complex finite element models where it uses a high order element, the program needs to use some numerical method to solve for the stiffness matrix.

There exist some variations of the S4R element with different properties as well

One variation is the S4RS element, this was developed to be more computational effective that requires less calculations. "By using one-point quadrature at the center of the element", in other words, it's a S4R element with an extra node at the centre. There are 2 main flaws with this element. 1) "It can perform poorly when warped, and in particular, it does not solve the twisted beam problem correctly". 2) "It does not pass the bending patch test in the thin plate limit" [3].

Another variation of the S4R element is the S4RSW element. This element was developed to deal with the shortcomings of the S4RS element, but to still be computational effective. In the S4RSW element there is an additional term added to the strain-displacement equation to remove the first shortcoming of the S4R element, and to handle the second shorthcoming there is used a shear projection in the calculation of the transverse shear.

The S8R elements is a second order element with twice as many nodes compared to the S4R elements. The S8R element can accurately describe shell behaviour with shear flexible shell theory, this will lead to a smooth displacement field. However, the S8R element require non negligible transverse shear flexibility to function properly. This makes the element suitable when using composite or sandwich shells. Note that the if the S8R element has an irregular mesh, the elements converges very poorly due to severe transverse shear locking [3]. This element type is therefore is recommended to use with regular meshes and thick shell application [3]. Note that this element also has reduced integration.

3.2 Mesh controls

When creating a mesh in Abaqus, and other software's for that matter, there exists an option for how the mesh should be created

Adding sweep as a mesh control tells the software to create the spiderweb mesh. It creates a mesh on one of the sides of the region, this is known as the source. Then Abaqus copies the nodes to the mesh, one layer at the time until the target side is reached (final side). This mesh control is essential at the crack tip in order to get an accurate result.

For the quad area there are 2 main algorithms to be used. Medial axis or Advancing front.

The medial axis mesh control takes the component that is to be meshed and decomposes it to a smaller simpler region. Then the algorithm uses structured meshing techniques to fill in the regions with elements. Note that if the region that is being meshed has a high number of elements and is relatively simple, this technique is faster than advancing front. The minimize mesh transition option, if used, may improve the quality of the mesh [3].

Advancing front starts at the boundary of the region where it generates quadrilateral elements and continues to create quadrilateral elements as it moves systematically to the interior region. This method will always follow seeding more closely than the medial axis method.

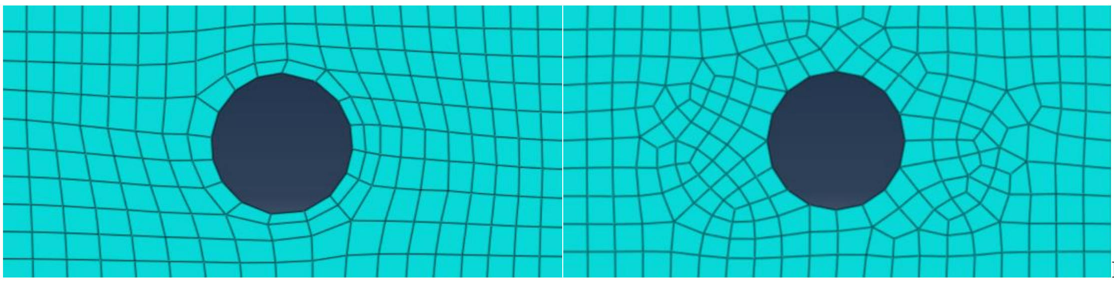

Figure 7: The picture on the left is Medial axis and the picture on the right is Advanced front, both algorithms produces acceptble meshes.

4. Numerical method

This chapter will contain the numerical method used to get the results.

4.1 Finite element method:

The first thing that needs to be considered when modelling with FEM and XFEM is whether 2D or 3D should be used, and the corresponding element that should be used.

The first thing that was compared is the K-value for FEM, XFEM and empirical value. To obtain this value, different method was used and is described under

FEM value: to obtain the K-value for FEM analysis, there are a couple things that need to be figured out such as, what types of element, 2D or 3D and the mesh design. One of the more versatile elements are the so called isoparametric elements. These elements have 4 sides that doesn't need to be a rectangle and are very useful in structural mechanics [3], due to the simple computer program formulation.

Mesh design of FEM analysis: "The design of a finite element mesh is as much an art as it is a science". One of the most important things, if the most important thing when generating a mesh for crack, is the crack tip. At the crack tip exist an elastic singularity $1/\sqrt{r}$, and to capture this is necessary to obtain accurate K-value.

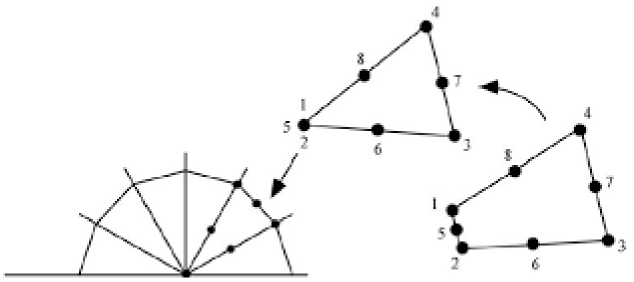

Figure 8: "Degeneration of a quadrilateral element into a triangle at the crack tip "[1]

In figure 8 the nodes "1,2,5" are tied together, and the mid-side nodes are moved 1/4 points. This modification to the element results in a $1/\sqrt{r}$ strain singularity, this enhances the numerical accuracy. This configuration of the mesh is often called spiderweb configuration.

When designing a mesh, it is also important, but not necessary, to be aware of the number of elements that are being used. Take figure 9 as an example.

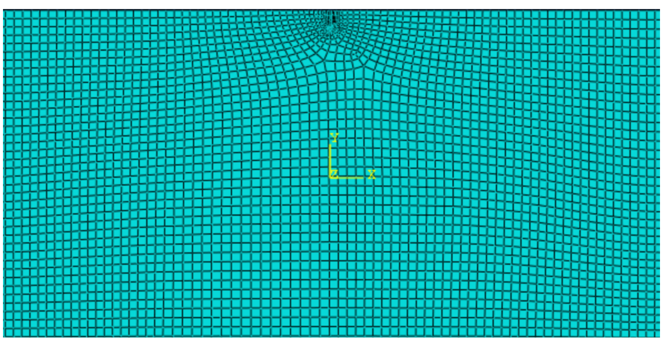

Figure 9: an example of non-optimized mesh

In figure 9 is an illustration of a non-optimized mesh. There are way too many elements on the sides in the figure. At the crack tip is the area of interest, so the mesh should be rather dense and refined at that area, but on the sides, it can be coarser since there isn't that much happening there. Note that this is not necessary to obtain a more accurate K-value, but it will save computational time. It is only saving time when the structures that are being analysed are rather big as well.

In figure 10 is an illustration of a more optimized mesh. In the figure there are fewer elements than in the figure above. If the mesh around the crack tip is focused at the crack tip, should the less elements in this example have little to no effect on the accuracy.

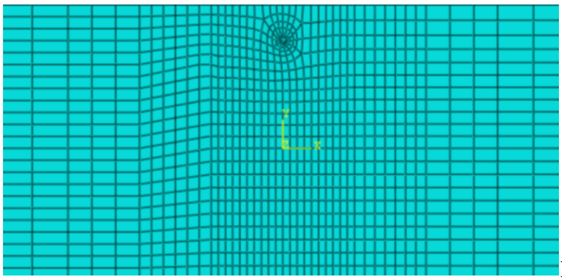
Figure 10: shows a more optimized mesh, the elements far from the crack are bigger.

When analysing growing cracks with FEM, the model often requires a special meshing strategy [1]. The spiderweb mesh is suitable when analysing stationary cracks, but when crack growth is included, the mesh needs re-meshing. The spiderweb mesh created is only suitable for some amount of crack growth before it needs to be updated. This re-meshing technique is appropriate for elastic problems since the strains and stresses are not history dependent. "Fatigue in an elastic body can be modelled by computing the stress intensity range at each time step, and then updating the crack dimensions (with a new mesh) based in the desired growth law".

When calculating the crack growth, the most used formula is the Paris law (2.1.2.3). Note that the Paris law only considers region II. Where C and m are material constants and ΔK_{eff} can be calculated from the following formula. There exists also an extended formula that considers all 3 regions.

$$\frac{da}{dN} = C\Delta K^{m}\left[\frac{\left(1-\frac{\Delta K_{th}}{\Delta K}\right)^{p}}{\left(1-\frac{k_{max}}{k_{Ic}}\right)^{q}}\right] \qquad (4.1.1)$$

Where p and q are material constants determined experimentally, and K_{Ic} is the fracture toughness.

There are two ways to determine the crack growth from Paris law, either use the Paris law or integrate it to get.

$$N = \frac{1}{C\left(\Delta\bar{\sigma}\sqrt{\pi}\right)^{m}} \int_{a_0}^{a_f} \frac{1}{Y(a)a^{\frac{m}{2}}} da \qquad (4.1.2)$$

One of the advantages of using this method is that the growth of the crack "a" can be determined. This is very practical since the finite element model will have less and less accuracy when the crack tip is away from the centre of the spiderweb mesh. Y(a) is a geometric depended factor and has this relationship with f(a/W) (see chapter 4.3)

$$Y(a) = f\left(\frac{a}{W}\right)\sqrt{\frac{W}{\pi a}} \qquad (4.1.3)$$

When there is a hole in the plate, there will be a mixed mode, for this case it will be mode I +II. The effective stress intensity factor can be calculated by the following formula

$$\Delta K_{eff} = \sqrt[4]{\Delta K_I^4 + 8\Delta K_{II}^4} \qquad (4.1.4)$$

ΔK_{eff} is used in Paris law. The angle which the crack will propagate follows this equation

$$\hat{\theta} = cos^{-1}\frac{3K_{II}^2 + \sqrt{K_I^4 + 8K_I^2K_{II}^2}}{K_I^2 + 9K_{II}^2} \qquad (4.1.5)$$

The increments are set to a value where the X and Y coordinates are calculated to find the new crack tip

4.2 Extended finite element method

The XFEM approach is a bit different when obtaining K-value, and analysing crack growth.

When obtaining the K-value with XFEM there is not necessary to create a spiderweb configuration, this is due to the enriched shape function. However, the mesh should nevertheless be refined around the crack and the crack tip. The accuracy with both FEM and XFEM is dependent on the number of elements around the area of interest. And same with XFEM the mesh can be coarser the longer away from the crack one gets.

When designing the XFEM mesh, the most convenient way to create is, is just to have the same sized square/rectangular elements all over the plate. But to reduce computational time, the mesh is a bit coarser away from the crack tip, and more refined at the crack tip.

When analysing a growing crack with XFEM, there is no need to update the mesh. Note that if the crack is growing it can be smart to create a mesh that takes this into account. So, the mesh is refined at the crack tip and at the path it is expected the crack will grow.

4.3 Calculation by empirical formula

For the most common test specimen, there exists formulas that can calculate the stress intensity factor, for this specimen the formula is [1]

$$f\left(\frac{a}{W}\right) = \frac{\sqrt{2\tan\left(\frac{\pi a}{2W}\right)}}{\cos\left(\frac{\pi a}{2W}\right)}\left[0.752 + 2.02\left(\frac{a}{W}\right) + 0.37\left(1 - \sin\left(\frac{\pi a}{2W}\right)\right)^3\right] \qquad (4.3.1)$$

Where W is the height of the plate, “a” is the crack length and f(a/w) is a function dependent on the ratio a/W. This expression is then used to calculate the stress intensity factor

$$K_I = \frac{P}{B\sqrt{W}} f\left(\frac{a}{W}\right) \qquad (4.3.2)$$

Where B is the thickness of the specimen and P is the applied load.

5. Results

5.1 FEM and XFEM for stationary cracks 2D

The figure under is the plate that has been analysed. The orange symbols in the middle and the sides are the supports. The middle support are fixed in U1 and U2 direction while the sides is fixed in only U2. The arrows on sides represent the line load. The plate is 80mmx40mm with a thickness of 1mm. The load is a tensile load of 100MPa on each side, and the crack is 2mm long.

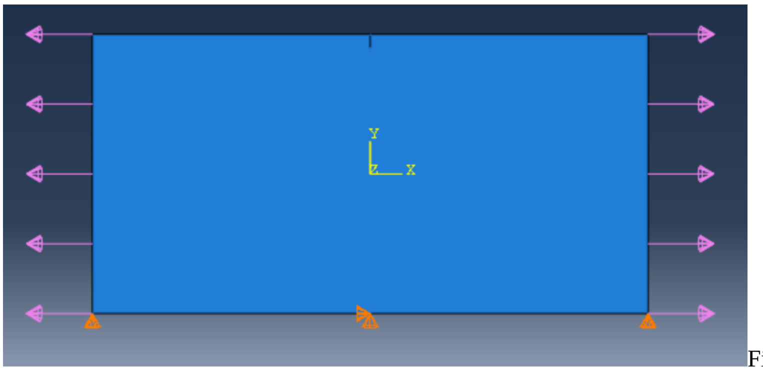

Figure 11: illustrates the plate with tensile forces, note that at the crack tip are some contour rings. There is only contour rings on the FEM analysis and not XFEM.

In the table under is the FEM stress intensity factor values, with different element numbers, compared to empirical values. Note for 64 elements, the K-value for crack 1 has increased instead of converged to the real value. This will be discussed in the discussion chapter.

Crack 1	FEM SIF	Empirical SIF	Deviation in %
8 elements	305.9	287.6	6.4%
16 elements	291.0	287.6	1.2%
32 elements	289.7	287.6	0.8%
64 elements	291.4	287.6	1.3%
Crack 2			
8 elements	435.4	423.9	2.7%
16 elements	425.5	423.9	0.4%
32 elements	419.0	423.9	-1.1%
64 elements	414.5	423.9	-2.2%

Table 1. This table looks at the stress intensity factor with FEM for 2 different crack lengths, where everything else is identical.

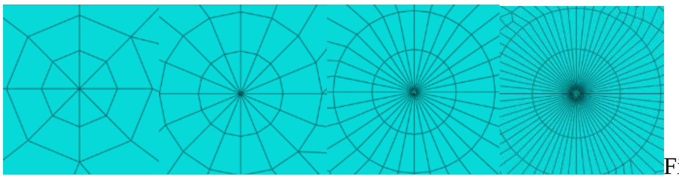
Figure 12: illustration of different number of elements used in the spider mesh

In the table under is the XFEM values, with different element size around the crack, and different number of blending elements. The plate is identical to the one used for FEM testing. The table under compares stress intensity factors for different meshes. The near element size is meant as the element near the crack but the blending elements. The smaller blending element is illustrated in figure 13

	XFEM SIF	Empirical SIF	Deviation in %
Near element size1	255,4	287.6	-11.2 %
Near element size 1, smaller blending elements	268.9	287.6	-6.5%
Near element size 0.75,	271.9	287.6	-5.5 %
Near element size 0.75, smaller blending elements	280.8	287.6	-2.4%

Table 2: looks at the stress intensity factor with XFEM, with some variations in the mesh.

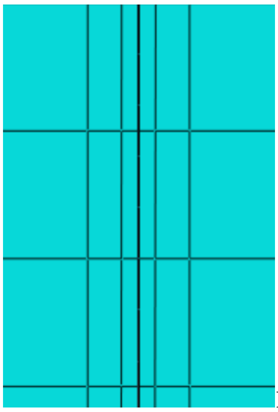
Figure 13: illustrates the mesh for the XFEM case. The thick line in the middle is the crack, the elements on the side is the blending elements and the rest of the elements is the mesh around

In the table under is the most accurate FEM and XFEM values compared to k-value from the formula for this instance.

	FEM	XFEM	Empirical
Most accurate k-value	289.7	280.8	287.6

Table 3: comparison between the best values from FEM and XFEM as well as empirical value.

5.2 Fatigue FEM

The plate that has been analysed is an 80mmx40mm plate with a thickness of 1. The plate has a tensile force on both sides with a magnitude of 110 MPa each. The crack has grown a total of 0.4 mm illustrated in the figure below. The crack has also grown toward the hole in the plate.

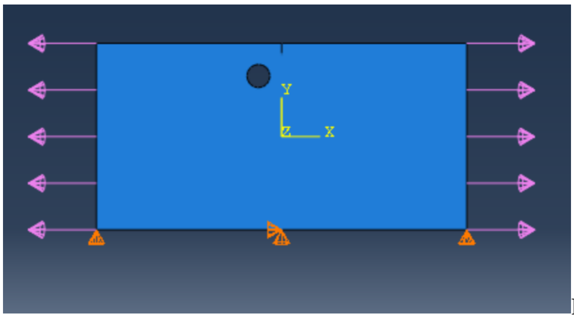

Figure 14: this is an illustration of the plate with a hole in it.

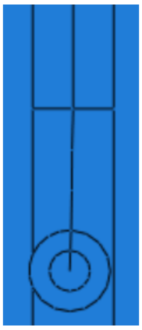
Figure 15: Illustrates the crack growth, the horizontal line is the end of the initial crack, the two parallel vertical lines are where the tangent lines of the first contour ring is (used to illustrate the small crack angle).

In the table below is the data for crack propagation. The numbers 1,2,3 and 4 are the iteration numbers. Number 1 is the crack growth between 2mm and 2.1mm, number 2 is the crack growth between 2.1mm and 2.2mm and so on. X and Y is the total crack growth in the respective directions. N is the number of cycles.

Crack/ iteration	Growth length [mm]	Angle [degrees]	Angle Abaqus	K1	K2	X	Y	N
1	0.1	1.516	2.03	402.18	5,3236	-0.002635	-0.09996	3875
2	0.1	0.369	0.47	422.58	1.3617	-0.003328	-0.19949	3555
3	0.1	1.033	1.33	419.68	3.7874	-0.005102	-0.29999	3244
4	0.1	1.970	3.54	442.02	7.605	-0.008530	-0.39997	2973

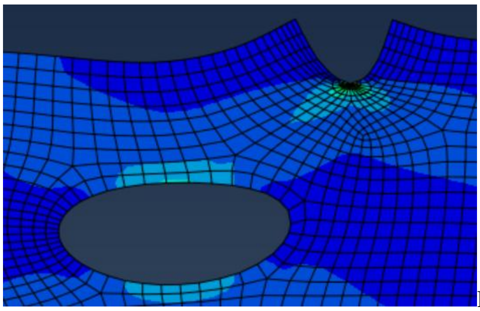
Figure 16: shows the stress field for the plate with a hole in it. Note that the stresses are higher towards the hole.

6. Discussion

One of the first things to take note of is that both XFEM and FEM are numerical methods, and will always have some deviation from analytical solution, but deviation can be negligible if the model is incredibly accurate. In fracture mechanics, there are many formulas based on experiments and not an analytical solution, so these solutions may also vary from the real solution.

The k-value based on the empirical value will have some deviation from the "true" stress intensity factor. This formula is based on the real world with errors. Software may be able to consider these small differences, but it is not done in this analysis.

In the results chapter for FEM and XFEM stationary cracks, the deviations for the 64 elements are greater than for the 32 elements. This is due to element distortion. When the contour rings are created, the selection of the radius is important. Consider a X-number of elements to describe the singularity and 2, different sizes, contour rings. The largest contour ring will have a smaller aspect ratio than the smaller. If this aspect ratio gets too small, the results can be inaccurate.

For "crack 2" in the chapter for stationary cracks the stress intensity factor deviates in another way than for crack 1. There are two possible explanations for this, either the mesh is somewhat different since this crack is longer and further away from the top. This could also be due to the smaller difference in length between the crack tip and the boundaries. Much of the theory in fracture mechanics is based on infinite plates with boundary conditions at the ends.

For the fatigue crack growth there is some error with the number of cycles. When calculating the number of cycles, the geometry correction factor "Y" is inside the integral from the modified Paris law. This is impossible to solve analytically in this case. The integral is solved numerically by Wolfram Alpha.

The geometry correction factor used in Paris law is based on a plate exposed to tensile load, not for the plate used in some of the models that have hole in it. The stress intensity factor calculated from the geometry correction factor (with no hole) is much lower than the stress intensity factor calculated with FEM for a plate with a hole. The latter is used for the calculation of the crack growth.

In the models above, there is used a line load instead of a concentrated load, but according to the empirical formula, there should be a concentrated load. Normal engineering practise says that these things is different sides of the same coin. However, if the line-load rotates due to crack opening, these situations may not be treated as the same, and the effect may be non negligible.

The angles for fatigue crack growth have variations, one that is calculated and one that is taken from Abaqus. These angles differ quite a bit, this is because the angle that is calculated has taken the averages over the 3 first stress intensity factors. If this is better or worse than just using the angle given by Abaqus depends. The angle Abaqus gives is only valid at that point, if one averages the stress intensity factor over some distance it can be more accurate. However, the accuracy of the stress intensity factor decreases the longer one gets from the crack. Note that the angle is the same if only the first k-values are used. The best thing to automate this so the increment can be as small as possible.

7. Conclusion

For the fatigue FEM situation there is an increase in the stress intensity factor, both for K1 and K2. This is due to the presence of the hole. The hole creates a different stress field around the hole that affects the crack. The stresses can't travel thru the hole, so some of them goes on the top side and some on the bottom, this creates more stresses around the hole which creates the higher stress intensity factor.

Finite element method is a more "straightforward" approach, than the XFEM. In FEM one may have to put in more work to get the mesh right, and if fatigue is included, update the mesh, which can be a bit tedious. However, the XFEM approach may seem simpler, but is not. The theory in XFEM is considerably more difficult than FEM theory. In the software, there are a lot more of option to consider. XFEM is nevertheless a much more practical approach for fatigue.

The difference in the stress intensity factor for FEM and empirical is 0.8% and 0.4%. This difference is probably from not dense enough mesh at the crack tip and could be fixed by creating a denser mesh with smaller contour rings. The FE method is a good approach to obtain stress intensity factors. The difference in the stress intensity factor for XFEM and empirical is -2.36%. This difference is rather large but should give a rough estimate of the stress intensity factor.

8. Further work

Create an algorithm that automatically generates the mesh to see how the crack will propagate when the increments are arbitrarily small. See if the stress intensity factor for the XFEM case could be reduced, or if there is something that prohibits it.

9. References

[1] T.L Anderson. (2017). *Fracture mechanics Fundamentals and Applications* (4. edition) 6000 Broken sound parkway: CRC Press

[2] Robert D. Cook, David S. Malkus, Michael E. Plesha and Robert J. Witt. *Concepts and applications of finite element analysis* (4. Edition). University of Wisconsin-Madison: John Wiley & Sons, INC

[3] Dassault systems (2011). Abaqus Theory Manual. Obtained from http://130.149.89.49:2080/v6.11/pdf_books/THEORY.pdf

[4] Prof. Dr. Eleni Chatzi. Introduction to the Extended Finite Element Method. Obtained from https://ethz.ch/content/dam/ethz/special-interest/baug/ibk/structural-mechanics-dam/education/femII/XFEM.pdf

[5] Yu. G. Matvienko (2012). Maximum average tangential stress criterion for prediction of the crack path. https://doi.org/10.1007/s10704-012-9715-1

[6] MIT (2021, 4.April). Contour integral evaluation. Obtained from: https://abaqus-docs.mit.edu/2017/English/SIMACAEANLRefMap/simaanl-c-contintegral.htm

[7] MIT (2021, 1. April). Prediction of the direction of crack propagation. Obtained from https://abaqus-docs.mit.edu/2017/English/SIMACAETHERefMap/simathe-c-crackpropdir.htm

[8] C. T. Sun & Z.-H. Jin (2012) *Fracture mechanics*. https://doi.org/10.1016/C2009-0-63512-1

[9] Chien H. Wu (1978). Maximum-energy-release-rate criterion applied to a tension-compression specimen with crack. Obtained from https://link-springer-com.ezproxy.uis.no/content/pdf/10.1007/BF00130464.pdf

[10] Zhuo Zhuang, Zhanli Liu, Binbin Cheng & Jianhui Liao. (2014). *Extended Finite Element Method* (1. Edition). Tsinghua University Press: Elsevier Inc