Experimental and numerical studies of impact behaviour of GFRP composites

by

Nils Arne Rakstad

Supervisor:
Jostein Hellesland, Alfred Andersen

RESEARCH REPORT
IN MECHANICS

UNIVERSITY OF OSLO
DEPARTMENT OF MATHEMATICS
MECHANICS DIVISION

UNIVERSITETET I OSLO
MATEMATISK INSTITUTT
AVDELING FOR MEKANIKK
Experimental and numerical studies of impact behaviour of GFRP composites

Nils Arne Rakstad

Supervisor: Jostein Hellesland, Alfred Andersen
Preface

This thesis is the final work for the degree Candidatus Scientiarum (master equivalent) at the Mechanics Division of the Department of Mathematics at the University of Oslo.

The work of this thesis has been done at SINTEF in Oslo with cand. scient. Alfred Andersen as supervisor. I would like to thank the Department of Polymer and Composite at SINTEF Material Technology who provided all equipment needed. Thanks to cand. scient. Reidar Friberg for excellent help with the logging equipment during the experiments. Special thanks goes to Alfred Andersen for good ideas and excellent supervising during the work.

Thanks to ABB Offshore Systems, Billingstad, who provided the materials used in the experiments and Brødrene Aa, Hyen, for manufacturing the laminates.

Simultaneous with this thesis, Sigve Takle has been writing his thesis about impact behaviour of sandwich plates. The same test equipment was used and the cooperation has been very helpful. There have been good discussions about the subjects.

I would like to thank my supervisor Professor Jostein Hellesland at the University of Oslo for comments and good advises.

My deepest gratitude and love goes to my girl friend, Ane Lerøy Sataøen. I will be eternally thankful for her understanding, support and love.

Nils Arne Rakstad

Oslo, 16th November 2003
Abstract

In this thesis, two main objectives have been to carry out impact experiments of fibre reinforced polymer composite plates and numerical simulations of these impacts. Two impact devices were used, a Rosand Instrumented Falling Weight device and one self constructed drop weight device, which were built for this thesis. In the Rosand device nine impacts tests were performed to study the effect of varying the energy and mass. The drop weight device built provided astonishing good result, considering the simply construction and equipment used. A lot of impact tests were conducted before the setup worked well and the three impacts in this thesis could be performed.

All the impact tests were simulated numerically using the finite element program AUTODYN and the results were satisfactory compared to the experiments. The impact durations were not identical for the simulations and the experiments, but the differences were in most cases small and insignificant. The simulated damage areas gave a good indication of the real damage from the experiments.

Static stress, strain and deflection simulations were done using the finite element program ANSYS. Stress and strain analyses were done for an applied deflection, known from the impact experiments. For the Rosand impact tests the stress distributions were about the same as those for the AUTODYN simulations, but for the drop weight impacts there was no similarity between the stress distributions in ANSYS and AUTODYN.
**Notation**

Principal symbols used in the thesis are listed here. Less frequently used symbols and, symbols that have different meanings in different contexts, are defined where they are used.

**Latin Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Area</td>
</tr>
<tr>
<td>$E$</td>
<td>Young’s modulus</td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
</tr>
<tr>
<td>$F = ma$</td>
<td>Newton’s 2. law. Force = mass $\times$ acceleration</td>
</tr>
<tr>
<td>$G$</td>
<td>Shear modulus</td>
</tr>
<tr>
<td>$M$</td>
<td>Mass</td>
</tr>
<tr>
<td>$M_i$</td>
<td>Mass of impactor</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$Q$</td>
<td>Vertical shear force in a midplane</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius</td>
</tr>
<tr>
<td>$V$</td>
<td>Volume fraction</td>
</tr>
<tr>
<td>$V_0$</td>
<td>Impact velocity</td>
</tr>
<tr>
<td>$W$</td>
<td>Weight fraction</td>
</tr>
<tr>
<td>$W_{mn}$</td>
<td>Fourier coefficient for deflection</td>
</tr>
<tr>
<td>$a, b$</td>
<td>Plate dimension</td>
</tr>
<tr>
<td>$k_{bs}$</td>
<td>Plate stiffness</td>
</tr>
<tr>
<td>$l$ and $w$</td>
<td>Delamination sizes</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$q_{mn}$</td>
<td>Fourier coefficient for force</td>
</tr>
<tr>
<td>$t$</td>
<td>Thickness or time</td>
</tr>
<tr>
<td>$u, v, w$</td>
<td>Displacement in $x, y$ and $z$ – direction, respectively</td>
</tr>
<tr>
<td>$x(t)$</td>
<td>Time dependent displacement used in equation of motion</td>
</tr>
<tr>
<td>$x, y, z$</td>
<td>Cartesian coordinates</td>
</tr>
</tbody>
</table>
Greek Symbols

\[ \begin{align*}
\Omega & \quad \text{Plate area} \\
\alpha & \quad \text{Slope in plate bending} \\
\gamma & \quad \text{Shear strain} \\
\delta & \quad \text{Elongation} \\
\epsilon & \quad \text{Strain} \\
\kappa & \quad \text{Curvatures} \\
\theta & \quad \text{Angle of fibre rotation about reference axes } x \\
\sigma & \quad \text{Stress} \\
\tau & \quad \text{Shear stress} \\
\omega & \quad \text{Frequency of the impact system}
\end{align*} \]

Matrices and Vectors

\[ \begin{align*}
[A] & \quad \text{Extensional stiffness matrix} \\
[B] & \quad \text{Coupling stiffness matrix} \\
[D] & \quad \text{Bending stiffness matrix} \\
[Q] & \quad \text{Stiffness matrix} \\
[S] & \quad \text{Compliance matrix} \\
[T] & \quad \text{Rotation matrix} \\
\{\sigma\} & \quad \text{Stress vector} \\
\{\epsilon\} & \quad \text{Strain vector}
\end{align*} \]

Subscripts

\[ \begin{align*}
1, 2, 3 & \quad \text{Material directions} \\
L & \quad \text{Longitudinal direction of a laminate} \\
T & \quad \text{Transverse direction of a laminate} \\
U & \quad \text{Ultimate, Failure criteria for stresses and strains} \\
c & \quad \text{Composite} \\
f & \quad \text{Fibre} \\
m & \quad \text{Matrix} \\
\text{max} & \quad \text{Maximum displacement} \\
x, y, z & \quad \text{Cartesian coordinate directions}
\end{align*} \]

Units

\[ \begin{align*}
G & \quad \text{Giga. } 10^9 \\
J & \quad \text{Joule. [Nm]} \\
M & \quad \text{Mega. } 10^6
\end{align*} \]
Contents

Preface i
Abstract ii
Notation iii
Contents v
List of Tables x
List of Figures xi

1 Introduction 1
  1.1 Background ........................................ 1
  1.2 The Problem ....................................... 3
  1.3 Outline ........................................... 4

2 Fundamentals of Composites 5
  2.1 Introduction ....................................... 5
  2.2 Behaviour of Unidirectional Composites ............. 5
    2.2.1 Introduction .................................. 5
    2.2.2 Nomenclature .................................. 6
    2.2.3 Volume and Weight Fractions ................. 6
    2.2.4 Longitudinal Stiffness and Strength .......... 8
    2.2.5 Transverse Stiffness and Strength ............ 9
    2.2.6 Prediction of Shear Modulus and Poisson’s Ratio ... 11
  2.3 Analysis of an Orthotropic Lamina ................ 11
    2.3.1 Introduction .................................. 11
    2.3.2 Hooke’s Law for Orthotropic Materials ........ 13
    2.3.3 Special Orthotropic Lamina ................... 14
    2.3.4 Stress-Strain for General Orthotropic Lamina ... 17
2.4 Analysis of Laminated Composites .......................... 19
   2.4.1 Introduction ................................................. 19
   2.4.2 Strain and Stress Variation in a Laminate .......... 19
   2.4.3 Synthesis of Stiffness Matrix ......................... 22
   2.4.4 Strengths for an Orthotropic Lamina .................. 24
2.5 Failure Modes for Composites ................................. 26
   2.5.1 Introduction ................................................ 26
   2.5.2 Fibre Breakage .............................................. 26
   2.5.3 Matrix Deformation and Cracking ..................... 26
   2.5.4 Fibre Debonding ............................................. 27
   2.5.5 Fibre Pullout ............................................... 27
   2.5.6 Delamination Cracks ...................................... 27

3 Analytical Solution for Plate Deflection and Impact Models 29
   3.1 Introduction ................................................... 29
   3.2 Equation of Equilibrium ..................................... 29
   3.3 Deflection of an Orthotropic Plate ....................... 32
   3.4 The Contact Law ............................................... 33
      3.4.1 Introduction .............................................. 33
      3.4.2 Contact Between Two Isotropic Elastic Solids .... 34
      3.4.3 Indentation of a Laminate ............................. 35
   3.5 Impact Models .................................................. 37
      3.5.1 Spring-Mass Model ...................................... 37
      3.5.2 Energy-Balance Model .................................. 39

4 Impact Experiments and Static Deflection Tests .......... 42
   4.1 Introduction ................................................... 42
   4.2 Strain Gauges .................................................. 42
      4.2.1 Strain Gauge Setup ..................................... 43
      4.2.2 The Low Pass Filter in the Strain Amplifier .... 43
   4.3 The Rosand Instrumented Falling Weight Impactor Tests 44
      4.3.1 Introduction .............................................. 44
      4.3.2 Setting Parameters ..................................... 45
      4.3.3 Test Setup ................................................ 45
      4.3.4 Instrumented Results ................................... 46
      4.3.5 Visual results ........................................... 48
   4.4 The Accelerometer Calibration ............................... 53
      4.4.1 Introduction .............................................. 53
      4.4.2 Measured Results ....................................... 53
   4.5 Drop Weight Impactor Tests .................................. 54
4.5.1 Introduction .............................................. 54
4.5.2 The Setup ............................................. 54
4.5.3 The 60 J Impact ........................................ 59
4.5.4 The 152 J Impact ...................................... 60
4.6 Static Deflection of a Plate ................................ 64
  4.6.1 Introduction and Setup ............................. 64
  4.6.2 Results from 1. Setup .............................. 64
  4.6.3 Results from 2. Setup .............................. 65
4.7 Summary .................................................. 67

5 AUTODYN Impact Simulations ................................. 69
  5.1 Introduction ............................................ 69
  5.2 Theory Overview ....................................... 69
    5.2.1 Lagrange Processor ............................. 70
    5.2.2 Shell Processor .................................. 71
    5.2.3 Processor Coupling ............................. 72
    5.2.4 Material Models ................................. 72
    5.2.5 Material Stress and Strain Failure ............ 73
  5.3 Material Properties and Material Models .............. 73
  5.4 Testing the Material Properties and the Material Models 77
  5.5 Simulations of the Rosand Impacts .................. 77
    5.5.1 60 J Impact Simulations ....................... 79
    5.5.2 40 J Impact Simulations ....................... 86
    5.5.3 20 J Impact Simulations ....................... 86
  5.6 Simulations of the Drop Weight Impacts .............. 88
    5.6.1 The 60 J Impact Simulation ................... 91
    5.6.2 The 152 J Impact Simulation ................... 93
  5.7 Summary ................................................ 96

6 ANSYS Simulations ........................................... 98
  6.1 Introduction ........................................... 98
  6.2 Comparing Element Types ............................. 98
    6.2.1 An Orthotropic Plate .......................... 99
    6.2.2 An Orthotropic Layered Plate ................ 102
  6.3 Simulated Plate Stiffness ............................ 104
  6.4 Stress and Strain Analyses ........................... 106
    6.4.1 The Clamped Plate ............................ 107
    6.4.2 The Simply Supported Plate ................... 109
  6.5 Summary ................................................ 115
# Comparison of Analytical Solutions, Simulations and Experiments

7.1 Introduction .............................................. 116
7.2 60 J Rosand Impacts ........................................ 116
  7.2.1 Energy and Deflection .............................. 117
  7.2.2 Damage Area and Damage Pattern .................. 120
7.3 40 J Rosand Impacts ........................................ 123
  7.3.1 Energy and Deflection .............................. 123
  7.3.2 Damage area ......................................... 126
7.4 20 J Rosand Impacts ........................................ 127
  7.4.1 Energy and Deflection .............................. 127
  7.4.2 Damage Area ......................................... 129
7.5 60 J Drop Weight Impact ................................... 129
  7.5.1 Energy and Deflection .............................. 132
  7.5.2 Strain Comparison ................................... 132
7.6 152 J Drop Weight Impact ................................ 133
  7.6.1 Energy and Deflection .............................. 133
  7.6.2 Damage and Damage Pattern ...................... 135
  7.6.3 Strain Comparison ................................... 135

# Summary and Conclusion

8.1 Summary .................................................. 138
  8.1.1 The Rosand Instrumented Falling Weight Impacts ... 139
  8.1.2 The Drop Weight Impacts ............................ 140
8.2 Conclusion ............................................... 141
8.3 Further Work ............................................ 142

References .................................................. 143

A Attaching Strain Gauges to Specimens .................. 146

B Plots from Impact Experiments .......................... 148
  B.1 Introduction ............................................ 148
  B.2 Plots from 40 J Rosand Instrumented Impacts ...... 148
  B.3 Plots from 20 J Rosand Instrumented Impacts ...... 150

C MAPLE Worksheets, ANSYS Files and PYTHON Scripts 152
  C.1 Introduction ............................................ 152
  C.2 MAPLE Worksheets ...................................... 152
    C.2.1 The Composite Package ............................ 152
    C.2.2 Deflection of a Layered Laminate ............... 154
C.2.3 Analytical Impact Model ......................... 156
C.3 ANSYS Input Files ................................. 157
  C.3.1 Strain Analyses for Clamped Plate .......... 157
  C.3.2 Nonlinear Solution for Applied Deflection . 161
C.4 PYTHON Scripts .................................. 164
  C.4.1 The *make plot* Script ....................... 164
  C.4.2 The *split columns* Script .................. 166
List of Tables

4.1 Plates impacted in the Rosand device. .......................... 46
4.2 Delamination area for impacted laminate plates .............. 52
4.3 Impact properties for plates used in manual drop weight tests .... 57

5.1 Properties for the projectile and the frame. ..................... 75
5.2 Parameters for laminate plate in AUTODYN simulation .......... 75
5.3 Material properties for the laminate plate ...................... 76
5.4 Simulation names for the 60 J Rosand impact. .................. 79
5.5 Material properties for simulation E60-3k and E60-3e .............. 82
5.6 Simulation names for 40 J impact onto clamped plates .......... 87
5.7 Velocity and mass for the three 20 J simulations. ............... 88
5.8 Material properties for the S3-a, S3-c and S3-d simulations. ... 91
5.9 Material properties for simulations S4-c and S4-k. ............... 95

6.1 Number of elements, nodes and DOF for each element type. .... 100
6.2 Orthotropic material properties .................................. 100
6.3 Deflection of square orthotropic plate for pressure load. ....... 102
6.4 Material properties for an unidirectional layer .................. 102
6.5 Number of elements, nodes and DOF for layered elements. ....... 102
6.6 Simulated deflection for layered elements. ...................... 103

7.1 Velocity and mass for the three 60 J simulations and tests. .... 117
7.2 Velocity and mass for the three 40 J simulations and tests. ...... 123
7.3 Velocity and mass for the three 20 J simulations and tests. ...... 127
List of Figures

2.1 Schematic representation of unidirectional composite ............... 6
2.2 Forces in an unidirectional composite ............................... 8
2.3 A simple model for predicting transverse properties ................. 10
2.4 Deformation behaviour of materials ..................................... 12
2.5 Components of the stress tensor ............................................. 14
2.6 A special orthotropic lamina .................................................. 15
2.7 Deformation behaviour of a special orthotropic lamina ................. 16
2.8 General orthotropic lamina ..................................................... 18
2.9 Bending of a line element ..................................................... 20

3.1 Moment and shear forces on a midplane of a plate ....................... 31
3.2 Pressure distribution in the contact zone .................................. 34
3.3 Spring-mass model ............................................................. 38
3.4 Displacement of plate and impactor ......................................... 38

4.1 The logging equipment ......................................................... 44
4.2 The Rosand Instrumented Falling Weight device .......................... 45
4.3 The Rosand Falling Weight setup .............................................. 46
4.4 Time Force diagram for 60 J impact .......................................... 47
4.5 Schematic representation of a impact ........................................... 48
4.6 Displacement and energy for 60 J impact ..................................... 49
4.7 Delamination between two lamina .............................................. 49
4.8 Delamination in side cut ........................................................ 50
4.9 Delamination shape ............................................................. 51
4.10 Different types of matrix cracks .............................................. 51
4.11 Delamination pattern on bottom side of 60 J impact ..................... 52
4.12 Acceleration from accelerometer test ....................................... 54
4.13 Setup for drop weight impacts ............................................... 55
4.14 Pictures of drop weight setup ............................................... 56
4.15 Comparison of different filters in strain amplifier ........................ 57
4.16 Strain measured at first 60 J impact ........................................ 58
4.17 Strain measured at second 60 J impact ................................. 59
4.18 60 J impact on laminate ..................................................... 61
4.19 Strain measured at 152 J impact ........................................... 62
4.20 152 J impact on laminate ................................................... 63
4.21 Models used in static deflection simulation ......................... 65
4.22 Force-deflection plots for a small plate ............................... 66
4.23 Force-deflection plots for a large plate .................................. 67

5.1 Grid before and after deformation ........................................ 70
5.2 Geometry of plate AUTODYN models .................................... 78
5.3 The uniform grid used in AUTODYN simulations .................... 78
5.4 Kinetic energy for different values for maximum shear stress ... 79
5.5 Failure plot for E60-3c ..................................................... 80
5.6 Energy plot for 60 J simply supported plate ......................... 81
5.7 Energy error for 60 J impact E60-3a ................................... 81
5.8 60 J simulations compared to the experiment ....................... 82
5.9 Failure plot for the 60 J simulation ..................................... 83
5.10 Simulation with material strain failure model ..................... 84
5.11 Failure plot for simulation E60-3h .................................... 85
5.12 Comparison of material failure models ............................... 85
5.13 Comparison of E60-1k, E60-2k and E60-3k ......................... 86
5.14 Three 40 J impacts with different masses ......................... 87
5.15 Three 20 J impacts with different masses ......................... 88
5.16 Geometry of frame used in simply supported plate simulation . 89
5.17 Grid used in AUTODYN simulation for simply supported plate. 90
5.18 Energy comparison from the simulations S3-a, S3-c and S3-d ... 91
5.19 Kinetic energy from 60 J drop impact simulation .................. 92
5.20 Comparison of the displacement for the simulations S3-k and S3-c ................................................................. 92
5.21 Energy plot for 60 J simply supported simulation ................ 93
5.22 Energy error plot for 60 J simulation .................................. 94
5.23 Failure for 60 J drop weight impact simulation S3-k ............ 94
5.24 Kinetic energy from 152 J impact simulations ................... 95
5.25 Displacement 152 J simulations on simply supported plate .... 96

6.1 Model used for static simulation in ANSYS ............................ 99
6.2 Different element types used in the ANSYS simulations ........... 101
6.3 Different layered elements used in ANSYS simulations .......... 104
6.4 Models used for static deflection simulations ..................... 105
6.5 Grid used in ANSYS for simulation of small plate ............... 105
6.6 Comparison of different elements and grid .................. 106
6.7 Model used in stress and strain analyses for clamped plate ... 107
6.8 Tension stress for clamped plate .......................... 108
6.9 In-plane shear stress for clamped plate ...................... 109
6.10 Inter-laminar shear stress for clamped plate ................. 110
6.11 Simulated strain in clamped plate ......................... 110
6.12 Tensile stress of simply supported plate .................... 111
6.13 Inter-laminar shear stress of simply supported plate ....... 112
6.14 ANSYS strain paths for simply supported plate ............. 113
6.15 Strain at paths for simply supported plate ................ 114
6.16 Strain plot for elements at path OA ....................... 114

7.1 60 J impact energy comparison .............................. 118
7.2 60 J impact deflection comparison ......................... 119
7.3 Damage from 60 J impact .................................. 120
7.4 Damage area in experiment and AUTODYN ................... 121
7.5 AUTODYN and ANSYS failure comparison .................. 122
7.6 40 J impact energy comparison ............................. 124
7.7 40 J impact deflection comparison ......................... 125
7.8 Damage from 40 J impact .................................. 126
7.9 Damage pattern for 40 J simulation ......................... 127
7.10 20 J impact energy comparison ............................ 128
7.11 20 J impact deflection comparison ......................... 130
7.12 Damage for 20 J experiment and simulation ................. 131
7.13 Energy comparison for 60 J drop weight impact ............ 131
7.14 Deflection comparison 60 J drop weight impact ............ 132
7.15 Strains from ANSYS and 60 J impact test ................... 133
7.16 Energy comparison for 152 J drop weight impact .......... 134
7.17 Deflection comparison 152 J drop weight impact .......... 134
7.18 Damage done by 152 J impact ............................. 135
7.19 Damage pattern for 152 J impact simulation ............... 136
7.20 Strains from ANSYS and 152 J impact test ................ 136

B.1 Force plot for the 40 J impacts ............................ 149
B.2 Displacement and energy plot for the 40 J impacts ....... 149
B.3 Force plot for 20 J impacts ............................... 150
B.4 Displacement and energy plot for 20 J ..................... 151
Chapter 1

Introduction

1.1 Background

The word composite implies that "it is a material composed of two or more distinct components" [27]. It is only when the constituent phases have significantly different physical properties and thus the composite properties are noticeably different from the constituent properties that we have come to recognize these materials as composites.

The idea of combining two or more different materials resulting in a new material with improved properties is very old. For more than 3000 years humans have been putting fibres into a binder or a matrix body. For example, in biblical times, the Pharaohs of Egypt and the ancient Incan and Mayan societies knew that plant fibres helped strengthen and prevent bricks and pottery from cracking. Leonardo da Vinci (1452-1519) conceived and used composite parts. He understood the advantages of laminating wood, metals, paper and fabric to produce a material with new characteristics and properties. This historic part is found in Richardson [27]. At the time around mid-1950s large investments were done in new composite materials. The driving forces for these investments were low weight and high strength materials for use in aero- and space structures. Glass fibre reinforced polymer became the most common used composite material and are still widely used.

Fibre reinforced composite materials offer enormous potential for use in a wide number of engineering applications. The applications ranging from sports to advanced aircraft and space shuttle structures. The superior stiffness and strength properties of long fibre composites can be utilized to manufacture complex components with lower weight at reduced cost.

Laminated fibre reinforced composites are known to be susceptible to damage resulting from accidental impact by foreign objects. Impact on aircraft
structures, for instance, from dropped tools, hail, and debris thrown up from the runway, poses a problem of great concern to designers. Such impacts can reasonably be expected during the life of the structure and can result in internal damage that is often difficult to detect and cause severe reductions in the strength and stability of the structure. This concern provided the motivation for intense research the last two decades. The study of impact on composite structures involves many different topics, including contact mechanics, structural dynamics, strength, stability, fatigue, damage mechanics and micromechanics. Impacts on laminated composite are simple events with many complicated effects.

A first step in gaining some understanding of the problem is to develop mathematical models for predicting the force applied by the projectile on the structure during impact and deflection of the structure.

Experiments have been and are still the best way to study impacts. Both low velocity and high velocity impacts are done. Low velocity impacts are typical used to study damage done by for example tool drop on a structure. A bird colliding with an airplane wing or a projectile fired from a gun are two examples of high velocity impacts. Many different instrumented impact machines are used in such experiments. They provide excellent results and are easy to use, but they are very expensive and the size of the specimen and environment often are limited. A lot of work has been done on these topics and hundreds of articles are written, such as [13, 14, 24, 33]. Abrate [1] summarizes many of the articles in his book.

During the last decades there has been a tremendous advancement of computer hardware, numerical algorithms and scientific software [20]. Engineers and scientists are now equipped with tools that make it possible to explore realworld applications of high complexity by computer simulations. The finite element method [11, 20] and the finite difference method [20] are the most used numerical methods in numerical engineering software. To perform impact si-
1.2 The Problem

In this thesis experimental impacts and numerical simulations of a glass fibre reinforced polymer (GFRP) composite laminate will be studied. The laminate consists of 4 multiaxial layers. A multiaxial layer consist of a $0^\circ$ fibre orientation layer and a $90^\circ$ fibre direction layer woven together. To make the laminate symmetric about the midplane the two bottom multiaxial layers are turned upside-down. The laminate can then be presented as $[0/90/0/90]_s$, where the subscript "$s$" indicate a symmetric layup. The thickness of the laminate is 5.2 mm.

Impacts test will be performed with a Rosand Instrumented Falling Weight machine and a self constructed impact device. The specimens are either $175 \times 175$ mm$^2$ or $300 \times 300$ mm$^2$. In Rosand device the specimen can not be larger than $175 \times 175$ mm$^2$, so an impact device for large specimen is needed and this drop weight impact device must be constructed. The Rosand device has a sensor logging the force during the impact. To receive the same information from the drop weight impacts the projectile has to be instrumented with an accelerometer. The plates will be instrumented with strain gauges, which can measure strain during the impact. Different energies and impact masses will be used to study their effects on damage and deflection. The nature of the damage will be characterized.

All impacts performed will be simulated using AUTODYN. AUTODYN is commercial software used for simulation of strong time dependent problems. Failure criteria for the material can be specified and failure plot for the simulation can be found. Comparison of the simulated damage area and the damage from the experiments will be carried out. Also energy and deflection plots will be compared. When knowing the deflection form, the impact stress and strain analyses will be performed using ANSYS.
1.3 Outline

In Chapter 2 the fundamental theory made use of in this thesis will be presented. Analytical expressions for material properties are found for unidirectional lamina and stress-strain relations are found for both a unidirectional lamina and a laminate. To understand the characteristic failures in a laminate during impact, a short introduction of the material failure models and the laminate strengths are given. From classic plate theory the equation of equilibrium for a plate is derived in Chapter 3. The contact law between two smooth solids and indentation of a laminate are also discussed. Analytical solutions of plate deflection for an orthotropic plate and two impact models are found. The two impact models are a spring-mass model and a energy-balance model. All the experiments performed are explained in Chapter 4. A short introduction to strain gauges and logging equipment is given before the details of two different impact setups are provided. The results from the impacts are discussed. In the last section of the chapter the static deflection experiments are described and compared to ANSYS simulations. In Chapter 5, AUTODYN simulations of the impact experiments are given. Material properties are discussed and a few impact tests are used to calibrate the material properties. Static simulations of plate deflection are done using ANSYS in Chapter 6. Element comparisons are done first to decide which element to use in the further simulations. Both layered and non-layered elements are tested. Force-deflection simulation and stress and strain analyses are done. Comparison of the experiments, simulations and the analytical models are carried out in Chapter 7.
Chapter 2

Fundamentals of Composites

2.1 Introduction

In the following sections the general theory made use of will be present. First material properties for a unidirectional lamina are derived. In the ANSYS simulations and the analytical solution the laminate will be modelled as 8 individual layers. Properties for each layer are then required. The material properties in z direction of the laminate could not be determined by material tests, so also here analytical expressions are needed. Stress and strain variations in a laminate are studied and will be used in the analytical solution of orthotropic plate deflection. The failure model used in AUTODYN simulations and the material failure modes in composite laminate will be introduced to better understand the simulations and material impact failures. Most of the theory is found in the textbooks [2, 26] and the articles [7, 8, 13].

2.2 Behaviour of Unidirectional Composites

2.2.1 Introduction

The oldest and one of the most widely used composite materials are fibre reinforced composites. Most of the structural elements or laminates made of fibrous composites consist of several distinct layers of unidirectional plies. Each ply is usually made of the same constituent materials (e.g., resin and glass), but an individual layer may differ from another layer in

1. relative volumes of the constituent materials

2. form of the reinforcement such as continuous or discontinuous materials
3. orientation of fibres with respect to common reference axes.

To analyze a structural element it is required to have complete knowledge of the properties of all individual layers. This theory follows [2].

### 2.2.2 Nomenclature

A unidirectional composite consist of parallel fibres embedded in a matrix. Several unidirectional layers can be stacked in a specific sequence of orientation to fabricate a laminate. Fig. 2.1 shows a unidirectional composite also called layer, ply or lamina. The directions in a ply are called *longitudinal direction* and *transverse direction* and they are, respectively, parallel and perpendicular to the fibres. For a typical ply the fibres are randomly distributed throughout the cross section and may be in contact with each other in some locations.

An unidirectional composite shows different properties in the longitudinal and transverse direction. Thus it is orthotropic with the axis 1, 2 and 3.

![Schematic representation of unidirectional composite](image)

**Figure 2.1:** Schematic representation of unidirectional composite

### 2.2.3 Volume and Weight Fractions

An important factor determining the properties of composites is the relative proportions of the matrix and reinforcing materials. The relative proportions can be given as the weight fractions or the volume fractions. The weight
fractions can be obtained during fabrication or by experiment after fabrication. A burnout test is commend used to determine the fibre fraction. The volume fractions are used in theoretical analysis of composite materials. It is thus desirable to determine the expressions for conversion between the weight fractions and volume fractions.

Let \( v_c, v_f \) and \( v_m \) be the volume and \( w_c, w_f \) and \( w_m \) be the weight for composite material, fibres and matrix material, respectively. The volume and weight fractions are defined as:

\[
v_c = v_f + v_m \\
V_f = \frac{v_f}{v_c}, \quad W_m = \frac{v_m}{v_c} \tag{2.1}
\]

and

\[
w_c = w_f + w_m \\
W_f = \frac{w_f}{w_c}, \quad W_m = \frac{w_m}{w_c} \tag{2.2}
\]

Eq. (2.3) can be written as

\[
\rho_c v_c = \rho_f v_f + \rho_m v_m \tag{2.5}
\]

where \( \rho \) is the density of the materials. Dividing both side of Eq. (2.5) by \( v_c \) yields

\[
\rho_c = \rho_f \frac{v_f}{v_c} + \rho_m \frac{v_m}{v_c} = \rho_f V_f + \rho_m V_m \tag{2.6}
\]

The density of the composite material can be written in terms of weight fractions

\[
\rho_c = \frac{1}{\left(W_f/\rho_f\right) + \left(W_m/\rho_m\right)} \tag{2.7}
\]

The conversion between the weight and volume fractions are

\[
W_f = \frac{w_f}{w_c} = \frac{\rho_f v_f}{\rho_c v_c} = \frac{\rho_f V_f}{\rho_c V_c} \tag{2.8}
\]

\[
W_m = \frac{\rho_m V_m}{\rho_c} \tag{2.9}
\]

And the inverse relations

\[
V_f = \frac{\rho_c}{\rho_f} W_f \tag{2.10}
\]

\[
V_m = \frac{\rho_c}{\rho_m} W_m \tag{2.11}
\]
2.2.4 Longitudinal Stiffness and Strength

A unidirectional composite lamina may be modelled by assuming fibres to be uniform in properties and diameter, continuous and parallel throughout the lamina. Also assuming perfect bonding, that is equal strain of matrix, fibres and composite

$$\epsilon_c = \epsilon_f = \epsilon_m$$  \hspace{2cm} (2.12)

The force on the composite, \(F_c\), is shared between the fibres, \(F_f\), and the matrix, \(F_m\).

$$F_c = F_f + F_m$$  \hspace{2cm} (2.13)

The cross-section areas are \(A_c\), \(A_f\) and \(A_m\). The force can then be written in term of stresses

$$F_c = \sigma_c A_c = \sigma_f A_f + \sigma_m A_m$$

or

$$\sigma_c = \sigma_f \frac{A_f}{A_c} + \sigma_m \frac{A_m}{A_c}$$  \hspace{2cm} (2.14)

With parallel fibres, the volume fraction are equal to the area fractions

$$V_f = \frac{A_f}{A_c}, \quad W_m = \frac{A_m}{A_c}$$  \hspace{2cm} (2.15)

Thus

$$\sigma_c = \sigma_f V_f + \sigma_m V_m$$  \hspace{2cm} (2.16)
2.2 Behaviour of Unidirectional Composites

Now Eq. (2.16) can be differentiated with respect to strain, which is the same for the composite, the fibres and the matrix.

\[
\frac{d\sigma_c}{d\epsilon} = \frac{d\sigma_f}{d\epsilon} V_f + \frac{d\sigma_m}{d\epsilon} V_m
\]  

(2.17)

d\sigma/d\epsilon can be replaced by the elastic module. Thus for linear elastic materials

\[
E_c = E_f V_f + E_m V_m
\]  

(2.18)

Eq. (2.18) are called rule of mixture For a composite with \( n \) materials Eq. (2.18) can be written

\[
E_c = \sum_{i=1}^{n} E_i V_i
\]

2.2.5 Transverse Stiffness and Strength

In the same manner as for longitudinal properties, a mathematical model for transverse properties can be derived. The fibres may be assumed to be uniform in properties and diameter, continuous and parallel throughout the composite. The composite is stressed in the transverse direction. Each layer in perpendicular to the direction of loading and has the same area on which the load acts as shown in Fig 2.3. That is

\[
\sigma_c = \sigma_f = \sigma_m
\]

Each layer is also assumed uniform in thickness so that the cumulative thickness of fibre layers and the matrix layers will be proportional to their respective volume fractions. The elongation of the composite, \( \delta_c \), in the direction of the load is the sum of the fibre elongation, \( \delta_f \), and the matrix elongation, \( \delta_m \).

\[
\delta_c = \delta_f + \delta_m
\]  

(2.19)

The elongation can be written as the product of the strain and its cumulative thickness, so that

\[
\delta_c = \epsilon_c t_c
\]

\[
\delta_f = \epsilon_f t_f
\]  

(2.20)

\[
\delta_m = \epsilon_m t_m
\]  

(2.21)

Substituting Eq. (2.20) into Eq. (2.19) gives

\[
\epsilon_c t_c = \epsilon_f t_f + \epsilon_m t_m
\]  

(2.22)
Figure 2.3: A simple model for predicting transverse properties.

Dividing both sides of Eq. (2.22) by $t_c$ and recognizing that the thickness is proportional to the volume fraction yields

$$
\epsilon_c = \frac{\epsilon_f t_f}{t_c} + \frac{\epsilon_m t_m}{t_c} = \epsilon_f V_f + \epsilon_m V_m
$$

(2.23)

If the fibres and the matrix are now assumed to deform elastically, the strain can be written in terms of the corresponding stress and the elastic modulus as follows:

$$
\frac{\sigma_c}{E_c} = \frac{\sigma_f}{E_f} V_f + \frac{\sigma_m}{E_m} V_m
$$

(2.24)

This can be simplified as

$$
\frac{1}{E_c} = \frac{V_f}{E_f} + \frac{V_m}{E_m}
$$

(2.25)

Eq. (2.25) can be generalized for a composite with $n$ number of materials

$$
E_c = \frac{1}{\sum_{i=1}^{n} (V_i/E_i)}
$$

Because the fibres are dispersed in the matrix material in a random fashion, the load will be shared between the fibres and the matrix. The assumption that the stress in the fibres and the matrix are equal is inaccurate. Another inaccurate in the solution arises due to the stresses in the fibres and matrix perpendicular to the load with no net resultant force on the composite in that direction. A mathematically rigorous solutions with a complete match of displacements
across the boundary between the fibre and the matrix is accomplished through the use of the theory of elasticity.

Halpin and Tsai have developed simple and generalized equations to approximate the results of more exact micromechanics analyses. These equations are found in [2].

### 2.2.6 Prediction of Shear Modulus and Poisson’s Ratio

The shear modulus, $G_{LT}$, and the two Poisson ratios, $\nu_{LT}$ and $\nu_{TL}$, are determined in a similar way as for $E_L$ and $E_T$ giving:

$$G_{LT} = \frac{G_f G_m}{G_m V_f + G_f V_m} \quad (2.26)$$

and

$$\nu_{LT} = \frac{\nu_f V_f + \nu_m V_m}{E_L \nu_T} \quad (2.27)$$

$\nu_{LT}$ is called major Poisson’s ratio and relates the longitudinal stress to the transverse strain. Minor Poisson’s ratio is $\nu_{TL}$ and relates the transverse strain to the longitudinal strain. The relation between major Poisson and minor Poisson is given by

$$\frac{\nu_{LT}}{E_L} = \frac{\nu_{TL}}{E_T}$$

### 2.3 Analysis of an Orthotropic Lamina

#### 2.3.1 Introduction

A single layer of a laminated composite material is generally referred to as a ply or a lamina. It usually contains a single layer of reinforcement, unidirectional or multi directional. A single ply is generally too thin to be directly used in any engineering application. Several plies are bonded together to form a structure termed a laminate. Properties and orientation of the plies in a laminate are chosen to meet the laminate design requirements. Properties of a laminate may be predicted by knowing the properties of its constituent plies. Behaviour of the laminate is governed by the behaviour of individual plies.

Fibre composites are among the class of materials called orthotropic materials. The behaviour of orthotropic materials lies between that of isotropic and that of anisotropic materials. The difference between these materials can best be explained through their response to tensile and shear loads. An isotropic material Fig. 2.4 (a) will under uniaxial load get an elongation in the load direction and a shortening in the perpendicular direction. There will be
Figure 2.4: Deformation behaviour of materials. Response to uniaxial tension and pure shear: (a) isotropic material; (b) anisotropic and general orthotropic; (c) special orthotropic material.
no change in the angle between two adjacent sides. A pure shear load will produce distortion of the specimen through changes in angle between its adjacent sides but will cause no change in lengths. The deformation behaviour of isotropic materials is direction independent and is characterized by "normal stress produce normal strain only but no shear strain" and "shear stresses produce shear strain only but no normal strain."

In the case of anisotropic materials, uniaxial tension will produce changes in lengths as well as in angle, shown in Fig 2.4 (b). Equal loads applied in different directions produce unequal changes in lengths and angle. Also, the deformation behaviour of anisotropic material is direction-dependent.

Deformation response of an orthotropic material, in general, is similar to that of the anisotropic material. That is, it is direction dependent and normal stresses and shear stresses alike, gives rise to normal strain as well as shear strain. In special cases, when the load in applied in some specific directions, the material response is similar to that of isotropic material. These directions with special behaviour are the axes of symmetry of the material. In a unidirectional composite longitudinal and transverse directions are the axes of symmetry.

### 2.3.2 Hookes’s Law for Orthotropic Materials

In general, the state of stress at a point in a body is described by the nine components of the stress tensor \( \sigma_{ij} \), as shown in Fig. 2.5. Correspondingly, there is a strain tensor, \( \epsilon_{ij} \), with nine components. The most general linear relationship that connects stress to strain is known as the generalized Hookes’s law [2] and can be expressed as

\[
\sigma_{ij} = E_{ijkl}\epsilon_{kl} \tag{2.28}
\]

where the 81 components of the fourth-order tensor \( E_{ijkl} \) are known as the elastic constants. For a orthotropic material \( E_{ijkl} \) in Eq. (2.28) will be reduced to 9 elastic constants.

\[
\begin{align*}
\sigma_1 &= \begin{bmatrix} Q_{11} & Q_{12} & Q_{13} & 0 & 0 & 0 \\ Q_{12} & Q_{22} & Q_{23} & 0 & 0 & 0 \\ Q_{13} & Q_{23} & Q_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & Q_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & Q_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & Q_{66} \end{bmatrix} & \epsilon_1 \\
\sigma_2 & \\
\sigma_3 & \\
\tau_{23} & \\
\tau_{31} & \\
\tau_{12} & 
\end{align*}
\]

where \([Q_{ij}]\) is the stiffness matrix and \( \epsilon_j \) are the engineering strain components. Engineering shear strain is twice the corresponding tensorial shear strain.
Figure 2.5: Components of the stress tensor.

To specialize the relations for a two-dimensional case, all the terms related to the $x_3$ axis may be dropped to simplify the stress-strain relation to

$$
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\tau_{12}
\end{bmatrix}
=
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\begin{bmatrix}
\epsilon_1 \\
\epsilon_2 \\
\gamma_{12}
\end{bmatrix}
$$

(2.30)

These fundamental relations are implemented in both AUTODYN and ANSYS, and will be used for modelling of orthotropic materials.

### 2.3.3 Special Orthotropic Lamina

Fig. 2.6 shows a *special orthotropic lamina*, where the reference axes coinciding with the axes of symmetry and designated as the longitudinal direction, L, and the transverse direction, T. Engineering constants for this lamina are:

- $E_L$ and $E_T$. Elastic modulus in longitudinal and transverse direction, respectively.
- $G_{LT}$. Shearing modulus
- $\nu_{LT}$ and $\nu_{TL}$. The major Poisson’s ratio and the minor Poisson’s ratio.
The significance of these constants can be explained by considering the deformation response of the lamina, as illustrated in Fig. 2.7 to the following states of stress:

i When $\sigma_L$ is the only nonzero stress ($\sigma_T = \tau_{LT} = 0$), the strain produced are

\[
\epsilon_L = \frac{\sigma_L}{E_L} \quad (2.31)
\]
\[
\epsilon_T = -\nu_{LT}\epsilon_L = -\nu_{LT}\frac{\sigma_L}{E_L} \quad (2.32)
\]
\[
\gamma_{LT} = 0 \quad (2.33)
\]

ii When $\sigma_T$ is the only nonzero stress ($\sigma_L = \tau_{LT} = 0$), the strain produced are

\[
\epsilon_T = \frac{\sigma_T}{E_T} \quad (2.34)
\]
\[
\epsilon_L = -\nu_{TL}\epsilon_T = -\nu_{TL}\frac{\sigma_T}{E_T} \quad (2.35)
\]
\[
\gamma_{LT} = 0 \quad (2.36)
\]

iii When $\tau_{LT}$ is the only nonzero stress ($\sigma_L = \sigma_T = 0$), the strain produced are

\[
\epsilon_T = 0 \quad (2.37)
\]
\[
\epsilon_L = 0 \quad (2.38)
\]
\[
\gamma_{LT} = \frac{\tau_{LT}}{G_{LT}} \quad (2.39)
\]
Figure 2.7: Deformation behaviour of a special orthotropic lamina.
2.3 Analysis of an Orthotropic Lamina

Superposition of these three states of stresses gives a most general state of stress on the lamina consisting of $\sigma_L$, $\sigma_T$ and $\tau_{LT}$.

\[
\begin{align*}
\epsilon_L &= \frac{\sigma_L}{E_L} - \nu_{TL} \frac{\sigma_T}{E_T} \\
\epsilon_T &= \frac{\sigma_T}{E_T} - \nu_{LT} \frac{\sigma_L}{E_L} \\
\gamma_{LT} &= \frac{\tau_{LT}}{G_{LT}}
\end{align*}
\]

(2.40)

This is the stress-strain relations of a special orthotropic lamina in terms of engineering constants.

2.3.4 Stress-Strain for General Orthotropic Lamina

A lamina referred to arbitrary axes is called a general orthotropic lamina. Consider an orthotropic lamina with its principal material axes oriented at an angle $\theta$ with the reference coordinate axes as shown in Fig. 2.8. The following transformation relations for transforming stresses and strains from $x - y$ to L-T axes:

\[
\begin{bmatrix}
\sigma_L \\
\sigma_T \\
\tau_{LT}
\end{bmatrix} = [T] \begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}
\]

(2.41)

and

\[
\begin{bmatrix}
\epsilon_L \\
\epsilon_T \\
\frac{1}{2} \epsilon_{LT}
\end{bmatrix} = [T] \begin{bmatrix}
\epsilon_x \\
\epsilon_y \\
\frac{1}{2} \epsilon_{xy}
\end{bmatrix}
\]

(2.42)

where $[T]$ is the transformation matrix given by

\[
[T] = \begin{bmatrix}
\cos^2 \theta & \sin^2 \theta & 2 \sin \theta \cos \theta \\
\sin^2 \theta & \cos^2 \theta & -2 \sin \theta \cos \theta \\
-\sin \theta \cos \theta & \sin \theta \cos \theta & \cos^2 \theta - \sin^2 \theta
\end{bmatrix}
\]

The angle $\theta$ is taken positive when the angle of the L-T axes measured from $x - y$ axes is in the counterclockwise direction.

The inverted form of Eq. (2.41) is

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} = [T]^{-1} \begin{bmatrix}
\sigma_L \\
\sigma_T \\
\tau_{LT}
\end{bmatrix}
\]

(2.43)
Figure 2.8: General orthotropic lamina.

The stress-strain relations referred to the L-T axes are given in matrix form by Eq. (2.30). When the tensorial rather than the engineering strains are used, the stiffness matrix of Eq. (2.30) has to be modified as follows:

$$
\begin{pmatrix}
\sigma_L \\
\sigma_T \\
\tau_{LT}
\end{pmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & 2Q_{66}
\end{bmatrix}
\begin{pmatrix}
\epsilon_L \\
\epsilon_T \\
\frac{1}{2}\gamma_{LT}
\end{pmatrix}
$$

(2.44)

Substituting Eq. (2.42) into Eq. (2.44) and then substituting the resulting equation in Eq. (2.43) yields

$$
\begin{pmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{pmatrix} =
[T]^{-1}
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & 2Q_{66}
\end{bmatrix}
[T]
\begin{pmatrix}
\epsilon_x \\
\epsilon_y \\
\frac{1}{2}\gamma_{xy}
\end{pmatrix}
$$

(2.45)

For the purpose of uniformity, a $[\bar{Q}]$ matrix similar to the $[Q]$ matrix of Eq. (2.30) is defined that relates engineering strains to the stresses referred to arbitrary axes. Then the $[\bar{Q}]$ matrix is defined by the equation

$$
\begin{pmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{pmatrix} =
\begin{bmatrix}
\bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\
\bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\
\bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66}
\end{bmatrix}
\begin{pmatrix}
\epsilon_x \\
\epsilon_y \\
\gamma_{xy}
\end{pmatrix}
$$

(2.46)

where
\[ \begin{align*}
\bar{Q}_{11} &= Q_{11} \cos^4 \theta + Q_{22} \sin^4 \theta + 2(Q_{12} + 2Q_{66}) \sin^2 \theta \cos^2 \theta \\
\bar{Q}_{22} &= Q_{11} \sin^4 \theta + Q_{22} \cos^4 \theta + 2(Q_{12} + 2Q_{66}) \sin^2 \theta \cos^2 \theta \\
\bar{Q}_{12} &= (Q_{11} + Q_{22} - 4Q_{66}) \sin^2 \theta \cos^2 \theta + Q_{12}(\cos^4 \theta + \sin^4 \theta) \\
\bar{Q}_{66} &= (Q_{11} + Q_{22} - 2Q_{12} - 2Q_{66}) \sin^2 \theta \cos^2 \theta + Q_{66}(\sin^4 \theta + \cos^4 \theta) \\
\bar{Q}_{16} &= (Q_{11} - Q_{12} - 2Q_{66}) \cos^3 \theta \sin \theta - (Q_{22} - Q_{12} - 2Q_{66}) \cos \theta \sin^3 \theta \\
\bar{Q}_{26} &= (Q_{11} - Q_{12} - 2Q_{66}) \cos \theta \sin^3 \theta - (Q_{22} - Q_{12} - 2Q_{66}) \cos^3 \theta \sin \theta
\end{align*} \]

The inverse stress-strain relations referred to arbitrary axes can now be written as

\[
\begin{bmatrix}
\epsilon_x \\
\epsilon_y \\
\gamma_{xy}
\end{bmatrix} =
\begin{bmatrix}
\bar{S}_{11} & \bar{S}_{12} & \bar{S}_{16} \\
\bar{S}_{12} & \bar{S}_{22} & \bar{S}_{26} \\
\bar{S}_{16} & \bar{S}_{26} & \bar{S}_{66}
\end{bmatrix}
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}, \tag{2.47}
\]

where the \([S]\) matrix is called the compliance matrix.

### 2.4 Analysis of Laminated Composites

#### 2.4.1 Introduction

Laminates are fabricated such that they act as single-layer material. The bond between two plies in a laminate is assumed to be perfect. That is infinitesimal thin and not shear deformable. Thus the plies can not slip over each other, and the displacements remain continuous across the bound. In this section equations are developed that relate the strain at any point in a thin laminate undergoing deformation to the displacements and curvatures. Then, recognizing the fact that the laminate consists of several plies with different directional properties, the variation of the stress across the thickness of the laminate are discussed. The relations derived in the two following sections are used in ANSYS composite shell elements and it will be used in the analytical solution for plate deflection.

#### 2.4.2 Strain and Stress Variation in a Laminate

Figure 2.9 shows the deformation of a section of a laminate in the \(x - z\) plane. The line \(ABCD\) is originally straight and perpendicular to the midplane of the laminate and remains straight and perpendicular to the midplane in the deformation state. This is equivalent to neglecting shearing deformations \(\gamma_{xz}\) and \(\gamma_{yz}\) and is also equivalent to assuming that the plies, that make up the
cross section, do not slip over each other. Further assume that the point \( B \) at the geometric midplane undergoes displacement \( u_0, v_0 \) and \( w_0 \) along \( x, y \) and \( z \) directions, respectively. The displacement \( u \) in the \( x \) direction of a point \( C \) that is located on the normal \( ABCD \) at a distance \( z \) from the midplane is given by

\[
u = u_0 - z\alpha
\]

(2.48)

where \( \alpha \) is the slope of the laminate midplane in the \( x \) direction, that is

\[
\alpha = \frac{\partial w_0}{\partial x}
\]

(2.49)

The displacement \( u \) in the \( x \) direction of an arbitrary point at a distance \( z \) from the midplane, can now be written

\[
u = u_0 - z\frac{\partial w_0}{\partial x}
\]

(2.50)

In the \( y \) direction the displacement, \( v \), can be found by similar reasoning

\[
v = v_0 - z\frac{\partial w_0}{\partial y}
\]

(2.51)

The displacement, \( w \), in the \( z \) direction of any point on \( ABCD \) is the displacement \( w_0 \) of the midplane plus the stretching of the normal. It is assumed that the stretching (or shortening) of the normal \((ABCD)\) is insignificant compared to the displacement, \( w_0 \). Thus the normal displacement of any point in

![Figure 2.9: Bending of a line element](image)

...
the laminate is taken equal to the displacement $w_0$ of the corresponding point at the midplane. The normal strains, $\varepsilon_z$, are then neglected. This reduces the laminate strains to $\varepsilon_x$, $\varepsilon_y$, and $\gamma_{xy}$. These strains can be obtained for the derived displacements, $u$ and $v$ as follows:

$$
\begin{align*}
\varepsilon_x &= \frac{\partial u}{\partial x} = \frac{\partial u_0}{\partial x} - z \frac{\partial^2 w_0}{\partial x^2} \\
\varepsilon_y &= \frac{\partial v}{\partial y} = \frac{\partial v_0}{\partial y} - z \frac{\partial^2 w_0}{\partial y^2} \\
\gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \frac{\partial u_0}{\partial y} + \frac{\partial v_0}{\partial x} - 2z \frac{\partial^2 w_0}{\partial x \partial y}
\end{align*}
$$

(2.52)

The preceding strain-displacement relations can be written in terms of the midplane strains and the plate curvatures as follows:

$$
\begin{align*}
\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} &= \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{bmatrix} + z \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix}
\end{align*}
$$

(2.53)

where the midplane strains are

$$
\begin{align*}
\begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{bmatrix} &= \begin{bmatrix} \frac{\partial u_0}{\partial x} \\ \frac{\partial v_0}{\partial y} \\ \frac{\partial u_0}{\partial y} + \frac{\partial v_0}{\partial x} \end{bmatrix}
\end{align*}
$$

(2.54)

and the plate curvatures are

$$
\begin{align*}
\begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix} &= - \begin{bmatrix} \frac{\partial^2 w_0}{\partial x^2} \\ \frac{\partial^2 w_0}{\partial y^2} \\ \frac{\partial^2 w_0}{\partial x \partial y} \end{bmatrix}
\end{align*}
$$

(2.55)

Equation (2.53) indicates that the strains in a laminate vary linearly across its thickness. Stresses in any lamina can be obtained by substituting Eq. (2.53) in the stress-strain relation (2.46) for the lamina as follows:

$$
\begin{align*}
\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}_k &= \begin{bmatrix} Q_{11} & Q_{12} & Q_{16} \\ Q_{12} & Q_{22} & Q_{26} \\ Q_{16} & Q_{26} & Q_{66} \end{bmatrix}_k \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{bmatrix} + z \begin{bmatrix} Q_{11} & Q_{12} & Q_{16} \\ Q_{12} & Q_{22} & Q_{26} \\ Q_{16} & Q_{26} & Q_{66} \end{bmatrix}_k \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix}
\end{align*}
$$

(2.56)
2.4.3 Synthesis of Stiffness Matrix

The stress in a laminate vary from layer to layer. Hence it is convenient to deal with a simpler but equivalent system of forces and moments acting on a laminate cross section. The resultant forces and moments acting on a laminate cross section are defined as follows [2]. Resultant force is obtained by integrating the corresponding stress through the laminate thickness, h:

\[
N_x = \int_{-h/2}^{h/2} \sigma_x \, dx \\
N_y = \int_{-h/2}^{h/2} \sigma_y \, dx \\
N_{xy} = \int_{-h/2}^{h/2} \tau_{xy} \, dx
\] (2.57)

and the resultant moment is obtained by integration through the thickness of the corresponding stress times the moment arm with respect to midplane:

\[
M_x = \int_{-h/2}^{h/2} \sigma_x z \, dx \\
M_y = \int_{-h/2}^{h/2} \sigma_y z \, dx \\
M_{xy} = \int_{-h/2}^{h/2} \tau_{xy} z \, dx
\] (2.58)

Consider a laminate consisting of \( n \) orthotropic layers. The force-moment system acting at the midplane of this laminate can be obtained by replacing the continuous integral in Eq. (2.57) and (2.58) by the summation of integrals representing the contribution of each layer as follows:

\[
\begin{align*}
\begin{bmatrix} N_x \\ N_y \\ N_{xy} \end{bmatrix} &= \int_{-h/2}^{h/2} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{bmatrix} \, dz = \sum_{k=1}^{n} \int_{h_{k-1}}^{h_k} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{bmatrix} \, dz \\
\end{align*}
\] (2.59)

and

\[
\begin{align*}
\begin{bmatrix} M_x \\ M_y \\ M_{xy} \end{bmatrix} &= \int_{-h/2}^{h/2} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{bmatrix} z \, dz = \sum_{k=1}^{n} \int_{h_{k-1}}^{h_k} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{bmatrix} z \, dz \\
\end{align*}
\] (2.60)

The resultant forces and moments can be directly related to the midplane strain and plate curvatures.
Observing that the stiffness matrix $[\bar{Q}]$ remains constant within a lamina and that the midplane strains and plate curvature remain constant for all the laminae. The result when inserting Eq. (2.56) into Eq. (2.59) and (2.60) can be written

$$
\begin{align*}
\begin{bmatrix} N_x \\ N_y \\ N_{xy} \end{bmatrix} &= \sum_{k=1}^{n} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix} \begin{bmatrix} e_x^0 \\ e_y^0 \\ \gamma_{xy}^0 \end{bmatrix} dz \int_{h_k}^{h_{k-1}} + \sum_{k=1}^{n} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix} \begin{bmatrix} e_x^0 \\ e_y^0 \\ \gamma_{xy}^0 \end{bmatrix} dz \int_{h_{k-1}}^{h} \\
\begin{bmatrix} M_x \\ M_y \\ M_{xy} \end{bmatrix} &= \sum_{k=1}^{n} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix} \begin{bmatrix} e_x^0 \\ e_y^0 \\ \gamma_{xy}^0 \end{bmatrix} dz \int_{h_k}^{h_{k-1}} + \sum_{k=1}^{n} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix} \begin{bmatrix} e_x^0 \\ e_y^0 \\ \gamma_{xy}^0 \end{bmatrix} dz \int_{h_{k-1}}^{h} z^2 \end{align*}
$$

(2.61)

(2.62)

By introducing definitions of three new matrices, Eq. (2.61) and (2.62) can be rewritten in a relatively simple form as follows:

$$
\begin{align*}
\begin{bmatrix} N_x \\ N_y \\ N_{xy} \end{bmatrix} &= \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix} \begin{bmatrix} e_x^0 \\ e_y^0 \\ \gamma_{xy}^0 \end{bmatrix} + \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix} \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix} \\
\begin{bmatrix} M_x \\ M_y \\ M_{xy} \end{bmatrix} &= \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix} \begin{bmatrix} e_x^0 \\ e_y^0 \\ \gamma_{xy}^0 \end{bmatrix} + \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{12} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix}
\end{align*}
$$

(2.63)

(2.64)

where

$$
A_{ij} = \sum_{k=1}^{n} (\bar{Q}_{ij})_{k} (h_{k} - h_{k-1})
$$

$$
B_{ij} = \frac{1}{2} \sum_{k=1}^{n} (\bar{Q}_{ij})_{k} (h_{k}^2 - h_{k-1}^2)
$$

$$
D_{ij} = \frac{1}{3} \sum_{k=1}^{n} (\bar{Q}_{ij})_{k} (h_{k}^3 - h_{k-1}^3)
$$

(2.65)
The matrices $[A]$, $[B]$ and $[D]$ are called the extensional stiffness matrix, coupling stiffness matrix and bending stiffness matrix, respectively. Equation (2.63) can no be written in short form.

$$\begin{pmatrix} N \\ M \end{pmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{pmatrix} \epsilon \phi \\ \kappa \end{pmatrix} \tag{2.66}$$

Composite laminate can be fabricated such that Eq. 2.66 are simplified. The laminate used in this thesis consist of 8 layers. The layer have either $0^\circ$ or $90^\circ$ fibre orientation. This cross stacking make $D_{16} = D_{26} = 0$. Because the laminate also have symmetric stacking, $[0, 90, 0, 90]$, $B_{ij} = 0$. The resultant forced $N$ are then independent of the curvature and the moment $M$ are independent of the midplane strain.

### 2.4.4 Strengths for an Orthotropic Lamina

Material strengths are experimentally obtained by subjecting suitable specimens to load that produce simple stress fields in the test specimen and by determining the load at which failure occurs. An example is ultimate tensile strength. This test produce only uniaxial tensile stress in the test section of the specimen. The strength model for a orthotropic material is considerably more complex then for a isotropic material. This because their strengths, like their elastic constant, are direction dependent. Thus, for an orthotropic material, an infinite number of strength values can be obtained even through uniaxial tests, depending of the direction of load application. For prediction purposes, they can be limited to five strengths in the principal material directions. These strengths are:

- Longitudinal tensile strength, $\sigma_{LU}$
- Transverse tensile strength, $\sigma_{TU}$
- Shear strength, $\tau_{LTU}$
- Longitudinal compressive strength, $\sigma'_{LU}$
- Transverse compressive strength, $\sigma'_{TU}$

Two widely used strengths theories for fibre composites are based on maximum stress and maximum strain. These two failure models are implemented and will be used in the AUTODYN software.
2.4 Analysis of Laminated Composites

Maximum Stress Theory

This theory states that failure will occur if any of the stresses in the principal material axes exceeds the corresponding allowable stress. To avoid failure the following inequalities must be satisfied:

\[
\sigma_L < \sigma_{LU} \\
\sigma_T < \sigma_{TU} \tag{2.67} \\
\tau_{LT} < \tau_{LTU}
\]

For compressive stresses, \(\sigma_{LU}\) and \(\sigma_{TU}\) in Eq. (2.67) must be replaced by the allowable compressive stresses:

\[
\sigma_L < \sigma'_{LU} \\
\sigma_T < \sigma'_{TU} \tag{2.68}
\]

The material is considered to have failed when any of the inequalities in Eq. (2.67) and (2.68) is violated. There is no interaction between the modes of failure in this criterion. Thus this is actually five subcriteria.

Maximum Strain Theory

This theory states that failure will occur if any of the strains in the principal material axes exceeds the corresponding allowable strain. Thus the following inequalities must be satisfied:

\[
\varepsilon_L < \varepsilon_{LU} \\
\varepsilon_T < \varepsilon_{TU} \tag{2.69} \\
\gamma_{LT} < \gamma_{LTU}
\]

If normal strains are compressive, \(\varepsilon_{LU}\) and \(\varepsilon_{TU}\) in Eq. (2.69) must be replaced by the allowable compressive strains:

\[
\varepsilon_L < \varepsilon'_{LU} \\
\varepsilon_T < \varepsilon'_{TU} \tag{2.70}
\]

This theory is similar to the maximum stress theory. All the stresses are replaced by the corresponding strains first to apply the maximum strain theory. If the material is assumed to be linearly elastic up to the ultimate failure, the
ultimate strain in Eq. (2.69) and (2.70) can be related directly to the strengths:

\[
\begin{align*}
\epsilon_{LU} &= \frac{\sigma_{LU}}{E_L} \\
\epsilon_{TU} &= \frac{\sigma_{TU}}{E_T} \\
\gamma_{LTU} &= \frac{\tau_{LTU}}{G_{LT}}
\end{align*}
\] (2.71)

2.5 Failure Modes for Composites

2.5.1 Introduction

Material failure for impact tests on composite laminate are very characteristic. The failure consist of several individual failure modes as described in this section.

A solid subjected to any kind of loading, static or impact, can absorb energy by two basic mechanisms: (1) creation of new surfaces and (2) material deformation. The material deformation occurs first. If the energy supplied is large enough, a crack may initiate and propagate, thus actuating the second energy-absorbing mechanism. In the case of brittle materials such as glass and other ceramics only a small amount of deformation takes place. The associated energy absorbed is also small. As a consequence, brittle materials exhibit a low energy-absorption capability. In ductile materials such as mild steel, aluminium and many other metallic materials, large plastic deformations take place during fracture. Thus large energies are absorbed during the fracture of ductile materials. It is, therefore, obvious that the total energy-absorbing capability or toughness of a material can be enhanced by increasing either the path of the crack during separation or the material-deformation capability.

There are various mechanisms involved during crack propagation [2, 13, 22].

2.5.2 Fibre Breakage

Whenever a crack has to propagate in the direction normal to the fibres, fibre breakage will eventually occur for complete separation of the laminate. Fibres will fracture when their fracture strain is reached.

2.5.3 Matrix Deformation and Cracking

The matrix material surrounding the fibres has to fracture to complete the fracture of the composite. Brittle materials can undergo only a limited defor-
mation prior to fracture. However, metal matrices may undergo extensive plastic deformation. Although cracking and deformation of the matrix materials both absorb energy, the energy required for plastic deformation is considerably higher than the surface energy contribution. Thus the contribution of metal matrices to the total impact energy of composites may be significant, whereas the contribution of polymer matrices may be relatively insignificant.

2.5.4 Fibre Debonding

During the fracture process the fibres may become separated from the matrix material by cracks running parallel to the fibres. In this process the chemical or secondary bonds between the fibres and the matrix material are broken. This type of cracking occurs when fibres are strong and the interface is weak.

2.5.5 Fibre Pullout

Fibre pullout occur when brittle or discontinuous fibres are embedded in a tough matrix. The fibres fracture at their weak cross sections that do not necessarily lie in the plane of composite fracture. The stress concentration in the matrix produced by the fibre breaks is relieved by matrix yielding, thus preventing a matrix crack that may join the fibre fractures at different points. In such cases the fracture may proceed by the broken fibres being pulled out of the matrix rather than fibres fracturing again at the plane of composite fracture.

The difference between fibre debonding and fibre pullout may be clarified at this point. Fibre debonding takes place when a matrix crack is unable to propagate across a fibre, whereas fibre pullouts are a results of the inability of a crack initiated at a fibre break to propagate into the tough matrix. The fibre pullouts are usually accompanied by extensive matrix deformation, which is absent in fibre debonding. Thus fibre debonding and fibre pullout may appear to be similar phenomena because of failure taking place at the fibre-matrix interface in both cases, but they are caused by mutually exclusive conditions. However, both phenomena do significantly enhance fracture energy.

2.5.6 Delamination Cracks

A crack propagating through a ply in a laminate may get arrested as the crack tip reaches the fibres in the adjacent ply. This process of crack arrestment is similar to the arrestement of a matrix crack at the fibre-matrix interface. Because of high shear stress in the matrix adjacent to the crack tip, the crack
may branch off and start running at the interface parallel to the plane of the plies. These cracks are called delaminations cracks, and whenever present they are responsible absorbing a significant amount of fracture energy. Delamination cracks frequently occur when laminates are tested in flexure, as in low velocity impact tests.
Chapter 3

Analytical Solution for Plate Deflection and Impact Models

3.1 Introduction

In this chapter the analytical solution of orthotropic plate deflection and two different impact models are derived. The analytical solution of plate deflection will later be used to verify static ANSYS models. Impact models are studied to see if they can provide useful information about force, contact duration and deflection. Two different impact models will be derived. One spring-mass model and one energy-balance model. In Section 7 the impact models are compared to the impact experiments done. Both the solution of plate deflection and the impact models were implemented in mathematical software MAPLE.

3.2 Equation of Equilibrium

In this section the equation of equilibrium for a plate is derived as in [30]. The equation will later be used to derive the equation of motion for an orthotropic plate. Following assumptions are made:

1. The load $q(x, y)$ is acting normal to the plate surface

2. Small deflection

3. The edges at the boundary are free to move in the plane of the plate, thus the reactive forces at the edges are normal to the plate

With these assumptions there are no strains in the the middle plane of the plate during bending. Studying the plate under pure bending in Figure 3.1,
there are vertical shearing forces $Q_x$ and $Q_y$ addition to the bending moment $M_x$, $M_y$ and the twisting moment $M_{xy}$. The shear forces are

$$Q_x = \int_{-h/2}^{h/2} \tau_{xz} \, dz \quad \text{and} \quad Q_y = \int_{-h/2}^{h/2} \tau_{yz} \, dz \quad (3.1)$$

and the bending moments are

$$M_x = \int_{-h/2}^{h/2} \sigma_{xz} \, dz, \quad M_y = \int_{-h/2}^{h/2} \sigma_{yz} \, dz \quad \text{and} \quad M_{xy} = \int_{-h/2}^{h/2} \tau_{xy} \, dz \quad (3.2)$$

The force equilibrium for the midplane, Fig. 3.1(a) will be

$$\frac{\partial Q_x}{\partial x} \, dx \, dy + \frac{\partial Q_y}{\partial y} \, dx \, dy + q(x, y) \, dx \, dy = 0$$

thus

$$\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} + q(x, y) = 0 \quad (3.3)$$

The moments of all the forces acting on the element with respect to the $x$ axis, will be

$$\frac{\partial M_{xy}}{\partial x} \, dx \, dy - \frac{\partial M_y}{\partial y} \, dx \, dy + Q_y \, dx \, dy = 0$$

there all higher order terms are neglected. Further simplification gives

$$\frac{\partial M_{xy}}{\partial x} - \frac{\partial M_y}{\partial y} + Q_y = 0 \quad (3.4)$$

Taking moment respected to the $y$ axis gives

$$-\frac{\partial M_{xy}}{\partial x} + \frac{\partial M_x}{\partial x} - Q_x = 0 \quad (3.5)$$

By eliminating the shear forces from Eq. (3.4-3.5) and inserting these in Eq. (3.3) the equation of equilibrium becomes

$$\frac{\partial^2 M_x}{\partial x^2} + \frac{\partial^2 M_y}{\partial y^2} - 2 \frac{\partial^2 M_{xy}}{\partial x \partial y} = -q(x, y) \quad (3.6)$$
Figure 3.1: (a) moment and (b) shear forces on an infinitesimal element of midplane of a plate.
3.3 Deflection of an Orthotropic Plate

The differential equation for a plate $\Omega \in [a, b]$ with orthotropic material is found by inserting Eq. (2.64) into (3.6). In this thesis a composites with symmetrical stacking of $0^\circ$ and $90^\circ$ lamina is studied, so $B_{ij} = 0$ and $D_{16} = D_{20} = 0$. Eq. (2.64) will then be

$$
\begin{bmatrix}
    M_x \\
    M_y \\
    M_{xy}
\end{bmatrix} =
\begin{bmatrix}
    D_{11} & D_{12} & 0 \\
    D_{12} & D_{22} & 0 \\
    0 & 0 & D_{66}
\end{bmatrix}
\begin{bmatrix}
    \kappa_x \\
    \kappa_y \\
    \kappa_{xy}
\end{bmatrix}
$$

(3.7)

Equation (3.7) combined with (2.55) inserted in (3.6) gives

$$
D_{11} \frac{\partial^4 w}{\partial x^4} + 2(D_{12} + 2D_{66}) \frac{\partial^4 w}{\partial x^2 \partial y^2} + D_{22} \frac{\partial^4 w}{\partial x^2 \partial y^2} = q
$$

(3.8)

For a static deflection of a simply supported rectangular plate according to classic plate theory, Navier solution will be an exact solution.

The boundary conditions for a simply supported plate are:

$$
w(0, y) = w(a, y) = w(x, 0) = w(x, b) = 0 \quad (3.9)\\nM_x(0, y) = M_x(a, y) = M_y(x, 0) = M_y(x, b) = 0 \quad (3.10)
$$

With the classic plate theory, the transverse displacement $w(x, y)$ and the transverse force $q(x, y)$ can be written as double Fourier series.

$$
w(x, y) = \sum_{m,n} W_{mn} \sin \frac{m\pi x}{a} \sin \frac{n\pi y}{b}
$$

(3.11)

$$
q(x, y) = \sum_{m,n} q_{mn} \sin \frac{m\pi x}{a} \sin \frac{n\pi y}{b}
$$

(3.12)

Substituting Eq. (3.11) and (3.12) into Eq. (3.9), (3.10) and (3.8) shows that both the equation of equilibrium and the boundary conditions are satisfied. Finding $W_{mn}$ by substituting Eq. (3.11) and (3.12)) into Eq. (3.8), multiplying through by $\sin \frac{m\pi x}{a} \sin \frac{n\pi y}{b}$, integrating over $\Omega$ and using the result

$$
\int_0^a \sin \frac{m\pi x}{a} \sin \frac{m'\pi x}{a} \, dx = \begin{cases} 
\frac{a}{2} & \text{when } m = m' \\
0 & \text{when } m \neq m'
\end{cases}
$$

yields the following expression

$$
W_{mn} \frac{\pi^4}{a^4} [D_{11} m^4 + 2(D_{12} + 2D_{66})n^2 r^2 + D_{22} n^4 r^4] = q_{mn}
$$

(3.13)
where $D_{11}, D_{12}, D_{22}$ and $D_{66}$ are found from Eq. (2.65). For a point load $F$ applied at the centre, the force Fourier coefficient $q_{mn}$ is

$$q_{mn} = \frac{4F}{ab} \sin \frac{m\pi}{2} \sin \frac{n\pi}{2}$$  

(3.14)

Equation (3.14) is also valid for a uniform pressure, $p$, applied at a small area, $4\alpha\beta$, such that $F = 4p\alpha\beta$. Inserting Eq. (3.14) into (3.13) and solving for for $W_{mn}$ gives the Fourier coefficient for Eq. (3.11).

### 3.4 The Contact Law

#### 3.4.1 Introduction

Local deformations in the contact region must be accounted for in the analysis in order to accurately predict the contact force history. The indentation, defined as the difference between the displacement of the projectile and that of the back face of the laminate, can be of the same order as or larger than the overall displacement of the laminate. One could consider the projectile and the structure as two solids in contact and then analyze the impact problem as a dynamic contact problem. However, this approach is computationally expensive and cannot describe the effect of permanent deformation and local damage on the unloading process. The unloading part of the indentation process can be modeled only using experimentally determined contact laws. To predict the contact force history and the overall deformation of the target, a detailed model of the contact region is not necessary. A simple relationship between the contact force and the indentation, called the contact law, has been used by Timoshenko [31] to study the impact of a beam by a steel sphere. This approach has been used extensively since then and is commonly used for the analysis of impact on composite structure.

Although the impact event is a highly dynamic event in which many vibration modes of the target are excited, statically determined contact laws can be used in the impact dynamics analysis of low-velocity impacts because strain rate and wave propagation effects are negligible with commonly used material systems. This section presents the basic results from contact mechanics needed to model the indentation of composite material during the impact by a foreign object. The objective of such models is the accurate predictions of the contact force history and the overall response of the structure.
3.4.2 Contact Between Two Isotropic Elastic Solids

Detailed accounts of the study of contact between two smooth elastic solid pioneered by Hertz are given in [29, 18]. Essential results from Hertz theory of contact as given here without derivation. For two isotropic bodies of revolution, contact occurs in a circular zone of radius \( a \) in which the normal pressure \( p \) varies as

\[
p = p_0 \left[ 1 - \left( \frac{r}{a} \right)^2 \right]^{1/2}
\]

where \( p_0 \) is the maximum contact pressure at the centre of the contact zone and \( r \) is the radial position of an arbitrary point in the contact zone.

Defining the parameters \( R \) and \( E \) as

\[
\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2} \quad (3.15)
\]

\[
\frac{1}{E} = \frac{1 - \nu_1^2}{E_1} + \frac{1 - \nu_2^2}{E_2} \quad (3.16)
\]

where \( R_1 \) and \( R_2 \) are the radii of curvature of the two bodies. The Young moduli and Poisson’s ratios of the two bodies are \( E_1, \nu_1 \) and \( E_2, \nu_2 \), respectively.

Without loss of generality, the subscript 1 is taken to denote properties of the indentator and subscript 2 identifies properties of the target. The radius of the contact zone \( a \), the relative displacement \( \alpha \), and the maximum contact

![Figure 3.2: Pressure distribution in the contact zone](image)
3.4 The Contact Law

pressure are given by

\[ a = \left( \frac{3FR}{4E} \right)^{1/3} \]  
(3.17)

\[ \alpha = \frac{a^3}{R} = \left( \frac{9F^2}{16RE^2} \right)^{1/3} \]  
(3.18)

\[ p_0 = \frac{3F}{2\pi a^2} = \left( \frac{6FE^2}{\pi^3 R^2} \right)^{1/3} \]  
(3.19)

The force indentation law is expressed as

\[ F = k_c \alpha^{3/2} \]  
(3.20)

where \( F \) is the contact force, \( \alpha \) is the indentation, and the contact stiffness \( k_c \) is given by

\[ k_c = \frac{4}{3} ER^{1/2} \]  
(3.21)

Equation (3.20) is usually referred to as Hertz contact law or the Hertzian law of contact and is found to apply for a wide range of cases, even if all the assumptions made in the derivation of the theory are not satisfied. For example Eq. (3.20) also applies for laminated composites, even though these are not homogeneous and isotropic materials. With composite materials, permanent deformations are introduced in the contact zone for relatively low force levels, but the material is assumed to remain linear elastic in the analysis. Matrix cracks, fibre fracture, and delaminations can be introduced in the contact zone, but overall the contact law follows Eq. (3.20).

The experimental determination of the contact law usually involves the use of a displacement sensor which directly measures the displacement of the indentor relative to the back face of the specimen. The plot of the applied force versus the indentation measured in this way follows the general form given by Eq. (3.20). Least-squares fits of the experimental data usually show that the exponent of \( \alpha \) in the equation is close to \( \frac{3}{2} \), and \( k_c \) is close to the value given by Eq. (3.21). The displacement of the indentor is the sum of the indentation and the deflection of the structure. The observed discontinuity corresponds to the reductions in the apparent stiffness of the structure caused by the introduction of delaminations.

3.4.3 Indentation of a Laminate

A more practical approach consists of determining the relationship between the contact force and the indentation experimentally. Typically during impact
the contact force increases to maximum value and then decreases back to zero. In some cases, multiple impacts and reloading occurs. Therefor, contact laws should include the unloading and reloading phases.

During the first loading phase, the contact law closely follows Hertz’s law of contact, Eq. (3.20). The parameter \( k_c \) in that equation is usually determined experimentally, but it can also be estimated if the radius of the indentator and the elastic properties of the impactor and the target are known using Eq. (3.21). \( R \) is given by Eq. (3.15) and \( E \) given by Eq. (3.16), where \( E_2 \) is the transverse modulus of the composite. Poisson’s ratio of the composite is often taken to be zero in Eq. (3.16).

Equation (3.20) describes the contact law during the loading phase of the indentation process. However, permanent indentation occurs even at relatively low loading levels, and the unloading phase of the process is significantly different from the loading phase. This phenomenon was observed by Crook [12] for the indentation of steel plates by spherical indentors. During the unloading phase, the contact law suggested by Crook is

\[
F = F_m \left[ (\alpha - \alpha_0)/(\alpha_m - \alpha_0) \right]^\frac{3}{2}
\]

where \( F_m \) is the maximum force reached before unloading, \( \alpha_m \) is the maximum indentation and \( \alpha_0 \) is the permanent indentation. \( \alpha_0 \) is zero when the maximum indentation remains below a critical value \( \alpha_{cr} \). When \( \alpha_m > \alpha_{cr} \),

\[
\alpha_0 = \alpha_m \left[ 1 - (\alpha_{cr}/\alpha_m)^\frac{2}{3} \right]
\] (3.22)

During subsequent reloading, the reloading curve is distinct from the unloading curve but always returns to the point where unloading began. The unloading curve is modeled by

\[
F = F_m \left[ (\alpha - \alpha_0)/(\alpha_m - \alpha_0) \right]^\frac{3}{2}
\] (3.23)

The parameter \( \alpha_0 \) does not necessarily correspond to the permanent indentation of the laminate, even though Eq. (3.23) indicates that \( F = 0 \) when \( \alpha = \alpha_0 \). \( \alpha_0 \) is selected so that Eq. (3.23) fits the experimental unloading curve using a least squares fit procedure. The parameter \( \alpha_0 \) is related to the actual permanent indentation \( \alpha_p \) and the maximum indentation \( \alpha_m \) during the loading phase by

\[
\alpha_0 = \begin{cases} 
\beta(\alpha_m - \alpha_p) & \text{when } \alpha_m > \alpha_{cr}, \\
0 & \text{otherwise.} 
\end{cases}
\]

The permanent indentation \( \alpha_p \) and the parameter \( \beta \) are determined from experiments. However, many experiments need to be performed in order to determine these parameters, whereas a single unloading curve is necessary to determine \( \alpha_{cr} \) when Eq. (3.22) is used.
3.5 Impact Models

Several types of mathematical models are used to study the impact of a structure by a foreign object [1, 15, 23, 25, 26]. In certain cases, the contact force history can be accurately determined by modelling the structure by an equivalent spring-mass system. When the structure behaves quasi-statically, it is also possible to calculate the maximum contact force. Assuming that when the contact force reaches its maximum, the sum of the strain energy in the structure and the energy required for indentation is equal to the initial kinetic energy of the projectile. This energy-balance approach is useful when one is interested not in the complete contact force history, but rather only in maximum contact force. When the dynamics of the structure must be more accurately accounted for, more sophisticated models are needed.

3.5.1 Spring-Mass Model

Spring-mass models, shown in Figure 3.3, are simple and provide accurate solutions for some types of impacts often encountered during tests on small-size specimens. The most complete model, consist of one spring representing the linear stiffness of the structure, \( K_{bs} \), and other spring for the nonlinear membrane stiffness, \( K_m \). A mass, \( M_2 \) representing the effective mass of the structure, the nonlinear contact stiffness, \( K_c \), and \( M_1 \), the mass of the projectile, \( K_{bs} \) is determined from experiments. Let \( x_1(t) \) and \( x_2(t) \) represent the displacement responses of the two masses at any time \( t \) after impact as shown if Figure 3.4. The transverse deflection of the plate is given by \( w = x_2(t) \) and the contact deformation is given by \( \alpha = x_1(t) - x_2(t) \). Throughout the analysis the impactor mass \( M_1 \) was assumed to be in contact with the plate. Applying Newton’s second law of motion, equations of equilibrium of the two degree-of-freedom spring-mass system are written as

\[
M_1 \ddot{x}_1 + F = 0 \quad (3.24)
\]

\[
M_2 \ddot{x}_2 + K_{bs}x_2 + K_m x_2^3 - F = 0 \quad (3.25)
\]

\( F \) is the contact force given by Eq. (3.20), which is a highly nonlinear function of the indentation \( x_1 - x_2 \), which can be expressed as

\[
F = F(x_1 - x_2) \quad (3.26)
\]

when \( x_1 > x_2 \). When \( x_1 < x_2 \), contact ceases, \( F = 0 \). Initial conditions are:

\[
\begin{align*}
x_1(0) & = 0 \\
\dot{x}_1(0) & = V_0 \\
x_2(0) = \dot{x}_2(0) & = 0
\end{align*}
\quad (3.27)
\]
Figure 3.3: Spring-mass model

Figure 3.4: Displacement of plate and impactor
where \( V_0 \) is the striker velocity. The coupled differential equations can be solve with numerical methods.

When geometric nonlinearities and the indentation are negligible, the model can be significantly simplified to a single degree of freedom system with the equation of motion

\[
M_1 \ddot{x} + K_{bs} x = 0
\]  

(3.28)

In that model, the effective mass of the structure is neglected and the structure and the projectile move together as soon as contact is made \((x_1 = x_2 = x)\). This is a well known single degree of freedom system \([6]\) with the general solution

\[
x = A \sin \omega t + B \cos \omega t
\]

where

\[
\omega^2 = K_{bs}/M_1
\]  

(3.29)

The constants \( A \) and \( B \) are determined from the initial conditions \((3.27)\), which gives the result

\[
x = \frac{V_0}{\omega} \sin \omega t
\]

Since the contact force \( F \) is equal to the force in the linear spring \( K_{bs} \), the contact force history is given by

\[
F = K_{bs} x = V(K_{bs} M_1)^{\frac{1}{2}} \sin \omega t
\]  

(3.30)

for \( \omega t < \pi \). When \( \omega t > \pi \) Eq. \((3.26)\) predicts a negative contact force, which is impossible. Therefore separation occurs for \( t = T_c = \pi/\omega \). The contact duration \( T_c \) increases with the mass of the projectile and decreases as the stiffness of the structure increases:

\[
T_c = \pi (M_1/K_{bs})^{\frac{1}{2}}
\]  

(3.31)

### 3.5.2 Energy-Balance Model

Another approach for analyzing the impact dynamics is to consider the balance of energy in the system. The initial kinetic energy of the projectile is used to deform the structure during impact. Assuming that the structure behaves quasi-statically, when the structure reaches its maximum deflection, the velocity of the projectile becomes zero and all the initial kinetic energy has been used to deform the structure. The overall deformation of the structure usually involves bending, shear deformation, and for large deformations, membrane stiffening effects. Local deformations in the contact zone also are to be
Analytical Solution for Plate Deflection and Impact Models

considered. For impacts that induce only small amounts of damage, the energy needed to create damage can be neglected. Therefore, the energy-balance equation can be written as

\[ \frac{1}{2} M V_0^2 = E_b + E_s + E_m + E_c \]

where the subscripts b, s and m refer to the bending, shear and membrane components of the overall structural deformation. \( E_c \) is the energy stored in the contact region during indentation. It is always possible to express the force-deflection relation in the form

\[ F = K_{bs} W + K_m W^3 \]

(3.32)

where \( K_{bs} \) is the linear stiffness including bending and transverse shear deformation effects, \( K_m \) is the membrane stiffness and \( W \) is the deflection at the impact point. Then

\[ E_b + E_s + E_m = \frac{1}{2} K_{bs} W_{\text{max}}^2 + \frac{1}{4} K_m W_{\text{max}}^4 \]

(3.33)

Both experimental and analytical studies of contact between smooth indentors and laminated composites indicate that during the loading phase, the contact law can be written as

\[ F = n \alpha^{\frac{2}{3}} \]

(3.34)

where \( \alpha \) represent the relation motion or indentation of the structure by the projectile:

\[ E_c = \int_0^{\alpha_{\text{max}}} F \, d\alpha = \frac{2}{5} n \alpha_{\text{max}}^{\frac{2}{3}} \]

(3.35)

Using Eq. (3.32), (3.34) and (3.35) the maximum indentation can be expressed in terms of the maximum displacement of the structure at the impact location:

\[ \alpha_{\text{max}} = \left( \frac{F}{n} \right)^{\frac{3}{2}} = n^{-\frac{2}{3}} \left( K_{bs} W_{\text{max}} + K_m W_{\text{max}}^3 \right)^{\frac{2}{3}} \]

After substitution into Eq. (3.35), the contact energy becomes

\[ E_c = \frac{2}{5} n^{-\frac{2}{3}} \left( K_{bs} W_{\text{max}} + K_m W_{\text{max}}^3 \right)^{\frac{2}{3}} \]

(3.36)

Using Eq. (3.33) and (3.36), the energy-balance becomes

\[ \frac{1}{2} M V_0^2 = \]

\[ \frac{1}{2} K_{bs} W_{\text{max}}^2 + \frac{1}{4} K_m W_{\text{max}}^4 + \frac{2}{5} n^{-\frac{2}{3}} \left( K_{bs} W_{\text{max}} + K_m W_{\text{max}}^3 \right)^{\frac{2}{3}} \]

(3.37)
This equation can be solved numerically for $W_{\text{max}}$ and the maximum contact force can then be obtained after substitution into Eq. (3.32). In order to examine the relative effects of the bending, shear, membrane and indentation components of the deformation, Eq. (3.37) can be written as

$$\frac{1}{2} MV_{0}^2 = \frac{1}{2} K_{bs} W_{\text{max}}^2 \cdot \left[ 1 + \frac{1}{2} \frac{K_{m}}{K_{bs}} W_{\text{max}}^2 + \frac{4}{5} \left( \frac{K_{bs} W_{\text{max}}}{n} \right)^{\frac{2}{3}} \cdot \left( 1 + \frac{K_{m}}{K_{bs}} W_{\text{max}}^2 \right)^{\frac{5}{3}} \right]$$  \hspace{1cm} (3.38)

Neglecting the membrane effects ($K_{m} = 0$), Eq. (3.38) will become

$$\frac{1}{2} MV_{0}^2 = \frac{1}{2} K_{bs} W_{\text{max}}^2 \left[ 1 + \frac{4}{5} \left( \frac{K_{bs} W_{\text{max}}}{n} \right)^{\frac{2}{3}} \right]$$  \hspace{1cm} (3.39)

or, in terms of maximum contact force,

$$\frac{1}{2} MV_{0}^2 = \frac{1}{2} \frac{F_{\text{max}}^2}{K_{bs}} + \frac{4}{5} \frac{F_{\text{max}}^{\frac{5}{2}}}{n^{\frac{2}{3}}}$$  \hspace{1cm} (3.40)

Also neglecting the effect of local deformation in the contact zone, represented by the second term on the right hand side

$$F_{\text{max}} = V_{0}(K_{bs} M)^{\frac{1}{2}}$$  \hspace{1cm} (3.41)

In this case, the maximum contact force increases linearly with the initial velocity of the projectile.
Chapter 4

Impact Experiments and Static Deflection Tests

4.1 Introduction

Three series of experiments were conducted in this thesis. The first series was done with a Rosand Instrumented Falling Weight impactor and the second series with a drop weight rig. In the Rosand machine the dart is instrumented with a force cell, such that energy, displacement and velocity are found. To receive information about the drop weight impacts, an accelerometer was attached on the projectile. All the plates used in the drop weight test were also instrumented with strain gauges. The third experiment was a static deflection test, done to determine the linear plate stiffness, $K_{ls}$.

4.2 Strain Gauges

Strain gauges are a very important tool when doing stress and strain analysis in material testing. They measure strain caused in the specimen due to applied load or deflection. They do this due to the principle that a change in length of the gauge is proportional to a change in electrical resistance through the gauge. The change in resistance can be measured and the strain can be found. A fundamental parameter of the strain gauge is its sensitivity to strain, expressed quantitatively as the gauge factor or k-factor, $K$. Gauge factor is defined as the ratio of fractional change in electrical resistance to the fractional change in length,

$$K = \frac{\Delta R/R}{\Delta L/L} = \frac{\Delta R/R}{\epsilon}$$
From the strain readings, and the element’s modulus of elasticity, the stress in the specimen can be calculated from Hooke’s law. A procedure for attaching a strain gauge to a specimen is found in Appendix A.

### 4.2.1 Strain Gauge Setup

The strain gauges were connected with wires to a switching box. From the switching box the signals were sent to the strain amplifier. The strain amplifier was a MGC AB12 from Hottinger Baldwin Messtechnik (HBM). As the resistance change of strain gauges is extremely small, it is indicated or recorded by means of an amplifier. The strain amplifier is designed to convert the small resistance change of the strain gauge into a voltage output, amplifying it to analog data. The strain amplifier was connected to a BNC-2090 Analog/Digital converter from National Instruments, which sending digital data to the computer. Figure 4.1 shows the setup of the logging equipment. The computer was also connected to the strain meter and from the software, CATMAN, all parameters in the strain meter were set. Parameters can be set directly on the strain meter, but using the software is easier and gives the user better control. The strain amplifier measure -10 V - 10 V and 10 V is set to largest expected strain which is found from static simulation in ANSYS to be 4.5% for 10 mm deflection. In this thesis 3 mm and 10 mm stain gauges from HBM was used. Both of them have advantages and disadvantages.

- Strain gauges with 3 mm measure length intercept better local deformation, but they have a lower grade of accuracy because of the small length. Since they are quite small, the user must be punctilious to get them parallel or perpendicular to the material axis. The k-factor for these gauges was 2.02

- The 10 mm strain gauges are more accuracy because of the bigger measure area, but they do not measure very local deformation. Attaching them is also pretty easy. The k-factor was 2.09

### 4.2.2 The Low Pass Filter in the Strain Amplifier

The strain amplifier has built in hardware filters. The filter could be either Butterworth or Bessel characteristic. Low pass filters are used to suppress unwanted higher-frequency interferences above a defined cutoff frequency. A Bessel characteristic filter set to 50 kHz for the strain gauges and 10 kHz for the accelerometer was used.
4.3 The Rosand Instrumented Falling Weight Impactor Tests

4.3.1 Introduction

The Rosand Instrumented Falling Weight Impactor Type 4 is a computer controlled fall weight device used to perform Charpy and other impact tests. The plates can be clamped or simply supported. Drop weight can be adjusted from 3 to 25 kg and maximum velocity is limited to 6 m/s, so largest possible kinetic energy is 450 J. The impactor is a steel dart with a replaceable tip, which can be flat, axe or spheric. For each tip type different sizes was available. In this thesis a 12.5 mm steel dart with a 12.5 mm diameter spherical tip was used. To avoid multiple impacts the dart is caught automatically after the first strike. A cell is logging the force on the dart and the logging start when the dart passes a optical cell. A trigger value can be adjusted to avoid logging to many points before the impact begin.
4.3 The Rosand Instrumented Falling Weight Impactor Tests

4.3.2 Setting Parameters

The drop height, $h$, has to be zeroed before starting the experiments. This is done by lowering the dart until the tip touches the plate and height is then set to zero. From potential energy, $E_p = mgh$, the control software calculates needed drop height, when impact weight, $m$, is known. The impact velocity, $V_0$, is found from kinetic, $E_k = \frac{1}{2}mV_0^2$. During the impact the computer reads and stores force at the impactor during the impact. Output available: time, force, velocity, acceleration, energy and displacement.

4.3.3 Test Setup

Figure 4.3 shows the setup of our experiments. A square plate with a side length of 175 mm was impacted. The plate was clamped between a circular steel plate and a aluminium frame. The steel plate had 42 mm light opening and the aluminium frame had curved edges 135 mm light opening.

The logging time was set to 15 ms and number of points was set to 4000. The room temperature was 20°C. Nine plates were tested. Three energy levels were used: 20, 40 and 60 J and for each energy level there were three different impact masses: 5.151, 7.151 and 10.151 kg. Table 4.1 shows impact energy, -mass and velocity for each test. Each plate was marked with a number also
Impact Experiments and Static Deflection Tests

![Diagram of the Rosand Falling Weight setup for a square plate. Lengths given in mm.](image)

Table 4.1: Plates tested in the Rosand Instrumented Falling Weight Impactor.

<table>
<thead>
<tr>
<th>Plate number:</th>
<th>20 J</th>
<th>40 J</th>
<th>60 J</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate number</td>
<td>4</td>
<td>5</td>
<td>6</td>
</tr>
<tr>
<td>Mass (kg)</td>
<td>5.151</td>
<td>5.151</td>
<td>5.151</td>
</tr>
<tr>
<td>Velocity (m/s)</td>
<td>2.787</td>
<td>3.941</td>
<td>4.827</td>
</tr>
<tr>
<td>Plate number</td>
<td>7</td>
<td>8</td>
<td>9</td>
</tr>
<tr>
<td>Plate number</td>
<td>10</td>
<td>11</td>
<td>15</td>
</tr>
<tr>
<td>Mass (kg)</td>
<td>7.151</td>
<td>7.151</td>
<td>7.151</td>
</tr>
<tr>
<td>Velocity (m/s)</td>
<td>2.365</td>
<td>3.345</td>
<td>4.096</td>
</tr>
<tr>
<td>Mass (kg)</td>
<td>10.151</td>
<td>10.151</td>
<td>10.151</td>
</tr>
<tr>
<td>Velocity (m/s)</td>
<td>1.985</td>
<td>2.807</td>
<td>3.438</td>
</tr>
</tbody>
</table>

4.3.4 Instrumented Results

In this section only the results from the 60 J energy level will be studied. The results for the two other energy levels looks very similar accept from different extremal values. For every graph of 60 J shown in this section a corresponding graph from 20 and 40 J can be found in Appendix B.

Figure 4.4 shows the time force plot for the 60 J impact tests. Observe the oscillation at the begin of the impact. For plate 9 and 15 where the impact masses was 7.151 and 10.151 kg, respectively, the fluctuations stops after 1 ms.
4.3 The Rosand Instrumented Falling Weight Impactor Tests

But for plate 6 with 5.151 kg impact mass the fluctuations do not stop during the impact. As seen oscillations are strongly dependent of the impact mass. For the first 1/2 ms all masses has the same frequency shown in Fig. 4.4, but with different amplitude.

Until the dart hits the plate at time 0, the force on the dart is zero. After the hit the force increase rapidly to the first peek at \( \approx 0.1 \) ms. At this peek the plate has started to deflect and therefore the force on the dart drops. The deflection of the plate tends to stop for the decreasing force, so the dart will hit the plate again. Now the force increase to the next peek. Again by deflection of the plate, the force drop. For the masses 7.151 and 10.151 kg the dart will be in contact with the plate for the rest of the impact. The 5.151 kg dart will oscillate through the whole impact.

A schematic representation of a impact is given in Fig. 4.5. At a) the impactor hits the plate and the displacement of the impactor and the force is 0. Figure 4.5 b) illustrate the plate at maximum deflection. The velocity of the impactor is then zero and the force is maximum. The plate has absorbed all the impact energy as plastic and elastic energy. Energy to friction is insignificant and can be neglected. The impactor leaves the plate in c). Some of the elastic energy in the plate has now been used to accelerate the impactor. Only gravity

---

Figure 4.4: Time Force digram for 60 J impact. Impact mass and velocity for each plate are found in Table 4.1. The first \( \frac{1}{2} \) ms is shown in the small figure.
force is now working on the impactor. In Figure 4.6 the energies of the plates are plotted. The energy in the plate before impact is zero. Under the impact the energy increase to a maximum, where the dart’s velocity is 0. From this point the plate use some of the elastic energy for the spring back of the dart.

When studying all the graphs it is observed that the impact duration increase for increasing mass. The frequency for the system is given by Eq. (3.29) and from this equation it is observed that the frequency of the system increase for decreasing mass. Studying the time-displacement plot in Fig. 4.6 the maximum deflection is nearly the same for all masses, but at different time. The time difference is about 1.5 ms between 5.151 and 10.151 kg.

4.3.5 Visual results

For low velocity impact, that do not result in complete penetration of the target, damage consist of delaminations, matrix cracking and fibre failures. Delamination is the debonding between adjacent plies and is very interesting, because the significantly reduce the strength of the laminate. Only at interfaces between plies with different fibre orientations delamination occur. If two adjacent plies have the same fibre orientation, there will be no delamination on the interface between them. Figure 4.7 illustrate the delaminated area be-
4.3 The Rosand Instrumented Falling Weight Impactor Tests

![Displacement and energy plot for 60 J Rosand impacts. Impact mass used for each plate is listed in Table 4.1](image)

Figure 4.6: Displacement and energy plot for 60 J Rosand impacts. Impact mass used for each plate is listed in Table 4.1

Between two plies. The delaminated area has a oblong or "peanut" shape with its major axes oriented along the fibre in the lower ply. A simple model can be used to explain why delamination appear in laminates with different fibre orientation in the plies. Each layer tends to deform in a particular way, but transverse, normal and shear stresses applied at the interfaces constrain the layup to behave as one plate. When these interlaminar stresses become too large under concentrated contact loads, delamination is introduced. The orthotropic behaviour of each ply and the mismatch in their bending stiffness is the cause of delamination.

![Impacted Lamina](image)

![Delamination on the Interface](image)

![Nonimpacted Lamina](image)

Figure 4.7: Delamination between a 90° lamina and a 0° lamina. The top lamina is impacted.

Figure 4.8 shows delamination in a side cut of a impacted plate with 8 plies
Figure 4.8: Side cut of delaminated area. Top layer in 0° direction. Six delamination areas occurred.

In a symmetric stacking, [0, 90, 0, 90]_s. The top layer is a 0° ply. This picture shows clearly that delamination is oriented along the fibres in the lower ply, since on 0/90 interfaces the delamination is very small, while it is large on 90/0 interfaces.

Lesser and Filippov [21] presented a simple model to explain the delamination between to layers with different fibre orientation. Equation (3.11) is used to calculate the transverse displacement of two simply supported rectangular plates consisting of a single composite. If two layers are stacked on top of each other but not bounded together, the two layers would separate under load because they deform differently. The difference between the displacement of the two layers has the same shape as the delaminations at the interface between the same two layers if they were bonded together. The idea behind this simple explanation is that when the two layers are bonded together, inter laminar stresses develop on the interface in order to force these layers to deform as a single layer. High inter laminar stresses are expected to cause delaminations.

To test this explanation, transverse deformation was calculated for two unidirectional glass-epoxy composite laminate. One with fibres in 0° and the other with fibres in 90° direction. The plates was 125 × 125 mm². Material properties for unidirectional glass-epoxy composite found in [2] are used.

\[ E_L = 38.6 \text{ GPa}, \quad E_T = 8.27 \text{ GPa}, \quad G_{LT} = 4.14 \text{ GPa}, \quad \nu_{LT} = 2.26. \]

Thickness of the plates was 2 mm and a point load \( F = 500 \text{ N} \) were applied at the middle of the plates. Figure 4.9 (a) shows the difference between the transverse displacement for the two plates. Maximum difference is \( \approx 2 \text{ mm} \). At Figure 4.9 (b) the contour of the difference has been plotted. The "peanut" shape is clearly shown in the middle.

Matrix cracks are arranged in a complicated pattern that are very difficult to predict. But matrix cracks do not significantly contribute to the reduction in residual properties of the laminate, so it not that important to predict the pattern. However, the damage process is initiated by matrix cracks which then
4.3 The Rosand Instrumented Falling Weight Impactor Tests

Figure 4.9: (a) difference in deflection of a 0° and a 90° unidirectional composite and (b) contour plot of the deflection difference.

Figure 4.10: Two types of matrix cracks: (a) tensile crack and (b) shear crack.
induce delamination at ply interfaces. There are two types of matrix cracks. Tensile crack and shear crack. These are illustrated in Fig. 4.10 (a) and (b), respectively. Tensile cracks appear when in plane normal stresses exceed the transverse tensile strength of the ply. Shear cracks are at an angle from the mid surface, which indicates that transverse shear stresses play a significant role in their formation.

Figure 4.11 shows delamination on the bottom side of Plate 9. The lengths $l$ and $w$ are parallel and perpendicular to the fibres on the bottom layer, respectively. In Table 4.2 the delamination sizes measured for each impacted plate are listed.

![Delamination pattern on bottom side of 60 J impact.](image)
4.4 The Accelerometer Calibration

4.4.1 Introduction

For the drop weight tests, an accelerometer was used to measure the acceleration. The piezoelectric accelerometer FA 101 is manufactured by FGP and measure ± 500 g. In Section 4.2.1 the logging equipment used is described. Before the accelerometer could be used, it had to be calibrated to ensure that the setup worked and the results were correct. The Rosand Instrumented Falling Weight device was used to do the calibration. The accelerometer was attached with very strong adhesive tape on top of the dart. Four tests was performed on two GFRP plates and each plate was impacted with same energy twice. On the first plate 20 J was used and on the second 10 J. The weight of the impactor was 3.151 kg and setup was the same as in Section 4.3.3.

4.4.2 Measured Results

Only the results from the first plate is shown here. The results from the second plate was almost identical only smaller maximum values. Figure 4.12 (a) and (b) shows the results for the first and second impact on plate 1, respectively. The plots shows a comparison of the Rosand acceleration, the accelerometer and a smoothed approximation of the accelerometer. On the fist impact the acceleration of the dart oscillate pretty much before reaching the maximum value. For the second impact this oscillations are gone. The oscillation of the dart is previously discussed in Section 4.3.4. Because of delamination and matrix cracking from the first impact the plate will become more elastic in the damaged area. There was no further visible damage on the plate after the second impact. Most of the force in the second impact was used for elastic deformation, which explain why there was no oscillation and that the acceleration is a bit higher than for the first impact.

To remove the high frequency oscillations a Bezier algorithm in GNUPLOT [19] was used to smooth the data.

Comparing the results from the accelerometer and Rosand shows that the measured data from the accelerometer are very good. For the first strike there are some differences before the maximum value, but these are insignificant. For the second strike the results are almost identical. Based on these results the data from the accelerometer are correct and can be used in the manual drop weight tests.

53
Impact Experiments and Static Deflection Tests

![Graphs showing acceleration versus time for two impacts](image)

Figure 4.12: Acceleration for accelerometer test 1 with 20 J impact energy: (a) first impact and (b) second impact.

### 4.5 Drop Weight Impactor Tests

#### 4.5.1 Introduction

Impacts on plates larger than $175 \times 175 \text{ mm}^2$ or non-flat objects, such as pipes, can not be performed in the Rosand impact machine. When doing impact test on rare or large targets, a drop weight impactor is used. Impact onto targets in water or other fluids is also possible. This drop devices are normally built just like the Rosand machine, but in much larger scale. An impactor, in free fall, is lead by two steel pipes down to the target. Both drop weight and height can be adjusted to receive the desired impact energy. There is also a force sensor logging the force on the dart during the impact. Since such machines are extreme expensive, one could not be bought for this project. Instead a much simpler drop weight device had to be constructed.

#### 4.5.2 The Setup

Figure 4.13 shows the setup for the drop weight test. A projectile was dropped through a Perspex tube onto a laminate plate, simply supported on a steel pipe frame. The plate was $300 \times 300 \text{ mm}^2$ and the diameter of the pipe was 40 mm. The light opening in the frame was $260 \times 260 \text{ mm}^2$. To support the Perspex tube a wood rig was built. This rig was fixed to the walls and the ceiling to make it complete rigid. The projectile was a solid steel cylinder with a spherical tip and both the cylinder and the spherical tip had 75 mm diameter. The length
of the impactor was 190 mm and the total weight 6.2 kg. An accelerometer, described in Section 4.4, was first mounted on top of the projectile.

![Diagram](a)

![Diagram](b)

Figure 4.13: (a) setup for drop weight test and (b) the projectile with the accelerometer lowered into the top.

The tube was 4 m high, so kinetic energy was limited to 240 J with the 6.2 kg projectile. To avoid contact between the plate and the floor during the impact, the frame was elevated. On two sides the frame were strapped to a u-beam on a Leca block. The tube was only 10 cm above the plate so the projectile returned into the tube after the impact. To avoid a 2. strike on the composite plate a thick rubber mat was placed onto the plate after the first impact. A nylon rope, with 4 mm diameter, was used to hoist the impactor.

After the setup was complete lots of impact tests were performed on sandwich and composite laminate plates. At these first tests a trigger value for the accelerometer was found, such that the logging started once the accelerometer was dropped. Because of poor results from the accelerometer, the accelerometer was lowered into the top of the projectile as illustrated in Fig. 4.13 (b). This to avoid that the accelerometer had any contacts with the tube during the fall. Silicon glue was used between the projectile and the accelerometer to damp high frequency vibrations. Four bolts was used to attach the accelerometer to the projectile, however the results did not become any better. Previously tests of the accelerometer, done in the Rosand impact machine, showed good results, so the problem had to be the tube. A lot of ventilation channels were machined out in the tube to hopefully reduce air flow problem around the projectile during impact. Figure 4.14 (c) shows the air channels. The result from the accelerometer was now very good, showing a approximate
constant acceleration for the projectile through the tube. Only one second was measured since the fall time for the impactor was only about 0.4 second. The logging frequency was 30000 Hz. The results from the accelerometer was measured in g. From Newton’s 2. law the force on the plate was found. To find the velocity and displacement of the projectile, the acceleration had to be integrated. This was done numerical with a simple but effective method called Rightsum Rule, that is

\[
\int_a^b f(x)dx \approx h \sum_{i=1}^n f(a + ih)
\]

(4.1)

where \( h = \frac{b-a}{n} \) is the log frequency. The error of this method is proportional with \( h \), \( |E| \leq O(h) \). A PYTHON [32] script was written to perform the numerical integration.

Three plates were impacted in the tests and Table 4.3 shows velocity and impact energy used for each plate. For S3 and S4 a second impact was done. By mistake the low pass filter was set to 100 Hz in the strain gauge amplifier instead of the 10 kHz filter. The 100 Hz filter smoothes the data, so small peak disappears and all maximum values are lowered. Figure 4.15 shows the acceleration plot for different filters. Since the strain gauges had the same filter all the impacts, a gauge was used to find the start of the impact. The gauge
Table 4.3: Impact properties for plates used in manual drop weight tests

<table>
<thead>
<tr>
<th>Plate number</th>
<th>Velocity</th>
<th>Drop Height</th>
<th>Energy</th>
</tr>
</thead>
<tbody>
<tr>
<td>S2</td>
<td>4.4 m/s</td>
<td>1.0 m</td>
<td>60 J</td>
</tr>
<tr>
<td>S3</td>
<td>4.4 m/s</td>
<td>1.0 m</td>
<td>60 J</td>
</tr>
<tr>
<td>S4</td>
<td>7.0 m/s</td>
<td>2.5 m</td>
<td>152 J</td>
</tr>
</tbody>
</table>

Figure 4.15: Comparison of different filters in the strain gauge amplifier. The strain gauge is plotted to show the start of the impact.
is also plotted in the figure. Only with the 1000 Hz and 10 kHz filter the acceleration measured starts at the right time. From the amplifier user manual [17] the delay for different filters are found. With the 10k Hz filter the delay is only 0.03 ms, while 100 Hz filter gives 1.8 ms delay compared to no filter. In the Fig. 4.15 the delay for the 100 Hz filter is clearly shown. Even though the 100 Hz smooths the acceleration, the results are not that bad and they provide a useful information of the impact. The maximum value is a bit low, but the duration of the impact is correct. When acceleration is integrated, to find for example velocity, the result will be smoothed anyway, so the results with 100 Hz filter can be used. Only when studying details such as the peak at the first 2 ms, the 100 Hz filter is useless.

Since the results from the impact were compared to simulations the smoothed plots were not satisfactory, so one 60 J and the 152 J impact were performed over again. When testing the setup, some plates was impacted many times. The results from these tests showed that the acceleration changed insignificant for the first impacts, which is also shown by Wu and Shyu [33] and Takle [28]. Also when studying the data from the strain gauges for the first and the second impact, the difference are insignificant, so the the results from the second impacts conducted on the plates are be very similar to the first ones.

![Graph](image)

**Figure 4.16:** (a) strain measured at first 60 J drop weight impact and (b) strain gauge location from centre of the plate. Gauge 7 is located on the same place as gauge 2, but on the opposite side of the laminate.
4.5.3 The 60 J Impact

Two 60 J drops was performed onto two different laminate plates. Both plates was instrumented with strain gauges. On the first plate, S2, strain gauges were attached near the centre of the plate to study local deflection near the impactor. Strain gauges of 3 mm length were used on this plate. For the second plate, S3, global deformation was studied by attaching 10 mm gauges.

![Graph](image)

Figure 4.17: (a) strain measured at second 60 J drop weight impact and (b) strain gauge location from centre of the plate. Gauge 7 is not shown in the figure, but is located 15 mm from the centre in the 0° direction.

Measured strains and gauge locations are showed in Fig. 4.16. The strain gauges were attached near the centre of the plate. Gauge 1, 3 and 5, which were located nearest to the centre, all failed during the impact, probably due to matrix cracks. On gauge 1 there was a visible crack following a matrix crack beneath. Gauge 2, 4 and 6 were expected, due to symmetry, to have about the same strain, but the plot in Fig. 4.17 (a) shows big differences. For gauge 2 and 4 the strains are the same until 4 ms. From that point gauge 2 increase fast to it’s maximum at 5 ms, while gauge 4 increase slower. Under gauge 2 there was delamination, which most likely is the explanation of the difference. Strain gauge 6 has a maximum at 2 ms. Since this gauge is located at the same distance from centre as gauge 2 and 4, but in 45° direction, the strain was supposed to be equal or greater than gauge 2 and 4. This did not happened and it looks like this strain gauge failed. For strain gauge 7 the result is also a bit strange. Gauge 7 was located on the same place as gauge 2, but on opposite
Impact Experiments and Static Deflection Tests

side of the laminate, so compression of this gauge was expected to be about
the same as the strain on gauge 2. This did not happen, so if the measured
data is correct, the bottom side with delamination deflect more than the upper
side of the plate.

From Fig. 4.17 the measured strains for plate S3 are studied. Gauge 1, 3
and 5 have almost the same strain, which is sensible because they have the
same distance, 35 mm to the centre of the plate. The distance from the centre
to gauge 2, 4 and 6 is also the same, 84 mm, giving them all the same strain.
For gauge 7, located 15 mm from the centre, the strain is higher then for
the other gauges. There was also delamination and matrix cracks under this
strain gauge. This indicates that the bending is symmetric and that the result
are independent of the orientation of the strain gauges with regard to fibre
orientation.

Figure 4.18 shows four plots from the impact on plate S3. (a) shows the
data measured with the accelerometer. A small peak is shown at about 1 ms.
This is the same as shown for the impacts in Rosand machine. When the
projectile hits the plate and the plate starts to deflect. The velocity of the
plate is then higher than the projectile and for a very short time there are no
contact between the dart and the plate. Obviously force on the dart will then
decrease, until it hit the plate again. There are also a small peak at about 3
ms, where there probably is loss of contact. At 4.18 (b) the energy curve for
the plate is drawn. This is calculated by first finding the kinetic energy of the
projectile and then subtract this from the initial energy of the system, since
all energy is conserved in the system. The plot show that half of the energy is
absorbed by the plate. The velocity of the projectile goes from 4.4 m/s, when
the impact starts, to -3.9 m/s when it leaves the plate. At 5.8 ms the velocity
is zero and at this point all the other graphs have their maximum values. It is
the point of maximum deflection, shown in (d) and then naturally the point
where the plate has absorbed maximum energy. The plots from the impact on
S2 is not shown here, while they are similar the plots of impact this impact.

4.5.4 The 152 J Impact

The impact energy on plate S4 was 152 J. The impactor was dropped from 2.5
m. A combination of 3 and 10 mm strain gauges was attached to the plate.
The setup is illustrated in Figure 4.19 (b). Gauge 1, 2 and 5 are 3 mm and
3, 4 and 6 are 10 mm. The small strain gauges were supposed to measure
the local deformations near the impactor and the large gauges to measure

60
4.5 Drop Weight Impactor Tests

Figure 4.18: Plot from impact on plate S3: (a) Acceleration of the dart, (b) Energy of the plate, (c) Velocity of the dart and (d) Displacement of the dart. The acceleration is measured with an accelerometer. From acceleration velocity and displacement are found by integrating numerical once and twice, respectively. A Right Sum rule, given in Eq. (4.1), is used for the numerical integration. The plate energy is found by subtracting the kinetic energy of the dart, $E_k = \frac{1}{2}mv^2$, from the impact energy, 60 J, for each timestep.
the global deformations. During the impact large delamination area occurred. Some large matrix cracks were also visible. Studying the data from the gauges in Fig. 4.19 a peak on gauge 1 occur at about 2 ms. This peak is probably because of a matrix crack spreading across the centre of the plate. The matrix cracks on the bottom side are parallel to the fibres. Gauge 1 is also parallel to the fibres. When studying the plate after the impact, a matrix crack has grown under gauge 1. Gauge 4 located 25 mm away fromcentre on the plate diagonal failed during the impact. As excepted, due to a symmetric bending, gauge 3, 5 and 6 gave the same results. These three gauges are located at the same distance from the centre, but in different orientations. Also the results from gauge 1 and 2 seems very sensible, showing a bit higher strain at gauge 1 then for gauge 2. Gauge 2 is located 10 mm further away from the centre then gauge 1. There was neither any delamination under gauge 2.

Acceleration (g) and calculated velocity and displacement of the projectile as well as calculated energy of the plate are shown in Fig. 4.20 (a), (b), (c) and (d), respectively. These plots are naturally similar to the S3 plots in Fig. 4.18, but since S4 was impacted with 152 J the graphs have higher maximum values. On the acceleration plot the second peak, at about 3 ms, is more conspicuous than for plate S3.
Figure 4.20: Plot from impact on plate S4: (a) Acceleration of the dart, (b) Energy of the plate, (c) Velocity of the dart and (d) Displacement of the dart. The acceleration is measured with an accelerometer. From acceleration velocity and displacement are found by integrating numerical once and twice, respectively. A Right Sum rule, given in Eq. (4.1), is used in the integration. The plate energy is found by subtracting the kinetic energy of the dart, \( E_k = \frac{1}{2}mv^2 \), from the impact energy, 152 J, for each timestep.
Since the plots from S2, S3 and S4 do look the same, the data measured with the accelerometer are assumed to be correct.

4.6 Static Deflection of a Plate

4.6.1 Introduction and Setup

In Section 3.5 analytical solutions of the impact are derived, but to use those expressions the plate stiffness, $K_{ls}$, is needed. From a simple experiment the plate stiffness is easily found. A plate is applied a slowly increasing point load at the centre and force and deflection are measured and the slope of the deflection-force curve is the plate stiffness.

For these tests a Zwick Z250 static test machine was used. The Zwick machine is used to perform mechanical material testing, for example tensile tests and 4 point bending experiment. A computer is connected to the machine and is used to set parameters and to process the data from the experiment. Maximum force available on the machine is 250 kN and the velocity can be adjusted from 0.001 - 500 mm/min. Three parameters were set in the tests:

- Maximum force: 14000 N.
- Maximum deflection: 15 mm.
- Velocity set: 10 mm/min.

A hemispheric piston, with 25 mm diameter, was used as indentator. Two setup were used:

1. Square plate with side length of 175 mm simply supported on the aluminium frame used in the Rosand impact tests.

2. Square plate with side length of 300 mm simply supported on the pipe frame used in the drop weight tests.

The results from the experiments are compared to an analytical solution for plate deflection from Eq. (3.13) and ANSYS simulations described in Section 6.3. The analytical solution is described in Section 3.3.

4.6.2 Results from 1. Setup

In Fig 4.22 a comparison of the test results, a linear approximation of the test results, ANSYS simulations and analytical calculation is shown. The difference
Figure 4.21: Models used in static deflection simulation: (a) the model is as in the experiments. The light opening of the frame is smaller than the plate. (b) only the part of the plate over the light opening is simulated. The length of the plate, \( l_1 \), was 175 and 300 mm for 1. and 2. setup, respectively. For the no frame simulation, \( l_2 \) was 135 and 260 mm. \( F \) is the force applied.

between the analytical calculation and the ANSYS simulation are insignificant and gives a plate stiffness \( K_{bs} \approx 1640 \text{ N/mm} \). The linear approximation gives a plate stiffness \( K_{bs} \approx 950 \text{ N/mm} \). The strain hardening take effect at 6 mm. Before the hardening the stiffness curve is approximately linear.

The simulations with frame model are more stiff then the no-frame models. The divagation between simulation/analytical and the experimental data can be attributed to the boundary conditions of the plate. Because of the small size of the plate there will be some buckling on the edges. Since the simulation and analytical solution fits very good, the model in the simulation can be assumed correct. The material properties are also correct since they are used in all other simulation too, with satisfactory results.

4.6.3 Results from 2. Setup

Figure 4.23 shows the results from the 2. setup compared with a ANSYS simulation and a analytical solution. The linear approximation is based on the experimental results up to deflection of 6 mm and fits very good to the simulation and analytical solution. Finding \( K_{bs} \approx 390 \text{ N/mm} \).
Figure 4.22: Force-deflection plots for 1. static plate bending experiment. The experiment and a linear approximation of the experiment are drawn with lines. An analytical solution is calculated with MAPLE. Two types of element were used in the ANSYS simulation. A volume element, SOLID45, and a shell element, SHELL63. For both elements two models were simulated. Figure 4.21 shows the difference between frame and no-frame models used. The analytical solution used the no-frame model.
Figure 4.23: Force-deflection plots for 2. static plate deflection experiment. The experiment and a linear approximation of the experiment are drawn with lines. A analytical solution is calculated with MAPLE. Two types of element were used in the ANSYS simulation. A volume element, SOLID45, and a shell element, SHELL63. For both elements two models were simulated. Figure 4.21 shows the difference between frame and no frame models used. The analytical solution used the no-frame model.

4.7 Summary

In the previous sections three different experiments have been done. One static deflection test and two impact experiments.

The static experiment is quite simple to perform when having the right test equipment. After the plate has been placed correct, such that the indentator hits the centre at the plate, a few parameters had to be set in the software controlling the rig. From the force-deflection plot the plate stiffness is found, which was needed in the analytical impact models.

Operating the Rosand Instrumented Falling Weight device is also simple. A computer controls the device and from a user-friendly interface parameters such as impact energy, drop weight and impact time are set. There are also a lot of other options which can be defined, but the default values were good enough. Only a few things have to be done manually: add desired drop weight, mount the right dart and place the specimen. From the force logged, the software calculates acceleration, velocity, displacement and energy. Because the machine is easy to operate, a lot of impacts can be performed rapidly with very good results. The drawback with device is that the specimen size is limited to $175 \times 175$ mm$^2$. 
Impact Experiments and Static Deflection Tests

For the drop weight test performed, there were a lot more to regard. As described these experiments required extensive testing of the setup, before the three experiments could be done. First the accelerometer was calibrated in the Rosand machine to ensure the log equipment setup worked and measured data were correct. After the calibration drop test through the tube could be conducted. Finally after a lot of drop test and many modifications of tube and impactor the impact results were satisfactory and the three experiments could be done. Even the setup seem perfect, one error was done when the low pass filter in the strain gauge amplifier were changed to 100 Hz, instead of the 10 kHz filter, by a mistake. This error could have been discovered if the impact data had been studied after the first impact, but since the setup seemed to work well, all three impacts were performed rapidly. A general conclusion is to carefully check the setup before doing the experiments. And if many similar experiments shall be done in the same setup, the user must verify results from the first experiments before continuing.
Chapter 5

AUTODYN Impact Simulations

5.1 Introduction

To simulate the impacts conducted on the composite laminates, the program AUTODYN was used. The AUTODYN software for nonlinear dynamics was first released by Century Dynamics in 1986 with the introduction of AUTODYN-2D. AUTODYN-3D, the three dimensional analog to AUTODYN-2D, was released in 1991. In this thesis the AUTODYN-3D v4.3 has been used. The AUTODYN programs are general-purpose engineering software packages that use finite difference, finite volume, and finite element techniques to solve a wide variety of nonlinear problems in solid, fluid and gas dynamics. This type of program is sometimes referred to as a "hydrocode". The phenomena to be studied with such a program can be characterized as highly time dependent with both geometric nonlinearities (e.g. large strains and deformations) and material nonlinearities (e.g. plasticity, failure, strain-hardening and softening, multi phase equations of state). The following theory is found in the AUTODYN Theory Manual [9] and the AUTODYN User Manual [10].

5.2 Theory Overview

The various numerical processors available in AUTODYN generally use a coupled finite difference/finite volume approach. The scheme allows alternative numerical processors to be selectively used to model different components of a problem. Individual structured meshes operated on by these different numerical processors can be coupled together in space and time to efficiently compute structure, fluid, or gas dynamics problems including coupled problems. AUTODYN includes the following numerical processors:
1. Lagrange Processor for modelling solid continua and structures
2. Shell Processor for modelling thin structural elements
3. Euler Processor for modelling fluids, gases, and large distortion
4. ALE (Arbitrary Lagrange Euler) processor for specialized flow models
5. SPH (Smooth Particle Hydrodynamics)

At present, all the above processors use explicit time integration.

### 5.2.1 Lagrange Processor

The Lagrange processor in 3D operates on a structured (I-J-K) numerical mesh of brick-type elements. The vertices of the mesh move with material flow velocity. Material remains within its initial element definition with no transport of material from cell to cell.

![Grid](image)

Figure 5.1: Grid before and after deformation

Because no transport of material through the mesh needs to be calculated the Lagrange formulations tends to become fast. Material interfaces, free surfaces, and history dependent material behaviour are generally easy to follow in the Lagrange framework. The major disadvantage of Lagrange is that if excessive material movement occurs, the numerical mesh may become highly distorted leading to an inaccurate and inefficient solution. Further, this may ultimately lead to a termination of the calculation. Rezoning the numerical mesh by re-mapping the distorted solution onto a more regular mesh is one approach to alleviate the mesh distortion problem. An other technique, erosion, are also available in AUTODYN and can be used to further extend the Lagrange formulation to highly distorted phenomenon. The erosion option can overcome many of the problems associated with using a Lagrange technique for high deformation situations. The user specifies a particular limit of the strain
within the element. If and when the limiting strain is reached, the element is eroded, i.e. transformed to a solid element to a free mass node disconnected from the original mesh thereby avoiding the mesh distortion problem.

Since the numerical algorithm used in AUTODYN is an explicit scheme there is a maximum time step of integration which must be observed if the numerical solution obtained is to be a reasonable representation of the true solution. The value of this time step depends on several parameters of the numerical method and solution so the local time step ensuring stability is calculated for each mesh point. The minimum value of all these local values is multiplied by a safety factor (currently a default value of 2/3 is built into the code) and this is chosen as the time step for the next update. In a Lagrangian mesh the time step must satisfy the CFL or Courant condition

$$\Delta t \leq \frac{d}{c}$$

where $d$ is a typical length of a zone (defined as the volume of the zone divided by the square of the longest diagonal of the zone and scaled by $\sqrt{2/3}$) and $c$ is the local sound speed. This ensures that a disturbance does not propagate across a zone in a single time step.

There are also two Von Neumann and Richtmeyer time steps ($\Delta t_2, \Delta t_3$) required for stability. The minimum value of $\Delta t = \min[t_1, t_2, t_3]$ must be found for all zones for the next time step of integration.

### 5.2.2 Shell Processor

The Shell processor is used to model thin structures where the use of a standard Lagrange would impose a very small time step on the calculation. The shell processor presupposes that the structure being modelled is "thin" such that a biaxial state of stress may be assumed. No wave propagation occurs across the shell but only along it’s length. Although thickness is considered in the shell formulation, it is not included in the geometric representation of the shell and does not enter into the time step calculation.

The shell subgrid is defined as a constant I=1 with J representing a string of nodes along the shell in 2D and J and K defining a surface of nodes in 3D. One goal for the simulations in this thesis was to study failure in the material. For shell this could not be done through the thickness, so shell elements were not used.
5.2.3 Processor Coupling

Subgrids may interact with each other (and themselves) in a number of ways depending on their processor types. The interaction feature makes AUTODYN powerful for solving "coupled" problems (e.g. impacts, explosions, fluid-structure interactions, etc.). Different numerical processors (solvers) can be applied optimally according to the phenomena being modelled and then linked together automatically in space and time. In this thesis the Lagrange/Lagrange (Lag./Lag.) interaction will be used:

Two Lagrange type subgrids (including Shell and ALE processors) may interact through either a join or a Lag./Lag. impact-slide surface. The join specification allows surfaces from different Lagrange subgrids to be joined node to node. Joined nodes are regarded as a single node in the calculation. This is very useful for constructing complex models. Lag./Lag. impact-slide surfaces allow the surfaces of Lagrange bodies to interact. Lagrange bodies are also tested for self-impact by default. Lag./Lag. interactions are used to simulate a wide variety of interaction phenomena including impact and penetration. For two subgrids defined to interact, the surface nodes of the second subgrid are tested to see if the would penetrate the surface of the first subgrid. This is done at each timestep. If any nodes penetrate, momentum conserving interactions are computed to prohibit penetration.

5.2.4 Material Models

The material models chosen in the problem will depend on the physical materials in the problem. Four basic types must be specified for each material. Only types used in the thesis are listed.

1. Equations of State
   - Linear: A bulk modulus and reference density are defined.
   - Orthotropic: Used to model orthotropic materials.

2. Yield Models
   - Elastic: No yield surface. A constant shear modulus is defined.
   - Von Mises: A constant yield surface and shear modulus are defined

3. Failure Mode
   - None: The material will never fail
5.3 Material Properties and Material Models

- Material Stress and Strain: The material fail due to stress or strain in the material principal directions.

- Erosion Model Erosion criteria. When a material is eroded it is transformed from solid element to a free mass node.

5.2.5 Material Stress and Strain Failure

These models allows different tensile and shear failure stresses and strain for each of the principal directions. Following failure in a cell, if the stress or strain reaches the user specified limiting value, the following occurs:

- The principal material stress in the direction of failure is set to zero.
- The shear modulus in the direction of failure is set to zero.
- The shear stress in the direction of failure is set to zero.
- The average stress (i.e. pressure) is recomputed, using the normal calculation

\[ p = -\left(\sigma_{11} + \sigma_{22} + \sigma_{33}\right)/3 \]

Post-failure behaviour is effectually isotropic:

- The orthotropic elastic incremental stress-strain relations are applied
- The average stress (pressure) is recomputed, using the calculation above
- If the cell is in compression the principal stresses are set equal to the average stress (pressure), i.e. \( \sigma_{ij} = -p \)
- If the cell is in tension all principal stresses, and therefor the average stress (pressure), are set to zero, i.e. \( \sigma_{ij} = -p = 0 \)

5.3 Material Properties and Material Models

Impacts on GFRP plates have been simulated earlier with AUTODYN by Feuerline [16]. The model used in that thesis gave results in good agreement to the instrumented impact experiments, so a similar model will be used here. Feuerline did only compare deflection from his simulations and to the experiments and since deflection is calculated by integrating the acceleration twice, any differences are smoothed. In this thesis also energy plots from the simulations as well as deflection will be compared to the experiments.
The plate was modelled with orthotropic material properties and Lagrange processor. Both the dart and the frame were also modelled with Lagrange processor, but as isotropic material. With ASTM standard material test performed by SINTEF [3], most of the material properties were known. Some of the material properties can not be measured experimentally, so they had to be calculated from analytical expression derived in Chapter 2.

Young’s module out of the plane, $E_3$, was found from Eq. (2.25).

$$E_3 = \frac{E_t E_m}{E_t V_m + E_m V_f}$$

Poisson’s ration was set to 0.2. This is a common value for composite laminate and vary $\nu$ from 0.16 to 0.25 had no influence on the simulation results. The shear modulus, $G_{12}$, is very nonlinear. Between 0 and 0.5% strain, it drops from 6 GPa to 4 GPa. In AUTODYN this nonlinearity could not be implemented, so the value 6 GPa was used. The inter-laminal shear modulus $G_{13}$ and $G_{23}$ could not be determined by experiments, so this value was calculated. A corresponding equation to (2.25) is available for $G_{13}$ and $G_{23}$,

$$G_{23} = G_{13} = \frac{G_t G_m}{G_t V_m + G_m V_f}$$

(5.1)

giving $G_{13} = G_{23} = 5.14$ GPa.

Maximum stress theory and maximum strain theory were used as failure model for the composite material, given by Eq. (2.67)-(2.70). The model was three dimensional, so maximum stress and strain values in $z$ direction were needed, but $\sigma_{3u}$ and $\epsilon_{3u}$ could not be determined by experiments, so this values had to be estimated. $\sigma_{3u}$ was set to be equal the maximum tensile value for the matrix, which is $\sigma_{3u} = 65.0$ MPa and from Eq. (2.24) $\epsilon_{3u}$ was found to be 4.05%. The maximum shear strain of the matrix is $\gamma_{um} = 2 - 8\%$. A value in between was chosen for the maximum inter laminar shear strain, $\gamma_u = 4\%$.

Ultimate inter laminar shear stresses, $\tau_{23u}$ and $\tau_{13u}$, were determined to 25 MPa in shear stress test. There was a crack initiation at 25 MPa, but the material collapsed at 40 MPa. Compared to other similar material, where crack initiation starts at 40 MPa, the ultimate values are low. Different values will be used in the simulations to study their effect on the results.

The residual shear stiffness fraction was set to 0.2, which is a default value. The maximum residual shear stress was estimated to 28 MPa, which is 50% of the maximum in-plane shear stress. Friction was neglected during the impact-simulations, because these results are only slightly different from the ones with. An orthotropic post-failure response was used to study the failure in each direction.
Table 5.1: Material properties for projectile and frame used in AUTODYN simulations.

<table>
<thead>
<tr>
<th></th>
<th>Dart:</th>
<th>Frame:</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Processor:</td>
<td>Lagrange</td>
<td>Lagrange</td>
<td></td>
</tr>
<tr>
<td>Equations of State:</td>
<td>Linear</td>
<td>Linear</td>
<td></td>
</tr>
<tr>
<td>Strength Model:</td>
<td>Von Mises</td>
<td>Von Mises</td>
<td></td>
</tr>
<tr>
<td>Failure Model:</td>
<td>Bulk Strain</td>
<td>Bulk Strain</td>
<td></td>
</tr>
<tr>
<td>Erosion Model:</td>
<td>None</td>
<td>None</td>
<td></td>
</tr>
<tr>
<td>Bulk Modulus K:</td>
<td>152.17</td>
<td>58.33</td>
<td>GPa</td>
</tr>
<tr>
<td>Shear Modulus G:</td>
<td>83.00</td>
<td>31.50</td>
<td>GPa</td>
</tr>
<tr>
<td>Yield Stress $\sigma_y$:</td>
<td>1000.00</td>
<td>104.00</td>
<td>MPa</td>
</tr>
<tr>
<td>Ultimate Strain:</td>
<td>1.00E20</td>
<td></td>
<td>kPa</td>
</tr>
</tbody>
</table>

Table 5.2: Input parameters for laminate plate in AUTODYN simulations.

<table>
<thead>
<tr>
<th></th>
<th>Laminate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Processor:</td>
<td>Lagrange</td>
</tr>
<tr>
<td>Equation of State:</td>
<td>Orthotropic</td>
</tr>
<tr>
<td>Strength Model:</td>
<td>Elastic</td>
</tr>
<tr>
<td>Failure Model:</td>
<td>Material stress and strain</td>
</tr>
<tr>
<td>Erosion Model:</td>
<td>None</td>
</tr>
<tr>
<td>Input Type</td>
<td>Plate</td>
</tr>
<tr>
<td>----------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>Young’s modulus 1</td>
<td></td>
</tr>
<tr>
<td>Young’s modulus 2</td>
<td></td>
</tr>
<tr>
<td>Young’s modulus 3</td>
<td>8.23</td>
</tr>
<tr>
<td>Poisson’s Ratio 12</td>
<td>0.20</td>
</tr>
<tr>
<td>Poisson’s Ratio 23</td>
<td>0.20</td>
</tr>
<tr>
<td>Poisson’s Ratio 31</td>
<td>0.20</td>
</tr>
<tr>
<td>Shear Modulus 12</td>
<td>6.00</td>
</tr>
<tr>
<td>Shear Modulus 23</td>
<td>5.14</td>
</tr>
<tr>
<td>Shear Modulus 13</td>
<td>5.14</td>
</tr>
<tr>
<td>Material Axes Option</td>
<td>x-y-z-space</td>
</tr>
<tr>
<td>Rotation Angle about 11</td>
<td>0</td>
</tr>
<tr>
<td>X-coord. for dirn (11)</td>
<td>1</td>
</tr>
<tr>
<td>Y-coord. for dirn (11)</td>
<td>0</td>
</tr>
<tr>
<td>Z-coord. for dirn (11)</td>
<td>0</td>
</tr>
<tr>
<td>Reference Temperature</td>
<td>0</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>0</td>
</tr>
<tr>
<td>Solid EOS</td>
<td>Linear</td>
</tr>
<tr>
<td>Tensile Failure Stress 11</td>
<td>482.00</td>
</tr>
<tr>
<td>Tensile Failure Stress 22</td>
<td>482.00</td>
</tr>
<tr>
<td>Tensile Failure Stress 33</td>
<td>65.00</td>
</tr>
</tbody>
</table>
5.4 Testing the Material Properties and the Material Models

In the previous section material properties was found for the laminate. Because some of the properties were calculated analytical or found a bit low, the effect of vary them are done in the two next sections. The run time for the simulations was very long. Material properties testing could than not be done for all impacts. To test the different material properties and material models the 60 J simulation with 10.151 kg impact mass was used. This simulation was called E60-3. Most of the other simulations were performed with the material properties giving the most satisfactory result for the simulation E60-3.

5.5 Simulations of the Rosand Impacts

The first series of simulations was perform to simulate the impacts conducted in the Rosand Impact machine. The material model used is discussed in the previously section. In the experiment the plate was clamped on a aluminium frame and since AUTODYN allows the user to easily simulate interruption between many subgrids with it’s processor coupling, so the frame was modelled and fixed in all directions. Due to symmetry only a quarter of the entire setup had to be simulated. Figure 5.2 (a) shows the geometry of the plate model used in AUTODYN simulation and (b) the aluminium frame. The projectile was a 52.5 mm long steel dart with 6.25 mm radius. The dart had a spherical tip with 6.25 mm radius. In the Rosand machine weights are placed above the dart. This was not done in the simulation, instead the density of the dart was increased to get the desired impact weight. Figure 5.3 shows the mesh in the AUTODYN simulations. The plate was modelled with 30 × 30 × 6 elements.

Material properties for the dart and the frame are found in Table 5.1. In the experiments nine plates was impacted, using three energy levels and three different masses at each energy level. The energy levels were 20, 40 and 60 J and the masses were 5.151, 7.151 and 10.151 kg. As mentioned the density of the dart was changed to receive the desired impact mass. The different masses with corresponding density are listed below.

\[
\begin{align*}
5.151 \text{ kg} & \iff 832.5 \text{ g/cm}^3 \\
7.151 \text{ kg} & \iff 1155.8 \text{ g/cm}^3 \\
10.151 \text{ kg} & \iff 1640.7 \text{ g/cm}^3
\end{align*}
\]
Figure 5.2: (a) geometry of plate model and (b) frame used in AUTODYN simulation. A quarter of the setup was modelled. All lengths are given in mm.

Figure 5.3: A uniform grid was used in AUTODYN simulations. The plate had $30 \times 30 \times 6$ elements.
5.5 Simulations of the Rosand Impacts

Table 5.4: Simulation names for the 60 J Rosand impact.

<table>
<thead>
<tr>
<th>Simulation name</th>
<th>Impact Mass</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>E60-1</td>
<td>5.151 kg</td>
<td>4.827 m/s</td>
</tr>
<tr>
<td>E60-2</td>
<td>7.151 kg</td>
<td>4.096 m/s</td>
</tr>
<tr>
<td>E60-3</td>
<td>10.151 kg</td>
<td>3.438 m/s</td>
</tr>
</tbody>
</table>

5.5.1 60 J Impact Simulations

Three 60 J Rosand impacts have been conducted in the experiments and all three impacts were simulated. In Table 5.4 simulation names with corresponding masses and velocities are listed.

For impact E60-3 a few simulations were performed to study the behaviour of the plate for different material properties. The simulations were compared to the impact experiment on plate P15 described in Section 4.3.4. First the effect of different values for $\tau_{3u}$ where studied and the values $\tau_{3u} = 25$, 30 and 40 MPa were used in simulation E60-3a, E60-3b and E60-3c, respectively. The kinetic energy of the darts are shown in Fig. 5.4. For decreasing value for $\tau_{3u}$

![Comparison of kinetic energy of the dart](image)

Figure 5.4: Comparison of kinetic energy of the dart for different values for maximum inter-laminar shear strengths, $\tau_{13u}$ and $\tau_{23u}$. 25 MPa for E60-3a, 30 MPa for E60-3b and 40 MPa for E60-3c. Only the first half of impact duration is plotted.

two phenomena appears. Both the impact time and the maximum deflection increase. When the shear stress criteria are decrease failure will occur earlier and when failure occur in an element, the shear modulus in direction of failure is set to zero. This will reduce the stiffness of the material and the deflection will increase.

Comparing the simulations to the experiment on plate P15 in Fig. 4.4
there are some differences. The simulation have minimum kinetic energy at about 6 ms, while the experiment shows that the time should be about 4 ms. This indicate that the failure criteria are have to be higher. In AUTODYN failure plots of the elements can be plotted and the failure mode was set to be orthotropic, so failure in each material direction can be studied. Figure 5.5 shows a failure plot for E60-3c at time 5.6 ms. The area under the dart is totally damaged and the material has plastic deformation. The plastic deformation do not appear in GFRP composite plates, because of the fibres. The fibres are much stronger than the matrix, so if they break, the entire laminate will collapse. In the experiments only delamination and matrix cracks occurred, but in the simulation also tensile failure occurred. Tensile failure in the simulation implicate fibre breakage in the plate. This did not happened in the experiment, so the tensile failure criteria had to be increased.

![Material Status](image)

Figure 5.5: Failure plot for E60-3c at time 5.6 ms. 60 J impact energy and i 10.151 kg impact mass. Ultimate inter-laminar shear stresses 40 MPa.

An energy plot from the 60 J impact simulation E60-3c is shown in Fig. 5.6. It is obviously that the total energy of the system change through the impact. For minimum energy of the dart, the energy in the plate is only 50 J and not 60 J as expected. The frame had zero total energy during the impact. Only plot for one impact is shown here, but the same happened for all simulations when using orthotropic material properties. For simulation of impact on an isotropic material, such as aluminium, there were no energy error.

The energy error plot for the energy in Fig. 5.6 for the simulation E60-3c is shown in Fig. 5.7. Both initial and current energy are plotted. Initial energy is constant at 60 J which is the impact energy of the impactor. For small energy error, the current energy should be about the same as initial energy, but for
Figure 5.6: Total energy of the impactor and plate for the impact simulation E60-3c. The total energy in the system is not constant. The plate do not absorb all the energy from the dart, so there is an energy error in the simulation.

Figure 5.7: Energy error for the 60 J impact simulation E60-3c. The energy error is plotted in both J and %.
Table 5.5: Material properties for the simulations E60-3k and E60-3e. All other properties unchanged from Table 5.3.

<table>
<thead>
<tr>
<th></th>
<th>E60-3k</th>
<th>E60-3e</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_{11U} = \sigma_{22U} = 550$ MPa</td>
<td>$G_{13} = G_{23} = 7.0$ GPa</td>
<td></td>
</tr>
<tr>
<td>$\tau_{13U} = \tau_{23U} = 50$ MPa</td>
<td>$\tau_{13U} = \tau_{23U} = 40$ MPa</td>
<td></td>
</tr>
</tbody>
</table>

orthotropic materials the error increase until the impactor have lost all it’s kinetic energy. The error for this simulation is then -10 J or 16 %. This is the energy difference in Fig. 5.6. Now the error decrease and after 11 ms the error is 11 %.

As shown the energy error in the simulation increased rapidly and the default value of 5% was to small. To be sure that the simulation did not stop, the energy error was set to 20%. ANKER-ZEMER Group in Oslo, who distribute AUTODYN in Norway, confirmed that the energy error become high when using orthotropic material models.

To prevent the plastic deformation showed in simulation E60-3c two new simulation were done. For E60-3k the ultimate stress failure criteria were increased and for E60-3e the shear modulus $G_{13}$ and $G_{23}$ were increased. Material properties for these two simulations are listed in Table 5.5 and the energy comparison is shown in Fig. 5.8.

![Figure 5.8: Simulation E60-3a, E60-3e and E60-3k compared with experiment. 60 J impact with 10.151 kg impact mass.](image)

The time used to reach minimum kinetic energy has been reduced about 1 ms from simulation E60-3a to E60-3k. E60-3e has it’s minimum inbetween. For the simulation E60-3k the time used to reach minimum energy of the
dart is only about 0.7 ms longer than in the experiment. Maximum deflection for E60-3c, E60-3e and E60-3k are -11.9, -11.2 and -10.8 mm, respectively. In the experiment the deflection was -10.1 mm, so the values found from the simulations are very satisfactory, but maximum deflection is reached a bit too late.

![Material Status](image)

**Figure 5.9:** Failure plot for the simulation E60-3k after 5.6 ms, 60 J and 10.151 kg impact mass. Material properties as in Table 5.3, but with $\sigma_{11U} = \sigma_{22U} = 550$ MPa and $\tau_{13U} = \tau_{23U} = 50$ MPa

The failure in simulation E60-3k is plotted in Fig. 5.9. All elements under the projectile have failed. Failure 23 and 31 dominate at the corner. A few elements have bulk failure, which is used when a element have failed in two or more directions. The elements at the bottom have tensile (Failed 11) failure. This failure implicate a fibre breakage, which did not happened in the experiments, so ultimate tension stress should be more than 550 MPa. A simulation was performed with $\sigma_{11U} = \sigma_{22U} = 650$ MPa, but the result was almost identical as for 550 MPa. There are also some in-plane failure, shown as failure 12 and the only in-plane failures occurred in the experiment were matrix cracks.

From the failure plot it is not possible to tell if the material failed due to stress or strain. Some simulation, with only strain failure criteria, were performed to check the failure caused by strain criteria. Different values for the shear strain criteria were used. Simulation E60-3g with 4%, E60-3h with
Figure 5.10: Energy plot from the simulations with material strain failure models. Maximum shear strains criteria were 2%, 3% and 4% for simulation E60-3g, E60-3h and E60-3i, respectively. 3% and E60-3i with 2%. Energy plot of these three simulation are found in Fig. 5.10. The differences between these three simulations are insignificant, so it seemed like the maximum shear strain values were too high. To see which failure occurred a failure plot of E60-3h is shown in Fig. 5.11. Almost all element that failed, failed in tension strain in 11 or 22 direction. Some of the elements have failed in two or more directions, shown as bulk failure. Only a couple of the elements have failed due to inter-laminar shear strain. Some new simulation were performed, where the maximum tension strain were increased, to make the material fail due to shear strain. Setting $\epsilon_{11U} = \epsilon_{22U} = 4\%$, $\epsilon_{33U} = 4.5\%$ and the maximum shear strains to 3% in E60-3m. The kinetic energy of the dart from this simulation are compared to the E60-3g simulation in Fig. 5.12. In simulation E60-3m the material was now complete elastic. Almost all energy are returned to the dart in the impact, so no energy have been used to damage the plate.

The simulations E60-1 and E60-2 were performed with the same material properties used for E60-3k. A comparison of the three three different simulations presented here. Figure 5.13 shows a comparison of three 60 J impacts simulations with different masses. The corresponding plots and masses are listed in Table 5.4. This figure can be compared to the experiments shown in Fig. 4.6. Notice that Fig. 4.6 shows the energy in the plate and not the kinetic energy of the dart as presented from the simulations. The impact duration decreases for decreasing mass. An analytical expression for the frequency of the
5.5 Simulations of the Rosand Impacts

Figure 5.11: Failure plot for the simulation E60-3h. 60 J impact with 10.151 kg impact mass. Material strain model and maximum shear strains set to 3%.

![Failure plot for the simulation E60-3h.](image)

Figure 5.12: Comparison of the material strain failure model, E60-3g and the no failure model, E60-3m. The no failure model is almost perfect elastic and the duration of the impact is too short compare to the strain failure model.

![Comparison of the material strain failure model, E60-3g and the no failure model, E60-3m.](image)
**AUTODYN Impact Simulations**

Figure 5.13: Comparison of E60-1k, E60-2k and E60-3k. 60 J impact with masses 5.151 kg, 7.151 kg and 10.151 kg.

system, \( \omega \), is found in Section 3.5

\[
\omega = \sqrt{\frac{K_b}{M_i}}
\]

When increasing the mass, the frequency will decrease and the impact duration increase. For the experiments the time difference between 5.151 kg and 10.151 kg was found to be 1.5 ms and the same difference is seen in the simulation. Simulation E60-2k are very close to E60-3k. In the experiments the 7.151 kg impact was in between the two others.

### 5.5.2 40 J Impact Simulations

From the 60 J simulations the material properties achieved the best results compared to the experiment were

- Tensile failure stress \( \sigma_{11U} = \sigma_{22U} = 550 \text{ MPa} \)
- Maximum shear stress \( \tau_{13U} = \tau_{23U} = 50 \text{ MPa} \)

The rest of the properties unchanged from Table 5.3. No other material properties were used for the three 40 J impact simulations. Figure 5.14 shows comparison of the three 40 J simulation done. The impact duration increase for increasing mass. This was explained for Fig. 5.13.

### 5.5.3 20 J Impact Simulations

Just as for the 40 J simulations, material properties found for 60 J are used.
5.5 Simulations of the Rosand Impacts

Figure 5.14: Three 40 J impacts with different masses. For E40-1k the impact mass was 5.151 kg, for E40-2k 7.151 kg and for E40-3k the mass was 10.151 kg.

<table>
<thead>
<tr>
<th>Simulation name</th>
<th>Impact Mass</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>E40-1</td>
<td>5.151 kg</td>
<td>3.941 m/s</td>
</tr>
<tr>
<td>E40-2</td>
<td>7.151 kg</td>
<td>3.345 m/s</td>
</tr>
<tr>
<td>E40-3</td>
<td>10.151 kg</td>
<td>2.807 m/s</td>
</tr>
</tbody>
</table>
AUTODYN Impact Simulations

- Tensile failure stress $\sigma_{11U} = \sigma_{22U} = 550$ MPa
- Maximum shear stress $\tau_{13U} = \tau_{23U} = 50$ MPa

The rest of the properties unchanged from Table 5.3. Three simulations were done with different impact masses and velocities as listed in Table 5.7.

<table>
<thead>
<tr>
<th>Sim. &amp; Exp. name</th>
<th>Impact Mass</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>E20-1</td>
<td>5.151 kg</td>
<td>2.787 m/s</td>
</tr>
<tr>
<td>E20-2</td>
<td>7.151 kg</td>
<td>2.365 m/s</td>
</tr>
<tr>
<td>E20-3</td>
<td>10.151 kg</td>
<td>1.985 m/s</td>
</tr>
</tbody>
</table>

The energy plot for the simulations are found in Fig. 5.15. Different mass gives different impact duration. The energy returned to the dart is about the same for all three simulations.

![Energy plot](image)

Figure 5.15: Three 20 J impacts with different masses. For E20-1k the impact mass was 5.151 kg, for E20-2k 7.151 kg and for E20-3k the mass was 10.151 kg.

5.6 Simulations of the Drop Weight Impacts

The second series of simulations were performed to simulate the drop weight impacts. A 5.2 mm thick and $300 \times 300$ mm² large plate was simply supported on a quadratic steel frame shown in Fig. 5.16. To simplify the model, the
5.6 Simulations of the Drop Weight Impacts

![Diagram of frame and steel pipe](image)

Figure 5.16: Top view of the frame and diameter of the steel pipe in the frame. The corners of the frame were not modeled and only top half of the frame was modeled.

corners of the frame were not modelled. This should not effect the simulation, because the corners of the plate would only bend up from the frame. Only the top half of the frame was modelled. The projectile was a solid steel cylinder with a spherical tip. Both the radius of the cylinder and the radius of the tip were 37.5 mm. Assuming no deformation of the steel pipe during the impact tests, the pipe frame was modelled as a solid steel cylinder. The frame was fixed, so that displacement of the frame did not happened. Due to symmetry, only a quarter of the setup was modelled. The uniform element mesh is shown in Fig. 5.17 and the plate had $36 \times 36 \times 6$ elements.

Two different impacts on the simply supported plate were simulated. One with 60 J impact energy and the second with 152 J impact energy.

- **60 J impact:**
  - Impact mass 6.2 kg
  - Impact velocity 4.4 m/s

- **152 J impact:**
  - Impact mass 6.2 kg
  - Impact velocity 7.0 m/s

89
Figure 5.17: Grid used in AUTODYN simulation for simply supported plate.
Table 5.8: Material properties for S3-a, S3-c and S3-d. Remaining properties unchanged from Table 5.3.

<table>
<thead>
<tr>
<th></th>
<th>S3-a</th>
<th>S3-c</th>
<th>S3-d</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\tau_{13U} = \tau_{23U} = 25$ MPa</td>
<td>$\tau_{13U} = \tau_{23U} = 30$ MPa</td>
<td>$\tau_{13U} = \tau_{23U} = 40$ MPa</td>
<td></td>
</tr>
</tbody>
</table>

### 5.6.1 The 60 J Impact Simulation

For this impact the velocity of the dart was 4.4 m/s. From the simulations performed for the Rosand impacts, the material properties listed in Table 5.3 achieved satisfactory results when the stress failure criteria were increased. Some simulations with different ultimate stress criteria were performed. First different values for $\tau_{13U}$ and $\tau_{23U}$ were used. The criteria for each simulation are listed in Table 5.8.

![Energy comparison of simulations S3-a, S3-c and S3-d](image)

Figure 5.18: Energy comparison of simulations S3-a, S3-c and S3-d. Material properties are listed in Table 5.8

Figure 5.18 shows a comparison of simulation with different values for maximum shear stress and it is seen that the differences are insignificant. There are no difference between 25 MPa and 30 MPa. For 40 MPa the path is a bit different. It reach minimum kinetic energy a bit earlier then for the two other plots.

For the Rosand impact simulations satisfactory results were achieved when

- $\sigma_{11U} = \sigma_{22U} = 550$ MPa
- $\tau_{13U} = \tau_{23U} = 50$ MPa
and the other properties unchanged from Table 5.3. These properties are used in the S3-k simulation.

![Graph](image1)

Figure 5.19: Energy comparison of the simulations S3-k and S3-c.

In Fig. 5.19 the plots from simulation S3-c and S3-k are compared. The two plots are almost identical until about 3.0 ms. They start, of course, at the same energy level and follow each other through the peak at 2 ms and down to 30 J at 3.0 ms. Simulation S3-c and S3-k have minimum at 6.0 and 5.7 ms, respectively and from there the energy increase to about 30 J for both the plots.

![Graph](image2)

Figure 5.20: Comparison of the displacement for the simulations S3-k and S3-c.

The displacement plot in Fig. 5.20 shows the same differences as for the energy plot. The plates have the same deflection until 3.0 ms. The value for
maximum deflection is about the same for the plots, but the impact duration is different.

![Energy Dart and Energy Plate](image)

Figure 5.21: Energy of the impactor and the plate for the impact simulation S3-c. The total energy in the system is not constant, so there is an energy error in the simulation.

Figure 5.21 shows total energy of the projectile and the plate for the S3-c simulation. As for the E60-3c simulation there is an energy error. The plate do not absorb all the energy. When the projectile have minimum kinetic energy the plate have only 52 J total energy and not 60 J. In Fig. 5.22 the energy error in J and %, initial and current energy are plotted. Initial energy is constant at 60 J which is the impact energy of the impactor. The maximum error of this simulation is 8 J or 13 %. At the end of the impact the error has decreased to 4%.

Figure 5.23 shows a failure plot for impact S3-k at time 5.6 ms. There are a lot of tension failure at the corner. In the experiment only delamination and matrix cracks occurred. Delamination is inter-laminar and shows as Failed 13 and 23 in the simulation. Matrix cracks are in-plane failure showed as Failed 12.

### 5.6.2 The 152 J Impact Simulation

For this simulations the impact velocity were 7.0 m/s. Two simulations were performed with the material properties used in the S3-c and S3-k simulations and they are listed in Table 5.9.

From Fig. 5.24 the energy results from the simulations can be studied. For S4-k the energy decrease faster then for S4-c, so the difference in impact
Figure 5.22: Energy error for 60 J impact S3-c. Energy error shown both in J and %.

Figure 5.23: Failure for 60 J drop weight impact simulation S3-k at time 5.7 ms.
Table 5.9: Material properties for simulations S4-c and S4-k. Other properties as in Table 5.3.

<table>
<thead>
<tr>
<th></th>
<th>S4-c</th>
<th>S4-k</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\tau_{13U} = \tau_{23U}$</td>
<td>40 MPa</td>
<td>$\tau_{13U} = \tau_{23U}$</td>
</tr>
<tr>
<td>$\sigma_{11U} = \sigma_{22U}$</td>
<td>550 MPa</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5.24: Kinetic energy of the dart from the simulations for 152 J impact on simply supported plate. The material properties for the two simulations are found in Table 5.9.
duration is about 0.5 ms for the two plots. S4-c do have lower failure criteria so more damage will occur and the stiffness of the plate will be reduced.

![Displacement plot](image)

Figure 5.25: Displacement from 152 J simulations on simply supported plate. The material properties for the two simulations are found in Table 5.9.

For the displacement plot in Fig. 5.25 the differences are clearly. The plate in simulation S4-c and S4-k has -27.8 and -25.4 mm displacement, respectively.

## 5.7 Summary

In the previously sections simulations of the impact experiments have been carried out. Most of the material properties had previously been determined by ASTM test at SINTEF, but some had to calculated from analytical expressions because they can not be determined experimentally. To calibrate the properties one 60 J Rosand impact was simulated. The ultimate in-plane tension stress and inter-laminar shear stress were too low, so they were increased and the best result, compare to the experiment, was achieved for

- $\sigma_{11U} = \sigma_{22U} = 550$ MPa
- $\tau_{13U} = \tau_{23U} = 50$ MPa

and the other properties unchanged from Table 5.3. The failure model was set to be a material stress and strain failure and without increasing the ultimate stresses, too much damage occurred in the material. This resulted in a too long impact duration and too large deflection. With the new failure criteria the simulation was satisfactory, even there was a impact duration error of
0.7 ms and some energy loss for the dart. The ultimate stresses could be set even higher, but the values would then be too high compared to similar GFRP composites.

For the 40 J and the 20 J Rosand impact simulations the material properties found for the 60 J impact simulation were used. In this chapter only impact duration for different impact masses were shown. Comparison of these simulations and the drop weight impact simulations will be carried out in Chapter 7.
Chapter 6

ANSYS Simulations

6.1 Introduction

To perform static simulation on the laminate plates ANSYS Structural 6.1 from ANSYS Inc. was used. ANSYS Structural is a numerical finite element method (FEM) [11] program used to perform linear or nonlinear static analyses of two and three dimensional structures. Dynamic analyses, such as natural frequency, harmonic response and random vibration, are also possible. The nonlinearity can be both geometric and material.

6.2 Comparing Element Types

ANSYS contains a large element library [4] which allows the user to choose suited elements for different analyses. When using the software for the first time, different elements should be compared to discover advantages and disadvantages and their restriction. Also when using new element types, they should be compared to well know elements.

For the static analyses in this thesis both three dimensional SOLID and two dimensional SHELL elements could be used. For solid elements variation of stress and strain through the height can be studied, but they have high CPU cost. Shell elements have only one node in the through thickness direction, so only in-plane stress and strain can be examined. One big advantage when using shell elements is the low CPU cost. The shell elements could be used since the thickness of the laminate is less than 1/10 of the length. For all element comparison simulations the simulations were performed without any nonlinear geometric effects. This was done to save run time and to simplify the models. In the ANSYS element library also layered elements are available.
for modelling multi layered material. When using these elements, each layer in a composite laminate can be modelled. The material properties for each layer must be specified.

### 6.2.1 An Orthotropic Plate

To verify the selected non-layered elements, a plate with orthotropic material properties was simulated. The geometry of the plate is shown in Fig. 6.1 and the model is a quarter of a simply supported square plate. In Table 6.2 the material properties are listed, which are discussed in Section 5.3. An analytical solution of this problem are know from Section 3.3 and will be used to verify the results from the simulations.

For SOLID45 and SOLID95 the mesh was $28 \times 28 \times 4$ elements and for SHELL63 $28 \times 28 \times 1$ elements. Number of nodes and degrees of freedom (DOF) for each element type are listed in Table 6.1.

The pressure $P$ was set to 125 MPa and applied at the corner. Displacement of point A was compared for the selected element types.

Three different element types were tested. Two SOLID elements and one SHELL element. A short description of the element types are given:

- **SOLID45**
  - 3-D Structural Solid
Table 6.1: Number of elements, nodes and degrees of freedom for each element type tested in orthotropic laminate plate.

<table>
<thead>
<tr>
<th>Element type</th>
<th>Elements</th>
<th>Nodes</th>
<th>DOF</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOLID45</td>
<td>28 x 28 x 5</td>
<td>3125</td>
<td>9375</td>
</tr>
<tr>
<td>SOLID95</td>
<td>28 x 28 x 5</td>
<td>11625</td>
<td>34875</td>
</tr>
<tr>
<td>SHELL63</td>
<td>28 x 28 x 1</td>
<td>625</td>
<td>1875</td>
</tr>
</tbody>
</table>

Table 6.2: Orthotropic material properties for the plates used in static simulation.

<table>
<thead>
<tr>
<th>Material Properties</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_x$</td>
<td>31.50</td>
<td>GPa</td>
</tr>
<tr>
<td>$E_y$</td>
<td>31.50</td>
<td>GPa</td>
</tr>
<tr>
<td>$E_z$</td>
<td>8.23</td>
<td>GPa</td>
</tr>
<tr>
<td>$\nu_{x}$</td>
<td>0.20</td>
<td>-</td>
</tr>
<tr>
<td>$\nu_{y}$</td>
<td>0.20</td>
<td>-</td>
</tr>
<tr>
<td>$\nu_{z}$</td>
<td>0.20</td>
<td>-</td>
</tr>
<tr>
<td>$G_{xy}$</td>
<td>6.00</td>
<td>GPa</td>
</tr>
<tr>
<td>$G_{yz}$</td>
<td>5.14</td>
<td>GPa</td>
</tr>
<tr>
<td>$G_{zz}$</td>
<td>5.14</td>
<td>GPa</td>
</tr>
</tbody>
</table>

- 8 nodes element
- Three degrees of freedom in each node: UX, UY and UZ
- The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities
- Figure 6.2 (a)

- SOLID95
  - 3-D Structural Solid
  - 20 nodes element
  - Three degrees of freedom in each node: UX, UY and UZ
  - The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities
  - Figure 6.2 (b)

- SHELL63
  - Elastic shell with both bending and membrane capabilities.
- 4 nodes element
- Six degrees of freedom in each node: UX, UY, UZ, ROTX, ROTY and ROTZ
- Figure 6.2 (c)

![Diagram of different element types](image)

(a) Eight nodes SOLID45  
(b) Twenty nodes SOLID95  
(c) Four nodes SHELL63

Figure 6.2: Different element types used in the ANSYS simulations.

As shown in Table 6.3 the different element types resulted in almost the same deflection. The two SOLID elements gave the same deflection, so using a 8 node or 20 nodes element is indifferent. Eight nodes SOLID45 elements were in the following simulations because the simulation was significant faster than for SOLID95 elements. Between the two SOLID elements and the SHELL63 element the difference is nearly 0.5 mm. The SHELL63 solution is closest to the analytical solution, which was expected since the SHELL elements are based on the same mathematical model as the analytical solution. For SHELL elements node solution through the thickness of the model is not possible, since they only have node in the through thickness direction. SOLID45 elements are therefore better suited for the simulations, because stress and strain distributions will be studied.
Table 6.3: Deflection of square orthotropic plate with pressure applied at the corner. The model is shown in Fig. 6.1.

<table>
<thead>
<tr>
<th></th>
<th>Analytic</th>
<th>SHELL63</th>
<th>SOLID45</th>
<th>SOLID95</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>10.945</td>
<td>10.787</td>
<td>11.302</td>
<td>11.362</td>
</tr>
</tbody>
</table>

Table 6.4: Material properties for an unidirectional layer

<table>
<thead>
<tr>
<th>Material Properties</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_T$</td>
<td>43.90</td>
<td>GPa</td>
</tr>
<tr>
<td>$E_L$</td>
<td>8.23</td>
<td>GPa</td>
</tr>
<tr>
<td>$E_Z$</td>
<td>8.23</td>
<td>GPa</td>
</tr>
<tr>
<td>$\nu_{LT}$</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>$\nu_{TL}$</td>
<td>0.04</td>
<td>-</td>
</tr>
<tr>
<td>$\nu_{LZ}$</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>$G_{LT}$</td>
<td>5.14</td>
<td>GPa</td>
</tr>
<tr>
<td>$G_{TL}$</td>
<td>5.14</td>
<td>GPa</td>
</tr>
<tr>
<td>$G_{LZ}$</td>
<td>5.14</td>
<td>GPa</td>
</tr>
</tbody>
</table>

### 6.2.2 An Orthotropic Layered Plate

To verify the layered elements an orthotropic plate was simulated. The geometry was the same as for the orthotropic plate, but the plate had 8 unidirectional layers, each with thickness 0.65 mm. The stacking was $[0/90/0/90]_S$. From the equations in Section 2.2 the material properties in longitudinal and transverse direction were calculated for each layer and the properties are listed in Table 6.4.

Also here three different elements were used, two SOLID elements and one SHELL element. A short description of the layered element types used:

- SOLID46
  - Layered version of Solid45

Table 6.5: Number of elements, nodes and degrees of freedom for each element type tested in layered laminate plate.

<table>
<thead>
<tr>
<th>Element type</th>
<th>Elements</th>
<th>Nodes</th>
<th>DOF</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOLID46</td>
<td>$28 \times 28 \times 5$</td>
<td>4205</td>
<td>12615</td>
</tr>
<tr>
<td>SOLID191</td>
<td>$28 \times 28 \times 5$</td>
<td>11625</td>
<td>34875</td>
</tr>
<tr>
<td>SHELL99</td>
<td>$28 \times 28 \times 1$</td>
<td>1825</td>
<td>10950</td>
</tr>
</tbody>
</table>
- Eight nodes
- Three degrees of freedom at each node: UX, UY and UZ
- Used to model layered thick shells or solids
- Figure 6.3 (a)

- SOLID191
  - Layered version of Solid95
  - Twenty nodes
  - Three degrees of freedom at each node: UX, UY and UZ
  - Used to model layered thick shells or solids
  - Figure 6.3 (b)

- SHELL99
  - Linear Layered Shell
  - Eight nodes element
  - Six degrees of freedom in each node: UX, UY, UZ, ROTX, ROTY and ROTZ
  - Figure 6.3 (c)

Table 6.6: Simulated deflection for layered elements.

<table>
<thead>
<tr>
<th></th>
<th>Analytic</th>
<th>SOLID46</th>
<th>SOLID191</th>
<th>SHELL99</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>14.123</td>
<td>14.936</td>
<td>15.009</td>
<td>15.010</td>
</tr>
</tbody>
</table>

The differences between the element types, listed in Table 6.6 are insignificant and the analytical solution is in good agreement with the simulations. Compared to the non-layered elements the deflection is a bit larger, as a result of the material properties and the model used. Most of the material properties used in non-layered simulation are determined by material tests, while the layered properties are all calculated from properties of the fibre and the matrix. In this model the layers are assumed to have exact the same height. Regard to the assumptions done in this layered model, the simulated deflection is in good agreement with the the non-layered simulation.
6.3 Simulated Plate Stiffness

In Section 4.6 the plate stiffness is found by experiment for two different plates and compared to an ANSYS simulation and an analytical solution. The ANSYS simulations used are described in this section. The first plate was $175 \times 175$ mm$^2$ and the second was $300 \times 300$ mm$^2$.

Simulation with both SOLID45 and SHELL63 elements were performed. For each element type two different models shown in Fig. 6.4 were used. In the first model, the plate is simply supported on a frame, which is smaller than the plate. In the second model the plate simulated just as large as the light opening of the frame. The second models was used because the analytical solution in Eq (3.13) use this model and a comparison of the analytical solution and the simulation could then be carried out.

For the small plate simulated, $l_1$ and $l_2$ were 87.5 mm and 67.5 mm, respectively. The mesh used for the SOLID45 elements was $28 \times 28 \times 5$ and $28 \times 28 \times 1$ for the SHELL63 elements. A SOLID45 element has eight nodes and three degrees of freedom (DOF) at each node. SHELL63 has four nodes and six DOF at each node. The number of degrees of freedom was then 15138 for the SOLID45 simulation and 5046 for the SHELL63 simulation. To verify the grid size some simulations with half the grid size were performed, but the
6.3 Simulated Plate Stiffness

Figure 6.4: Side view of the two models used for static deflection simulations: (a) for a plate simply supported on a frame and (b) the plate is just as big as the light opening of the frame.

Figure 6.5: Grid used in ANSYS for simulation of small plate: (a) $28 \times 28 \times 5$ SOLID45 elements and (b) $28 \times 28 \times 1$ SHELL63 elements.
results did not change, so the grid size used was good enough. A comparison between 8 nodes SOLID45 and 20 nodes SOLID95 elements showed also the same results, so SOLID45 elements was chosen to decrease the run time of the simulations. The comparison is shown in Fig. 6.6 and the differences are insignificant.

![Graph showing force vs. deflection for 28 SOLID45 and 56 SOLID45 elements, and 28 SOLID95 elements.]

Figure 6.6: Comparison of deflection-force for $28 \times 28 \times 5$ SOLID45 and SOLID95 elements and $56 \times 56 \times 8$ SOLID45 elements for static deflection simulation. Model used is shown in Fig. 6.4 (a).

For the simulations of the large plate, where $l_1 = 150$ and $l_2 = 130$, the mesh was $36 \times 36 \times 5$ and $36 \times 36 \times 1$ for the SOLID45 and the SHELL63 model, respectively.

For each plate 6 simulations were done. The pressure applied was increased for every simulation. In Section 4.6 the results from all these static plate deflection simulations are presented.

### 6.4 Stress and Strain Analyses

In this section the stress and strain distributions for plates in the impact tests are found by static deflection. Maximum deflection known from the impact experiments was applied to the plate and stress and strain were then found. The stress patterns were compared to the impacts simulation performed in
AUTODYN and the strains were be compared to the strains measured with gauges in the impact tests.

Geometric nonlinear simulations were used. ANSYS employs the Newton-Raphson’s method [20] to solve nonlinear problems. In this approach, the load is subdivided into a series of load increments. The load increments can be applied over several load steps.

First the plate used in Rosand Impact machine were simulated and second simulation for the simply supported plate in the drop weight test.

### 6.4.1 The Clamped Plate

To perform stress and strain analyses on the plate tested in the Rosand Instrumented Weight Falling device, the model shown in Fig. 6.7 was used. The plate was meshed with $48 \times 48 \times 4$ elements, so the number of DOF was 36015. In the test set up, the plate was clamped on an aluminium frame. This could be simulated in ANSYS with contact analysis, but to save run time this was avoided. Instead the problem was simplified by allowing no displacement in $z$ direction for the nodes at the patterned area in Fig 6.7. From the experiments of the 60 J impacts, maximum deflection was 12 mm. This deflection was applied at the corner nodes. Number of substeps in the nonlinear analyses was set

![Figure 6.7: The model used in stress and strain analyses for clamped plate. Two paths are used for strain analyses. One from the centre to the edge, OA and one from the centre to the corner, OB.](image)
to be minimum 20 and maximum 100. The simulation converged after 22 steps. The failure criterion for normal stresses $\sigma_{11U}$ and $\sigma_{22U}$ was set to 550 MPa in

![Element solution of in-plane tension stress in $x$ direction for the clamped plate. Only stresses at the corner exceeded the failure criterion of the material. Applied deflection is 12 mm.](image)

Figure 6.8: Element solution of in-plane tension stress in $x$ direction for the clamped plate. Only stresses at the corner exceeded the failure criterion of the material. Applied deflection is 12 mm.

AUTODYN simulations. Compression stresses in $x$ direction higher than 550 MPa in the ANSYS simulation are shown in Fig. 6.8. On the bottom of the plate there are corresponding high tension stresses, not shown here. Element solution is used instead of node solution, because node solution is smoothed over the element boarders.

The in-plane shear stresses are plotted in Fig. 6.9. Maximum values are found along the diagonal of the plate. The failure criteria used in AUTODYN were 55.9 MPa. There is a pretty large area with in-plane shear stress higher than 55.9 MPa.

The inter-laminar shear stress $\tau_{xz}$ is plotted in Fig. 6.10. Only near the corner high stresses occur. Delamination occurred in the experiments, because of high inter-laminar shear stress. The failure criterion for $\tau_{xz}$ and $\tau_{yz}$ was set to 50 MPa. This is a high criterion compared to the values found in experiments, but the value was set high to avoid too much damage occur in AUTODYN simulation. Only a few elements exceeded this value for inter-laminar shear. $\tau_{yz}$ is identical to $\tau_{xz}$ only mirrored about the diagonal. Comparing the delami-
nation from the experiments with this inter-laminar shear stress, the failure area looks too small. In Section 7 these stress plots will be compared to the damage areas found in AUTODYN simulation and in the experiments.

Figure 6.11 shows the normal strain along the diagonal on both top and bottom and the strain along line OA in Fig. 6.7. The point O is the centre of the whole plate. The strain near the centre are very high. Almost 9% at the top and 8% at the bottom, which are much higher than the ultimate strain in the material. 10 mm away from the centre the strain along OA and OB has dropped to 2%. These two paths do have the same strain until 40 mm, so the deflection is symmetric.

6.4.2 The Simply Supported Plate

Figure 6.4 (a) shows the model of the plates used in the static simulations of the drop weight tests. \( l_1 \) was 150 mm. Instead of applying force, displacement of the centre nodes was set to be maximum deflection from the experiment. The plate was modelled with \( 48 \times 48 \times 4 \) SOLID45 elements.

Two impacts were conducted in the experiments. One 60 J and one 152 J impact. Maximum deflection of the plates were calculated to be 15 mm and 23
Figure 6.10: Element solution of inter-laminar shear stress in \(xz\)-direction for clamped plate. A small area near the corner have inter-laminar shear stress higher than the failure criterion. The deflection is 12 mm

Figure 6.11: Simulated normal strains on top and bottom along the diagonal OB and at line OA shown in Fig. 6.7.
mm. Only the 152 J impact are present here. The 60 J impact do look much the same, only with smaller damage areas and lower stresses and strains.

![Element normal stress in x direction near the corner for static simulation of simply supported plate. Deflection applied is 23 mm, which is the maximum deflection in the 152 J drop weight experiment.](image)

Figure 6.12: Element normal stress in $x$ direction near the corner for static simulation of simply supported plate. Deflection applied is 23 mm, which is the maximum deflection in the 152 J drop weight experiment.

The normal stress in $x$ direction can be studied in Fig. 6.12. In AUTODYN the failure criteria for normal stress in $x$- and $y$ direction were 550 MPa. Only a few elements near the corner have compression stress higher than the failure criteria. All those elements are placed near the corner in the model. There were no fibre breakage in the plate after the impact, so this stress simulation seems to be right.

Figure 6.13 shows the inter-laminar shear stress in the plate for 23 mm deflection. Only inter-laminar shear stresses higher than 50 MPa are plotted, which is the failure criteria used in AUTODYN. This area is too small compare to the delamination occurred in the impact test.

In the experiments the plates were instrumented with strain gauges. The strain for maximum deflection are found in these simulations and will in Chapter 7 be compared with the maximum strain measured at the strain gauges. In ANSYS strain in $x$, $y$ and $xy$ can be obtained directly. A path can be defined and node solution along the path can be written to a file. In Fig. 6.14 three paths used in the simulation are shown. The diagonal at the bottom of the
Figure 6.13: Element inter-laminate shear stress in $xz$-direction for static simulation of simply supported plate. Deflection applied is 23 mm, which is the maximum deflection in the 152 J drop weight experiment.
plate $OB_b$, the diagonal at the top of the plate, $OB_t$ and one path at the at the middle of the plate $OA$ at bottom. Notice that the figure shows the entire plate and not only a quarter as modelled in the simulations.

Figure 6.14: Paths on simply supported plate used to obtain strains in ANSYS simulation.

To obtain the normal strain along the diagonal $OB$ in Fig. 6.14 the following expression derived in [5] was used

$$\epsilon_n = \frac{\epsilon_x + \epsilon_y}{2} + \frac{\epsilon_x - \epsilon_y}{2} \cos 2\theta + \frac{1}{2} \gamma_{xy} \sin 2\theta \quad (6.1)$$

For the line $OB \theta = 45^\circ$ and since $\cos 90 = 0$ and $\sin 90 = 1$ and Eq. (6.1) will become

$$\epsilon_n = \frac{\epsilon_x + \epsilon_y}{2} + \frac{1}{2} \gamma_{xy} \quad (6.2)$$

Equation (6.2) was implemented in a PYTHON script reading $\epsilon_x$, $\epsilon_y$ and $\epsilon_{xy}$ from a file and calculating $\epsilon_n$. The normal strain $\epsilon_n$ along $OB_b$, $OB_t$ and the strain $\epsilon_x$ along $OA$ are plotted in Fig. 6.15 against distance from the centre. For the first 80 mm strains at $OB_b$ and $OB_t$ are almost identical. This deflection symmetry were also showed on the strain gauge measurements. The strain at the centre are about 3.3%, but drops to less than 1% at 25 mm. From 25 mm to 110 mm the strains are about constant. At the top of the plate the strain has a negative sign at at the centre, which implicit compression. The strain goes from compression to tension at 25 mm and stops at 1%. Both the top and bottom of the plate do now have the almost same strain. In Fig. 6.16 the strain profile at path $OA$ is plotted. The high strain at the centre of the plate are clearly shown, both the compression at the top and the tension at the bottom.
Figure 6.15: Simulated normal strains at paths showed in Fig. 6.14. There are high strain near the centre, but they drop fast. For path OA and OB the strains are almost identical until 80 mm from the centre.

Figure 6.16: Strain plot for the elements at path OA from Fig. 6.14. Near the centre there is high tension strain at the bottom and compression at the top.
6.5 Summary

In the previous sections static simulations of composite plates have been performed. First static deflection simulations were conducted to test different element types. Both non-layered and layered SOLID and SHELL elements were used in the simulations. The deflection was about the same for SOLID and SHELL elements. Available analytical solutions showed about the same deflection as the simulations. SHELL elements do only have one node through the thickness, so stress and strain at top and bottom can not be found. SOLID45 elements was therefore chosen for the others simulations.

Stress and strain analyses were performed for both the clamped plate and the simply supported plate. Instead of load, maximum deflection known from the impact experiments was applied. These stress analyses were done to check if static simulation in ANSYS could describe the damage area simulated using AUTODYN. Only stresses higher than the failure criteria used in the AUTODYN simulations were plotted. For both the clamped and the simply supported plate these high inter-laminar shear stress area were too small. Special for the simply supported plate, where almost none elements exceeded the failure criterion. The delamination between layers in a composite material occur due to different deflection form of layers with different fibre orientation. In ANSYS the composite was modelled as one layer with orthotropic material properties, so interface effects between distinct layers, such as delamination, can not be simulated.

Strains along three paths for the plate were also found. There were very high strains near the centre of the plates, but they dropped fast. The strain showed a symmetric deflection of the plate.
Chapter 7

Comparison of Analytical Solutions, Simulations and Experiments

7.1 Introduction

Total eleven different impact experiments have been done, nine Rosand impacts and two drop weight impacts. All the impacts have been simulated using AUTODYN and the simulations have been explained in Chapter 5, but there only one simulation was compared to the experimental impact results. That comparison was done to calibrate the material properties, which were used in the other simulations. In this chapter comparison of the simulation, analytical solution and the experiment for all impacts will be done. Differences in energy, deflection and damage will be studied and commented. There will also be comparison of stress distribution from the static ANSYS simulations and the dynamic AUTODYN simulations.

7.2 60 J Rosand Impacts

The first series of simulation were done for the 60 J Rosand impacts and Table 7.1 shows velocity and mass for each simulation and experiment. Impact 60-3 was simulated in Section 5.5.1 to calibrate the material properties and it was found that ultimate stresses needed to be increased. With the following values

- Tensile failure stress $\sigma_{1U} = \sigma_{2U} = 550$ MPa
- Maximum shear stress $\tau_{13U} = \tau_{23U} = 50$ MPa
Table 7.1: Velocity and mass for the three 60 J simulations and tests.

<table>
<thead>
<tr>
<th>Sim. &amp; Exp. name</th>
<th>Impact Mass</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>60-1</td>
<td>5.151 kg</td>
<td>4.827 m/s</td>
</tr>
<tr>
<td>60-2</td>
<td>7.151 kg</td>
<td>4.096 m/s</td>
</tr>
<tr>
<td>60-3</td>
<td>10.151 kg</td>
<td>3.438 m/s</td>
</tr>
</tbody>
</table>

and the others properties unchanged from Table 5.3 the simulations were in good agreement with the experiments. These properties were used for the two other 60 J impact simulations.

7.2.1 Energy and Deflection

Figure 7.1 shows the energy comparisons of the simulations and the experiments for all three 60 J impacts and they have one common error. The impact duration of the simulations is about 1 ms too long compared to the tests. Simulation 60-3 will be used to explain this time difference. When an element fails in stress or strain the shear module is set to zero and the element will lose some stiffness. Figure 5.9 shows damage for a large number of elements in this simulation and the stiffness of the structure is then seriously reduced. It is seen in the failure plot that there are not only delamination failures, but also tensile failures and in-plane failures, and because of all these failures there are large local deformations of the elements under the projectile. In the experiments the fibres did not break during the impact, so this local effect did not happen and therefore the projectile was stopped faster than in the simulations. The failure plots for the two other 60 J simulations did look about same as for the one discussed here, so for all these simulation the failure criteria seems to be too low. One simulation with 650 MPa ultimate tensile stresses was ran, however this did not change the results significantly. Even higher failure criteria could of course be used, to make the simulations similar to the tests, but the properties would then be too high compare to similar glass-epoxy laminates.

Even though there is a time difference and the energy returned to the projectile slightly too low, the AUTODYN simulations are satisfactory compared to the impact tests.

In Fig. 7.2 the deflections from the simulations, the analytical solutions and the experiments are compared for the three 60 J impacts. The analytical solutions are found from the mass-spring impact model in Section 3.5.

The time difference, discussed for the energy plots, are naturally also shown here. For the experiments the deflection is also smaller than for the simulations and the analytical solutions. Largest difference between simulation and
Comparison of Analytical Solutions, Simulations and Experiments

(a) 5.151 kg impact mass

(b) 7.151 kg impact mass

(c) 10.151 kg impact mass

Figure 7.1: Energy comparison between all 60 J simulations and tests for Rosand impacts. For all the plots the impact duration in the simulation is too long compared to the experiment.
Figure 7.2: Displacement comparison between all 60 J simulations, analytical solution and tests for Rosand impacts. The analytical solution and the simulation have larger deflection than the experiment, but the differences are quite small.
experiment at this energy level is about 2 mm. The simulation and the analytical solution have almost the same deflection, but the impact duration for the analytical solutions are better compared to the experiments.

7.2.2 Damage Area and Damage Pattern

In this section the damage from the experiments will be compared to the damage from the AUTODYN simulations. The impact damages in a laminate composite are very complicated and it is therefore interesting to see how AUTODYN simulates the damage. A comparison of stress distribution in ANSYS and the failure plot from the dynamic simulation is performed to see if static simulations can be used to predict damage of an impact.

Figure 7.3 shows the damage pattern for the 60 J impact with 7.151 kg impact mass and the area is 72 mm long and 46 mm wide. From the experiments it was found that damage area only depend of impact energy, so therefore only one 60 J impact is present here. In Fig 7.4 the delamination in the plate is

Figure 7.3: Damage at plate 9. Impact energy 60 J and impact mass 7.151 kg.

compared to the damage area from the AUTODYN simulation. The matrix cracks in the plate are not visible on the picture, but the area of in-plane failures (Failed 12) from the simulation predicts these cracks. There are too few inter-laminar failures in the simulation, so the delaminated area is about half of the delamination in the experiment.
Figure 7.4: (a) damage area on bottom side from experiment and (b) from AUTODYN simulation. Impact energy 60 J and impact mass 7.151 kg.

Figure 7.5 shows a comparison of the failure plot from AUTODYN and three stress plots from ANSYS, where only stresses in $x$-direction exceeding the AUTODYN failure criteria are plotted. The failure plot from AUTODYN (a) is for maximum deflection at about 5.5 ms. In (b) the compression stresses in $x$-direction are shown and a few elements on the top exceeding the failure criterion. On the bottom a corresponding area have high tension stresses, but this is not shown here, but in the AUTODYN plot only one element failed due to $x$ tension stress. The in-plane shear stresses in (c) do look more like the AUTODYN plot. Along the diagonal a lot of elements exceed the failure criterion, which is shown in both ANSYS and AUTODYN simulation. For the inter-laminar shear stress the static simulation (d) are in good agreement with the dynamic simulation. Elements near the corner failed due to inter-laminar shear for both simulations and this area is about the same for ANSYS and AUTODYN, but it is too small compared to the experiment. None of the simulations managed to recreate the delamination area from the experiment exactly. As said before, delamination occurs because of high stresses between layers with different fibre orientation. In both ANSYS and AUTODYN the laminate is modelled as one orthotropic layer, so there are no interfaces effects in the simulations. Even if the size of delamination area was not correct simulated, the damage plot from AUTODYN gives a good indication of the size of the damage. Also from the static ANSYS simulation, damage area can be partial shown.
Figure 7.5: (a) AUTODYN failure plot, (b) ANSYS tensile stress, (c) ANSYS in-plane shear stress and (d) ANSYS inter-laminear shear stress.
7.3 40 J Rosand Impacts

Table 7.2 shows the velocity and mass for each 40 J simulation and experiment. Material properties used in the simulations are the same as for the 60 J Rosand impacts.

Table 7.2: Velocity and mass for the three 40 J simulations and tests.

<table>
<thead>
<tr>
<th>Sim. &amp; Exp. name</th>
<th>Impact Mass</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>40-1</td>
<td>5.151 kg</td>
<td>3.941 m/s</td>
</tr>
<tr>
<td>40-2</td>
<td>7.151 kg</td>
<td>3.345 m/s</td>
</tr>
<tr>
<td>40-3</td>
<td>10.151 kg</td>
<td>2.807 m/s</td>
</tr>
</tbody>
</table>

7.3.1 Energy and Deflection

In Fig. 7.6 the energy plots for the different masses at 40 J can be studied and it is seen that the simulations are now in very good agreement with the experiments. For the impact 40-1, with 5.151 kg impact mass, the impact duration is slightly too long for the simulation, but the time used to reach minimum kinetic energy of the dart is less than 0.5 ms longer than for the test, so this difference is insignificant. In the two other simulations the impact duration is identical to the experiments. For all the simulations the energy of the dart is insignificant lower until it reach minimum energy, so the plate tends to be too stiff or the damage criteria are too high. From minimum kinetic energy to the end of the impact the simulations of 40-2 and 40-3 follow the experimental impact, while for 40-1 the energy in the simulation increase to slow. The energy returned to the dart is almost equal the tests for all three simulations.

The displacement comparison can be studied in Fig. 7.7. For the 5.151 kg impact shown in Fig. 7.7 (a) the simulation and the experiment are very close to each other, except the time difference discussed for Fig. 7.6. The deflection found by the analytical solution is slightly larger than for the simulation and experiment. At the 7.151 kg the differences between the plots are insignificant. The experiment and the analytical solution are nearly identical, while the simulated deflection is smaller. Maximum deflection is reached at the same time for all the three plots. The last 40 J impact was done with 10.151 kg impact mass. The results here are pretty much the same as for the 7.151 kg impact. Largest deflection is found for the analytical solution and the simulation has the smallest deflection. All three solutions have same impact duration.
Comparison of Analytical Solutions, Simulations and Experiments

Figure 7.6: Energy comparison of all the 40 J simulations and the Rosand impact experiments.
7.3 40 J Rosand Impacts

Figure 7.7: Displacement comparison between all 40 J simulations, analytical solution and tests for Rosand impacts.
For this energy level there are no big differences, neither for the energy nor the deflection comparisons. Because of lower energy of the projectile, not enough damage are done to make the material behave plastic as seen for the 60 J impact.

### 7.3.2 Damage area

The damage in the 40 J impact experiments was smaller than for the 60 J impacts. Figure 7.8 shows the damage pattern for the 7.151 kg impact, which is 6.2 cm long and 3.8 cm wide at the bottom layer.

![Damage from 40 J impact with 7.151 kg impact mass.](image)

Damage from the AUTODYN simulation can be studied in Fig. 7.9 and compared to the damage in the experiments the damage is too small. The failure plot shows a lot of in-plane failures (Failed 12) and there are also some tension failure, but in the experiment only a few matrix cracks were visible near the centre and there were no fibre breakages. As for the 60 J impacts, the simulation do not provides a correct damage area, but it a good indication.

Also for this energy level static stress analysis were done in ANSYS by applying maximum deflection found from impact test. Inter-laminar shear stresses higher than failure criteria used in AUTODYN were plotted. The elements exceeded the criteria were located about the same as failed in AUTODYN. For the in-plane shear stress the ANSYS simulation also gave about the same
failure as AUTODYN. The area failed was now less than for the 60 J impact. So again a static stress analyses predicted the damage pattern.

![Material Status](image)

(a) (b)

Figure 7.9: Damage pattern for 40 J simulation: (a) perspective view and (b) bottom view.

## 7.4 20 J Rosand Impacts

The last Rosand impacts where conducted with 20 J impact energy. Mass and velocity are listed in Table 7.3 and the material properties used in the

<table>
<thead>
<tr>
<th>Sim. &amp; Exp. name</th>
<th>Impact Mass</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>20-1</td>
<td>5.151 kg</td>
<td>2.787 m/s</td>
</tr>
<tr>
<td>20-2</td>
<td>7.151 kg</td>
<td>2.365 m/s</td>
</tr>
<tr>
<td>20-3</td>
<td>10.151 kg</td>
<td>1.985 m/s</td>
</tr>
</tbody>
</table>

simulations are equal those used in the 60 J Rosand impact simulations.

### 7.4.1 Energy and Deflection

Figure 7.10 shows energy comparison of the 20 J impacts. Unlike the 40 J and the 60 J, the simulations do now have shorter impact duration than the tests.
Figure 7.10: Energy comparison between all 20 J simulations and tests for Rosand impacts
The time differences are about 1 ms. For these tests less damage occurred than for the two other energy levels. In the 60 J simulations they were set high to avoid that too much damage occurred in the material. For these 20 J simulations the failure stresses and strains used are too high. Few elements are damaged and the material will not become weaker. The energy returned to the dart after the impact is also too high, which shows that the material is too elastic.

For the deflection plots in Fig. 7.11 the material modelling problem is shown again. The deflections from the simulations are smaller than for the tests and the analytical solutions. Maximum difference is found for the 7.151 kg impact and is about 2 mm. One new simulation was done for the 5.151 kg impact. For that simulation called E20-1c, tensile failure stresses were set to 482 MPa and maximum out-of-plane shear stresses were set to 40 MPa, which are the values from Table 5.3. The deflection from this simulation is plotted in Fig. 7.11 (a). The deflection is now very similar to the experiment. These material properties were also used for a simulation of the 10.151 kg 60 J impact in Section 5.5.1, but there too much damage occurred and the impact duration become too long compared to the test.

### 7.4.2 Damage Area

Figure 7.12 (a) shows the damage done at 20 J impact with 5.151 kg impact mass. The visible damage area is 4.3 mm long and 2.9 mm wide. The failure plot from AUTODYN is shown in Fig. 7.12 (b). Again the failure in the AUTODYN simulation is in good agreement with the damage in the experiments, nevertheless the simulated damage area is slightly too small.

### 7.5 60 J Drop Weight Impact

For this impact the material properties used were the same as for the 60 J Rosand impact. That is

- \( \sigma_{11U} = \sigma_{22U} = 550 \text{ MPa} \)
- \( \tau_{13U} = \tau_{23U} = 50 \text{ MPa} \)

and the other properties unchanged from Table 5.3.
Figure 7.11: Displacement comparison between all 20 J simulations, analytical solution and tests for Rosand impacts
Figure 7.12: (a) damage for 20 J drop weigh impact and (b) failure plot from AUTODYN simulation.

Figure 7.13: Energy comparison for tests and simulation of 60 J drop weight impact.
7.5.1 Energy and Deflection

In Fig. 7.13 the simulated kinetic energy of the dart is compared to the experiment. At the beginning the simulation has perfect match with the experiment, however after 3 ms the slop of the simulated energy decrease while the test do not, as a result the simulation reach minimum kinetic energy 1 ms later than the experiment.

![Graph showing deflection comparison](image)

Figure 7.14: Deflection comparison for experiment, simulation and analytical solution of 60 J drop weight impact.

For the deflection plot in Fig. 7.14 the simulation and the experiments are very close to each other. Maximum deflection is almost identical for those two plots. The analytical solution has slightly larger deflection.

7.5.2 Strain Comparison

The plate was instrumented with strain gauges before the impact. In the static ANSYS simulations strain analysis was performed for maximum deflection known from the impact experiment. The strain found in ANSYS will here be compared to maximum strain from the strain gauges.

Figure 7.15 shows the location of the strain gauges in mm from the centre O and their maximum strain measured compared to the static strain analysis done in ANSYS. Gauge 1, 2 and 6 have maximum strain very close to the strain found in ANSYS, while gauge 5 have a little lower strain. The strain analysis done in ANSYS is in good agreement with the maximum strain measured in the test.
7.6 152 J Drop Weight Impact

The 6.2 kg impactor was dropped from 2.5 m, so the impact energy was 152 J. For the simulation the material properties are equal those used in the 60 J drop weight impact simulation.

7.6.1 Energy and Deflection

For this simulation the difference between the simulation and the tests are more clearly. At 2 ms the test has a peak, same as shown for all other experiments. For the simulation this peak is indistinct. The slope of the plot from simulation is also different from the experiments, so time used to reach minimum kinetic energy is about 4.5 ms for the experiment and 5.9 ms for the simulation.

Large differences are also show in the for the displacement plot in Fig. 7.17. The three plots do have the same on-loading path, but after 3.5 ms differences appear. Deflection for the tests stops at -22.5 mm, while the simulation and the analytical solution have -25.3 and -27.9 mm, respectively.
Comparison of Analytical Solutions, Simulations and Experiments

Figure 7.16: Energy comparison for experiment and simulation of 152 J drop weight impact.

Figure 7.17: Deflection comparison for, simulation and analytical solution of 152 J drop weight impact.
7.6.2 Damage and Damage Pattern

Figure 7.18 shows the damage done on the plate for the 152 J drop weight test. The area in 15 cm long and 10 cm wide. Outside the delaminated area there is a large area with matrix cracks. The matrix cracks are too small to be visible on the picture.

![Damage done by 152 J impact](image_url)

Figure 7.18: Damage done by 152 J impact

The damage pattern from AUTODYN simulation is shown in Fig. 7.19. Compared to the size of the damage in Fig. 7.18, the size of this damage pattern seems correct. Stress distributions were found in ANSYS simulations for maximum deflection known from the experiment and stresses exceeding the AUTODYN stress failure criteria were plotted. These stress plots were shown in Section 6.4.2 and comparing them with the AUTODYN failure plot gave no sensible results. The areas with stresses exceeding the failure are insignificant compare to the AUTODYN failure plot.

7.6.3 Strain Comparison

As for the 60 J drop weight impact, the plate was instrumented with strain gauges and their maximum strain will here be compared to strain found in ANSYS. In ANSYS the deflection of the plate was set to be equal maximum deflection from the impact experiment. The strain comparisons are shown in
Figure 7.19: Damage pattern for 152 J impact simulation. Perspective view in (a) and bottom view in (b).

Figure 7.20: (a) Strain gauges located along the paths OA and OB. (b) Normal strain found in ANSYS along line OA and OB compared with maximum strain from strain gauges in 152 J drop weight impact experiment.
Fig. 7.20 and it is seen that the strains from the ANSYS simulation are similar to the measured strains.
Chapter 8

Summary and Conclusion

8.1 Summary

In this thesis the two main objectives have been to carry out impact experiments and to perform numerical simulations of composite laminate. The plates were $175 \times 175$ mm$^2$ or $300 \times 300$ mm$^2$.

For the small plates a Rosand Instrumented Falling Weight Impactor device was used to perform impacts. The user must add desired drop weight and changing indentator manually, but the rest of the machine can be controlled from a graphical user interface. Force is logged by a sensor above the indentator and the control program calculates acceleration, velocity, displacement and energy. Nine plates were impacted with different masses and energy levels. In the setup the plates were clamped on a aluminium frame.

To be able to conduct the experiments on the larger plates a new drop weight device was constructed. The impact device was made of a ventilated Perspex tube supported by a massive wood rig. A solid steel cylinder with a spherical tip was used as a indentator. An accelerometer were mounted into the top of the projectile and connected with wires to a fast logging equipment. Force, velocity and displacement as well as energy were calculated from the acceleration. The accelerometer and the logging equipment were calibrated in the Rosand device before used in these drop weight impacts. All the plates were instrumented with strain gauges and the gauges were connected to the same logging equipment as used for the accelerometer. Three plates were impacted, two with 60 J and one with 152 J. The mass of the projectile was 6.2 kg, so the drop heights were 1 m and 2.5 m for the two energy levels, respectively. For these impacts the plates were simply supported on a frame made of steel pipes.

All the impacts have been simulated numerical using AUTODYN, which is
a commercial program package used to simulate strongly time dependent problems. Since AUTODYN provide a very good processor coupling, interaction between frame, plate and dart was simulated. Failure models are implemented in the software and the material stress and strain failure model was used.

ANSYS was chosen to perform the static simulations. For both plate sizes the plate stiffness were found from linear force-deflection simulations. These plate stiffnesses were used in the analytical impact solutions and the values found in the simulations were compared to the static deflection experiments. Stress and strain distributions were also found for the two plates, but instead of load, maximum deflection from the impact experiments was applied. Stress distributions exceeding the stress failure criteria used in the AUTODYN failure model were plotted and compared to the failure plots from AUTODYN.

The results from the two impact setups compared with simulations and will be discussed separately.

### 8.1.1 The Rosand Instrumented Falling Weight Impacts

Material properties for the laminate were known from material tests done by SINTEF. Simulations of a 60 J impact showed that the failure criteria were to low, so ultimate tensile stress and inter-laminaer stress were increased to receive better results. For all 60 J impact simulations the impact duration was slightly to long compared to the experiments, but the time differences were only about 0.7 ms so the results were satisfactory. Maximum deflection were a little larger in the simulations than in the experiments. The analytical impact solutions had about the same maximum deflection as the simulations, but the impact duration was longer than the experiments.

For the 40 J energy level the differences were insignificant between the simulations and the experiments. There was slightly less energy of the projectile in the simulations than in the experiments, which made the deflection a little too small. The analytical deflections for all three impacts at this energy level were very close to the experiments.

In the 20 J simulations the failure criteria used for the two other energy levels were too high. The kinetic energy of the dart decreased to fast compared to the experiments and naturally the deflection became too small. If the criteria was lowered to their original value, known from the material properties tests, the simulations became similar to the experiments. The simulation errors were confirmed by the analytical solutions, which were almost identical to the experiments.

Failure plots for all energy levels were compared to the real damage done in the plates and generally the damage areas in the simulations were a bit too
small, however they gave a very good indication of the damage. Delamination is a complicated damage, which occur between layers with distinct fibre orientation. AUTODYN do not know anything about the layers in the plate and will therefore not be able to simulate this damage correctly.

The stress distributions in the ANSYS simulations were compared to the failure plot from AUTODYN simulations and it is seen that the areas of interlaminar shear failure and in-plane failure are about the same.

8.1.2 The Drop Weight Impacts

Two 60 J impact and one 152 J impact were done in the self constructed drop weight device. The data present from these impacts are from a second impact on the plates, because a wrong low-pass filter was used during the first impacts. The low-pass filter removes frequencies above a specified value, so when the filter value were set to low the measured data became very smooth. Some details from the impact were therefore gone, such as the initial loss of contact peak, and since the impacts should be compared to simulation, a second impact had to be done. From the setup tests it was noticed that the measured acceleration did not change significantly from the first to the second impact on a plate, so comparing the second impacts with the simulations would make no big difference.

The AUTODYN simulations of the impacts, showed the same time difference errors as seen for the Rosand impact simulations. For the 60 J simulation the impact duration was about 1 ms longer than the experiment, however both energy and displacement from the simulation are in good agreement with the test. Larger differences were found for the 152 J impact. The energy plot from the simulation is close to the experiment, but in the displacement plot there are quite huge differences. These differences are due to the damage done in the model, because for increasing impact energy more element will fail and then the strength of the model are reduced. To avoid this the failure criteria can be set higher, but the properties would then be abnormal high compare to similar GFRP composites.

In either of these impacts simulations AUTODYN managed to recreate the exact damage done in the tests, but the size of the damage zones are about correctly predicted, as seen for the Rosand impacts simulations.

In the static ANSYS simulation stress and strain distributions were found in the plates, when a maximum deflection from the experiments were applied. The strain distributions were compared to the maximum strain measured with gauges during the impact tests and found to be about the same, however comparison of the stress distribution and the failure plot from AUTODYN
gave no sensible results. Only a few elements had stresses exceeding the failure criteria, while in the AUTODYN simulations a lot of elements failed.

8.2 Conclusion

Results of the present study allows one to draw the following conclusions:

1. A drop weight impact device do not have to be a expensive fabricated machine. From the drop weight device built in this thesis astonishing good results were received. This device is very simple, but a lot of testing and careful modifications had to be done before the results were satisfactory and reliable. The Perspex tube, which was supported by a massive wood rig and got air channels machined out to remove air before the projectile during the fall. An accelerometer were mounted into the top of the projectile and it was connected with wires to a logging equipment. This great setup allows the user to perform impacts on targets with rare geometry, such as pipes, and the environment can be changed, for example targets in a fluid.

2. AUTODYN is simple to use compare to other simulations programs as eg LS-DYNA and provides good results of the impact simulations. The impact duration were not exact for all the simulations, but results were satisfactory for both the energy of the projectile and the deflection of the plate compared to the experiments. Exact results could have been derived for all simulations, but then the material properties had to be tuned different for each impact. Same material properties were used for all impact simulations to study effects of different impact energies and masses. The failure plots from the simulations shows slightly less damage than in the experiments, however the simulations indicates how large the damage will be and provides therefore useful information to the user.

3. Static simulation of the laminate plates can be done in ANSYS to receive information about deflection the plate and strains on the surfaces. The plate stiffness found in these simulations were in good agreement with the value found in experiments.

4. From the analytical spring-mass model impact duration and deflection of the plate were found and both were close to the experiments. This model can however not be used without knowledge of the plate stiffness, which can be found from a static simulation or a simple experiment.
8.3 Further Work

Impact behaviour of composites is an advance topic with a lot of interesting problems to study. The work done in this thesis are basic impact studies and the following points would be a natural continuation:

- Perform impact on laminate plates surrounded by water. Composites are common in high speed crafts and offshore subsea structures and impacts on composites in water are interesting. The drop weight device used in this thesis allows the user to place the target in a water tank. The tank should be large enough such that boundary effects can be neglected.

- Strength analyses of the structural part after impacts. If a structure is impacted and the strength is heavily reduced the part has to be exchanged. Strength reduction for known impacts can be found.

- Full scale AUTODYN simulation. Simulation of impact on a full scale model or a part of a structure should be done. This would require powerful computers.
References


143
REFERENCES


REFERENCES


145
Appendix A

Attaching Strain Gauges to Specimens

It is important to be very accurately when attaching the strain gauges to the specimen. If not they can fall of and the result are useless.

- The area where the gauges would be glued needed to be sanded extensively to create a smooth surface. When the surface in smooth enough, clean with spirit and let it dry.

- Mark where the strain gauges are supposed to be fixed.

- Attach the upper side of the strain gauges to a bit of adhesive tape. The strain gauges should be handled with a pincette. Attach one end of the adhesive tape to the plate, so the strain gauges are in right place. It is important to orientate the strain gauges parallel or perpendicular to the fibers in the laminate, if they not are supposed to be rotated in 45 degrees. If the strain gauges are distorted, the results will not be correct due to material axes.

- Add glue to the strain gauge and press, with a bit of teflon (teflon would not stick with the glue), on top of the strain gauges. Remove the adhesive tape before the glue has got stiff.

- Before solder wires to the strain gauges, use a scalpel to clean the lead contacts on strain gauges. Also check with a ohmmeter that the resistance is correct in the strain gauge.

- A hint when solder: Place a droplet of tin on each of the lead contacts on the strain gauge and solder the wires to the droplet. This will be a
stronger connection than solder the wires direct to the lead contacts. Now check that the resistance is correct through the wires.
Appendix B

Plots from Impact Experiments

B.1 Introduction

In Chapter 4 only plots from the 60 J impacts performed in the Rosand Impact device were presented. Here the corresponding plots for the 20 J and 40 J impacts can be studied.

B.2 Plots from 40 J Rosand Instrumented Impacts

Three plots are showed for the 40 J Rosand impacts.
B.2 Plots from 40 J Rosand Instrumented Impacts

Figure B.1: Force plot for the 40 J impacts. Impact mass is 5.151 kg, 7.151 kg and 10.151 kg for plate 5, 8 and 11, respectively.

Figure B.2: Displacement and energy plot for the 40 J impacts. Impact mass is 5.151 kg, 7.151 kg and 10.151 kg for plate 5, 8 and 11, respectively.
B.3 Plots from 20 J Rosand Instrumented Impacts

Figure B.3: Force plot for 20 J impacts. Impact mass is 5.151 kg, 7.151 kg and 10.151 kg for plate 4, 7 and 10, respectively.
Figure B.4: Displacement and energy plot for 20 J impacts. Impact mass is 5.151 kg, 7.151 kg and 10.151 kg for plate 4, 7 and 10, respectively.
Appendix C

MAPLE Worksheets, ANSYS Files and PYTHON Scripts

C.1 Introduction

Here some of the MAPLE worksheets, ANSYS input files and PYTHON script used can be studied.

C.2 MAPLE Worksheets

C.2.1 The Composite Package

A MAPLE package named Composite was implemented. This package is used to calculate the matrices $[\bar{Q}]$ and $[D]$ for a laminate. The laminate layup must be specified in an array.
C.2 MAPLE Worksheets

> Composite := module()
> export Q_hat, D_
> local Q_find;
> option package;

> Q_hat:=proc(Ex,Ey,nuxy,Gxy,theta)
> local a,Q_h;
> a:=Q(Ex,Ey,nuxy,Gxy);
> Q_h:=array(1..3,1..3);
> Q_h[1,1]:=a[1,1]*cos(theta)^4+a[2,2]*sin(theta)^4+2*(a[1,2]+2*a[3,3])*sin(theta)*cos(theta)^2;
> Q_h[1,2]:=(a[1,1]+a[2,2]-4*a[3,3])*sin(theta)*cos(theta)^2+a[1,2]*sin(theta)^4+cos(theta)^4;
> Q_h[1,3]:=(a[1,1]-a[1,2]-2*a[3,3])*cos(theta)*sin(theta)^3-(a[2,2]-a[1,2]-2*a[3,3])*cos(theta)^3*sin(theta);
> Q_h[2,2]:=(a[1,1]+a[2,2]-4*a[3,3])*cos(theta)*sin(theta)^3+(a[1,1]+a[2,2]-4*a[3,3])*cos(theta)^3*sin(theta)^2;
> Q_h[2,3]:=(a[1,1]-a[1,2]-2*a[3,3])*cos(theta)*sin(theta)^3+(a[2,2]-a[1,2]-2*a[3,3])*cos(theta)^3*sin(theta)^2;
> Q_h[3,3]:=(a[1,1]+a[2,2]-4*a[3,3])*cos(theta)*sin(theta)^3+(a[1,1]+a[2,2]-4*a[3,3])*cos(theta)^3*sin(theta)^2;
> Q_h[2,1]:=Q_h[1,2];
> Q_h[3,1]:=Q_h[1,3];
> Q_h[3,2]:=Q_h[2,3];
> evalm(Q_h);
> end proc;

> Q_find:=proc(El,Et,nult,Glt,i)
> local q;
> q:='q';
> q:=Q_hat(El,Et,nult,Glt,Data[i+1,1]);
> end proc;

> D_:=proc(El,Et,nult,Glt)
> local n;
> with(linalg,rowdim):
> n:=rowdim(data)-1;
> evalm(1/3*add(Q_find(El,Et,nult,Glt,i)*(data[i+1,2]^3-data[i,2]^3),i=1..n));
> end proc;

153
C.2.2 Deflection of a Layered Laminate

This worksheet calculates the deflection of an 8 layered laminate.

> restart;
> libname:="P:\maple\lib",libname:
> with(Composite);
> [D_, Q, Q_hat]

> a:=135;
> b:=a;
> k:=100;
> P:=15820;
> E[L]:=43.9E3;
> E[T]:=8.23E3;
> nu:=0.20;
> G[LT]:=6E3;

> data:=array([[0,-4*0.65],[0,-3*0.65],[Pi/2,-2*0.65],[0,-1*0.65],[Pi/2,0.65],[Pi/2,0.65],[0,2*0.65],[Pi/2,3*0.65],[0,4*0.65]]);
\[ \text{data} := \begin{bmatrix} 0 & -2.60 \\ 0 & -1.95 \\ \frac{1}{2} \pi & -1.30 \\ 0 & -0.65 \\ \frac{1}{2} \pi & 0 \\ \frac{1}{2} \pi & 0.65 \\ 0 & 1.30 \\ \frac{1}{2} \pi & 1.95 \\ 0 & 2.60 \end{bmatrix} \]

\[ \text{D} := \begin{bmatrix} 386678.9350 & 19432.45192 & 0. \\ 19432.45192 & 228760.7486 & 0. \\ 0. & 0. & 70304.0000 \end{bmatrix} \]

\[ q_{mn} := (m, n) \rightarrow 4 \frac{P \sin\left(\frac{1}{2} m \pi\right) \sin\left(\frac{1}{2} n \pi\right)}{ab} \]

\[ W_{mn} := (m, n) \rightarrow 9 \frac{q_{mn}(m, n) a^4}{\pi^4 (D_{1,1} m^4 + \frac{2(D_{1,2} + 2 D_{3,3}) m^2 n^2 a^2}{b^2} + \frac{D_{2,2} n^4 a^4}{b^4})} \]

\[ w := (x, y) \rightarrow \sum_{m=1}^{k} \left( \sum_{n=1}^{k} W_{mn}(m, n) \sin\left(\frac{m \pi x}{a}\right) \sin\left(\frac{n \pi y}{b}\right) \right) \]

\[ w(67.5, 67.5) \rightarrow 14.12323387 \]

155
C.2.3 Analytical Impact Model

The impact models implemented in MAPLE. This worksheet calculates the solutions for the 152 J drop weight impact.

\[ E := 152 \]
\[ M := 6.2 \]
\[ V := \sqrt{2E/M} \]

\[ E := 152 \]
\[ M := 6.2 \]
\[ V := 7.002303768 \]

\[ K := 3900/(10/1000.) \quad \text{#K\_big} \]
\[ K := 390000.000 \]

\[ \omega := \sqrt{K/M} \]
\[ \omega := 250.8051551 \]

\[ P_{\text{max}} := (V, K, M) \rightarrow V \sqrt{K/M} \]

\[ P_{\text{max}} := (V, K, M) \rightarrow V \sqrt{K/M} \]

\[ V := 7.002303768 \]

\[ K_{\text{big}} := 390000.000 \]

\[ \omega := 250.8051551 \]

\[ P_{\text{max}} := (V, K, M) \]
\[ 10888.52607 \]

\[ t_{c} := (M, K) \rightarrow \text{evalf}(\pi \sqrt{M/K}) \]
\[ t_{c} := (M, K) \rightarrow \text{evalf}(\pi \sqrt{M/K}) \]

\[ t_{m} := t_{c}(M, K) \quad \text{#second} \]
\[ t_{m} := .01252602904 \]

\[ x := t \rightarrow V/\omega \sin(\omega t) \times 1000 \]
\[ x := t \rightarrow 1000 \frac{V \sin(\omega t)}{\omega} \]

\[ \text{plot}(x, 0..t_{c}(M, K), \text{labels}=["Time (s)", "Displacement (mm)"]) \]
C.3 ANSYS Input Files

Here some ANSYS input files are listed.

C.3.1 Strain Analyses for Clamped Plate

!############
!Define variables
!############
deflection=-12
width=87.5
num_elem=36
num_elem_height=5
nm_substep=2
min_nm_substep=1
substep_size=2
filename=deflection

!############
!Output file
!################################################
/OUTPUT, filename, TXT
/COM Pressure Displacement Force_z
/OUTPUT

/PREP7

!################################################
!Element type
!################################################
ET, 1, SOLID45

!################################################
!Material properties
!################################################
MP, EX, 1, 3.15E4
MP, EY, 1, 3.15E4
MP, EZ, 1, 8.23E3
MP, PRXY, 1, 0.2
MP, PRYZ, 1, 0.2
MP, PRXZ, 1, 0.2
MP, GXY, 1, 6E3
MP, GYZ, 1, 5.14E3
MP, GXZ, 1, 5.14E3

!################################################
!Geometry
!################################################
K, 1
K, 2, width
K, 3, width, width
K, 4, 0, width

!################################################
!Generate keypoints and make volume
!################################################
KGEN, 2, ALL,, 0, 0, 5.2
V, 1, 2, 3, 4, 5, 6, 7, 8

!################################################
!Meshing
!################################################
LSEL, S, LENGHT,, width
LESIZE, ALL,, num_elem
LSEL, S, LENGHT,, 5.2
C.3 ANSYS Input Files

LESIZE, ALL,, num Elem Height
MAT, 1
VEH, 1

FINISH

!##############
!Solution with non-linear geometry
!##############
/SOLU
NLGEOM, on

!##############
!Boundary conditions
!Clamped by frame
!##############
NSEL, S, LOC, X, 0, 22
NSEL, R, LOC, Y, 20, width
NSEL, R, LOC, Z, 0
NSEL, S, LOC, X, 0, 22
NSEL, R, LOC, Y, 20, width
NSEL, R, LOC, Z, 5.2
NSEL, S, LOC, Y, 0, 22
NSEL, R, LOC, X, 20, width
NSEL, R, LOC, Z, 0
NSEL, S, LOC, Y, 0, 22
NSEL, R, LOC, X, 20, width
NSEL, R, LOC, Z, 5.2
D, ALL, UZ, 0

NSEL, S, LOC, X, width
DSYM, SYMM, X
NSEL, S, LOC, Y, width
DSYM, SYMM, Y

!##############
!Applying pressure at corner elements
!##############
NSEL, S, LOC, X, width
NSEL, R, LOC, Y, width
NSEL, R, LOC, Z, 5.2
ESLN, ALL
NSLE, S
NSEL, R, LOC, Z, 5.2
ESLN, ALL
NSLE, ALL
D, ALL, UZ, deflection
!###################################
!Setting number of iterations
!###################################
ALLSEL
AUTOTS, ON
PRED, ON
LNSRCH, ON
NSUBST, substep_size, nm_substep, min_nm_substep
KBC, 0
NCNV, 2
NEQIT, 50
OUTRES, ALL, ALL

SOLVE
FINISH

!###################################
!Result.
!Write strain for 3 paths to file
!###################################
/POST1
SET,,,,,,LAST

ALLSEL
PATH, diag_bot, 2,,num_elem
PPATH, 1,,width,width,0
PPATH, 2,,0,0,0
PDEF, EPTO, EPTO, X, NOAVG
PDEF, EPTOY, EPTO, Y, NOAVG
PDEF, EPTOX, EPTO, XY, NOAVG
/OUTPUT, diag_bot.txt
PRPATH, EPTO, EPTOY, EPTOXY
/OUT

PATH, diag_top, 2,,num_elem
PPATH, 1,,width,width,5.2
PPATH, 2,,0,0,5.2
PDEF, EPTO, EPTO, X, NOAVG
PDEF, EPTOY, EPTO, Y, NOAVG
PDEF, EPTOX, EPTO, XY, NOAVG
/OUTPUT, diag_top.txt
PRPATH, EPTO, EPTOY, EPTOXY
/OUT

PATH, 0A, 2,,num_elem
PPATH, 1,,width,width,0
PPATH, 2,,width,0,0
C.3 ANSYS Input Files

PDEF,EPTOX,EPTOX,X,NOAVG
PDEF,EPTOY,EPTOY,Y,NOAVG
/OUTPUT,OA.txt
PRPATH,EPTOX,EPTOY
/OUT

C.3.2 Nonlinear Solution for Applied Deflection

!#################
!Define variables
!#################
pressure=150
width=150
num_elem=36
num_elem_height=5
mm_substep=100
min_mm_substep=20
substep_size=50
filename=large_plate

!#################
!Output file
!#################
/OUTPUT,filename,TXT
/COM Pressure Displacement Force_z
/OUTPUT

/PREP7

!#################
!Element
!#################
ET,1,SOLID45

!#################
!Material properties
!#################
MP,EX,1,3.15E4
MP,EF,1,3.15E4
MP,EZ,1,8.23E3
MP,PRXY,1,0.2
MP,PRYZ,1,0.2
MP,PRXZ,1,0.2
MP,GXY,1,6E3
MP,GYZ,1,5.14E3
MP,GXZ,1,5.14E3

!#################
!Geometry
!!!!!!!!!!!!
K,1
K,2,width
K,3,width,width
K,4,0,width

!!!!!!!!!!!!
!Generate keypoints and make volume
!!!!!!!!!!!!
KGEN,2,ALL,,0,0,5.2
V,1,2,3,4,5,6,7,8

!!!!!!!!!!!!
!Meshing
!!!!!!!!!!!!
LSEL,S,LENGHT,,width
LESIZE,ALL,,,num_elem
LSEL,S,LENGHT,,,5.2
LESIZE,ALL,,,num_elem_height
MAT,1
VMESH,1

FINISH

!!!!!!!!!!!!
!Solution with non-linear geometry
!!!!!!!!!!!!
/SOLU
NLGEOM,on

!!!!!!!!!!!!
!Boundary conditions
!Simply supported on frame
!!!!!!!!!!!!
NSEL,S,LOC,X,20,22
NSEL,R,LOC,Y,20,width
NSEL,R,LOC,Z,0
D,ALL,UZ,0
NSEL,S,LOC,Y,20,22
NSEL,R,LOC,X,20,width
NSEL,R,LOC,Z,0
D,ALL,UZ,0

NSEL,S,LOC,X,width
DSYM,SYMM,X
NSEL,S,LOC,Y,width
DSYM,SYMM,Y
!################
!Applying pressure at corner elements
!################
NSEL,S,LOC,X,width
NSEL,R,LOC,Y,width
NSEL,R,LOC,Z,5.2
ESLN,ALL
nsle,s
nse,l,loc,z,5.2
esl,n,all
SFE,ALL,6,PRES,,pressure

!################
!Setting number of iterations
!################
ALLSEL
AUTOTS,ON
PRED,ON
LNSRCH,ON
NSUBST,substep_size,nn_substep,min_nn_substep
KBC,0
NCNV,2
NEQIT,50
OUTRES,ALL,ALL

SOLVE
FINISH

!################
!Result. Write force and displacement
!for each substep to file
!################
/POST1
*GET,sub_step,ACTIVE,0,SOLU,NCMSS
*DO,i,1,sub_step
SET,,,,,,,i
NSORT,U,Z,,1
*GET,DMAx,SORT,,MAX
DAMAX=ABS(DMAX)
FSUM
*GET, FORCE, FSUM, 0, ITEM, FZ
FORCE=4*FORCE
FORCE=ABS(FORCE)
/OUTPUT,filename,TXT,,APPEND
*WRITE,DMAx,FORCE,
%f %f
/OUTPUT
C.4 PYTHON Scripts

Two of the PYTHON scripts written during the work with this thesis.

C.4.1 The make plot Script

A script was written to generate plots using GNUPLOT. The script reads a data file, writes a GNUPLOT input file and runs GNUPLOT, which generates a file containing the plot.

```python
#!/usr/bin/python
import sys,string,os,getopt,math

xlabel=0
ylabel=0
makefilename=0
legends_set=0

if len(sys.argv)>1:
    options,arg=getopt.getopt(sys.argv[1:],'m:f:x:y:1:',['"legends="","file=",
"xlabel="","ylabel="","makefilename="'])
columns=arg
else:
    print 'Usage: %s filename [-x "xlabel"] [-y "ylabel"] [-l "legend1,legend2,.."] [-m "outfile"] 1:2,3..' %sys.argv[0]
sys.exit(1)

for option,value in options:
    if option in ('-f','--file'):
        infilename=value
        infile=open(infilename,'r')
        labels=string.split(infile.readline())
        labels=map(string.capitalize,labels)
        number_col=len(labels)
    elif option in ('-x','--xlabel'):
        xlabel=value
    elif option in ('-y','--ylabel'):
        ylabel=value
    elif option in ('-l','--legends'):
        legends=string.split(value,',')
        legends_set=1
    elif option in ('-m','--makefilename'):
        filename=value
        makefilename=1
```

164
xaxe=string.split(cols[0],':')[:1]
yaxe=string.split(string.split(cols[0],':')[1],',')

if not xlabel:
xlabel=labels[int(xaxe)-1]
if not ylabel:
ylabel=labels[int(yaxe[0])-1]

if makefilename:
gnufilename=filename+".gnuplot"
figfilename=filename+".tex"
else:
gnufilename=infilename+".gnuplot"
figfilename=infilename+".tex"

start_x=1e10
start_y=1e10
stop_x=-1e10
stop_y=-1e10

for line in infile:
    stop_val=map(float,string.split(line))
    if stop_val[int(xaxe)-1] < start_x:
        start_x=stop_val[int(xaxe)-1]
    if stop_val[int(xaxe)-1] > stop_x:
        stop_x=stop_val[int(xaxe)-1]
    for item in yaxe:
        if stop_val[int(item)-1] < start_y:
            start_y=stop_val[int(item)-1]
        if stop_val[int(item)-1] > stop_y:
            stop_y=stop_val[int(item)-1]

gnufile=open(gnufilename,"w")
gnufile.write("####
set xtics nomirror
set ytics nomirror
set border 3
set key below
set data style lines
set terminal pslatex color
set output "/%(figfilename)s"
set xlabel "/%(xlabel)s"
set ylabel "/%(ylabel)s"
"#### vars()
if len(yaxe)>1:
s="plot [% (start_x)g; % (stop_x)g] [% (start_y)g; % (stop_y)g]" %vars()
1=0
for graf in yaxe:
columns=str(xaxe)+'</'+str(graf)
title=str(labels[int(graf)])
if legends_set:
title=str(legends[i])
i+=1
s+=""
'(%filename)s' using % (columns)s title '(%title)s' lw 2,""
%(vars())
gnufile.write(s[::1])
else:
gnufile.write(""";plot [% (start_x)g; % (stop_x)g] [% (start_y)g; % (stop_y)g]
'(%filename)s' using % (columns)snotitle w 1 lw 2
""";%(vars()));
gnufile.close();
cmd='gnuplot %s' %gnufilename
os.system(cmd)
print "Saved plot in file: %s" %(figfilename)
print "and gnuplot file in %s" %(gnufilename)
infile.close();

C.4.2 The split columns Script

In the ANSYS output files space is missing between the columns if a column contains a
negative number. This script use regular expression to find all numbers in a line and write
them to a new file readable for the make plot script.

#!/usr/bin/python
import sys,string,re,shutil

try:
    filename=sys.argv[1]
except:
    print 'Usage %s filename' %sys.argv[0]
sys.exit(1)

shutil.copy2(filename, filename+'.bak')
infile=open(filename+'.bak','r')
outfile=open(filename,'w')

pattern='-?[\d. \d+[Ee][+-]\d+|\d+\.\d*|\d\.\d+'].'
for line in infile:
    find_start=string.split(line)
    if len(find_start)>0 and find_start[0]=='S':

166
```python
outfile.write(line[:-1])
match=re.findall(pattern,line)
match.append('
')
output=string.join(match)
outfile.write(output)
line=infile.readline()
outfile.close()
infile.close()
```