Bimaterial Interface Cracks
Analytical solutions and a FEM analysis using Cohesive Zone Modeling.

Stijn Boere

June 2, 2006

supervisors: Dr. ir. B.A.E. van Hal
            Dr. ir. P.J.G. Schreurs
## Contents

1 Introduction 2

2 Linear Elastic Fracture Mechanics 3
   2.1 Crack in a homogenous material 4
   2.2 Crack along a bimaterial interface 7
   2.3 Linear finite element analysis 9
   2.4 Optimization of the model size 14

3 Cohesive Zone Modeling 17
   3.1 Mechanical behavior 17
   3.2 Cohesive zone elements 19
   3.3 Solution strategies 20

4 Simulations with Cohesive Zone Modeling 24
   4.1 FE model 24
   4.2 Results 26
   4.3 COD approximation function 29

5 Conclusions 31

Bibliography 31

Appendices 32

A Geometry correction factor of a bimaterial 33
Chapter 1

Introduction

This project deals with the modeling of fracture between two material layers using the finite element method (FEM). Integrated circuits for example contain these stacked material layers, referred to as bimaterials. It is of great importance to be able to predict the failure of such bimaterials. The theory of Linear Elastic Fracture Mechanics describes the stress and displacement fields near a bimaterial crack. These results are easily verified by a finite element analysis. A disadvantage of this method however is that crack nucleation can not be simulated and an initially imposed crack is required.

Cohesive zone modeling deals with these problems. The implementation of cohesive zone elements in the Finite Element Method is straightforward. Crack propagation itself is an outcome of the analysis and there is no need for an initially imposed crack. Cohesive zone elements introduce a non-linear constitutive relation which accounts for the gradual decohesion of the bimaterial interface. The disadvantage of cohesive zone modeling are the difficulties in the solution procedure caused by this non-linear behavior.

For the shape function in a cohesive zone element, a linear interpolation of the field variable of interest is used. However, the displacement field and the corresponding stress field near a crack tip is non-linear. Therefore, the linear shape function is a poor approximation of the actual situation. Knowledge of the geometry of the crack tip can help to implement other shape functions that interpolate the field variables more realistically. Therefore the goal of this project is:

Finding a simple function that describes the crack opening displacement along a bimaterial interface as a function of the distance to the crack tip using a finite element model with cohesive zone elements.

The analysis of this problem is done with Matlab using the FEM Artemis code.

First, two analytically well known problems are examined to gain insight into the validity of linear elastic FEM solutions. A crack in a homogenous material and a crack in a bimaterial are considered. For this purpose, the displacement field and the corresponding stress field near the crack tip is examined. In these cases, analytical solutions are available, which are used to validate the numerical results. Next, Cohesive Zone Modeling is introduced which accounts for the material softening. The difficulties in the solution procedure caused by non-linearities are considered. Next, the model and the results of the non-linear analysis are presented. In this case no analytical solution is available and therefore a function will be fitted onto the resulting data of the displacement field near the crack tip. Finally a conclusion will be drawn on which function is most suitable for use in the shape functions.
Chapter 2

Linear Elastic Fracture Mechanics

Cracks in bimaterials have been studied extensively using the theory of fracture mechanics. Three fracture modes are recognized. Mode I: the crack opening mode, mode II: the sliding mode and mode III: the tearing mode. Since this project assumes a plane strain situation, fracture mode III does not occur. Two different cases are considered, cracks in homogeneous materials and cracks along a bimaterial interface. The following section gives the analytical asymptotic solutions of the displacement and stress fields. The FEM results are compared to the analytical solutions. This comparison provides insight into the range in which the analytical solution is valid. Subsequently, the boundary effects are examined and the model size is optimized for the simulations which are carried out in the next chapters.
2.1 Crack in a homogenous material

2.1.1 Problem definition

Figure 2.1 shows a strip of infinite length and width \( w \). A plane strain situation is assumed. A crack of length \( 2a \) is located in the center of the strip. The material is linear elastic with Young’s modulus \( E \) and poisson ratio \( \nu \). The strip is subjected to a load \( \sigma_\infty \) far away from the crack. The displacement and stress fields near the tip of the crack are examined.

![Figure 2.1: Crack in a homogeneous material](image)

2.1.2 Analytical solution of stress field

The analytical solutions are a function of the cylindrical coordinates \( \{r, \theta\} \). The following solutions hold for the different stress components defined in a cartesian coordinate system,

\[
\sigma_{xx} = \frac{K_I}{\sqrt{2\pi r}} \left( \cos \left( \frac{1}{2} \theta \right) \left( 1 - \sin \left( \frac{1}{2} \theta \right) \sin \left( \frac{3}{2} \theta \right) \right) \right) \quad (2.1a)
\]

\[
\sigma_{yy} = \frac{K_I}{\sqrt{2\pi r}} \left( \cos \left( \frac{1}{2} \theta \right) \left( 1 + \sin \left( \frac{1}{2} \theta \right) \sin \left( \frac{3}{2} \theta \right) \right) \right) \quad (2.1b)
\]

\[
\sigma_{xy} = \frac{K_I}{\sqrt{2\pi r}} \left( \cos \left( \frac{1}{2} \theta \right) \sin \left( \frac{1}{2} \theta \right) \cos \left( \frac{3}{2} \theta \right) \right) \quad (2.1c)
\]

\[
\sigma_{zz} = \nu (\sigma_{xx} + \sigma_{yy}) \quad (2.1d)
\]

These expressions result from the complex function theorem [Schreurs]. The solutions are asymptotic approximations and are only valid near the crack tip because higher order terms are neglected in the derivation of the solutions.
Here, the main interest goes to the stress situation in front of the crack tip along the line \( y = 0 \). The amount of data is reduced substantially, which simplifies the comparison between the analytical and the FEM analysis. For \( \theta = 0 \), Equations (2.1a) - (2.1d) reduce to

\[
\begin{align*}
\sigma_{xx} &= \frac{K_I}{\sqrt{2\pi r}} \\
\sigma_{yy} &= \frac{K_I}{\sqrt{2\pi r}} \\
\sigma_{xy} &= \frac{K_I}{\sqrt{2\pi r}} \\
\sigma_{zz} &= \nu(\sigma_{xx} + \sigma_{yy})
\end{align*}
\] (2.2a - 2.2d)

In the expressions above, \( K_I \) represents the stress intensity factor (SIF) in the crack opening mode. This factor characterizes the magnitudes of the stress field which depends on the crack size \( a \) and geometrical boundaries. \( K_I \) is given by

\[
K_I = c\sigma_\infty\sqrt{\pi a}.
\] (2.3)

Parameter \( c \) in Equation (2.3) is a correction factor which depends on the width of the strip. In case of an infinite plate \( (w = \infty) \), \( c \) equals 1. Otherwise, this factor is given by

\[
c = \left(\frac{2w}{\pi a}\tan\frac{\pi a}{2w}\right)^{\frac{1}{2}}.
\] (2.4)

### 2.1.3 Analytical solution of crack opening displacement

The analytical solution for the displacement field in mode I is given by

\[
\begin{align*}
u_x &= \frac{K_I}{2G}\left(\frac{r}{2\pi}\cos\left(\frac{1}{2}\theta\right)\left(\kappa - 1 + 2\sin^2\left(\frac{1}{2}\theta\right)\right)\right) \\
u_y &= \frac{K_I}{2G}\left(\frac{r}{2\pi}\sin\left(\frac{1}{2}\theta\right)\left(\kappa + 1 - 2\cos^2\left(\frac{1}{2}\theta\right)\right)\right).
\end{align*}
\] (2.5a - 2.5b)

In this relation, \( G \) and \( \kappa \) represent the shear modulus and the Kolosov constant respectively, which are given by

\[
\begin{align*}
G &= \frac{E}{2(1 + \nu)} \\
\kappa &= \frac{3 - \nu}{1 + \nu}
\end{align*}
\] (2.6a - 2.6b)
The crack opening displacement (COD) is defined as the separation vector between two points that initially coincided before crack propagation occurred. Due to symmetry in the model, the x component of the COD is zero. The y component of the crack opening displacement is defined as

\[ COD_y = 2u_y(\theta = 0) = \frac{K_I}{G} \sqrt{\frac{r}{2\pi}} (\kappa + 1). \]  

(2.7)

Figure 2.2 plots the \( COD_y \) as a function of the distance to the crack tip \( r \). Figure 2.3 plots the \( \sigma_{yy} \) component of the stress field in front of the crack tip as a function of the distance to the crack tip \( r \). These results are obtained with a stress \( \sigma_\infty = 100 \) Mpa and material constants \( E = 130 \) Gpa and \( \nu = 0.3 \). The crack length is 60 mm and the width of the strip \( w \) is infinite.

![Figure 2.2: Analytical solution \( COD_y \)](image1)

![Figure 2.3: Analytical solution \( \sigma_{yy} \)](image2)
2.2 Crack along a bimaterial interface

2.2.1 Problem definition

Again a strip of infinite length is examined. The strip consists of two materials with material constants \( E_1, \nu_1 \) and \( E_2, \nu_2 \). The strip is subjected to the same loading and boundary condition as used before. The displacement and stress fields near the tip of an interface crack of length \( 2a \) are examined.

![Figure 2.4: Linear crack along a bimaterial interface](image)

2.2.2 Analytical solution of stress field

The analytical solution for the \( \sigma_{yy} \) component of the stress for \( \theta=0^\circ \) is given by [Hutchinson]

\[
\sigma_{yy} = \frac{\Re\left(\hat{K}r^{i\epsilon}\right)}{\sqrt{2\pi r}}
\]  

(2.8)

where \( \Re\{\bullet\} \) denotes the real part of the complex quantity. The complex stress intensity factor \( \hat{K} \) in this expression is defined as

\[
\hat{K} = \hat{c}\sigma_\infty\sqrt{\pi a}(1 + 2i\epsilon)(2a)^{-i\epsilon}
\]  

(2.9)

In this expression, \( \hat{c} \) represents a correction factor which depends on the strip’s geometry. For an infinite plate, this factor equals 1. Contrary to the correction factor for a crack in a homogeneous strip however, there is no analytical solution for this factor stated in literature for a strip of finite width. In Appendix A efforts will be made to obtain this correction factor by fitting the analytical results onto the FEM data. However due to inaccuracies in the FEM data this has not succeeded.
Therefore, from now on the correction factor will be set equal to 1 as an assumption. The bimaterial parameter $\epsilon$ is given by

$$\epsilon = \frac{1}{2\pi} \ln \frac{1 - \beta}{1 + \beta},$$

where $\beta$ represents the Dundurs’ elastic mismatch parameter. This parameter solely depends on the elastic properties of the considered material.

$$\beta = \frac{\mu_1(\kappa_2 - 1) - \mu_2(\kappa_1 - 1)}{\mu_1(\kappa_2 + 1) + \mu_2(\kappa_1 + 1)}.$$ (2.10)

### 2.2.3 Analytical solution of crack opening displacement

The analytical solution of the COD is given by [Hutchinson]

$$COD_y = \Re \left\{ \frac{8}{(1 + 2i\epsilon) \cosh \pi \epsilon E^*_E \left( \frac{r}{2\pi} \right) \frac{1}{r}} \right\}. \quad (2.12)$$

In this relation, the equivalent Young’s modulus $E^*_E$ is expressed as

$$E^*_E = \left( \frac{1}{2} \left[ \frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \right] \right)^{-1}. \quad (2.13)$$

Figure 2.5 plots the COD as a function of the distance to the crack tip $r$. Figure 2.6 plots the $\sigma_{yy}$ component of the stress in front of the crack tip as a function of the distance to the crack tip $r$. These results are obtained with an infinite plate. Table 2.1 gives the model parameters and the material properties which are used.

<table>
<thead>
<tr>
<th>Load</th>
<th>$\sigma_\infty$</th>
<th>100 [Mpa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Crack length</td>
<td>a</td>
<td>60 [mm]</td>
</tr>
<tr>
<td>Elastic Moduli</td>
<td>$E_1$</td>
<td>200 [Gpa]</td>
</tr>
<tr>
<td></td>
<td>$E_2$</td>
<td>50 [Gpa]</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>$\nu_1$</td>
<td>0.3 [-]</td>
</tr>
<tr>
<td></td>
<td>$\nu_2$</td>
<td>0.3 [-]</td>
</tr>
<tr>
<td>Kolosov constants</td>
<td>$\kappa_1$</td>
<td>1.8 [-]</td>
</tr>
<tr>
<td></td>
<td>$\kappa_2$</td>
<td>1.8 [-]</td>
</tr>
<tr>
<td>Dundurs elastic mismatch parameter</td>
<td>$\beta$</td>
<td>0.17</td>
</tr>
<tr>
<td></td>
<td>$\epsilon$</td>
<td>-0.055</td>
</tr>
</tbody>
</table>

Table 2.1: Model parameters and material properties
2.3 Linear finite element analysis

2.3.1 Model Description

Figure 2.7 shows a schematic representation of the strip with the applied boundary conditions. Symmetry conditions are imposed on the left boundary. The left bottom node is fixed. The set of boundary conditions suppress rigid body motion. The crack is modeled by disconnecting the nodes of the elements on the crack faces. Figure 2.7 shows the non-uniform mesh distribution. The mesh is refined in the zone where high stress gradients are expected, i.e. near the crack tip.

Table 2.2 lists the parameters used in the FEM analysis.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strip width</td>
<td>( \frac{1}{2} w )</td>
</tr>
<tr>
<td>Strip length</td>
<td>L</td>
</tr>
<tr>
<td>Crack length</td>
<td>a</td>
</tr>
<tr>
<td>Elastic Moduli</td>
<td></td>
</tr>
<tr>
<td>( E_1 )</td>
<td>200 Gpa</td>
</tr>
<tr>
<td>( E_2 )</td>
<td>50 Gpa</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>( \nu_1 )</td>
</tr>
<tr>
<td></td>
<td>( \nu_2 )</td>
</tr>
<tr>
<td>Load</td>
<td>( \sigma )</td>
</tr>
</tbody>
</table>

Table 2.2: Parameters of FE model
2.3.2 Results of a homogeneous strip

The elastic modulus $E = E_1$ and Poisson ratio $\nu = \nu_1$. Figure 2.8 shows a plot of the deformed mesh, and Figure 2.9 shows a contour plot of the $\sigma_{yy}$ component of the stress. Figure 2.10 shows a close up of the asymptotic stress field near the crack tip. This figure clearly illustrates the familiar dog bone shape of the stress contours. There is a minor influence of boundary effects near the horizontal boundaries.
Figure 2.8: FEM Deformed Mesh

Figure 2.9: FEM Stress field

Figure 2.10: Close up of asymptotic stress field
The FEM results are compared to the analytical solution of the stress field of Equation (2.2b) and the COD of Equation (2.5b). These relations assume a strip of infinite length. The FEM model has a finite length, the analytical solutions are therefore an approximation. Obviously, when the length of the strip is increased this approximation becomes more accurate.

To gain knowledge on the influence of the meshsize, several simulations are carried out. Figures 2.11 and 2.12 show the COD and stress field from the analytical and numerical results. A parameter $h$ is introduced which is a measure of the size of the smallest element in the mesh. The Finite Element solution converges to the analytical solution for the subsequent mesh refinements. The analytical solution is asymptotic approximation. Near the crack tip, where $r$ approaches zero, the analytical solution becomes more accurate. The FEM solution in this range however is relatively less accurate. Far away from the crack tip, the analytical solution becomes highly inaccurate due to the neglected higher order terms. Consequently, these figures give an indication of the range in which the analytical solution is valid. This is illustrated in the figure by the vertical line.

![Figure 2.11: Comparison of the COD<sub>y</sub>](image1)

![Figure 2.12: Comparison of $\sigma_{yy}$](image2)
2.3.3 Results of a bimaterial interface

Figure 2.13 shows a contour plot of the stress field near the crack tip in a bimaterial. The familiar dog bone shape has clearly been distorted due to the different materials properties of the layers. This figure also shows some disturbance in the stress field caused by the variation in element size.

![Figure 2.13: FEM stress field](image)

Again the stress field obtained from the FEM analysis and the analytical results are compared. The analytical solution is not corrected for the finite strip width ($\hat{c} = 1$). Figures 2.14 and 2.15 show the analytical and FEM results of the COD and the stress field. This figures give a good indication of the range in which the asymptotic solution is valid as shown in the figure as a vertical line.
2.4 Optimization of the model size

The width of the model is reduced to reduce computing time, which becomes important in case non-linearities are introduced. The correction factor which accounts for the finite width of the model is very small compared to the inaccuracies in the FEM analysis. Therefore the width of the model can be reduced without affecting the accuracy of the FEM solutions. The strip’s width is reduced from 600 mm to 420 mm.

Figure 2.9 illustrates that there is a minor influence of boundary effects at the horizontal boundaries. The length of the model can be reduced to reduce computing time. However, minimizing the length of the strip will affect the comparison between the analytical and numerical results. A criterion is formulated, which gives an acceptable deviation from the exact solution, to be able to minimize the length of the strip without affecting the accuracy by a large amount. The initial strip of size 600x420 mm is taken as a reference. Several simulations are performed with decreasing strip length. A relative stress error measure in node \( i \) is defined as

\[
e_{rel}^{i} = \frac{\sigma_{i} - \sigma_{ref}^{i}}{\sigma_{ref}^{i}}
\]  

(2.14)

Figure 2.16 shows several error plots.
Large relative errors appear along the crack side because the reference stress levels are very small and zero on the crack faces. This will effect the overall accuracy rate. However, interest goes to the stress field in front of the crack tip. Therefore, only the nodes in front of the crack tip are considered. The Root Mean Square relative error in these nodes is determined. This value is set to a maximum of 5%. As a result the strip length can be reduced to 475 mm instead of 600 mm, which will lead to a mean deviation from the reference solution of 4.4%.

The modified model of the strip is plotted in Figure 2.17 and 2.18.
Figure 2.17: Modified mesh

Figure 2.18: Modified deformed mesh
Chapter 3

Cohesive Zone Modeling

Recall that the analytical solution for the stress at a crack tip goes to infinity close to the crack tip. Obviously this cannot be observed in reality since the material cannot be loaded beyond its yield strength. If the yield strength is reached, the material begins to deform plastically. To account for the plastic behavior, Cohesive Zone Modeling is used. Cohesive zone elements are located at the crack interface. These elements account for the irreversible non linear processes of the entire model, which consist of linear elastic elements.

3.1 Mechanical behavior

Figure 3.1(a) shows a bimaterial interface. Two material points \( P_1 \) and \( P_2 \) are considered that initially coincide. Figure 3.1(b) shows the cracking of the bimaterial.

![Figure 3.1](image)

Figure 3.1: Cracking along a bimaterial interface

The separation of of the two material faces is accompanied by the traction vector \( \vec{t} \). The constitutive relation between the separation and the traction is defined in equivalent quantities. The effective separation \( \lambda \) is defined as
\[ \lambda = \sqrt{\eta^2 \delta_s^2 + \delta_n^2}, \quad (3.1) \]

where \( \eta \) defines the ratio between the maximum shear traction and the maximum normal traction.

The following traction-separation law relates the effective separation to the effective traction.

\[ \tau = \sigma_c \frac{\lambda}{\lambda_c} e^{1-\lambda/\lambda_c} \quad (3.2) \]

In this relation, \( \sigma_c \) is the maximum cohesive traction and \( \lambda_c \) is the characteristic effective separation where the traction reaches its maximum.

The traction vector \( \mathbf{t} \) is defined as follows

\[ \mathbf{t} = \frac{\tau}{\lambda} \left\{ \begin{array}{c} \beta^2 \delta_s \\ \delta_n \end{array} \right\} \quad (3.3) \]

Figure 3.2: Exponential traction separation path according to Equation (3.2)

Figure 3.2 gives a graphical representation of the traction separation law. The traction increases with increasing separation until the characteristic effective separation \( \lambda_c \) is reached. After this point the traction decreases with increasing separation, i.e. softening takes places. Eventually with increasing separation, the traction vanishes and complete decohesion occurs.
3.2 Cohesive zone elements

The implementation of the cohesive zone modeling is discussed briefly. Consider the quadrilateral CZ element depicted in Figure 3.3. Initially the element is collapsed. Upon loading, the element deforms and the normal and the tangential separation between the element nodes is determined as illustrated in Figure 3.3.

Equation 4.1 introduces the effective separation on the left and right element side

\[ \lambda_i = \sqrt{\beta^2 \delta_{si}^2 + \delta_{ni}^2} \]  

(3.4)

The effective separation along the edge is approximated by linear interpolation. The traction vector \( \mathbf{t}(s) \) is determined using the separation approximation and the effective traction using Equation (3.3). The following equilibrium equation is obtained from simple kinematic relations and by application of the principle of virtual work.

\[ \mathbf{f}_i = \mathbf{f}_i(\mathbf{u}) = \mathbf{f}_e \]  

(3.5)

Due to the non-linearities in the traction separation law, the nodal forces \( \mathbf{f}_i \) are a non-linear function of the nodal displacements \( \mathbf{u} \). \( \mathbf{f}_e \) represents the external force vector.

The nodal displacements \( \mathbf{u} \) are determined using a global solution procedure. This procedure requires a tangential stiffness matrix \( \mathbf{K} \).

\[ \mathbf{K}(\mathbf{u}) = \frac{\partial \mathbf{f}_e(\mathbf{u})}{\partial \mathbf{u}} \]  

(3.6)

The tangential stiffness matrix is also a non-linear function of the nodal displacement vector \( \mathbf{u} \).
3.3 Solution strategies

In the previous section a non-linear cohesive zone element is introduced. This poses some numerical difficulties. Therefore special attention is paid to the solution strategies to solve these equilibrium equations. In order to illustrate the numerical difficulties associated with cohesive zone modeling, a series connection of a linear spring (spring I) and a non-linear spring (spring II) is considered (see Figure 3.4). These springs illustrate the effect of non-linearities in a combined system. The non-linear spring can be seen as a analog of the cohesive zones and the linear spring can be seen as an analog for linear plane strain elements.

![Multiple spring systems](image)

Figure 3.4: Multiple spring systems

Three different non-linear springs are examined, \(a\), \(b\) and \(c\). Figure 3.4 also shows the behavior of the entire system in the three different situations. Spring \(b\) causes a snap-through point in the load-displacement response of the combines spring system. Spring \(c\) even causes a snap-back point in the load-displacement response due to the extreme softening after the maximum load is reached.

The non-linearities necessitate the use of an iterative solution procedure such as the Newton Raphson iteration procedure. An incremental procedure is generally followed, in which the load is applied in steps. Within each load step the solution procedure iterates until the solution is approximated with a specified accuracy. Such incremental iterative solution procedure allows one to trace a non-linear equilibrium path, for example the path \(a\) for the combines spring system of Figure 3.4.
### 3.3.1 Newton Raphson solution strategy

The unknown solution vector \( \mathbf{u} \) must satisfy the equilibrium equation given by

\[
f_i(\mathbf{u}) = \alpha \bar{f}_e
\]  

(3.7)

In this equation, \( f_i(\mathbf{u}) \) represents the internal force vector, \( \alpha \) is a scalar and \( \bar{f}_e \) represents the unit force vector. The scalar load factor \( \alpha \) is increased after an increment has converged. The iteration procedure consist of the following steps

1. **Compute residual:**
   \[
   r(u^{(i-1)}) = \alpha \bar{f}_e - f_i(u^{(i-1)}),
   \]  
   (3.8a)

2. **Solve iterative update:**
   \[
   K(u^{(i-1)})d\mathbf{u}^{(i)} = r(u^{(i-1)}),
   \]  
   (3.8b)

3. **Update solution:**
   \[
   u^{(i)} = u^{(i-1)} + d\mathbf{u}^{(i)} = u^{(c)} + D\mathbf{u}^{(i)}
   \]  
   (3.8c)

These steps are repeated until convergence occurs. In this relation, \( r \) represents the residual or unbalanced force and \( K \) represent the tangential stiffness matrix. \( u^{(c)} \) is the solution of the previous load increment, and \( D\mathbf{u} \) is the total incremental update. Figure 3.5 gives a graphical representation of the Newton Raphson procedure.

![Figure 3.5: Newton Raphson solution strategy](image)

(a) Newton Raphson trace successful

(b) Newton Raphson trace fails

Figure 3.5: Newton Raphson solution strategy

Figure 3.5(b) also shows the disadvantage of the full Newton Raphson Method. The procedure can not pass load limit points due to the fixed load factor \( \alpha \). Figure 3.5(b) shows that the solution for \( \alpha = \alpha_{n+1} \) does not exist. The Newton Raphson solution strategy is therefore not suitable for the spring systems b and c from Figure 3.4. A more enhanced solution procedure is required such as Crisfield’s cylindrical arc-length method discussed next.
3.3.2 Crisfield’s cylindrical arc length method

In Crisfield’s cylindrical arc-length method both the load and the displacement are varied within a load increment. The load variation is expressed as

$$\alpha^{(i)} = \alpha^{(i-1)} + d\alpha^{(i)}$$ (3.9)

where $d\alpha^{(i)}$ is a priori unknown. The iterative solution update $du$ is defined as

$$du^{(i)} = d\hat{u}^{(i)} + d\bar{u}^{(i)}d\alpha^{(i)}$$ (3.10)

The iterative procedure consists of the following steps, which are repeated until convergence.

compute residual: \[ r(u^{(i-1)}) = \alpha^{(i-1)} \hat{f}_e - f_i(u^{(i-1)}) \] (3.11a)

solve iterative update due to residual: \[ K(u^{(i-1)})d\hat{u}^{(i)} = r(u^{(i-1)}) \] (3.11b)

solve iterative update due to unit force: \[ K(u^{(i-1)})d\bar{u}^{(i)} = \bar{f}_e \] (3.11c)

solve constraint equation: \[ \|Du^{(i)}\| = L \rightarrow c_1d\alpha^{(i)} + c_2d\alpha^{(i)} + c_3 = 0 \] (3.11d)

update solution: \[ u^{(i)} = u^{(i-1)} + du^{(i)} = u^{(c)} + Du^{(i)} \] (3.11e)

The parameters $c_1$, $c_2$ and $c_3$ in the constraint equation depend on the iterative update and the specified arc-length $L$.

Figure 3.6 illustrates the arc-length method and illustrates how this method is able to pass limit points. However, if these limit points become too sharp the method suffers from convergence problems. Sharp limit points for example occur in case of strong deformation localization. This localization causes problems in the arc-length method due to the improper updating of the iterative load factor $d\alpha$. The global deformation vector $d\hat{u}$ controls this updating procedure. In case of localization, most elements show small deformation which mask the large deformation in the zones where the actual deformation takes place. In other words, the load factor updating procedure is insensitive to localized deformation. Therefore, the procedure needs to be further adjusted.
3.3.3 Weighted subplane control

The weighted subplane control technique adds more weight to the localized deformation. The update process of the iterative load update $d\alpha$ in Equation (3.11d) changes substantially. However, the details are far beyond the scope of this project and are therefore not discussed further. The weighted subplane control enables the procedure to pass sharper limit points. The simulation discussed in the next chapter have all been obtained using weighted subplane control.
Chapter 4

Simulations with Cohesive Zone Modeling

4.1 FE model

Figure 4.1 shows the mesh that is used to simulate crack propagation along a bimaterial interface. The model size and mesh size are identical to the model used in the linear elastic analysis in Chapter 2. Also the same boundary condition are imposed as seen in Figure 2.7. Cohesive zone elements are placed along the bimaterial interface. These cohesive zones are not visible in Figure 4.1 because they are initially collapsed.

![Undeformed Mesh](image)

Figure 4.1: Undeformed Mesh
In chapter 2 the model size has been changed to reduce computing time. In a linear elastic analysis computing time is rather low and is thus of minor importance. However, the analysis now becomes non-linear, which is accompanied by a huge increase of computational cost. These computational costs are kept within reasonable limits by reducing the overall length of the strip.

The reduced strip length has also another implication, namely the reduction of the deformation localization. This can be illustrated by the spring system in Figure 4.2. When the length of the strip is reduced, the stiffness increases. In analogy, this results in a steeper slope in the mechanical behavior of the linear spring as illustrated. The steeper slope causes the limit points to become less sharp as illustrated in Figure 4.2. In other words, the overall behavior of the system becomes less brittle. This has a positive effect on the solution procedure because the localization effect reduces.

Figure 4.2: Reducing length of the model
4.2 Results

Figure 4.3 shows the deformation of the mesh at different stages of the crack propagation. The deformations are exaggerated for viewing purposes. The model is subjected to the same load as in the analytical analysis, $\sigma_\infty = 100$ Mpa. The bimaterial already fails however, before this load is reached.

Figure 4.3: (1): $\lambda \sigma_\infty = 10.9$ MPa  (2): $\lambda \sigma_\infty = 9.8$ MPa  (3): $\lambda \sigma_\infty = 8.0$ MPa  (4): $\lambda \sigma_\infty = 7.0$ MPa
Figure 4.4 plots the force-displacement response \((\lambda \sigma_{\infty} - u_y)\) for the top left node. The figure clearly illustrates the snap-back behavior. The oscillations in the softening branch are caused by a too coarse mesh discretization. Each cycle corresponds to the failure of a single integration point along the interface.

Figure 4.4: Load-Displacement behavior of top left node

Figure 4.5(a) shows the COD\(_y\) at the different stages of the crack propagation. When a snapshot is taken every time an element starts to break, the shape of the crack stays the same. Also the stress field around the crack tip shifts during the crack propagation. The shape of the stress field however remains constant in the subsequent snapshots. These stress fields are shown in Figure 4.5(b)
A function is fitted onto the numerical data. This function is a function of the distance to the crack tip \( r \). For this purpose, the data is shifted so that the crack tip is located at \( r = 0 \). This shifting is rather arbitrary because the crack tip cannot be pointed out exactly. Therefore in the remaining analysis, multiple plots are analyzed with different shifting.
4.3 COD approximation function

4.3.1 Fitting Method

A simple function has to be fitted onto the FEM data of the Crack Opening Displacement as a function of the distance to the crack tip. The Least Squares Fitting method is used for this purpose, which is implemented in Matlab’s Curve Fitting toolbox. This method minimizes the sum of the squares of the errors between the data points $COD_{yi}$ and the fitting curve $COD_y(r_i)$ where $i$ denotes the node number. The Summed Square Error (SSE) is a measure for the accuracy of the fit and is defined as

$$SSE = \sum_{i=1}^{n} (COD_{yi} - COD_y(r_i))^2$$  \hspace{1cm} (4.1)

A disadvantage of this method is that large errors have a relative high influence on the outcome. Therefore, attention should be paid to the range of points that is used in the fitting process. Figure 2.11 shows that the analytical solution begins to differ from the numerical solution some 15 mm away from the crack tip. Because the shape of the crack near the crack tip is required, only the data points in the range till 15 mm from the crack tip are used. In case more nodes are used, the shape of the fitted curve cannot reflect the shape near the crack tip.

4.3.2 Multiple COD approximation functions

The following functions are fitted through the numerical data with the degrees of freedom (DOF’s) a, b and c

$$COD_y = ar$$  \hspace{1cm} (4.2a)
$$COD_y = a\sqrt{r}$$  \hspace{1cm} (4.2b)
$$COD_y = ar^2 + br$$  \hspace{1cm} (4.2c)
$$COD_y = a\sin(br)$$  \hspace{1cm} (4.2d)
$$COD_y = ar^3 + br^2 + cr$$  \hspace{1cm} (4.2e)

The linear function of Equation (4.2a) is currently used to approximate the shape of the crack. The fitting functions become increasingly complex. Obviously, an extremely complex function with a lot of degrees of freedom can be fitted onto the shape of the crack tip with very high accuracy. However, such a function would not be of any use for implementation in the shape function of a cohesive zone element. Therefore, a simple function is sought with an as small number of degrees of freedom as possible.
4.3.3 Fitting results

Figure 4.6 shows results of the fitting process. The black data points are used for fitting, the remaining data points are grayed out.

![Figure 4.6: Multiple fittings onto the numerical FEM data](image)

Table 4.1 shows the SSE for every fit and the number of DOF that are used. The most suited COD approximation function is chosen based on this data.

<table>
<thead>
<tr>
<th>Fitting function</th>
<th>Number of DOF’s</th>
<th>SSE $[mm^2]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$COD_y = ar$</td>
<td>1</td>
<td>6.807e-5</td>
</tr>
<tr>
<td>$COD_y = a\sqrt{r}$</td>
<td>1</td>
<td>3.32e-6</td>
</tr>
<tr>
<td>$COD_y = ar^2 + br^2$</td>
<td>2</td>
<td>6.55e-6</td>
</tr>
<tr>
<td>$COD_y = asin(br)$</td>
<td>2</td>
<td>1.177e-5</td>
</tr>
<tr>
<td>$COD_y = ar^3 + br^2 + cr$</td>
<td>3</td>
<td>1.634e-6</td>
</tr>
</tbody>
</table>

Table 4.1: Fitting characteristics

The function must reach a high accuracy with as few DOF’s as possible. The square root function of Equation (4.2b) gives good results on those criteria. The accuracy of the square root fit is significantly better than the accuracy of the linear fit. Only the third order polynomial of Equation (4.2e) has a higher accuracy. This function has 2 DOF’s more and therefore becomes less suitable. For these reasons, the square root fit is chosen to be best suitable for modeling the shape of the crack tip.
Chapter 5

Conclusions

The comparison between the analytical solutions and FEM simulations of a crack in a homogeneous material gives good results. The stress field shows the familiar dog bone shape. The boundary effects are of minor importance. This comparison indicates the range in which the analytical solution is valid.

The comparison between the analytical solution and FEM simulations of a crack along a bimaterial interface gives acceptable results. There is no geometry correction factor stated in literature to account for the width of the strip. This factor is of minor importance however, because it is relatively small as has been observed in the analysis of the homogeneous material. The crack becomes asymmetrical and the dog bone shape of the stress contours is distorted due to the different material properties of the layers.

Cohesive Zone Modeling is introduced to simulate crack propagation. Cohesive zones have a non-linear constitutive relation which accounts for the material softening during cracking. This non-linear behavior causes difficulties in the solution procedure. To overcome these difficulties, Crisfield’s cylindrical arc-length solution procedure with weighted subplane control has been used.

The shape of the crack tip along a bimaterial interface is examined using a FE model with cohesive zone elements. A function is fitted onto the data of the displacement field. The geometry of the crack tip exhibits a square root dependency with respect to the distance to the crack tip

$$COD_y = a\sqrt{r}, \quad (5.1)$$

with a scaling parameter $a$. This square root dependency models the shape of the crack tip with high accuracy and few degrees of freedom. In the future, this knowledge of the shape of the crack tip can be implemented into the shape function of the cohesive elements.
Bibliography


Appendix A

Geometry correction factor of a bimaterial

The correction factor $\hat{c}$ for the correction of the strip width for bimaterials is not stated in literature. Efforts are made to obtain this parameter by fitting the analytical solution onto the FEM data. First, a valid strategy to obtain this correction factor is required. Therefore, first the correction factor $c$ of a homogeneous material, which is known theoretically, will be fitted to the FEM data. The obtained correction factor is compared to the analytical one.

To obtain the c factor for the bulk material the following function will be fitted through the FEM data

$$\text{COD}_y = \frac{cK_I}{G} \sqrt{\frac{r}{2\pi}}(\kappa + 1) \quad \text{(A.1)}$$

The fits are made using a least squares fitting method with $c$ as degree of freedom. In this fitting process not all data points of the FEM data are used because the analytical solution is only valid near the crack tip. Only the nodes that lie in a range of 15 mm from the crack tip are used in the fitting process. This Least Square fit results in a correction factor $c$ of 0.980. The value of the analytical solution of the correction factor is 1.017. This result is disappointing. Apparently, the FEM data has not yet fully converged to the exact solution. To obtain a better approximation of the correction factor, further mesh refinement is required. The exact solution will never be reached due to the stress singularity. For this reason it is concluded that this method is not suitable to obtain the correction factor for a bimaterial.