INTERPHASE ELEMENTS CONNECTING STRUCTURAL FINITE ELEMENTS
-FORMULATION, IMPLEMENTATION AND VERIFICATION

By
Kent Salomonsson

SUBMITTED IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE OF
MASTER OF SCIENCE IN ENGINEERING PHYSICS
AT
CHALMERS UNIVERSITY OF TECHNOLOGY
GOTHENBURG, SWEDEN
DECEMBER 2002

© Copyright by Kent Salomonsson, 2002
The undersigned hereby certify that they have read and recommend to the Faculty of Engineering Physics for acceptance a thesis entitled “Interphase elements connecting structural finite elements -formulation, implementation and verification” by Kent Salomonsson in partial fulfillment of the requirements for the degree of Master of Science in Engineering Physics.

Dated: December 2002

Supervisor: ________________________________
Ragnar Larsson

Readers: ________________________________
Ulf Stigh

Svante Alfredsson
Author: Kent Salomonsson
Title: Interphase elements connecting structural finite elements - formulation, implementation and verification

Department: Engineering Physics
Degree: M.Sc Convocation: June Year: 2002

Permission is herewith granted to Chalmers University of Technology to circulate and to have copied for non-commercial purposes, at its discretion, the above title upon the request of individuals or institutions.

Signature of Author

THE AUTHOR RESERVES OTHER PUBLICATION RIGHTS, AND NEITHER THE THESIS NOR EXTENSIVE EXTRACTS FROM IT MAY BE PRINTED OR OTHERWISE REPRODUCED WITHOUT THE AUTHOR'S WRITTEN PERMISSION.

THE AUTHOR ATTESTS THAT PERMISSION HAS BEEN OBTAINED FOR THE USE OF ANY COPYRIGHTED MATERIAL APPEARING IN THIS THESIS (OTHER THAN BRIEF EXCERPTS REQUIRING ONLY PROPER ACKNOWLEDGEMENT IN SCHOLARLY WRITING) AND THAT ALL SUCH USE IS CLEARLY ACKNOWLEDGED.
To Mia.
Abstract

Starting from a structural mechanics approach, two interphase elements are created to model adhesive joints: The line structural element, to be used with Timoshenko beam elements and the surface structural elements, to be used with Mindlin shell/plate elements. An implementation of these elements into a FORTRAN code is presented as well as an implementation of the elements into ABAQUS.

A number of tests are made using the interphase elements to test their performance. First the double cantilever beam test (DCB), using both the line structural element and the surface structural element. In the DCB-test the height of the bottom adherend is varied when using the line structural element; the width of the adherends is varied when using the surface structural element. Secondly, the end notched flexure (ENF) test is analyzed with the line structural element with a varying initial crack length. The final test is the single lap joint (SLJ) specimen. This is analyzed both with the line structural element and with the surface structural element. When using the line structural element, the bond length is varied. The bond length is kept constant when using the surface structural element and instead, the width is varied.

The results of these tests are verified by comparing with analytical beam solutions. Stress distributions concerning the three dimensional analysis of the SLJ-test is verified by comparing with other finite element solutions.

**Keywords:** FEM; Adhesive; Interphase; Interface; DCB; ENF; SLJ;
Acknowledgements

First of all I would like to thank my examiner Ragnar Larsson for making it possible to complete my thesis in an area I find most interesting.

I would also like to extend my deepest gratitude to Ulf Stigh and Svante Alfredsson for all the jokes, small talks and for their ability to explain advanced theory on a simple level.

I would like to thank all the graduate students at the Department of Engineering Science at the University of Skövde for all the laughs.

Finally, a gigantic ”Thank YOU!!!” to my girlfriend Mia for her patience over the past five years.
# Table of Contents

Abstract ........................................ v
Acknowledgements ................................ vi

1 Introduction ................................... 9

2 Theory ........................................ 11
   2.1 Structural mechanics approach ............... 11
   2.2 Interface formulation ............................ 12
   2.3 Weak formulation ................................. 15
   2.4 Finite Element Formulation ..................... 16
       2.4.1 Line structural element (4 nodes, linear) ... 19
       2.4.2 Surface structural element (8 nodes, linear) 22
   2.5 Transformation ................................ 24
       2.5.1 Line structural element ....................... 26
       2.5.2 Surface structural element .................... 28
       2.5.3 Summary of transformation .................... 32

3 Implementation .................................. 33
   3.1 Fortran ........................................ 33
   3.2 Abaqus ......................................... 35
       3.2.1 Line structural element ....................... 36
       3.2.2 Surface structural element .................... 38

4 Analysis and Verification ...................... 41
   4.1 Double Cantilever Beam (DCB) specimen .......... 41
       4.1.1 Line structural element for 2D analysis .......... 42
       4.1.2 Surface structural element for 3D analysis ...... 45
   4.2 End Notched Flexure (ENF) specimen ............... 48
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.2.1</td>
<td>Line structural element for 2D analysis</td>
<td>49</td>
</tr>
<tr>
<td>4.3</td>
<td>Single Lap Joint (SLJ) specimen</td>
<td>56</td>
</tr>
<tr>
<td>4.3.1</td>
<td>Line structural element for 2D analysis</td>
<td>56</td>
</tr>
<tr>
<td>4.3.2</td>
<td>Surface structural element for 3D analysis</td>
<td>62</td>
</tr>
</tbody>
</table>

### 5 Conclusions and future work

### Bibliography

Page 72
Chapter 1

Introduction

Adhesive joining is one of the main contenders among technological methods to connect substrates into structures. As an example the automotive industry today welds their chassis. This is where adhesive joining becomes a strong contender towards welding. Particularly when combining spot welding and adhesive joining the stiffness of the body will increase substantially.

Upon being a strong contender towards welding, research on the mechanical properties of adhesives is needed. Questions like ”How does the adhesive behave when it is exerted to a specific load?” and ”How does one find these properties?” are to be answered. The most trivial method to get answers to these questions would be to perform tests on a prototype. A different method is to simulate the load case using the finite element method (FEM). This method saves time and at the same time the costs are lowered.

A lot of effort has been made to model adhesive joints and the constitutive laws that govern them. One of the first to study adhesive joints was Goland and Reissner [7]. They presented closed-form solutions for elastic models of single- and double-lap joints of unit width assuming plane strain conditions.
When numerical methods like the finite element method were available, more complex adhesive joints could be modelled. Carpenter [5] proposed an element for the analysis of adhesive joints of unit width. Using different parameters he could switch between a number of adhesive models. Because of the complicated behavior of certain adhesives, nonlinear elements were created. Reddy and Roy [11] are among the first to study adhesive joints using nonlinear finite elements.

The research on adhesive joints progressed and interface finite elements were created for adhesive layers (see Stigh [13]). He proposed that interface elements could be used to model the adhesive joints, connecting solid structures. The interface elements were used to lower the number of elements in the structure. When using interface elements, a discontinuity in the displacement field appears over the adhesive layer. This problem can be solved using regularized strong discontinuity (see Jansson and Larsson [9]).

Even though a lot of elements for adhesive joints have been developed over the years, they are not available in any of the commercial finite element analysis programs.

In order to find the properties of the adhesive, there are different standard methods available. The DCB-test (Double Cantilever Beam), the SLJ-test (Single Lap Joint) and the ENF-test (Edge Notched Flexure) are some of these methods. In the SLJ-test, both peel and shear stresses appear. If only shear stresses are to be analyzed, the obvious choice would be the ENF-test e.g Chai [6]. For peel stresses the choice would be the DCB-test e.g Stigh and Andersson [12]. These test geometries are analyzed within the present thesis. The results are verified by comparing with analytical solutions provided by Alfredsson and Högberg [2].
2.1 Structural mechanics approach

The adhesive is considered to be a thin weak layer. Klarbring [10] assumes a linear displacement field in the adhesive layer and concludes that the dominating deformation modes are modes I and II, see figure 2.1.

Figure 2.1: Deformation modes of an infinitesimal adhesive element.

Mode I corresponds to uniaxial straining with peel stress, $\sigma$, and mode II corresponds to pure shear with shear stress, $\tau$. 
In this thesis the peel and shear stresses are modelled linearly according to

\[ \sigma = \frac{\bar{E}}{t} w \quad \tau = \frac{G}{t} v \]

where \( \bar{E} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \) is the effective Young’s modulus, \( t \) is the thickness of the adhesive layer, \( w \) is the vertical relative displacement, \( G \) is the shear modulus and \( v \) is the longitudinal relative displacement.

The in-plane stress in the adhesive layer is given by

\[ \sigma_t = \frac{\nu}{1-\nu} \sigma \]

Due to the fact that the adhesive has an extent in the vertical direction, there are stresses present in the longitudinal direction as well as in the vertical direction to get equilibrium. Because of this, the boundary conditions specifying the in-plane and shear stresses are inaccurate at the free edge of the adhesive layer. One way to discard this inaccuracy is to neglect the thickness of the adhesive, while letting it have some stiffness corresponding to the thickness. This gives a displacement discontinuity over the adhesive layer which can, mathematically, be handled as a regularized strong discontinuity, see Jansson and Larsson [9].

With the displacement discontinuity problem solved, it is straightforward to use this methodology when generating interface elements. The interface elements can then be connected to finite structural elements such as beam and shell elements, which completes the structural mechanics approach.

2.2 Interface formulation

Consider an adhesive joint as depicted in figure 2.2. In order to analyze this joint using the finite element method one has to define the controlling differential equations. In the adherends the divergence of the stress tensor is zero, due to the fact that no
Figure 2.2: A part of an adhesive joint specimen. Adherend $(r)$ is denoted $V^{(r)}$ and the adhesive layer (the interface) is denoted $V^{(I)}$.

Internal (body) forces are present. From now on $r=1,2$ indicates adherend $r$,

$$\sigma_{ij,\cdot j}^{(r)} = 0 \quad \text{in } V^{(r)} \quad (2.2.1)$$

Boundary conditions are specified on the interface boundary,

$$\sigma_{ij}^{(r)} \cdot n = \bar{t}_i^{(r)} \quad \text{on } S_t^{(r)} \quad (2.2.2)$$

$$u_i = \bar{u}_i \quad \text{on } S_u^{(r)} \quad (2.2.3)$$

where $S_u$ is the boundary with prescribed displacements and $S_t$ is the boundary with prescribed traction. Interface conditions are also necessary to specify. These are

$$t^{(1)}_i = \sigma_{ij}^{(1)} \cdot n^{(1)}_j = -p_i \quad \text{on } S^{(I)} \quad (2.2.4)$$

$$t^{(2)}_i = \sigma_{ij}^{(2)} \cdot n^{(2)}_j = +p_i \quad (2.2.5)$$
Here $S^{(I)}$ is the adhesive interface boundary and $p_i$ corresponds to the traction on the adherend boundaries due to Newton’s third law (see figure 2.3).

Once the boundary and interface conditions are specified, it is necessary to introduce the constitutive relations describing the material behavior during load.

\[
\sigma_{ij}^{(r)} = \sigma_{ij}^{(r)}(\epsilon) \tag{2.2.6}
\]

\[
p_i = p_i(w) = p_i(w_j) = p_i(u_j^{(1)} - u_j^{(2)}) \tag{2.2.7}
\]

Where $w_i$ is defined as $w_i = u_i^{(1)} - u_i^{(2)}$. As can be seen in the constitutive relations Eqs. (2.1.6) and (2.1.7) the stress tensor in each adherend is a function of the strain and the stress vector, $p_i$, acting on the adhesive layer is a function of the relative displacement, $w_j$. Due to the fact that the stress vector is a function of the relative displacement, an approximative relation between these variables have to be used. These approximations originates from experimental data. In this thesis, only linear elastic relations are studied.
2.3 Weak formulation

To obtain the weak form of a differential equation it is customary to multiply the equilibrium equation with a weight function and then integrate over the surface (2D) or volume (3D). The weight functions \( v_i \) satisfy the essential boundary conditions on \( S_u^{(r)} \). The governing space for the weight functions is the space of all continuous and piecewise differentiable functions. These functions are also denoted virtual displacements in the thesis.

Eq. (2.2.1) multiplied with \( v_i \) and integrating by parts yields

\[
\int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_i dV = \int_{V^{(r)}} (\sigma_{ij}^{(r)} \cdot v_i)_j dV - \int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV
\]

\[
= \int_{\partial V^{(r)}} \sigma_{ij}^{(r)} \cdot v_i \cdot n_j dS - \int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV
\]

\[
= \int_{V^{(r)}} t_i^{(r)} \cdot v_i dV - \int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV \quad (2.3.1)
\]

Now the weak formulation implies \( \int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV = 0 \), which yields

\[
\int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV = \int_{\partial V^{(r)}} t_i^{(r)} \cdot v_i dS \quad (2.3.2)
\]

Use of the boundary and interface conditions and summation over the two adherends \( r=1,2 \) yields

\[
\sum_{r=1}^{2} \int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV = \sum_{r=1}^{2} \int_{S^{(r)}} t_i^{(r)} \cdot v_i dS - \int_{S^{(t)}} p_i(v_i^{(1)} - v_i^{(2)}) dS
\]

where \( v_i^{(r)} \) is the virtual displacements value at the interface between adherend \( r \) and the adhesive.

Now form the residual which is given by

\[
R = \sum_{r} \int_{S^{(r)}} t_i^{(r)} \cdot v_i dS - \int_{V^{(r)}} \sigma_{ij}^{(r)} \cdot v_{ij} dV - \int_{S^{(t)}} p_i(v_i^{(1)} - v_i^{(2)}) dS \quad (2.3.3)
\]

Changing from tensor notation to matrix notation the contribution from the adhesive is

\[
R^{(t)} = -\int_{S^{(t)}} (v^{(1)} - v^{(2)})^T \cdot p dS \quad (2.3.4)
\]
where \( \mathbf{u}^{(1)} - \mathbf{u}^{(2)} \) is the virtual relative displacement and \( p = p(w) \) is the stress vector which is a function of the relative displacement \( w = \mathbf{u}^{(1)} - \mathbf{u}^{(2)} \), cf Eq. (2.2.7).

### 2.4 Finite Element Formulation

The displacements on the interface surfaces, \( \mathbf{u}^{(r)} \), can be described using the local degrees of freedom \( \nu^{(r)} \). These local degrees of freedom are depicted in figure 2.4 for two points, one in each adherend. Keeping in mind that figure 2.4 illustrates the local degrees of freedom for a line structural element to be used in a 2D analysis, this obviously expands to a surface structural element to be used in a 3D analysis. As an example, there are six active degrees of freedom in each node in the 3D case, but only three in the 2D case. The rotation is assumed to be small e.g. \( |\nu_3^{(r)}| << 1 \).

![Figure 2.4](image-url)

Figure 2.4: The local degrees of freedom \( \nu_i^{(r)} \) for a line structural element (2D analysis).
The displacements of points on the interfaces can be calculated by multiplying the local degrees of freedom with a metric matrix $A^{(r)}$.

\[ u^{(r)} = A^{(r)} \cdot \nu^{(r)} \] (2.4.1)

Due to the fact that $\nu^{(r)}$ are the local degrees of freedom they are interpolated from the nodal degrees of freedom $\tilde{a}^{(r)}$,

\[ \nu^{(r)} = N^{(r)} \cdot \tilde{a}^{(r)} \] (2.4.2)

where $N^{(r)}$ are the shape functions for adherend $r$.

Inserting this into Eq. (2.4.1), the displacements at the interface surfaces can be calculated as

\[ u^{(r)} = A^{(r)} N^{(r)} \cdot \tilde{a}^{(r)} \] (2.4.3)

Now, the weight functions, $\psi^{(r)}$, are chosen according to the Galerkin method,

\[ \psi^{(r)} = A^{(r)} N^{(r)} \cdot \zeta^{(r)} \] (2.4.4)

where the constants $\zeta^{(r)}$ are chosen arbitrarily.

Using Eqs. (2.4.3) and (2.4.4), one can write the relative displacement vectors as follows

\[ w = u^{(1)} - u^{(2)} = A^{(1)} N^{(1)} \cdot \tilde{a}^{(1)} - A^{(2)} N^{(2)} \cdot \tilde{a}^{(2)} = \left( A^{(1)} N^{(1)} - A^{(2)} N^{(2)} \right) \cdot \begin{pmatrix} q^{(1)} \\ q^{(2)} \end{pmatrix} \]

Similarly for the weight function difference

\[ \psi^{(1)} - \psi^{(2)} = A^{(1)} N^{(1)} \cdot \zeta^{(1)} - A^{(2)} N^{(2)} \cdot \zeta^{(2)} = \left( A^{(1)} N^{(1)} - A^{(2)} N^{(2)} \right) \cdot \begin{pmatrix} \zeta^{(1)} \\ \zeta^{(2)} \end{pmatrix} \]
Introducing the matrix
\[
\mathcal{G} = \begin{pmatrix}
\mathcal{A}^{(1)} \mathcal{N}^{(1)} & -\mathcal{A}^{(2)} \mathcal{N}^{(2)}
\end{pmatrix}
\] (2.4.5)
we may thus write
\[
w = \mathcal{u}^{(1)} - \mathcal{u}^{(2)} = \mathcal{G} \cdot \mathcal{q} \quad , \quad \mathcal{v}^{(1)} - \mathcal{v}^{(2)} = \mathcal{G} \cdot \mathcal{c}
\] (2.4.6)
where
\[
\mathcal{q} = \begin{pmatrix}
\mathcal{q}^{(1)} \\
\mathcal{q}^{(2)}
\end{pmatrix} \quad , \quad \mathcal{c} = \begin{pmatrix}
\mathcal{c}^{(1)} \\
\mathcal{c}^{(2)}
\end{pmatrix}
\] (2.4.7)
Inserting the weight function difference result into Eq. (2.3.4) gives
\[
R^{(I)} = -\int (\mathcal{G} \mathcal{c})^T \cdot \mathcal{p} \cdot dS = -\mathcal{c}^T \int \mathcal{G}^T \mathcal{p} \cdot dS
\] (2.4.8)
Thus, the residual can be written as, \(R^{(I)} = -\mathcal{c}^T \cdot \mathcal{F}^{(I)}\), where the nodal force vector is given by
\[
\mathcal{F}^{(I)} = \begin{pmatrix}
\mathcal{F}^{(1)} \\
\mathcal{F}^{(2)}
\end{pmatrix} = \int_{S^{(I)}} \mathcal{G}^T \mathcal{p} \cdot dS
\] (2.4.9)
Here, the internal adhesive forces acting on the nodes on the two adherends are given by
\[
\mathcal{F}^{(1)} = \int (\mathcal{A} \mathcal{N})^{(1)}^T \cdot \mathcal{p} \cdot dS \quad \mathcal{F}^{(2)} = -\int (\mathcal{A} \mathcal{N})^{(2)}^T \cdot \mathcal{p} \cdot dS
\] (2.4.10)
The tangent stiffness is described by
\[
\mathcal{K}_t = \frac{\partial}{\partial \mathcal{q}} \mathcal{F}^{(I)} = \frac{\partial}{\partial \mathcal{q}} \int_{S^{(I)}} \mathcal{G}^T \mathcal{p} \cdot dS = \int_{S^{(I)}} \mathcal{G}^T \left( \frac{\partial}{\partial w} \mathcal{p} \right) \left( \frac{\partial}{\partial \mathcal{q}} w \right) \cdot dS
\] (2.4.11)
Introducing the stiffness matrix on the material level,
\[
\mathcal{D} = \frac{\partial}{\partial \mathcal{w}} \mathcal{p}
\] (2.4.12)
and using Eq. (2.4.6), the tangent stiffness takes the form,

\[
\approx_{t} K = \int_{S(t)} \approx_{t} G^{T} \approx_{t} D \approx_{t} G \cdot dS 
\]  

(2.4.13)

To simplify the calculations the tangent stiffness matrix is divided into blocks

\[
\approx_{t} K = \begin{pmatrix}
\approx_{(11)} K & \approx_{(12)} K \\
-\approx_{(21)} K & \approx_{(22)} K
\end{pmatrix}
\]  

(2.4.14)

where \( \approx_{(ij)} K = \int_{S} \approx_{(ij)} C \cdot dS \) and

\[
\approx_{(ij)} C = \left( \approx_{(ij)} A, \approx_{(ij)} N \right)^{T} \cdot \approx D \cdot \left( \approx_{(j)} A, \approx_{(j)} N \right)
\]  

(2.4.15)

In these calculations, the indices (ij) within parenthesis define the sub-matrix ij. This is not to confuse with the component ij of a matrix. Until otherwise specified, when indices appears within a parenthesis in subscript notation it defines a sub-matrix.

All of these calculations are done in the elements local system. Section 2.5 describes how to transform the elements from the global system to the local system and vice versa.

### 2.4.1 Line structural element (4 nodes, linear)

For a line structural element, \( \nu^{(r)} \) contains the displacements of the mean axis of the adherend and the rotation of the adherend cross section, see figure 2.4.

The metric matrix in Eq. (2.4.1) is given by

\[
\approx A^{(r)} = \begin{pmatrix}
1 & 0 & \frac{H(r)}{2} \\
0 & 1 & 0
\end{pmatrix}
\]  

(2.4.16)
where $H_{(1)} = +H_1$ and $H_{(2)} = -H_2$ are the heights of the adherends 1 and 2. The interface displacements for a line structural element are thus

$$
\begin{pmatrix}
u_1^{(r)} \\ \nu_2^{(r)}
\end{pmatrix} = \begin{pmatrix} 1 & 0 & H_{(r)} / 2 \\ 0 & 1 & 0 \end{pmatrix} \begin{pmatrix} u_1^{(r)} \\ u_2^{(r)} \\ \nu_3^{(r)}
\end{pmatrix}
$$

In order to generate the nodal force vector $\tilde{F}^{(J)}$ and the tangent stiffness matrix $\tilde{K}_t$ for the line structural elements, shape functions based on 2-node Timoshenko beam elements are used. The deflection and the cross sectional rotation is interpolated independently of each other and the shape function matrix takes the form,

$$N^{(r)} = \begin{pmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 \end{pmatrix}, \quad r = 1, 2 \quad (2.4.17)$$

where $N_1 = \frac{1-\xi}{2}$ and $N_2 = \frac{1+\xi}{2}$. Having defined the metric matrix, $A^{(r)}$, and the shape functions, $N^{(r)}$, we can construct $\tilde{C}^{(ij)}$ of Eq. (2.4.15), for the line structural elements,

$$\tilde{C}^{(ij)} = \begin{pmatrix} N_2^2 \cdot E \approx (ij) & N_1 N_2 \cdot E \approx (ij) \\ N_2 N_1 \cdot E \approx (ij) & N_2^2 \cdot E \approx (ij) \end{pmatrix} \quad (2.4.18)$$

where

$$E \approx (ij) = A^{T} \cdot D \cdot A^{(j)} \quad (2.4.19)$$

Let the constitutive matrix $D$ be represented by

$$D = \begin{pmatrix} D_{11} & D_{12} \\ D_{21} & D_{22} \end{pmatrix} \quad (2.4.20)$$

For a linear elastic adhesive

$$D_{11} = G, \quad D_{22} = \bar{E}, \quad D_{12} = D_{21} = 0$$
and the four matrices \( \approx \) takes the form

\[
E_{11} = \begin{pmatrix}
D_{11} & D_{12} & \frac{H_2}{2} D_{11} \\
D_{21} & D_{22} & \frac{H_2}{2} D_{21} \\
\frac{H_2}{2} D_{11} & \frac{H_2}{2} D_{12} & (\frac{H_2}{2})^2 D_{11}
\end{pmatrix}
\]

\[
E_{12} = \begin{pmatrix}
D_{11} & D_{12} & -\frac{H_2}{2} D_{11} \\
D_{21} & D_{22} & -\frac{H_2}{2} D_{21} \\
\frac{H_2}{2} D_{11} & \frac{H_2}{2} D_{12} & -\frac{H_2}{2} \frac{H_2}{2} D_{11}
\end{pmatrix}
\]

\[
E_{21} = \begin{pmatrix}
D_{11} & D_{12} & \frac{H_2}{2} D_{11} \\
D_{21} & D_{22} & \frac{H_2}{2} D_{21} \\
-\frac{H_2}{2} D_{11} & -\frac{H_2}{2} D_{12} & -\frac{H_2}{2} \frac{H_2}{2} D_{11}
\end{pmatrix}
\]

\[
E_{22} = \begin{pmatrix}
D_{11} & D_{12} & -\frac{H_2}{2} D_{11} \\
D_{21} & D_{22} & -\frac{H_2}{2} D_{21} \\
-\frac{H_2}{2} D_{11} & -\frac{H_2}{2} D_{12} & -\frac{H_2}{2} \frac{H_2}{2} D_{11}
\end{pmatrix}
\]

The nodal forces, \( \tilde{F}^{(1)} \) and \( \tilde{F}^{(2)} \) cf Eq. (2.4.10), for the line structural element takes the form

\[
\tilde{F}^{(1)} = \int_{-1}^{1} \begin{pmatrix}
b \cdot J \cdot N_1 \cdot p_1 \\
b \cdot J \cdot N_1 \cdot p_2 \\
b \cdot J \cdot N_1 \cdot H_1/2 \cdot p_1 \\
b \cdot J \cdot N_2 \cdot p_1 \\
b \cdot J \cdot N_2 \cdot p_2 \\
b \cdot J \cdot N_2 \cdot H_1/2 \cdot p_1
\end{pmatrix} \cdot d\xi
\]

\[
\tilde{F}^{(2)} = \int_{-1}^{1} \begin{pmatrix}
-b \cdot J \cdot N_1 \cdot p_1 \\
-b \cdot J \cdot N_1 \cdot p_2 \\
-b \cdot J \cdot N_1 \cdot H_2/2 \cdot p_1 \\
-b \cdot J \cdot N_2 \cdot p_1 \\
-b \cdot J \cdot N_2 \cdot p_2 \\
-b \cdot J \cdot N_2 \cdot H_2/2 \cdot p_1
\end{pmatrix} \cdot d\xi
\]

where \( b \) is the width of the adherends and \( J \) is the jacobian. These nodal forces are integrated using 2 point Gaussian integration in one dimension. The weight function \( W(I) = 1 \) for the integration points \( I = 1, 2 \) are also used when integrating. For the line structural elements the jacobian is \( J = h/2 \), where \( h \) is the element length.

The evaluation of the stiffness matrices \( \approx \) is done by integrating

\[
K_{ij} \approx \int_{-1}^{1} \begin{pmatrix} C_{ij} (\xi) \cdot b \cdot J \cdot d\xi
\end{pmatrix}
\]

This integral can be solved exactly for the linear case, but preparation has been made for nonlinear cases.
2.4.2 Surface structural element (8 nodes, linear)

In this thesis, only planar elements are used to model surface structures and only non-distorted elements are used.

An analysis with a surface structural element obviously have displacements and rotations in the x,y and z directions; thus six local degrees of freedom in each node. This will generate a 3 by 6 metric matrix \( \mathbf{A}^{(r)} \) given by

\[
\mathbf{A}^{(r)} = \begin{pmatrix}
1 & 0 & 0 & 0 & 0 & \frac{H(r)}{2} \\
0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & -\frac{H(r)}{2} & 0 & 0
\end{pmatrix}
\]  

(2.4.22)

corresponding to the 2 by 3 metric matrix for the line structural elements and where \( H(1) = +H_1 \) and \( H(2) = -H_2 \) are the thicknesses of the adherends 1 and 2. The interface displacements can now be calculated in the same manner as for the line structural elements

\[
\begin{pmatrix}
u_1^{(r)} \\
u_2^{(r)} \\
u_3^{(r)}
\end{pmatrix} = \begin{pmatrix}
1 & 0 & 0 & 0 & 0 & \frac{H(r)}{2} \\
0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & -\frac{H(r)}{2} & 0 & 0
\end{pmatrix} \begin{pmatrix}
u_1^{(r)} \\
u_2^{(r)} \\
u_3^{(r)} \\
u_4^{(r)} \\
u_5^{(r)} \\
u_6^{(r)}
\end{pmatrix}
\]

Shape functions, based on 4-node Mindlin shell elements are used. The deflection and the rotation is interpolated independently of each other and the shape function matrix for adherend \( r \) then takes the form,

\[
\mathbf{N}^{(r)} = \begin{pmatrix}
N_1 \cdot \mathbf{I} \\
N_2 \cdot \mathbf{I} \\
N_3 \cdot \mathbf{I} \\
N_4 \cdot \mathbf{I}
\end{pmatrix}, \quad r = 1, 2
\]  

(2.4.23)

where \( \mathbf{I} \) is a 6 by 6 unity matrix, and where

\[
N_1 = \frac{(1 - \xi)(1 - \eta)}{4} , \quad N_2 = \frac{(1 + \xi)(1 - \eta)}{4} \\
N_3 = \frac{(1 + \xi)(1 + \eta)}{4} , \quad N_4 = \frac{(1 - \xi)(1 + \eta)}{4}
\]  

(2.4.24)
From Eq. (2.4.15) we can form \( \approx_{(ij)} C \)

\[
\begin{pmatrix}
N_1^2 \cdot E & N_1 N_2 \cdot E & N_1 N_3 \cdot E & N_1 N_4 \cdot E \\
N_2 N_1 \cdot E & N_2^2 \cdot E & N_2 N_3 \cdot E & N_2 N_4 \cdot E \\
N_3 N_1 \cdot E & N_3 N_2 \cdot E & N_3^2 \cdot E & N_3 N_4 \cdot E \\
N_4 N_1 \cdot E & N_4 N_2 \cdot E & N_4 N_3 \cdot E & N_4^2 \cdot E
\end{pmatrix}
\]

(2.4.25)

where

\[
\approx_{(ij)} E = \approx_{(i)} A^{T} \cdot \approx_{(j)} D \cdot \approx_{(j)}
\]

(2.4.26)

The differences between the line structural element and the surface structural element obviously lies in that the metric matrix, \( \approx_{(r)} A \), is here a 3 by 6 matrix and the constitutive matrix, \( \approx_{(r)} D \), is a 3 by 3 matrix with

\[
D_{11} = D_{33} = G, \quad D_{22} = E, \quad D_{12} = D_{21} = D_{13} = D_{31} = D_{23} = D_{32} = 0
\]

The matrices \( \approx_{(ij)} E \) and the nodal forces, \( \approx_{(1)} F \) and \( \approx_{(2)} F \), will not be given here due to the size of the matrices.

The numerical integration is done in the same way as for the line structural elements, but now with integration in two dimensions using 2 x 2 Gauss point integration. The jacobian is given by \( J = \frac{bh}{4} \), where b is the element width and h is the element length. The jacobian is constant since the element is non-distorted.

The surface structural element does not have any stiffness for in-plane deformation. Moreover, the stiffness in the drilling degrees of freedom, corresponding to \( \nu_5 \), is zero. This is not a problem since the element is used together with shell elements that provides this kind of stiffness.
2.5 Transformation

Consider a transformation of the global degrees of freedom, \( \tilde{\nu}^{(r)} \), to the local degrees of freedom, \( \nu^{(r)} \) (see figure 2.5),

\[
\nu^{(r)} = T^{(r)} \tilde{\nu}^{(r)}
\]

(2.5.1)

where the transformation matrix \( T^{(r)} \) is an orthogonal matrix.

Transformation of the nodal degrees of freedom is done as for the local degrees of freedom

\[
a^{(r)} = L^{(r)} \tilde{a}^{(r)}
\]

(2.5.2)

with \( a \) as in Eq. (2.4.7) and \( r = 1, 2 \). Here \( L^{(r)} \) contains as many blocks of \( T^{(r)} \) along the diagonal as there are nodes in each element at one of the adherends.

Figure 2.5: Local and global degrees of freedom.
Once the nodal force vector, $\tilde{F}^{(r)}$ cf Eq. (2.4.9), has been calculated in the local system it has to be transformed into the global system.

\[
\tilde{F}^{(r)} = L^{(r)} \tilde{F}^{(r)}
\]

(2.5.3)

The total nodal force vector is then transformed from the local system to the global system by

\[
\bar{F} = \begin{pmatrix}
L^{(1)} & 0 \\
0 & L^{(2)}
\end{pmatrix}^{T} \begin{pmatrix}
\tilde{F}^1 \\
\tilde{F}^2
\end{pmatrix} = L^{T} \cdot \tilde{F}
\]

(2.5.4)

The tangent stiffness matrix will also have to be transformed into the global system.

\[
\bar{K} = L \cdot K \cdot L
\]

(2.5.5)

This implies that

\[
\bar{K}^{(r)} = \begin{pmatrix}
\bar{K}^{(11)} & -\bar{K}^{(12)} \\
-\bar{K}^{(21)} & \bar{K}^{(22)}
\end{pmatrix}
\]

where

\[
\bar{K}^{(ij)} = L^{T} \cdot K^{(ij)} \cdot L^{(j)}
\]

(2.5.6)

with

\[
\bar{K}^{(ij)} = \int_{\bar{C}^{(ij)}} \bar{C}^{(ij)} \cdot dS
\]

(2.5.7)

and

\[
\bar{C}^{(ij)} = L^{T} \cdot C^{(ij)} \cdot L^{(j)}
\]

(2.5.8)
2.5.1 Line structural element

Consider a part of an adhesive joint. For the line structural elements there are two nodes per adherend as shown in figure 2.6. The nodes are oriented in the midplane of the adherends, sharing the adherend nodes to reduce the degrees of freedom.

Figure 2.6: Illustration of line structural element.
For an adherend oriented in one dimension at an angle $\theta$ to the x direction, see figure 2.5, the transformation matrix is given by

$$T^{(r)} \approx \begin{pmatrix}
\cos \theta & \sin \theta & 0 \\
-sin \theta & \cos \theta & 0 \\
0 & 0 & 1
\end{pmatrix} \quad (2.5.9)$$

Due to the above statement that $L^{(r)} \approx T^{(r)}$ contains as many blocks of $T^{(r)}$ along the diagonal as there are nodes in one element at one of the adherends, the transformation matrix, $L^{(r)}$, is given by

$$L^{(r)} \approx \begin{pmatrix}
T^{(r)} & 0 \\
0 & T^{(r)}
\end{pmatrix} \quad (2.5.10)$$

The line structural elements have two nodes in each adherend. The nodes have three degrees of freedom, which implies that the transformation matrix, $L^{(r)}$, is a 6 by 6 matrix.

Transformation of the nodal degrees of freedom, from the local system to the global system, is done using Eqs. (2.5.9) and (2.5.2)

$$\bar{a}^{(r)} = \begin{pmatrix}
\bar{a}_1^{(r)} \\
\bar{a}_2^{(r)}
\end{pmatrix} = \begin{pmatrix}
T^{(r)} & 0 \\
0 & T^{(r)}
\end{pmatrix} \cdot \begin{pmatrix}
q_1^{(r)} \\
q_2^{(r)}
\end{pmatrix} = L_{(r)}^T \cdot \bar{q}^{(r)} \quad (2.5.11)$$

where $a_i^{(r)}$ denotes the nodal degrees of freedom associated with node i on adherend (r). Both the tangent stiffness matrix, $K_{i,r}$, and the total nodal force, $F$, are to be transformed according to Eq. (2.5.4) and (2.5.5). Similarly as for the degrees of freedom, the transformation of the nodal forces acting on each node can be calculated by

$$\bar{F}^{(r)} = \begin{pmatrix}
\bar{F}_1^{(r)} \\
\bar{F}_2^{(r)}
\end{pmatrix} = \begin{pmatrix}
T^{(r)} & 0 \\
0 & T^{(r)}
\end{pmatrix} \cdot \begin{pmatrix}
F_1^{(r)} \\
F_2^{(r)}
\end{pmatrix} = L_{(r)}^T \cdot F^{(r)} \quad (2.5.12)$$
where \( F_i^{(r)} \) are the nodal loads associated with \( a_i^{(r)} \). Both \( a_i \) and \( F_i \) contain three components because of the number of degrees of freedom in each node. The matrices of the transformed tangent stiffness matrix is given by Eq. (2.5.7), here with

\[
\bar{C}_{(ij)} = \begin{pmatrix}
N_2 \bar{E}_{(ij)} & N_1 N_2 \bar{E}_{(ij)} \\
N_2 N_1 \bar{E}_{(ij)} & N_2^2 \bar{E}_{(ij)}
\end{pmatrix}
\]  

(2.5.13)

for the line structural elements, where

\[
\bar{E}_{(ij)} = T^T \cdot E_{(i)} \cdot T^{(j)}
\]

(2.5.14)

using \( E_{(ij)} \) from Eq. (2.4.19).

### 2.5.2 Surface structural element

The adherends are modelled by 4-node shell elements in the three dimensional analysis. Thus, the surface structural element has four nodes in each of the adherends as depicted in figure 2.7. These elements are connected to nodes in the adherend midplane.

![Figure 2.7: Illustration of the surface structure element.](image)

Instead of having three degrees of freedom in each node, as for the line structural elements, there are six degrees of freedom in each node for the surface structural elements. This changes the dimension of the transformation matrix \( T^{(r)} \), as well as
the dimension of $L^{(r)}$, as can be seen below.

To be able to generate the correct transformation matrix $T^{(r)}$ the rotation matrix $R^{(r)}$ is introduced. This rotation matrix is calculated by successive multiplications of the rotation matrices around each cartesian axis.

$$R = R_x R_y R_z$$

$$R_x = \begin{pmatrix}
1 & 0 & 0 \\
0 & \cos \psi & -\sin \psi \\
0 & \sin \psi & \cos \psi
\end{pmatrix}$$

$$R_y = \begin{pmatrix}
\cos \phi & 0 & -\sin \phi \\
0 & 1 & 0 \\
\sin \phi & 0 & \cos \phi
\end{pmatrix}$$

$$R_z = \begin{pmatrix}
\cos \alpha & -\sin \alpha & 0 \\
\sin \alpha & \cos \alpha & 0 \\
0 & 0 & 1
\end{pmatrix}$$

and the total rotation matrix is thus given by

$$R^{(r)} \approx R = \begin{pmatrix}
c \phi c \alpha & -c \phi s \alpha & -s \phi \\
c \psi s \alpha - s \psi s \phi c \alpha & c \psi c \alpha + s \psi s \phi s \alpha & -s \psi c \phi \\
s \psi s \alpha + c \psi s \phi c \alpha & s \psi c \alpha - c \psi s \phi s \alpha & c \psi c \phi
\end{pmatrix}$$

(2.5.15)

where the abbreviations $s$ and $c$ are used for sine and cosine. The angles $\alpha$, $\phi$ and $\psi$ describe rotations about $z$-, $y$- and $x$-directions.

The six local degrees of freedom are

$$\nu^{(r)} = \begin{pmatrix}
\nu_1 \\
\nu_2 \\
\nu_3 \\
\nu_4 \\
\nu_5 \\
\nu_6
\end{pmatrix}$$

(2.5.16)
where \( \nu_i \) with \( i=1,2 \) and 3 correspond to translations and \( i=4,5 \) and 6 correspond to rotations. The transformation matrix, \( T^{(r)} \), to be used in \( L^{(r)} \) is thus given by

\[
\tilde{T}^{(r)} = \begin{pmatrix} R^{(r)} & 0 \\ 0 & R^{(r)} \end{pmatrix}
\]  
(2.5.17)

and this implies that the transformation matrix, \( L^{(r)} \), in this case is given by

\[
\tilde{L}^{(r)} = \begin{pmatrix} \tilde{T}^{(r)} & 0 & 0 & 0 \\ 0 & \tilde{T}^{(r)} & 0 & 0 \\ 0 & 0 & \tilde{T}^{(r)} & 0 \\ 0 & 0 & 0 & \tilde{T}^{(r)} \end{pmatrix}
\]  
(2.5.18)

Here \( \tilde{L}^{(r)} \) is obviously a 24 by 24 matrix, corresponding to 4 nodes on each adherend. Hence the four blocks of \( \tilde{T}^{(r)} \) along the diagonal.

Given the transformation matrix \( \tilde{L}^{(r)} \), the nodal degrees of freedom can be transformed accordingly as

\[
\tilde{\alpha}^{(r)} = \begin{pmatrix} \tilde{\alpha}_1^{(r)} \\ \tilde{\alpha}_2^{(r)} \\ \tilde{\alpha}_3^{(r)} \\ \tilde{\alpha}_4^{(r)} \end{pmatrix} = \begin{pmatrix} \tilde{T}^{(r)} & 0 & 0 & 0 \\ 0 & \tilde{T}^{(r)} & 0 & 0 \\ 0 & 0 & \tilde{T}^{(r)} & 0 \\ 0 & 0 & 0 & \tilde{T}^{(r)} \end{pmatrix} \begin{pmatrix} \alpha_1^{(r)} \\ \alpha_2^{(r)} \\ \alpha_3^{(r)} \\ \alpha_4^{(r)} \end{pmatrix} = \tilde{L}^{(r)} \cdot \alpha^{(r)}
\]  
(2.5.19)

where the indices 1, 2, 3 and 4 indicate the nodes in each of the adherends.
The nodal force vector is transformed in the same manner as for the nodal degrees of freedom

\[
\tilde{F}^{(r)} = \begin{pmatrix}
\tilde{F}_1^{(r)} \\
\tilde{F}_2^{(r)} \\
\tilde{F}_3^{(r)} \\
\tilde{F}_4^{(r)}
\end{pmatrix} = \begin{pmatrix}
T^{(r)} \\
0 \\
0 \\
0
\end{pmatrix}
\begin{pmatrix}
T^{(r)} \\
T^{(r)} \\
T^{(r)} \\
T^{(r)}
\end{pmatrix}^T
\begin{pmatrix}
\tilde{F}_1^{(r)} \\
\tilde{F}_2^{(r)} \\
\tilde{F}_3^{(r)} \\
\tilde{F}_4^{(r)}
\end{pmatrix} = \tilde{L}_{(r)}^T \cdot \tilde{F}^{(r)} \quad (2.5.20)
\]

Both \(a_i\) and \(l_i\) contain six components because of the number of degrees of freedom in each node. The matrices of the tangent stiffness matrix is given by Eq. (2.5.7) with

\[
\tilde{C}^{(ij)} = \begin{pmatrix}
N_1^2 \tilde{E}^{(ij)} & N_1N_2 \tilde{E}^{(ij)} & N_1N_3 \tilde{E}^{(ij)} & N_1N_4 \tilde{E}^{(ij)} \\
N_2N_1 \tilde{E}^{(ij)} & N_2^2 \tilde{E}^{(ij)} & N_2N_3 \tilde{E}^{(ij)} & N_2N_4 \tilde{E}^{(ij)} \\
N_3N_1 \tilde{E}^{(ij)} & N_3N_2 \tilde{E}^{(ij)} & N_3^2 \tilde{E}^{(ij)} & N_3N_4 \tilde{E}^{(ij)} \\
N_4N_1 \tilde{E}^{(ij)} & N_4N_2 \tilde{E}^{(ij)} & N_4N_3 \tilde{E}^{(ij)} & N_4^2 \tilde{E}^{(ij)}
\end{pmatrix} \quad (2.5.21)
\]

for the surface structural elements, where

\[
\tilde{E}^{(ij)} = T_{(i)}^T \cdot \tilde{E}^{(ij)} \cdot T_{(j)}^T \quad (2.5.22)
\]

using \(\tilde{E}^{(ij)}\) from Eq. (2.4.26).
2.5.3 Summary of transformation

To summarize the transformation chapter, the algorithm below shows the methodology of how to implement a transformation in the FORTRAN finite element code.

1. Calculate the necessary angles for the transformation.

2. Transform the global degrees of freedom into local degrees of freedom.

3. Calculate the nodal force vector and the tangent stiffness matrix locally.

4. Transform the nodal force vector from the local system to the global system.

5. Transform the local tangent stiffness matrix to the global tangent stiffness matrix.

\[ q^{(r)} = \tilde{L}^{(r)} \cdot \tilde{q}^{(r)} \]

\[ \tilde{F}^{(r)} = \tilde{L}^{(r)} \cdot F^{(r)} \]

\[ \tilde{K} = \tilde{L}^{T} \cdot \tilde{K} \cdot \tilde{L} \]
Chapter 3

Implementation

3.1 Fortran

The basic FORTRAN file for the ABAQUS calculations begins with a user element subroutine in which the different parameters, such as the displacements U and the stiffness matrix AMATRX are defined.

It is preferable to have all of the calculations written down before the FORTRAN implementation begins. This simplifies the rest of the work considerably.

Due to the fact that the FORTRAN code controls the FE calculations for a single element, the element parameters have to be defined. The most important ABAQUS parameters are RHS, AMATRX, PROPS, U, COORDS and SVARS. These parameters are sent between ABAQUS and the FORTRAN code during the calculations. The element stiffness matrix has the parameter name AMATRX, the nodal force vector RHS, the element properties PROPS, the node displacements U, the node coordinates COORDS and the peel and shear stresses and also the separation in SVARS. There are of course more parameters in the calculations but they are not brought up at this stage.
The user element properties such as the adherend height, the adherend width, Young’s modulus, Poisson’s number and the adhesive thickness are the parameters in PROPS. The rest of the ABAQUS parameters are calculated in the FORTRAN code. To do this the FORTRAN programmer has to define suitable variables, preferably the same variable names as in the calculation written down before programming.

Matrix manipulations is one of the difficulties in FORTRAN. The reason is that when for example multiplying two matrices, it has to be done with loops over the indices of the matrices.

This example illustrates the problem. Let M and N be the two 3 by 3 matrices that we want to multiply.

\[
M = \begin{pmatrix}
a1 & a2 & a3 \\
b1 & b2 & b3 \\
c1 & c2 & c3 \\
\end{pmatrix} \quad N = \begin{pmatrix}
d1 & d2 & d3 \\
e1 & e2 & e3 \\
f1 & f2 & f3 \\
\end{pmatrix}
\]

The correct multiplication of these two matrices generates a 3 by 3 matrix MN with for example MN_{11} = a1d1 + a2e1 + a3f1. In a mathematics program like MATLAB these multiplications are done simply by typing M*N.

Using FORTRAN the same multiplication is done by looping over the matrices indices. The matrix MN is a zero 3 by 3 matrix going into this loop.

\[
\begin{align*}
\text{DO} & \ I = 1, 3 \\
\text{DO} & \ K = 1, 3 \\
\text{DO} & \ L = 1, 3 \\
\text{MN}(I,K) &= \text{MN}(I,K) + \text{M}(I,L) \times \text{N}(L,K) \\
\text{ENDDO} \\
\text{ENDDO} \\
\text{ENDDO}
\end{align*}
\]
Multiplying three matrices is done by successive matrix multiplications, i.e. first multiply two matrices and then multiply this product with the third matrix.

Matrices are not always as small as 3 by 3, but it is always possible to divide large matrices into smaller matrices which gives a better overview of large matrix multiplications.

Basically matrix manipulations is the only difficulty when implementing a user element in FORTRAN, the rest of the calculations are straight forward.

### 3.2 Abaqus

Once the user element has been implemented in the FORTRAN code, the next step in solving the problem is to implement the input data into ABAQUS.

This is done by generating an input file. In this input file the geometry of the desired structure is defined. By geometry meaning coordinates of the nodes. In ABAQUS there are a lot of simplifying commands that can be used. Commands like ELGEN and NGEN are two of these simplifying commands that is commonly used when generating the geometry of the structure. The command ELGEN is really short for "element generation" and NGEN is short for "node generation".
\begin{section}{Line structural element}

The command NGEN is used in the input file by first defining the coordinates of the corner nodes, for example when defining the node coordinates of an adherend in one dimension, the code looks like this

\begin{verbatim}
*NODE
 1, 0, 0
10, 1, 0
\end{verbatim}

These two lines in the input file will generate node 1 in \((x,y)=(0,0)\) and node 10 in \((x,y)=(1,0)\). Now using the command NGEN in this manner,

\begin{verbatim}
*NGEN, NSET=BEAM1
 1,10,1
\end{verbatim}

will generate 8 additional nodes equally distributed between nodes 1 and 10, thus a total of 10 nodes in the interval \([0,1]\).

When the nodes are in place it is time to generate the elements. This is done by first choosing the element type and then using the already mentioned command ELGEN.

\begin{verbatim}
*ELEMENT, TYPE=B21, ELSET=ELBEAM1
 1,1,2
*ELGEN, ELSET=ELBEAM1
 1,9,1,1
\end{verbatim}

This part of the input file will generate 9 elements of the type B21 between nodes 1 and 10. The element type B21 is a 2-node linear beam element. There are a lot of
different types of elements in the ABAQUS library that can be used in combination with the user element or by itself as in the example above. When implementing a user element, the procedure is almost identical. The main difference lies in what type of element that is supposed to be implemented. Already having implemented a user element in FORTRAN we can write,

\*USER ELEMENT, TYPE=U1, NODES=4, PROPERTIES=6, COORDINATES=3, VARIABLES=8

1, 2, 6
\*ELEMENT, TYPE=U1, ELSET=ELIM ** elnr, nod1, nod2, nod3, nod4

1, 1, 2, 11, 12
\*ELGEN, ELSET=ELIM

1,9, 1,1
\*UEL PROPERTY, ELSET=ELIM

** E, nu, t=thickness, b=width, H1=beam height 1, H2=beam height 2

2.0E3, 0.4, 0.2, 5., 6., 5.

and the user element is thus implemented in the ABAQUS input file. The line structural elements have two nodes in each adherend and in total four nodes. The numbers 1, 2, 6 describes the active degrees of freedom in each node, which corresponds to

Figure 3.1: Two rows of beam elements and the location of the fifth line structural element.
There are eight output variables, VARIABLES=8, corresponding to shear and peel deformation and shear and peel stress in the two integration points. The eight variables are placed in SVARS, where SVARS(1) contain the shear stress in Gauss point 1, SVARS(2) the peel stress in Gauss point 1, SVARS(3) the shear deformation in Gauss point 1, SVARS(4) the peel deformation in Gauss point 1. Analogous treatment is done for Gauss point 2.

Figure 3.1 shows a user element with nodes 5,6,15 and 16. To generate the 9 user elements as in the code above, one has to generate new beam elements with nodes 11 to 20 as shown in figure 3.1. When this has been done the line structural elements are connected sharing the same node numbers as the beams. In the code, user element 1 is built up by nodes 1,2,11 and 12. Thus, with the command ELGEN 9 user elements are generated. Finally the properties of the user element have to be defined.

### 3.2.2 Surface structural element

The surface structural elements are implemented in ABAQUS almost exactly as the line structural elements, with the difference that each element contains eight nodes instead of four as for the line structural elements. Before implementing the surface structural elements, the connecting shell elements has to be implemented.

First, the coordinates of the corner nodes of the geometry have to be defined. Lets say, node 1 in \((x,y,z)=(0,1,0)\), node 10 in \((x,y,z)=(1,1,0)\), node 11 in \((x,y,z)=(0,1,1)\) and node 20 in \((x,y,z)=(1,1,1)\), thus in the ABAQUS input file

```abaqus
*NODE
  1, 0, 1, 0
  10, 1, 1, 0
  11, 0, 1, 1
  20, 1, 1, 1
```
This is obviously not enough, nodes have to be generated between these four created nodes. Once again using the command NGEN

*NGEN, NSET=SHELL1
1,10,1
11,20,1

This is actually two lines created exactly as for the line structural elements, but with one line translated in the z direction. Thus a surface in the x-z plane with corner nodes 1, 10, 11 and 20. To create the shell elements, the command ELGEN is used.

*ELEMENT, TYPE=S4R, ELSET=EL SHELL1
1,1,2,12,11

*ELGEN, ELSET=EL SHELL1
1,9,1,1

Here, the shell elements S4R are used and these are 4-node quadrilateral shell elements with reduced integration. These are not the only shell elements that can be used in ABAQUS, there are a lot more elements to choose from. With 10 nodes in the x direction, 9 shell elements are created using ELGEN. Element 1 has the corner nodes 1, 2, 12 and 11. User elements, like the surface structural elements, are implemented in the same fashion as the line structural user elements described previously.

*USER ELEMENT, TYPE=U1, NODES=8, PROPERTIES=5, COORDINATES=3, VARIABLES=24
1, 2, 3, 4, 5, 6

*ELEMENT, TYPE=U1, ELSET=ELIM ** elnr, nod1, nod2, nod3, nod4, nod5, nod6, nod7, nod8
1, 1, 2, 12, 11, 21, 22, 32, 31
*ELGEN, ELSET=ELIM
1, 9, 1, 1
*UEL PROPERTY, ELSET=ELIM
** E, nu, t=thickness, H1=beam height 1, H2=beam height 2
2.0E3, 0.4, 0.2, 6., 5.

For the surface structural elements there are 24 variables, three stresses and three deformations in each of the four Gauss points. These variables are placed in SVARS in the same manner as for the line structural element implementation. The surface structural elements have four nodes in each adherend and this comes to eight nodes total. Here, there are six degrees of freedom in each nodes. Hence, 1, 2, 3, 4, 5 and 6 in the code above. Figure 3.2 shows element 1 of the surface structural user elements with nodes 1, 2, 12, 11, 21, 22, 32 and 31. To generate the 9 user elements as in the code above, one has to generate new shell elements with nodes 21 to 30 and 31 to 40 as shown in figure 3.2. When this has been done the surface structural elements are connected sharing the same node numbers as the shell elements. The properties of the user element are defined similarly as for the line structural elements, except that no element width can be specified.

Figure 3.2: Two rows of shell elements and the location of the first surface structural element.
Chapter 4

Analysis and Verification

4.1 Double Cantilever Beam (DCB) specimen

The DCB specimen in figure 4.1 consists of two beams, called adherends when bonded by an adhesive. These adherends, which are placed in a sandwich fashion with the adhesive bonding them, are subjected to the loads $F$ in opposite vertical directions. The adhesive bond length is much larger than the height of the adherends.

Figure 4.1: The DCB specimen.
The symmetry plane of the DCB specimen is obviously in the adhesive midplane and by the use of this symmetry in the finite element method, it is possible to model only half of the structure to get the correct solution. In this thesis, however, a model of the entire structure is analyzed. In the following subsections the DCB-test is analyzed for both the line structural elements and the surface structural elements. The results of the analysis is then analyzed and verified.

As was mentioned above the DCB-test considers peel stresses but shear stresses occur if the specimen is antisymmetric. These shear stresses are small compared to the peel stresses and this is why the DCB-test is used when peel stresses are to be analyzed.

### 4.1.1 Line structural element for 2D analysis

Four different DCB specimens, as depicted in figure 4.1, were analyzed using the line structural elements. The difference in these specimens was the bottom beam height, which was varied to catch the shear stresses that become more apparent when the symmetry in the structure is broken.

The DCB specimen depicted in figure 4.1 is a symmetric structure and as already mentioned, this structure have a dominating peel stress effect. When the symmetry is broken by varying the bottom beam height, shear stresses appear to compensate the difference in stiffness. The effect of this broken symmetry is seen in analyzing figures 4.2 and 4.3. Figure 4.2 illustrates the peel stress in the adhesive, both for a totally symmetric (TS) and with a clamped bottom beam (CBB) DCB specimen. These specimens were analyzed using ABAQUS and the line structural elements and the results was then implemented in MATLAB to be compared with an analytical solution provided by Alfredsson and Högbäck [2]. It is seen that the finite element solution follows their analytical solution very well.
The input data used for the TS and CBB specimens are

- TS specimen: Beam width = 5 mm, Beam height = 5 mm, Beam length = 100 mm
- CBB specimen: Beam width = 5 mm, Beam height = ∞, Beam length = 100 mm

The adhesive bonding the beams in all specimens had a length of 50 mm, giving 50 mm of initial crack length. The loads, \( F = 100 \) N, were applied as depicted in figure 4.1. The number of nodes used when analyzing the two specimens in ABAQUS were 1000 nodes, with the initial crack region having 150 nodes and the adhesive 850 nodes respectively.

![Diagram](image)

Figure 4.2: Peel stress in DCB specimens.
The shear stress behavior is depicted in figure 4.3 for the two specimens described and similarly as for the peel stress behavior the finite element solution follows the analytical solution very well. There is however a slight difference between the finite element solution and the analytical solution in the shear stress far from the crack tip for the CBB solution. The analytical solution shows higher shear stress than the finite element solution.

![Figure 4.3: Shear stress in DCB-specimens.](image)

The TS and CBB specimens are considered to be extreme cases and for this reason it is unnecessary to plot the results of the two other specimens with different bottom beam height. Two other geometries were tested with the lengths 15 and 25 mm. The results agrees well with the analytical solution of Alfredsson and Högberg [2].
4.1.2 Surface structural element for 3D analysis

Two DCB specimens were analyzed using the surface structural elements. Here, using the surface structural elements, the width of the beams is varied, thus showing the anticlastic bending effect. It is an important fact that the peel stress varies over the width of the beams. The peel stress as function of the z-coordinate is shown in figure 4.4 for a DCB specimen with a beam height of 5 mm, a width of 5 mm and a length of 100 mm, that is to say the same structure as TS in the previous section.

![Graph showing peel stress over width](image)

Figure 4.4: Peel stress in the adhesive over the width.

Figure 4.4 illustrates the peel stress at the crack tip, giving a maximum of the anticlastic bending effect for this linear elastic analysis. The difference in peel stress between the center and the edge is approximately 17 MPa. The analytical solution

\[ \Delta \sigma = \frac{3E\nu_b bFL}{EH^3 t} \]  

(4.1.1)

see Andersson [3], with the parameters \( F = 100 \) N (applied force), \( \nu_b = 0.3 \) (Poisson’s number), \( b = 5 \) (the width), \( E = 200 \) GPa (Young’s modulus of adherends), \( H = 5 \) mm (the height), \( L = 50 \) mm (free length), \( \bar{E} \approx 4.3 \) GPa, \( t = 0.2 \) mm gives approximately 19 MPa, which compares favorably with the finite element solution.

The peel stress as a function of the length coordinate \( x \) is shown in figure 4.5 for
the center and edge of the finite element solution and also for an analytic solution provided by Alfredsson and Högberg [2]. It is obvious that the finite element solution and the analytical solution share the same wave number, even though they are not identical. It should be remembered that the analytic solution is based on beam theory.

Figure 4.5: Peel stress in the adhesive over the adhesive joint length.
A change in width of the DCB specimen, from 5 mm to 25 mm, similarly gives an anti-clastic bending effect, see figure 4.6. The height of the beams is kept at 5 mm. The result of this test is that the difference in peel stress between the center and the edge is almost identical to the previous DCB specimen, that is 20 MPa. The maximum peel stress is however much lower, which is explained by the fact that an equally large applied force acts over a much larger surface generates lower stress. The difference in peel stress at the edge and the center divided by the center peel stress is in this case approximately 57 %. This quotient is almost five times as large as for the previous DCB specimen.

![Graph showing peel stress over width](image)

**Figure 4.6:** Peel stress in the adhesive over the width.

Similarly as for the line structural element solution, the peel stress as a function of x is plotted in figure 4.7. The analytical solution, Alfredsson and Högberg [2], compares favorably with the finite element solution.
4.2 End Notched Flexure (ENF) specimen

To get a correct idea of the adhesive behavior one has to consider both peel and shear stress. Shear deformation is analyzed using the ENF-test. Instead of having two opposite loads pulling the adherends apart, the ENF-test in figure 4.8 consists of a sandwich structure in which the bottom adherend is placed on two rigid supports. The top adherend lies entirely on the adhesive layer bonding the two adherends. A load is applied symmetrically between the supports. Even though this test is made for shear stress analysis, peel stresses occur but they are much smaller than the shear
stresses. The ENF-test was only analyzed using the line structural elements.

4.2.1 Line structural element for 2D analysis

Three different geometries were studied, varying the initial crack length, \(a\). The initial crack length was varied from zero initial crack to one fourth of the beam length.

The beams used in all of the structures had the dimensions

- Length, \(L = 900 \text{ mm}\)
- Height, \(H = 32 \text{ mm}\)
- Width, \(b = 32 \text{ mm}\)

and the different structures were made up by different initial crack length \(a = 0, L/5, L/4\). These geometries are denoted ENF1, ENF2 and ENF3 in the sequel. The adhesive thickness in all structures was 0.2 mm.

ENF1 is made with zero initial crack length and obviously this is a totally symmetric structure returning an antisymmetric shear stress distribution, see figure 4.9. In the point of the applied force, the shear stress changes sign. The integral of the shear stress over the entire domain should be zero in order to get equilibrium. In figure 4.9, the results of the ABAQUS simulation is compared to an analytic solution provided by Alfredsson [1]. The results are nearly identical and this verifies that the line structural elements are a good approximation to be used in simulating structures of this type.

In the second geometry, ENF2, the initial crack length is \(L/5\), which gives a non-symmetric structure. The shear stress results of the ENF2 geometry is depicted in figure 4.10. The shear stress is much much larger at the crack tip. Further away from the crack tip, the shear stress behavior is virtually exactly the same as for the ENF1
geometry. The phenomenon that the behavior over large parts of the specimen, is almost the same as for ENF1 is not that hard to comprehend realizing that nothing has been changed but the initial crack length. As for the ENF1 structure, the result of this structures ABAQUS simulation is also compared with the analytical solution provided by Alfredsson [1].

Finally the last geometry that was analyzed, the ENF3 geometry with the initial
crack length $L/4$. Similarly as for the ENF2 geometry, this is a nonsymmetric geometry which gives rise to a larger shear stress at the crack tip. The ABAQUS simulation results are similar to the ENF2 geometry results with the simple difference that the shear stress behavior is contracted to a larger crack length. Even here, the ABAQUS simulation results are in good agreement with the analytic solution provided by Alfredsson [1] (see figure 4.11).

![Shear stress distribution for the ENF3 geometry.](image)

Figure 4.11: Shear stress distribution for the ENF3 geometry.

Some numerical errors occurred while transporting the ABAQUS simulation results into Matlab. These errors came from ABAQUS which did not give enough significant numbers in the output data. The errors can be seen in figures 4.9-4.11 as some fluctuations in the ABAQUS simulated results.
A comparison between all the shear stress distributions, that was simulated using ABAQUS, for the different structures are depicted in figure 4.12. It is seen that the shear stress distribution is the same some length from the crack tip.

The shear stress behavior in figure 4.12 is compared with the Jouravsky shear equation
\[ \tau = \mu \frac{V}{A} \]
where \( V = \frac{F}{2} \) is the cross sectional shear force, \( A = 2bH \) is the cross sectional area and \( \mu = \frac{3}{2} \) is the cross sectional factor for a rectangular cross section. It is seen in figure 4.12 that the line structural elements gives results which are in good agreement with the analytic Jouravsky shear equation.

Figure 4.12: Comparison of the shear distributions of the different geometries.
What about the peel stresses in these geometries? The peel stresses are much smaller than the shear stresses, as was mentioned before, but they are present in the analysis. Figure 4.13 shows the peel stress for the ENF1 geometry. This structure is totally symmetric which gives a symmetric peel stress distribution.

For the ENF2 geometry, the initial crack length is $L/5$ and this generates a maximum peel stress of approximately 10 MPa, see figure 4.14. This is same maximum value as for the ENF1 geometry.

![Figure 4.13: Peel stress distribution for the ENF1 geometry.](image)

There is no difference in maximum peel stress when analyzing the ENF3 structure. The main difference is the initial crack length, see figure 4.15.
The difference between the ENF1 and the two other geometries, concerning the peel stresses, is that the peel stresses at the crack tip are almost zero when an initial crack is present. For the ENF1 geometry this is not the case, noting that in figure 4.13 the peel stress is present at the ends and in the middle. With no initial crack, the upper adherend lies entirely on the adhesive bonding the two adherends. This is why the reaction forces give rise to peel stresses at the ends. With an initial crack, boundary conditions are employed on the specimen according to figure 4.8, thus if these boundary conditions had been employed with no initial crack the peel stress at the left end would also been zero. It is obvious that the peel stress is present in the middle where the load is applied and at the ends of the specimen where there are reaction forces due to the load.

![Peel stress distribution for the ENF2 geometry.](image)

Figure 4.14: Peel stress distribution for the ENF2 geometry.

When an initial crack is present the shape of the peel stress over the adhesive joint from the ENF1 geometry is the same but with the difference that the peel stress is zero close to the crack tip.

The quotient between the peel and shear stress with total symmetry, is almost 1. The ENF2 structure has an initial crack length equal to L/5 and this initial crack
generates a quotient of 1/5. For the ENF3 structure, with an initial crack length of \( L/4 \), the quotient is 1/6. Even though the quotient is not very small, is it possible to neglect the peel stress in the analysis? The answer to this question is fairly simple. The interesting part of the entire structure is in the region of the crack tip and in this region, the shear stress is much larger than the peel stress and this is why the peel stress can be neglected in the analysis of the ENF tests.

Figure 4.15: Peel stress distribution for the ENF3 geometry.
4.3 Single Lap Joint (SLJ) specimen

A test combining peel and shear stresses is the SLJ-test and it is illustrated in figure 4.16. The main difference from the DCB- and the ENF-tests is that neither the peel stress nor the shear stress can be neglected in the analysis.

The SLJ-test is made for three different geometries using the line structural elements and two geometries using the surface structural elements. For the geometries, using the line structural elements, the beam data is kept constant and the adhesive joint length is varied. The difference when using the surface structural elements instead of the line structural elements, is that instead of varying the adhesive joint length, the width of the geometries is varied to catch the anti-clastic bending effect.

It is not obvious how the deformed geometry will appear when the load $F$ is applied. The expected result is an anti-symmetrically deformed geometry considering a vertical line in the middle between the supports.

4.3.1 Line structural element for 2D analysis

Denote the analyzed geometries SLJ1, SLJ2 and SLJ3. All of the geometries share the beam data, the load and the adhesive thickness, which is not changed in any of these three geometries.
• Beam length: L=500 mm
• Beam height: H=32 mm
• Beam width: 32 mm
• Adhesive thickness: $t=0.2$ mm
• Load: $F=10.7$ kN

The SLJ1 geometry had an adhesive joint length $a=50$ mm, which is a fairly small length compared to the other dimensions, and with the load $F$ applied to the geometry the peel stress distribution for the SLJ1 geometry is given in figure 4.17.

![Peel stress distribution of the SLJ1 geometry.](image)

Figure 4.17: Peel stress distribution of the SLJ1 geometry.
The peel stress distribution, in figure 4.17, is compared to an analytic solution provided by Alfredsson and Högberg [2]. Again, the results of the ABAQUS solution is very close to the analytic solution.

The maximum peel stress in the adhesive joint is approximately 24 MPa, the maximum shear stress is approximately 12 MPa and the peel stress is thus almost two times the shear stress. The maximum values of the two stresses are at the edges of the adhesive joint bonding the beams. Here it is obvious that neither the peel stress nor the shear stress can be neglected.

The distribution of the shear stress for the SLJ1 geometry is depicted in figure 4.18 together with an analytic solution provided by Alfredsson and Högberg [2].

Even here, it is almost not possible to distinguish the ABAQUS solution from the analytic solution.
For the second geometry, SLJ2, the adhesive joint length is extended to $a=100$ mm. This obviously creates a larger adhesive region to distribute the load. Because of this, the peel and shear stresses are lower. The effect of a lower maximum peel stress is not the only effect of making the adhesive joint longer as is seen in figure 4.19, where the peel stress distribution is depicted and it is also compared to an analytic solution provided by Alfredsson and Högborg [2].

![Peel stress distribution of the SLJ2 geometry.](image)

Figure 4.19: Peel stress distribution of the SLJ2 geometry.

The distribution of the shear stress for the SLJ2 geometry is shown in figure 4.20. The shear stress distribution keep a lot of its shape, but with a smaller maximum and also a smaller minimum value. It is hard to see the effect of the more and more constant shear stress distribution some length into the adhesive joint from the edges. This effect is easier to see in the shear stress distribution for the SLJ3 geometry, see figure 4.22.
In the third geometry analyzed, the SLJ3 geometry, the adhesive joint length is $a=200$ mm. This length is almost half of the entire beam length and would give even lower maximum values both for the peel and shear stresses. The distribution of the peel stress for the SLJ3 geometry is depicted in figure 4.21 both for the ABAQUS solution and an analytic solution provided by Alfredsson and Högborg [2].

Figure 4.20: Shear stress distribution of the SLJ2 geometry.

Figure 4.21: Peel stress distribution of the SLJ3 geometry.
An effect is that the peel stress, see figure 4.21, starts with a tensile stress. Inside the adhesive joint the peel stress is compressive. This effect is due to the length of the adhesive joint. The line of symmetry lies vertically between the supports, giving a symmetric peel stress distribution. This symmetry is seen in all of the analyzed geometries. The way that the ABAQUS solution follows the analytic solution is also here to great satisfaction. The shear stress distribution for the SLJ3 geometry is depicted in figure 4.22 together with an analytic solution provided by Alfredsson and Högberg [2].

Figure 4.22: Shear stress distribution of the SLJ3 geometry.
4.3.2 Surface structural element for 3D analysis

Two different geometries were analyzed using the surface structural elements, these are denoted SLJ4 and SLJ5. The beam length is \( L = 100 \text{ mm} \) and the height is \( H = 5 \text{ mm} \) in both geometries. The adhesive thickness is the same as in the analysis using the line structural elements, i.e \( t = 0.2 \text{ mm} \). The adhesive joint length, \( a \), is 50 mm in the analysis of both geometries. Here, the width of the beams is varied to catch the anti-clastic bending effect and also to study if there are any variation in the shear stress distribution across the beam width. Finally, the applied load, \( F \), on both geometries is 100 N.

The SLJ4 geometry has a beam width of 5 mm, giving a square cross section. With this width, the anti-clastic bending effect is not large. Three plots are made on different stress distributions for the SLJ4 geometry. Figure 4.23 illustrates the peel stress distribution at the center and at the edge over the adhesive joint length together with an analytic beam solution provided by Alfredsson and Högberg [2].

![Figure 4.23: Peel stress distribution of the SLJ4 geometry.](image_url)
Secondly, the results of the peel stress distribution across the width of the adhesive joint at the end of the joint is shown in figure 4.24. It shows the anti-clastic bending effect on the geometry due to the load $F$. The difference in peel stress at the edge and the center divided by the center peel stress is in this case approximately 11%, thus the same result as for the DCB specimen with the same width.

![Peel stress distribution over the width at the crack tip of the SLJ4 geometry.](image)

Figure 4.24: Peel stress distribution over the width at the crack tip of the SLJ4 geometry.

A plot of the peel stress distribution across the width and along the length of the joint is shown in figure 4.25. It is seen, analyzing both figure 4.23 and figure 4.25 that the peel stress distribution is approximately constant over the width in the central part of the adhesive joint. The difference in the peel stress distribution becomes more and more apparent when analyzing the adhesive joint further out to the edges. In these regions the peel stress differs because of the already mentioned anti-clastic bending effect.

The peel stress distribution given in figure 4.25 are similar to results by Andruet et al [4] even though the input data in this analysis is different from theirs. It is explained by Andruet et al [4] that the peel stress distribution is due to the Poisson effects and the anti-clastic bending of the beams.
Andruet et al. [4] used a nonlinear finite element to model the adhesive. They used 9-node shell elements in the adherends, which coupled the three dimensional adhesive elements. The adhesive elements were in fact three dimensional solid brick elements with nodes offset to the midplane of the shell.

![Peel stress distribution over the entire adhesive joint of the SLJ4 geometry.](image)

Figure 4.25: Peel stress distribution over the entire adhesive joint of the SLJ4 geometry.

For the shear stress distribution, see figure 4.26, the shape is almost identical to the analysis that was done for the line structural elements. There is nevertheless a slight change in shear stress in that the maximum shear stress is lower at the center than at the edge. It is difficult to see this effect in figure 4.26, but a surface plot over the entire adhesive region, illustrated in figure 4.27, shows the change in maximum shear stress.
Figure 4.26: Shear stress distribution of the SLJ4 geometry.

Figure 4.27: Shear stress distribution over the entire adhesive joint of the SLJ4 geometry.
The SLJ5 geometry has a beam width that is five times the width of the SLJ4 geometry, i.e. 25 mm. This implies that the anti-clastic bending effect would be more apparent when analyzing both peel and shear stresses. It was mentioned that the previous geometry, SLJ4, gave similar results as Andruet et al. [4]. The SLJ5 geometry has a width of 25 mm and an adhesive joint length of 50 mm, giving a larger adhesive area compared to the SLJ4 geometry. In the analysis of the SLJ5 geometry, the results are even more similar and the reason is that Andruet et al. [4] analyzed a geometry that had a square adhesive region (a width of 25.4 mm and an adhesive joint length of 25.4 mm).

An illustration of the peel stress distribution over the adhesive joint is given in figure 4.28, both for the center of the width and at the edge. These results are compared to an analytic solution provided by Alfredsson and Högberg [2].

![Peel stress distribution of the SLJ5 geometry.](image_url)
The anti-clastic bending effect is seen when analyzing the peel stress across the beam width at the beginning of the overlap. Here, see figure 4.29, compared to the SLJ4 geometry, the peel stress drops rapidly close to the edges. Even with this rapid decrease in peel stress close to edges, the difference between the center peel stress and the edge peel stress remains the same as for the SLJ4 geometry.

Figure 4.29: Peel stress distribution over the width at the crack tip of the SLJ5 geometry.

The peel stress distribution over the entire adhesive region of the SLJ5 geometry is illustrated in figure 4.30. As seen in figure 4.30, the peel stress distribution over the entire region has the same shape as for the SLJ4 geometry (except for the rapid decrease at the edges).
A difference between the SLJ4 and the SLJ5 geometry is, however, given by the shear stress distribution. The effect of this difference is only seen in a surface plot of the shear stress distribution over the entire adhesive region, see figure 4.32. The results of the shear stress distribution over the adhesive joint length is depicted in figure 4.31. The difference in the shear stress distribution is that close to the edges, the shear stress makes a dip, see figure 4.32. The same phenomenon is present in the results by Andruet et. al [4].
Figure 4.31: Shear stress distribution of the SLJ5 geometry.

Figure 4.32: Shear stress distribution over the entire adhesive joint of the SLJ5 geometry.
Chapter 5

Conclusions and future work

In this thesis, interphase elements are created to be connected to finite structural elements such as Timoshenko beam elements and Mindlin shell/plate elements. Two elements are created, the line structural element and the surface structural element. Three standard tests, the DCB-, ENF- and the SLJ-test, are analyzed to verify the interphase element. The results of these tests are compared to analytical beam solutions.

The results of the shear stress behavior in all three tests, using the line structural element, is almost identical to the analytical beam solutions. This indicates that the line structural element gives a good approximation of the shear stress behavior when analyzing structures in two dimensions. Even the results of the peel stress behavior is almost the same as the results from the analytical beam solutions in all the tests. There are, however, some fluctuations in the peel stress behavior of the ENF-test. These fluctuations could be due to numerical noise, but the noise is in fact small and can, for that reason, be neglected.

The anti-clastic bending effect can easily be observed using the surface structural element in the three tests. This is an important effect that can not be neglected. Plots of the peel and shear stress variation in the longitudinal direction compared to
analytical beam solutions show that the surface structural element gives good results.

Both the line structural element and the surface structural element are based on linear elastic theory. Further development would be: to extend the capability of the elements so that a nonlinear material model can be used; transform the existing elements to geometrically nonlinear elements; introduce plasticity and damage of the adhesive layer etc.
Bibliography


