Numerical simulation of crack growth in fibre reinforced composites

M.L.C. Morcus
DC no.: 2010.057
Numerical simulation of crack growth in fibre reinforced composites
Master Traineeship at Queen Mary, University of London

DC no.: 2010.057

M.L.C. Morcus
ID: 0638101

London, June 2010

University of Technology Eindhoven
Den Dolech 2
5612 AZ Eindhoven

Supervisor:
Prof. Dr. H. Nijmeijer

Queen Mary, University of London
Mile End Road
London, E1 4NS

Supervisor:
Dr. P. Wen
Preface
This report was completed during my master internship of mechanical engineering at the Eindhoven University of Technology in cooperation with Queen Mary University of London. It represents the work realized in a period of three months.

Background information about fibre reinforced composites and fracture mechanics can respectively be found in Chapter 2 and 3. Readers especially interested in the results of the Finite Element Analysis, are referred to Chapter 5.

The project that I started more than three months ago, would probably never have led to the work I am about to present now, if there had not been these helping hands, that guided me through several difficulties.
I would like to thank Dr. Oluwamayokun Adetoro, without his assistance and expertise the understanding and usages of the finite element package ABAQUS, would not have been so successful. Also I would like to thank Dr. Henri Huijberts for arranging this assignment and taking care of the administrational matters during my stage. Furthermore, I am deeply grateful to Dr. Pihua Wen, for offering me the possibility to be part of this project. Moreover I would like to thank him for the advice and discussions we had in the past months and for the support and encouragement throughout my work. My special thanks goes out to my supervisor of the Eindhoven University of Technology, Prof. Dr. Henk Nijmeijer, for the guidance throughout the project and for his interest in my work.

Martijn Morcus
London, United Kingdom
June 2010
Summary

With the aid of finite element analysis, pre-existing cracks in fibre reinforced composites can be analyzed. To do so, a number of simulations have been done in order to obtain a visualization of how a crack grows inside one of the constituents. By calculating the stresses and displacements around the crack tip, a stress intensity factor (SIF) also called K-value, can be calculated. This factor describes the stress field around the tip and is an important parameter. If the stress intensities for two different shaped cracks in the same material are equal, there is exact similitude in extension. Hence, one should expect the two cracks to respond in the same manner. Because of the similitude argument, any crack in any body of this material will cause fracture at that same value of the stress intensity factor. Therefore it is a practical value to work with in order to predict the crack growth in a composite. If the SIF is known, reliable predictions can be made with regard to crack growth.

However one complexity enters the problem and needs more attention, the interface between the fibre and matrix. In the area around this border, the regular crack tip singularities will not apply anymore and other definitions and formulas have to be taken into account. Consequently the K-value is not consistent any longer. During finite element simulations this problem has been observed. In the results this effect was shown in different graphs. Yet, one cannot conclude with certainty how to interpret the K-values around the interface. Nevertheless, certain theoretical relations are confirmed by the results of the finite element analysis.

Once the K-values are known and trustworthy, regular crack growth relations, like Paris law, can be applied to calculate permitted stresses and the safe-operating lifetime. Still safety factors have to be taken into account, but overdimensioning of designs will be smaller.
### Symbols

- **a** [m] \hspace{1cm} Crack length
- **C** \[ m\text{mMPa}^{-1}(m^{-3/2})^m \] \hspace{1cm} Paris law coefficient, with Paris law exponent \( m \) in the unit (red)
- **E** [Nm\(^{-2}\)] \hspace{1cm} Young’s modulus
- **K** [Nm\(^{-3/2}\)] \hspace{1cm} Stress intensity factor
- **m** [-] \hspace{1cm} Paris law exponent
- **N** [-] \hspace{1cm} Number of cycles
- **r** [m] \hspace{1cm} Distance to crack tip
- **U** [m] \hspace{1cm} Displacement
- **θ** [radians] \hspace{1cm} Angle to crack tip
- **κ** [-] \hspace{1cm} Kolosov’s constant
- **μ** [Nm\(^{-2}\)] \hspace{1cm} Modulus of rigidity
- **ν** [-] \hspace{1cm} Poisson’s ratio
- **σ** [Nm\(^{-2}\)] \hspace{1cm} Stress
- **σ_{ys}** [Nm\(^{-2}\)] \hspace{1cm} Yield stress
- **τ** [Nm\(^{-2}\)] \hspace{1cm} Shear stress
1. Introduction

Fibre reinforced composites find wide application in several branches of engineering, for example civil wind power, aerospace, marine and automobile industry. Given their strength, low weight and high toughness, composites are exceedingly suitable for many different uses. Due to the fact that composites are often homogenous and anisotropic on a macroscopic scale, material properties differ from point to point in the material. These varieties of properties in different directions are however an advantage during the structural design and optimization of products. It creates a large number of degrees of freedom for given constraints simultaneously, such as minimum weight and maximum dynamic stability.

Despite the attractiveness of using composite materials, they are not being used as much as they could be. Often when used, it is mostly in a low stress application or with such large safety factors that they are not fully used to their potential. The basic reason for this is the uncertainty that exists in determining the strength and safe-operating lifetime in service conditions.

The fracture mechanics of composite structural materials comes with a variety of complexities. While composites are also heterogeneous on microscopic scale, crack growth generally does not behave in the relatively simple way in which for example metals usually do. Furthermore, the mechanical properties associated with the failure processes are often affected by various environmental parameters, such as time-dependency, sensitivity to temperature and moisture. Therefore the mathematical predictive models for fibre composites will be challenging.

Components made from fibre reinforced composites may eventually fail because of pre-existing defects such as microcracks, voids and interface debonding. The reliability of such components may be predicted and eventually increased on the basis of analysis done to the effect of pre-existing failures. These analyses can be carried out in the framework of computational experiments. In theory it should be possible to calculate the stress- and displacement field at the crack tip and determine growth graphs. To confirm this, a part of a fibre reinforced composite with a pre-existing crack inside the fibre will be modeled in a FE-package. A cubic formed composite material will be loaded with a static load and the crack length will be varied. Stresses and displacements at the crack tip will be evaluated to determine growth graphs.

In this project the possibility of using stress intensity factors to determine crack growth graphs of fibre reinforced composites is explored. By means of finite element analysis the K-values of different sized cracks are calculated.

Therefore first some backgrounds of composites have been studied. In Chapter 2 some general properties of fibre reinforced composites are reviewed. Chapter 3 goes into the linear elastic fracture mechanics of cracks, used to interpret the results from finite element simulations. Amongst others crack tip stresses and crack growth are discussed.
1. Introduction

The model will be made with the FE program ABAQUS, a description of the model with its parameters and boundary conditions can be found in Chapter 4. The calculation methods and results of the simulations along with the graphs are discussed in Chapter 4 as well. The report will finish with general conclusions and recommendations.
2. Fibre reinforced composites

Composites have unique advantages over monolithic materials, for example such as high strength, high stiffness, long fatigue life and low density. Depending on the purpose of the use of the material, it can be adapted to improve all kinds of properties. However composites also have some limitations that conventional materials do not have.

2.1 Fibre composites

It is commonly accepted that a composite material consists of at least two constituents that are chemically distinct on a macroscopic scale. Ordinarily, one constituent is a discontinuous phase that is bonded to, or embedded in, a continuous phase. The former is termed the reinforcement, the latter the matrix. In case of fibre composites, the reinforcement is a fibre. Together they form the composite that possess properties of both constituents [1].

Fibre materials can be created in various ways, which influences the material behaviour of the composite. Amongst others the fibres can be continuous and aligned or ‘chopped’ and distributed randomly in the matrix, see figure 2-1.

![Classification of composite material systems](image)

Depending on the orientation of the reinforcement, the physical and mechanical properties will be different. Whereas these properties with conventional materials, such as metal, are relatively easy to determine, the properties of fibre reinforced materials are more complex. Since there are two or more materials involved, sometimes orientated in different ways, the absorption of stresses and impacts will not always be straightforward. With numerical models the global properties of the materials can be approached relatively accurately. However when the mechanics become more complex, for example
with fracture mechanics, these numerical models should be handled with care and mostly be evaluated. Therefore it is complicated to apply fracture mechanics on composite structures.

2.2 Fracture mechanics in fibre composites
The simplest way of analyzing a fibre composite is to regard it as a homogeneous continuum and to make no distinction between fibre and matrix. One can consider the material as quasi-homogeneous, where low variability exists from point to point on a macroscopic scale. The material is then treated as a single material with suitably averaged properties of its constituents. For this approach a lot of ‘mixed’ formulas can be applied. In that case the different properties will be joined together to get one overall property. This can be useful for general design qualities.

In the area of fracture mechanics, these ‘mix rules’ are not satisfying anymore, basically because a composite is not homogeneous and not isotropic at every point in the material. Even for homogeneous materials the fracture mechanics has prescribed models which are bounded to specialized conditions. Nevertheless, it can be proven that for anisotropic materials in some cases isotropic based fracture mechanics is useful. Although research in this area is still developing, the fracture mechanics used in conventional materials is often used for composites as well [2].

2.3 Crack propagation in fibre composites
If a premade large cut is absent, it can be assumed that failure in a fibre composite starts from small natural defects in the material. These defects may be broken fibres, flaws in the matrix and/or debonded interfaces. In figure 2-2 several appearances prior to a crack are schematically displayed [1].

![Figure 2-2: Schematic representation of micromechanical failure nearby the crack tip in a fibre composite.][1]

As can be seen in this figure, a crack can start in the fibre, in the matrix or simultaneously in both. Assume the crack will grow until it reaches the interface with the other constituent; the crack can proceed further in several ways. It proceeds
- through the interface into the other constituent;
- along the interface, not affecting the other component;
- on the other side of the constituent and leave the intermediary material intact, this is called fibre/matrix bridging;
- not at this location, but at another location due to stress changes.
2. Fibre reinforced composites

Obviously these options can and will appear in combination at and around a crack tip. The way of crack development is matrix and fibre dependent. For this, a numerical fracture analysis could be used to predict which option is likely to happen. Still, even with reliable numerical models, it remains an approximation of reality and the material behaviour can be different from the theoretical approach.
3. Fracture mechanics

When material properties and associated mechanical variables can be assumed to be continuous functions of spatial coordinates, analysis of mechanical behaviour can be done with continuum mechanics. Unless the material behaviour is observed on a large scale, large enough to average out small scale discontinuities, these principles can be used. However when the discontinuity inside the materials, such as micro cracks and voids, become too large to average out, different approaches of study need to be used. Amongst others these are studied in the field of fracture mechanics.

In short some basics principles of fracture mechanics related to the problem will be described in this chapter. The basics of continuum mechanics are assumed to be background knowledge.

3.1 Modes

A crack in a solid can be stressed in three different modes, as illustrated in figure 3-1. The opening, sliding and the tearing modes are respectively called mode I, II and III. Crack surface displacements are in the plane of the crack and parallel to the leading edge of the crack. Often a combination of these three modes is necessary to describe a crack. However different formulas will apply for these situations. In this report only a pure mode I crack will be discussed, since the investigated model is only loaded with normal stresses [3].

![figure 3-1: Three modes of loading](image)

3.2 Crack tip stresses

Consider a mode I crack of length $2a$ in an infinite plate, as in figure 3-2. The plate is subjected to a tensile stress of $\sigma$ at infinity. An element $dx\,dy$ of the plate at a distance $r$ from the crack tip and at an angle $\theta$ with respect to the crack plane is observed. It experiences normal stresses $\sigma_x$ and $\sigma_y$ in $x$ and $y$ directions and a shear stress, $\tau_{xy}$. Notice that $a$ represents only half the crack length, these stresses can shown to be [3]:

\[
\sigma_x = \sigma \sqrt{\frac{a}{2r}} \cos \left( \frac{\theta}{2} \right) \left[ 1 - \sin \left( \frac{\theta}{2} \right) \sin \left( \frac{3\theta}{2} \right) \right] \\
\sigma_y = \sigma \sqrt{\frac{a}{2r}} \cos \left( \frac{\theta}{2} \right) \left[ 1 + \sin \left( \frac{\theta}{2} \right) \sin \left( \frac{3\theta}{2} \right) \right] \\
\tau_{xy} = \sigma \sqrt{\frac{a}{2r}} \sin \left( \frac{\theta}{2} \right) \cos \left( \frac{\theta}{2} \right) \cos \left( \frac{3\theta}{2} \right) \tag{3-1}
\]
The stresses are proportional to the external stress, $\sigma$, and vary with the square root of half the crack size. They tend to infinity at the crack tip, where $r$ is small. The typical distribution of the stresses $\sigma_y$ as a function of $r$ (at $\theta=0$) is illustrated in figure 3-3.

The equations (3.1) are only valid for a limited area around the crack tip; this is shown in figure 3-3. In this graph it seems that for a large $r$ the stresses approach zero, while it should be $\sigma$. The reason of this disability is the accuracy of the (3-1). Each of the equations represents only the first term of a series. Near the crack tip these first terms give a sufficiently accurate description of the crack tip stress field. The following terms are small compared to the first; further away, more terms will have to be taken into account [3].

### 3.3 Linear elastic fracture mechanics and stress intensity factor

The original form of (3.1) can be generalized and written as, [8]:

$$\sigma_{yj}(r, \theta) = \sum_n e_n r^{\lambda_n} f_n(\theta) = \varepsilon_1 r^{\lambda_1} f_1(\theta) + \varepsilon_2 r^{\lambda_2} f_2(\theta) + ...$$

(3-2)

Where the constants $e_n$, the functions $f_n$ and the exponents $\lambda_n$ have to be found. If the stresses are singular at least one of the $\lambda_n$ is negative, in which case $\sigma_y \to \infty$ as $r \to 0$. If $\lambda_1 < 0$, and assuming that $\lambda_1 < \lambda_2 < ...$, the stresses given by (3-2) are dominated by the first term [8]. After some mathematical manipulation, using an airy stress function and the biharmonic equation, the stress near the crack tip can be stated as:

$$\sigma_{yj} = K_1 (2\pi r)^{-1/2} f_1(\theta)$$

(3-3)

In which $f_1(\theta)$ is a known function of $\theta$ and $K_1 = \sigma \sqrt{\pi a}$. Note that the constant $e_1$ in (3-2) is equal to $\sigma \sqrt{a/2}$.

The factor $K_1$ is known as the stress intensity factor (SIF), where the subscript I stand for mode I. The stress field at the crack tip is known when the stress intensity factor is known. For a crack twice as big as $2a$, but with a load twice as small, the K-value will be the same. Therefore also the stress distribution around the crack tip will be the same. Equation (3-3) is an elastic solution; it seems that the stresses become infinite at the crack tip. In reality this cannot occur, plastic deformation taking place at the crack tip keeps the stresses finite. To estimate the size of this crack tip plastic zone a distance $r_p$ can be
obtained, where the $\sigma_y$ is larger than the yield stress, $\sigma_{ys}$, see figure 3-4 a. Substituting $\sigma_y = \sigma_{ys}$ into (3-3) for $\theta = 0$, it follows that:

$$\sigma_y = \frac{K_I}{\sqrt{2\pi r_p}} = \sigma_{ys}, \text{ so, } r_p^* = \frac{K_I^2}{2\pi \sigma_{ys}^2} = \frac{\sigma_y^2}{2\sigma_{ys}^2}$$

(3-4)

In reality the plastic zone is somewhat larger, see figure 3-4 b. In this project only linear elastic fracture mechanics (LEFM) is used. LEFM is only valid if the applied stresses are small enough that general plastic yielding does not occur. In other words, the crack tip plastic zone should be small compared to the size of the crack. These requirements are also known as ‘small-scale yielding’ conditions. According to (3-4) the size of the plastic zone is proportional to $K_I^2/\sigma_{ys}^2$ [3].

Crack extensions will occur when the stresses and strains at the crack tip reach a critical value. Since $K_I$ is a measure for all stresses and strains in this area, one can pose that fracture must be expected to appear when $K_I$ reaches a critical value, $K_{Ic}$. This $K_{Ic}$ has been determined experimentally for quite a few materials. Provided that several conditions are taken into account, it appears to be a material property. On that basis the fracture strength of cracks of any size in the same material can be predicted. It is however slightly more complicated, since the above stress intensity factor ($K_I$) is only valid for an infinite plate and with this specific crack inside. In other situations, for example with a finite plate or with other shaped cracks, an extra dimensionless factor $\beta$ must be added to (3-3). This factor depends upon geometry and has been calculated for many geometries. In case of an infinite plate, this factor appears to be $\beta = 1$. Herewith the general expression for the stress intensity factor becomes:

$$K_I = \beta \sigma \sqrt{a}$$

For a design it is important to keep the $K$-value below the critical value ($K_{Ic}$). The point that they are equal; the crack will start to grow.
3. Fracture mechanics

3.4 Crack growth

During the development of fracture mechanics, experiments have shown that crack length $a$ can be expressed as an exponential function of the number of cycles, $N$. These numbers of cycles represent the load changes on the material. The crack growth will be very slow until a particular number of cycles ($N_f$) is reached, which result in fast growth, leading to failure. The development from an initial crack $a_i$, to the critical crack length $a_c$, can be seen in figure 3-5.

\[ \frac{da}{dN} = C (\Delta K)^m \]  

(3.5)

The Paris law is widely used and represents a straight line on a double-logarithmic axes, see figure 3-6. The values of $C$ and $m$ are material dependent.

For low and high values of $K$, the Paris law will not describe the crack growth rate accurately. For lower values of $K$, the crack growth is much slower and for higher values, the growth is much faster than predicted by the Paris law, see figure 3-6. In order to describe the crack growth more precisely, several other laws, as an extension of the Paris law, were published, but will not be discussed here [4].

However the Paris law does not completely fulfill the desire to predict cracks from the start to the end, it is a powerful tool to have an indication of how fast a crack will grow. So if the $K$-value of a crack is known, accurate predictions could be done about the crack development.
4. Finite element analysis

A part of a fibre reinforced composite with a pre-existing crack inside the fibre is modeled in the finite element package ABAQUS. By applying a load, the stresses and displacements at the crack tip can be calculated. The stress intensity factor, specific for this crack and geometry, can be determined. The parameters and the results of the finite element analysis are described in this chapter.

4.1 Experimental setup

A cubic unit is generated from composite material; it represents a part of a unidirectional and continuous fibre composite. This cell measures 10x10x10 mm$^3$ and has one fibre in the centre, green in figure 4-1. The cube is subjected to a tensile load at the top and is restricted at the bottom. In the middle of the fibre a pre-existing failure is present, in figure 4-1 the origin of the coordinate system. This crack starts to grow through the fibre into the matrix. It is assumed that this growth is in one plane (ZX-plane) simultaneously in positive and negative x-direction.

For the sake of simplicity the model is reduced to a 2D cross-cut through the middle of the cube. From this section only a quarter of the plane is simulated due to symmetry, indicated in red in figure 4-1.

![Figure 4-1: 3D-model, cell of fibre reinforced composite](image)

4.2 2D-Model properties

The dimensions of the quarter plane are 5x5 mm$^2$, with the fibre at the right side of the plane. With a volume content of 7%, the diameter of the fibre in the 3D-model is 3mm. The cube is fixed at the bottom and loaded with a static tensile force divided equally over
4. Finite element analysis results

the upper plane. This results in the following application of boundary conditions for the 2D-model, visualized in figure 4-2. The cross-cut is subjected to a tensile load of 10 N/mm at the top edge, in y-axis direction (purple). At the bottom edge of the plane, with the exception of the crack opening, the 2nd degree of freedom is fixed (displacement in y-axis direction, orange). The displacement in the x-axis direction and rotation into z-axis direction of the upper right corner is restricted (red). Due to symmetry, the displacement of the right side of the model is restricted in the x-axis direction (light blue). It represents the centre of the fibre and thus a symmetry-axis.

![figure 4-2: 2D-model with boundary conditions and load](image)

4.3 Model overview

During this study several models are used in which several simulations are done. The basic geometry of all the models is the same; the 2D geometry is reduced from the 3D-model. This results in the geometry, boundary conditions and load in each model. In total six models are used in which element shapes, types and material properties are adapted.

For every model several simulations are done, in which only the crack length varies. Each simulation the crack is manually enlarged by shortening the boundary condition at the bottom. In the first simulation the displacement in the y-direction is applied over almost the complete bottom edge, except for a small part which represents the crack. Next simulation this boundary condition is shortened, suggesting the crack has grown. The crack starts at the middle of the fibre and grows along the x-axis into the matrix. Therefore the model has to be re-meshed every time a new simulation is done with a larger crack. These simulations are done for all the models.
4. Finite element analysis results

The first model is made with quadratic quadrilateral elements with collapsed distorted quadratic quadrilateral elements around the crack tip. The material represents an epoxy matrix composite with glass fibre reinforcements. This first model is called the ‘standard’ model with ‘standard epoxy’ material. The second and third models are equipped with the same elements. However, the Young’s modulus of the matrix is adapted in respectively a lower and higher modulus as the standard epoxy.

For so far all elements used in the models are plane stress elements. To explore the influence of this choice, model four is equipped with plane strain elements instead. The material is the same as used in the first model.

In all above mentioned models, the calculation method to determine the stress intensity factor is the same. This is done by using the displacements of the nodes from the first layer of elements around the crack tip. A more detailed explanation of this method is discussed in section 4.6.

Finally, two more models are created, provided with linear quadrilateral elements. Around the crack tip collapsed linear quadrilateral elements are applied, all elements have the plane stress properties. Concerning the material, model five is equipped with the standard epoxy composite. Model six consist out of one material, only matrix material with the normal Young’s modulus, see table 4-2.

The calculation of the K-value is based on a method commonly known as stress and displacement matching in conventional literature. The description of this method is done in section 4.6.

A typical deformation of the model is represented in figure 4-3. Table 4-1 gives an overview of the different models and adapted properties.
4. Finite element analysis results

<table>
<thead>
<tr>
<th>Model</th>
<th>Plane stress/strain</th>
<th>E-modulus Matrix</th>
<th>Quadratic/linear</th>
<th>Calculation method for K-value</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>Plane stress</td>
<td>Normal</td>
<td>Quadratic</td>
<td>Displacements first layer nodes</td>
</tr>
<tr>
<td>II</td>
<td>Plane stress</td>
<td>Lower</td>
<td>Quadratic</td>
<td>“ “</td>
</tr>
<tr>
<td>III</td>
<td>Plane stress</td>
<td>Higher</td>
<td>Quadratic</td>
<td>“ “</td>
</tr>
<tr>
<td>IV</td>
<td>Plane strain</td>
<td>Normal</td>
<td>Quadratic</td>
<td>“ “</td>
</tr>
<tr>
<td>V</td>
<td>Plane stress</td>
<td>Only Matrix</td>
<td>Linear</td>
<td>Extrapolation</td>
</tr>
<tr>
<td>VI</td>
<td>Plane stress</td>
<td></td>
<td>Linear</td>
<td>Extrapolation</td>
</tr>
</tbody>
</table>

table 4-1: Overview different FE models

4.4 Material properties

The model is split into two different constituents, a matrix and fibre part. In figure 4-1 and 4-2 this is visible by the green and grey colours, representing respectively fibre and matrix material. Table 4-2 shows the properties of the constituents used in the models. The first model is equipped with the material properties of materials 1 and 2 of table 4-2. Two other materials are added, with a higher and lower Young’s modulus, \( E \), for the matrix. Notice that the Poisson’s ratio, \( \nu \), is the same. The matrix with the normal E-modulus approximate an epoxy matrix composite reinforced with glass fibre reinforcement, the other matrixes are fictive [5].

<table>
<thead>
<tr>
<th>Material</th>
<th>( \nu ) [-]</th>
<th>( E ) [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Fibre</td>
<td>0,26</td>
<td>72e3</td>
</tr>
<tr>
<td>2. Matrix normal E-modulus</td>
<td>0,37</td>
<td>3,79e3</td>
</tr>
<tr>
<td>3. Matrix lower E-modulus</td>
<td>0,37</td>
<td>2,0e3</td>
</tr>
<tr>
<td>4. Matrix higher E-modulus</td>
<td>0,37</td>
<td>4,5e3</td>
</tr>
</tbody>
</table>

table 4-2: Material properties of matrix and fibre

4.5 Element shapes and types

The materials in the models are assumed to be linearly elastic, therefore the stresses at the crack tip is proportional to \( 1/\sqrt{r} \), [7]. The steep stress gradient at the tip can only be described with a lot of small elements. However, refining the mesh will not lead to convergence, further refinement always results in different solutions. The reason for this mesh dependency is the singularity in the stress field, which is not described by the element interpolation functions. In order to realize this singularity in the model, the mesh adjoint to the crack tip consists out of collapsed elements in a rosette-form. Furthermore this mesh form provides an equally divided stress transition to the surrounding elements.

In case of quadratic elements, this first band of elements is distorted as displayed in figure 4-4. The two mid-side nodes are repositioned manually in ABAQUS towards the one corner node, such that they provide a 1:3 ratio of the element side. It can be proved that they describe the \( 1/\sqrt{r} \) field singularity at the crack tip [6].

The linear elements do not describe the \( 1/\sqrt{r} \) singularity, but a \( 1/r \) singularity. Nevertheless, these models have been created to see what the effect is of this difference.
in singularity. Notice that the method of calculating the K-value has nothing to do with this. Also with quadratic elements it is possible to use the extrapolation method.

Figure 4-4 presents a typical final mesh of a model. The regions in which the model is divided are still recognizable, but the transition is not abrupt. Notice that the crack in this example is inside the matrix.
4. Finite element analysis results

4.6 Calculation methods for K-value

Every simulation results in an output of stresses and displacements. After simulating the different models, several methods are used to calculate the stress intensity factor.

In this research the K-value has been calculated by using its definition, which relates the K-value to stresses or displacements. This is called stress and displacement matching and can be done in two ways, either by extrapolation or by using formulas.

Stress and displacement matching by extrapolation

The stress field in front of the crack tip can be calculated by finite element method. As in Chapter 3 the shape of this field is expected to be as shown in figure 4-6, the continuous line. However it is known that for linear elastic material the stress field should be infinite at the crack tip, FEM predicts a finite stress value at the crack tip. If plotted the analytical solution of Chapter 3 above the FEM solution, it is clear that they are similar in the ‘matching zone’ of figure 4-6. This region is not too close to the crack tip, nor too far from the tip.

![Matching true stress field (circles), analytical (dashed line) and FEM solution (continuous line)](image)

The analytical solution is obtained considering the first term of (3-2) only; therefore it is reliable at short distances to, and inaccurate far away from, the crack tip. The opposite is true for FEM. The stress intensity factor can be obtained equating the analytical expression to the FEM solution in the ‘matching zone’.

From (3-3) $K_I$ can be calculated as:

$$K_I = \frac{\sigma_y \sqrt{2\pi r}}{\cos\left(\frac{\theta}{2}\right) + \frac{1}{2} \sin \theta \sin\left(\frac{3\theta}{2}\right)}$$  \hspace{1cm} (4-1)

In (4-1) $\sigma_y$ and $r$ can directly be deduced from the FEM output. For the crack angle $\theta$ the value 0 will be used, which corresponds with a crack along the x-asis, so $\sin(0) = 0$ and $\cos(0) = 1$.

Herewith $K_I$ can be calculated in every single node in line of the crack plane. These K-values plotted as function of the distance to the crack tip, $r$, result in a typical graph like figure 4-7. As shown here, the dependency on $r$ is approximately linear, when the nodes taken into account are not to close or to far from the crack tip. If one extrapolates
the straight line, then the intersection of this line with the vertical $K_I$-axis gives an accurate estimate of the SIF at the crack tip.

\[ K_I = \frac{2\mu u_y}{\sqrt{2\pi r}} \left[ \sin \left(\frac{\theta}{2}\right) \left(\kappa + 1 - 2\cos^2 \left(\frac{\theta}{2}\right)\right) \right] \]  

(4-2)

With the factors $\kappa$ and $\mu$

$\mu = \frac{E}{2(1+\nu)}$ and $\kappa = \frac{3-\nu}{1+\nu}$ for plain stress

$\kappa = 3-4\nu$ for plain strain

Again the crack angle is fixed, this time at $\theta = \pi$ otherwise the displacement is zero due to the boundary conditions, so $\sin(\pi) = 0$ and $\cos(\pi) = -1$. The graphs distinguish a little from the graphs determined by using the stress components, but the estimated SIF is approximately the same.

These methods are known as ‘stress and displacements matching’. In theory the accuracy between stress- and displacement matching should be equal, in practice there is however a numerical difference between both. The stresses calculated by FEM are in fact a result of the displacement field. Therefore some numerical errors occur during calculations of stresses and thereby also some small errors in the estimated $K_I$-value. Nevertheless both methods can be used for stress intensity factor calculations [7].
4. Finite element analysis results

Stress and displacement matching by using formulas

Another way to retrieve K-values is by using only the first layer of elements with their nodal stresses and displacements. This method is based on the same basics as with stress and displacement matching by extrapolation. Quadratic elements are required of which the mid-nodes of the first elements nearby the crack tip are placed differently, see section 4.5.

The following formulas of (4-3) can be used [6]:

\[
K_I = \frac{\sqrt{2\pi E}}{(12-3\nu)\sqrt{r}} \left[ 8U_y^{22} - U_y^{43} \right] \text{ for plain stress}
\]

\[
K_J = \frac{\sqrt{2\pi E}}{12(1-\nu^2)\sqrt{r}} \left[ 8U_y^{22} - U_y^{43} \right] \text{ for plain strain}
\]

Where \(U_y\) is the displacement in y-direction of the specific node, referring to figure 4-4. With this method no extrapolation is needed, the K-values follow straightforward from the formula. Actually this method is more accurate as with the conventional stress or displacement matching [7]. Due to moving the two middle nodes, the stress field shows the \(1/\sqrt{r}\) singularity at the crack tip. Whereas with a collapsed linear quadrilateral element, used with extrapolation, the singularity is \(1/r\) [4].

In (4-3) the y-displacements are used, it is however also possible to use the stresses at the crack tip.

Other methods to calculate the K-values make use of the J-integral, energy release rate or matching local strain energy. These methods are not used in these simulations and therefore not discussed here.

4.7 K-value graphs

In order to have a reference graph first model VI of table 4-1 has been tested. The K-values have been calculated and plotted into a K-a graph, see figure 4-8. In this graph the K-values grow with increasing the crack length. It is clear that until 2,5 mm the K-values are low and increase very rapidly. After 2,5 mm this increase seems to be linear and at the end the K-values grow again exponential.

The data of this graph, each crack length with corresponding K-value, can be found in appendix A. Also the data for the other graphs mentioned in this chapter are stated in that appendix.
4. Finite element analysis results

![K-a Graph](image)

**K-a Graph**

This ‘basic graph’ for one component material can be used as a reference when analyzing models I to III of *table 4-1*. Since these models consist of two constituents, some differences are expected. Still, the overall relation between crack length and K-value is likely to show some similarities to *figure 4-8*.

In *figure 4-9* model I-III is displayed, using the different material properties for the matrix. As can be seen in this graph, a distinction can be made between the cracks in the fibre and in the matrix phase. It is clear that the curve of the crack in the fibre corresponds with the K-a graph of a monolithic material, *figure 4-8*. More remarkable is the pattern of the matrix part of this graph. Around the interface the matrix crack distinguishes visibly from the reference graph. The K-value drops from 3.1mm to 4mm and increases afterwards, like the one constituent material, to infinity. At first sight this implies that a crack of 4mm is less likely to grow than a crack of 3.1mm. This is rather odd and is explained by the interface between fibre and matrix at 3mm. At this interface the normal singularities are not appropriate anymore. Therefore the reliability on K-values is not stable in a certain range around the interface. The cause of this irregularity is explained in *section 4.9*.

Furthermore, the difference between the models equipped with higher and lower Young’s modulus for the matrix and the ‘standard’ E-modulus can be seen. The ‘weaker’ material has higher K-values than the ‘stronger’ material. Given that a lower K-value indicates less chance of fracture, a crack in the material with a higher Young’s modulus has less potential to grow than with a lower E-modulus. This is according to the expectations; stronger material corresponds to a lower K-value and vice versa.
4. Finite element analysis results

To verify the form of the graph, the models IV and V are used. These models use different element types or calculation methods. The results are reproduced in figure 4-10. Model IV is equipped with plain strain quadratic elements and the K-values are calculated by using the displacement of the first layer nodes. Model V is simulated with linear plain stress elements and for calculation the extrapolation method is used. For both models all the other parameters are the same as in model I-III.

For evaluation also model III is reproduced in figure 4-10. Notice the influence of different element type and calculation method used in models IV and V. There are some not negligible differences of 5-10% between the models. These disabilities can be explained as the approximation of the reality. Plane strain and plane stress are both an idealisation of the reality. Further more finite element method is also an approximation. However, as can be seen the shape of all the curves are similar. Since different calculations methods and element types are used this implies that the form of the graphs is correct. The actual values deviate perhaps a little from the models.
4. Finite element analysis results

### 4.8 Verification LEFM

To verify linear elastic fracture mechanics can be used, the approximate plastic zone can be calculated. As stated in Chapter 3 this can be calculated with (3-4):

\[ r_p^* = \frac{K_I^2}{2\pi\sigma_{yy}} \]

The yield stress of a composite is not a hard digit, but can be estimated on the basis of different material properties and seems to be in a range of \(\sigma_{yy} = 70\text{--}140\text{MPa} \). Herewith the approximation of the plastic zone \(r_p^*\) is at the largest of the order \(4\cdot10^{-6}\text{mm} \). Even if the real plastic zone will be larger, see section 3.3, the smallest \(r\) used for calculations is in the order \(10^{-5}\text{mm} \). Therefore it can be concluded that LEFM is allowed to use for these calculations.

### 4.9 Interface between fibre and matrix

As simulation shows, in figure 4-9 and 4-10, the K-values at the interface of fibre and matrix differ from monolithic material behaviour. This is expected since material properties change in this area. The singularity at the crack tip in homogeneous material is known as \(r^{-1/2}\). At the interface a new type of singularity is encountered, which depends on the elastic constants of the materials and on the angle at which the crack is oriented with respect to the interface.

The term dominating the stress field series expansion of (3-2) is of order \(r^2\) as the crack tip is approached, see Chapter 3. The eigenvalue \(\lambda\) may be real or complex; in the latter
4. Finite element analysis results

case it may be written as $\lambda = \text{Re}(\lambda) + i \text{Im}(\lambda)$ and $\lambda$ is the smallest root of the following equation (or has the smallest $\text{Re}(\lambda)$ in case of a complex root) [8]:

$$(1 - \beta^2) \cos(\lambda \pi) - 2(\beta - \alpha)(1 - \beta)(1 + \lambda)^2 - (\alpha + \beta^2) = 0$$

(4-4)

In which $\alpha$ and $\beta$ are Dundurs’ parameters, a representation of the ratio between the constituents in the composite, as is further explained in appendix B.

Note that (4-4) is only a reduction of the original equation, given in appendix C. Equation (4-4) is valid for a straight crack approaching an interface at an angle of $\pi/2$, see figure 4-11, which is the case in this study.

figure 4-11: Crack touching, and normal to, an interface

Equation (4-4) produces two equal real roots in the range $-1 < \lambda < 0$ for each combination of $\alpha, \beta$. In this project $\alpha$ and $\beta$ are respectively -0.90 and -0.28, for calculations see appendix B. The corresponding eigenvalue is found by using Dundurs’ parameters in figure 4-12, which leads to $\lambda = -0.87$.

figure 4-12: Graph to determine the eigenvalue with the aid of Dundurs’ parameters

The singularity at the interface in this case would be $r^{-0.87}$, if $r \to$ the interface. This results in an accelerated increase of the K-value nearby the interface, visible in the graphs.
4. Finite element analysis results

At the other side of the interface, in the matrix, the K-value does not start as high as it ends in the fibre, while the distance to the interface is equal, 0.1mm. This can be ascribed to the different properties in the two constituents.

The reason why this change in singularity appears has to do with the attendance of both materials nearby the interface. The path of the stress field is influenced by the weaker or stronger material. If the stress reaches the other material, it has more or less ‘freedom’ to displace the molecules of that material. Therefore it reaches more of the material or the opposite, less of the material. Consequently the field looks different from the regular stress field in full material.

4.10 Crack growth graph

If the K-value is calculated, Paris law can be used for crack growth prediction. The stress intensity factor is related to the crack size $a$ with a polynomial function. This function can be determined from the graphs of figure 4-9 and 4-10 and is stated in the form as:

$$K_I(a) = c_4a^4 + c_3a^3 + c_2a^2 + c_1a + c_0$$  \hspace{1cm} (4-5)

In which $c_i$, with $i = 1, 2, 3, 4$, has to be determined.

Integrating the Paris law, (3-5), between two certain crack lengths, the numbers of cycles can be plotted to the crack length. This integration is stated as.

$$N(a) = \frac{1}{C} \int_{a_0}^{a_1} \frac{1}{K_I^n(a)} da$$  \hspace{1cm} (4-6)

The relation between the number of cycles, $N$, and crack length, $a$, is plotted in a figure 4-13. This is only an example of a N(a)-graph, it is accomplished from the fibre data of model V.

![Graph](image)

**figure 4-13: N(a) of model V**

Note that the parameters $C$ and $m$ have to be known for this graph. Since these are material constants, they have to be determined by experiments [3].
5. Conclusions and recommendations

Fracture mechanics applied on fibre reinforced composites has many unknown parameters. Therefore fibre composites are not always used to the fullest of their potential. During this project a finite element analysis of a fibre reinforced composite has been made. From the results of these analyses some remarks and general conclusions can be stated.

5.1 Conclusions

Theoretical the stress intensity factor can be linked to crack growth graphs. By calculating the stresses at the crack tip the K-value can be determined by the use of formulas. Combining these stress intensity factors for different crack lengths, the relation between K-values and crack lengths is found to apply in the Paris law. The Paris law integrated delivers a growth prediction graph of the material.

This study shows that the K-value is a useful parameter to predict crack growth inside fibre reinforced composites and can be determined with stress and displacement fields obtained by finite element analysis. The model used in this project shows results in accordance with the theory. The theory states that the stress intensity factor implies the possibility of crack growth in a material; the higher the K-value, the more chance of cracking. If the K-value reaches a critical value, the crack starts to grow. Several graphs show a clear picture of how the stress intensity factor evolves as a function of the crack size in a constituent. Indeed it is confirmed that if a crack length is enlarged, the K-values rise.

Different simulations and calculations show approximately the same results. This implies that the path of the graphs is correct. However, there are some deviations between the models around 5-10%. This can be explained by the setup of the finite element model. Finite element method is an approximation of the reality and so are the element types used in these models. Plain strain and plain stress are an idealisation of the actual material behaviour. Consequently there will be some differences between the models.

A remarkable occurrence is notable at the interface between fibre and matrix. Far enough from the border between fibre and matrix, the crack tip presents the material behaviour of a monolithic material. The K-value grows as the crack grows; the pattern is roughly the same as a K-a graph from a monolithic material. However, at the interface the curve distinguishes from the normal one. This can be explained by the singularity of the stress field around the interface. Inside the two constituents, not close to the interface, the singularity will be \( r^{-0.5} \), whereas the singularity is different nearby the fibre/matrix border. The calculated singularity in this study is around \( r^{-0.87} \), therefore the increase of the K-value accelerates. After the interface the singularity will again change in \( r^{-0.5} \). This change in singularity is caused by the two different materials nearby the interface, influencing the path of the stress field. If the crack is just inside the matrix, the stress does not follow the normal pattern like in full material. Since the fibre is a stronger material the vicinity of the fibre influences the stresses around the crack tip. This causes the differences in singularity.
5. Conclusions and recommendations

The change in material behaviour in the region of the interface makes it fragile to use the K-values in the Paris law around this area.

Furthermore, three different materials are simulated, with a higher and lower Young’s modulus and the same Poisson’s ratios. The results are consistent with the theory; the stress intensity factor of the stronger material is smaller as from the weaker material. Consequently the chance of cracking with the higher Young’s modulus is lower, corresponding to what is commonly known.

5.2 Recommendations

It is recommended to find out how the singularity changes exactly around the fibre and matrix interface. Since the change of singularities the K-value calculations are not reliable in this area. The stress intensity factor is calculated with the idea of a $r^{-0.5}$ singularity at all crack lengths. While in fact the singularity changes smoothly near the interface to $r^{-0.87}$. Therefore formulas have to be found to calculate K with this change of singularity as a function of $r$. Then the reliability of the K-value will increase around the interface and therefore the crack growth graphs are more trustworthy.

During this study several differences were noticed between plain stress and plain strain models. Although the shape of the K-a graphs are equal, there are still some deviations between the two. This seems to be obvious since both are an approximation of reality. However, if K-values are used for crack growth prediction graphs, it should be investigated if the accuracy is sufficient enough for its purpose. Otherwise still a too large safety factor must be applied.

This can be done by calculating the stress intensity factors in a different way. For example with the J-integral the K-value can be calculated due to conversion relations. If these results are the same, they are probably correct.
References


Appendix A: Original graph data

In section 5.2 several graphs were represented, the graph data can be found in this appendix.

Table A-1: The $K_I$-values of a crack varying from 0.10 to 9.90 mm in the same material, original matrix of epoxy.

<table>
<thead>
<tr>
<th>$a$ (mm)</th>
<th>$K_I$ ($Nm^{-3/2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>3.98</td>
</tr>
<tr>
<td>0.75</td>
<td>10.93</td>
</tr>
<tr>
<td>2.00</td>
<td>18.70</td>
</tr>
<tr>
<td>3.00</td>
<td>24.40</td>
</tr>
<tr>
<td>3.50</td>
<td>27.36</td>
</tr>
<tr>
<td>4.00</td>
<td>30.50</td>
</tr>
<tr>
<td>5.00</td>
<td>37.39</td>
</tr>
<tr>
<td>6.00</td>
<td>45.48</td>
</tr>
<tr>
<td>7.00</td>
<td>55.68</td>
</tr>
<tr>
<td>9.00</td>
<td>102.00</td>
</tr>
<tr>
<td>9.90</td>
<td>329.90</td>
</tr>
</tbody>
</table>

Table A-2: The $K_I$-values of a crack varying from 0.10 to 9.90 mm in three different materials.

Table A-3: The $K_I$-values of a crack varying from 0.10 to 9.90 mm in ‘standard’ epoxy, calculated with linear elements or plain strain condition.

<table>
<thead>
<tr>
<th>$a$ (mm)</th>
<th>$K_I$ ($Nm^{-3/2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>8.93</td>
</tr>
<tr>
<td>0.25</td>
<td>13.59</td>
</tr>
<tr>
<td>0.50</td>
<td>17.87</td>
</tr>
<tr>
<td>0.75</td>
<td>22.35</td>
</tr>
<tr>
<td>1.00</td>
<td>27.88</td>
</tr>
<tr>
<td>1.25</td>
<td>31.03</td>
</tr>
<tr>
<td>1.50</td>
<td>35.91</td>
</tr>
<tr>
<td>2.00</td>
<td>51.01</td>
</tr>
<tr>
<td>2.50</td>
<td>76.98</td>
</tr>
<tr>
<td>2.90</td>
<td>160.99</td>
</tr>
<tr>
<td>3.10</td>
<td>53.47</td>
</tr>
<tr>
<td>3.25</td>
<td>44.47</td>
</tr>
<tr>
<td>3.50</td>
<td>40.05</td>
</tr>
<tr>
<td>3.75</td>
<td>38.74</td>
</tr>
<tr>
<td>4.00</td>
<td>38.59</td>
</tr>
<tr>
<td>4.25</td>
<td>34.52</td>
</tr>
<tr>
<td>4.50</td>
<td>39.56</td>
</tr>
<tr>
<td>5.00</td>
<td>41.63</td>
</tr>
<tr>
<td>5.50</td>
<td>39.60</td>
</tr>
<tr>
<td>6.00</td>
<td>47.93</td>
</tr>
<tr>
<td>7.00</td>
<td>57.53</td>
</tr>
<tr>
<td>8.00</td>
<td>65.80</td>
</tr>
<tr>
<td>9.00</td>
<td>108.62</td>
</tr>
<tr>
<td>9.90</td>
<td>361.30</td>
</tr>
</tbody>
</table>

Table A-1

<table>
<thead>
<tr>
<th>$a$ (mm)</th>
<th>$K_I$ ($Nm^{-3/2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>9.08</td>
</tr>
<tr>
<td>0.25</td>
<td>14.85</td>
</tr>
<tr>
<td>1.00</td>
<td>30.64</td>
</tr>
<tr>
<td>2.00</td>
<td>56.13</td>
</tr>
<tr>
<td>2.50</td>
<td>84.83</td>
</tr>
<tr>
<td>2.90</td>
<td>178.25</td>
</tr>
<tr>
<td>3.10</td>
<td>57.10</td>
</tr>
<tr>
<td>3.25</td>
<td>47.38</td>
</tr>
<tr>
<td>3.50</td>
<td>42.64</td>
</tr>
<tr>
<td>3.75</td>
<td>41.24</td>
</tr>
<tr>
<td>4.00</td>
<td>41.09</td>
</tr>
<tr>
<td>4.50</td>
<td>42.16</td>
</tr>
<tr>
<td>5.00</td>
<td>44.43</td>
</tr>
<tr>
<td>6.00</td>
<td>51.30</td>
</tr>
<tr>
<td>7.00</td>
<td>61.75</td>
</tr>
<tr>
<td>9.00</td>
<td>117.15</td>
</tr>
<tr>
<td>9.90</td>
<td>390.20</td>
</tr>
</tbody>
</table>

Table A-2

<table>
<thead>
<tr>
<th>$a$ (mm)</th>
<th>$K_I$ ($Nm^{-3/2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>9.37</td>
</tr>
<tr>
<td>0.25</td>
<td>15.30</td>
</tr>
<tr>
<td>1.00</td>
<td>31.72</td>
</tr>
<tr>
<td>2.00</td>
<td>58.88</td>
</tr>
<tr>
<td>2.50</td>
<td>90.78</td>
</tr>
<tr>
<td>2.90</td>
<td>203.46</td>
</tr>
<tr>
<td>3.10</td>
<td>60.12</td>
</tr>
<tr>
<td>3.25</td>
<td>48.98</td>
</tr>
<tr>
<td>3.50</td>
<td>43.55</td>
</tr>
<tr>
<td>3.75</td>
<td>41.85</td>
</tr>
<tr>
<td>4.00</td>
<td>41.54</td>
</tr>
<tr>
<td>4.50</td>
<td>42.41</td>
</tr>
<tr>
<td>5.00</td>
<td>44.43</td>
</tr>
<tr>
<td>6.00</td>
<td>51.28</td>
</tr>
<tr>
<td>7.00</td>
<td>61.65</td>
</tr>
<tr>
<td>9.00</td>
<td>117.01</td>
</tr>
<tr>
<td>9.90</td>
<td>390.15</td>
</tr>
</tbody>
</table>

Table A-3
Appendix B: Dundurs’ Parameters

The state of stress in homogeneous material undergoing plane deformation depends on only two elastic parameters, for example Young’s modulus, $E$, and Poisson’s ratio, $\nu$. Despite two materials in a bi-material joint, which both have their own two elastic parameters, Dundurs showed that the stress state in a bi-material joint undergoing plane deformation also depends on only two parameters, $\alpha$ and $\beta$.

If $\mu_1, \nu_1$ and $\mu_2, \nu_2$ are respectively the modulus of rigidity and Poisson’s ratio of material ‘1’ and material ‘2’, Dundurs’ parameters are then given by:

$$\alpha = \frac{\mu_2 (\kappa_1 +1) - \mu_1 (\kappa_2 +1)}{\mu_2 (\kappa_1 +1) + \mu_1 (\kappa_2 +1)}$$

$$\beta = \frac{\mu_2 (\kappa_1 -1) - \mu_1 (\kappa_2 -1)}{\mu_2 (\kappa_1 +1) + \mu_1 (\kappa_2 +1)}$$

(B.1)

Where $\kappa_i$ is Kolosov’s constant: $\kappa_i = (3-\nu_i)/(1+\nu_i)$ in plane stress, $\kappa_i = 3-4\nu_i$ in plane strain, of material $i$.

These two parameters enable one to generalize solutions and reduce the complexity of formulae. An $\alpha$-$\beta$ plane can be constructed and is shown in figure B-1. It provides a convenient means of classifying composite materials and for displaying results that depend on elastic constants. The magnitude of $\alpha$ and $\beta$ describe the degree of mismatch between the materials. It is obvious that the origin then represents no mismatch, being identical materials.

The parallelogram may be divided into two by the straight line $\alpha = \beta$, along which $\mu_1 = \mu_2$. To the left of this line, $\mu_1 > \mu_2$, material ‘1’ is more rigid than material ‘2’, and vice versa [8].

Figure B-1: The $\alpha$-$\beta$ plane
Appendix C: Straight cracks terminating the interface

Consider a mode I crack in an infinite plate, the plate is subjected to a tensile stress of \( \sigma \) at infinity. An element \( dx \, dy \) exists at a distance \( r \) from the crack tip and at an angle \( \theta \) with respect to the crack plane. The normal stresses and shear stress this element experiences can in general be described in a series expansion of the form:

\[
\sigma_y(r, \theta) = \sum_n \varepsilon_n r^\lambda_n f_n(\theta) = \varepsilon_1 r^{\lambda_1} f_1(\theta) + \varepsilon_2 r^{\lambda_2} f_2(\theta) + \ldots \tag{C-1}
\]

In case this crack exists in a bi-material and touches the interface at a certain angle \( \theta \), see figure C-1, the eigenvalue \( \lambda \) can be calculated with the following equation C-2:

\[
0 \equiv \left[ A \beta^2 - (2A - B) \beta + A - B + 1 \right] \alpha^2 + \left[ (-2A + B + C) \beta^3 + (4A - 2B - C + D + 2) \beta^2 - (2A - B + C) \beta + C - D \right] \alpha + (A - B - C + D + E + 1) \beta^4 - (2A - B - C) \beta^3 + (A + C - D - 2E) \beta^2 - C \beta + E \tag{C-2}
\]

Where \( \alpha \) and \( \beta \) are Dundurs’ parameters and:

\[
A(\theta, \lambda) = 4(1 + \lambda)^4 \sin^2 \theta + \sin^2 \left[ (1 + \lambda)(2\theta - \pi) \right]
\]

\[
B(\theta, \lambda) = 4(1 + \lambda)^2 \sin^2 \theta + 2 \sin^2 \left[ (1 + \lambda)(2\theta - \pi) \right]
\]

\[
C(\theta, \lambda) = 4(1 + \lambda)^2 \sin^2 \theta \left\{ \sin^2 \left[ (1 + \lambda)\theta \right] + \sin^2 \left[ (1 + \lambda)(\theta - \pi) \right] - 1 \right\} \tag{C-3}
\]

\[
D(\theta, \lambda) = 2 \left\{ \sin^2 \left[ (1 + \lambda)\theta \right] + \sin^2 \left[ (1 + \lambda)(\theta - \pi) \right] - 1 \right\}
\]

\[
E(\theta, \lambda) = \cos^2 \left[ \lambda \pi \right]
\]

The eigenvalue \( \lambda \) may be real or complex and is the smallest root of equation C-2. In case of a complex root, the smallest value of Re(\( \lambda \)) [8].

![Figure C-1: The crack tip touching the interface at an angle \( \theta \)]