Copyright Statement

The digital copy of this thesis is protected by the Copyright Act 1994 (New Zealand).

This thesis may be consulted by you, provided you comply with the provisions of the Act and the following conditions of use:

- Any use you make of these documents or images must be for research or private study purposes only, and you may not make them available to any other person.
- Authors control the copyright of their thesis. You will recognise the author's right to be identified as the author of this thesis, and due acknowledgement will be made to the author where appropriate.
- You will obtain the author's permission before publishing any material from their thesis.

To request permissions please use the Feedback form on our webpage. http://researchspace.auckland.ac.nz/feedback

General copyright and disclaimer

In addition to the above conditions, authors give their consent for the digital copy of their work to be used subject to the conditions specified on the Library Thesis Consent Form.
NUMERICAL MODELLING OF FIBRE-REINFORCED THERMOPLASTIC SHEET FORMING

by

G. Richard Christie

A thesis submitted in partial fulfilment of the requirements for the degree of Doctor of Philosophy in Engineering

Department of Mechanical Engineering,
University of Auckland, New Zealand

September 1997
Abstract

Continuous Fibre Reinforced Thermoplastics (CFRTPs) combine high strength, stiffness, impact and chemical resistance with possibilities for efficient part production by various thermoforming processes. Sheet forming of molten CFRTP laminates has generated much interest, but problems of buckling, wrinkling and predicting fibre distribution have meant a deeper understanding of these processes is needed.

The first part of this study looks at the problem of gross buckling in homogeneous “trellis” flows of bidirectional laminates. Modelling the molten composite as a Newtonian fluid reinforced by inextensible fibres, linear stability analysis is used to determine the growth rate of small out-of-plane imperfections. Buckling is predicted when fibre tensions are negative, indicating that laminates must be kept in tension during forming to reduce such defects.

A new approach to Grid Strain Analysis is presented, which uses surface fitting to determine the deformations occurring in sheet forming. The new method improves analysis of smooth, inhomogeneous deformations, and allows greater flexibility in viewing the results. The technique has been used to visualise deformations in a blister fairing made from cross-ply PLYTRON laminates. Arrow diagrams produced from the part demonstrate the tendency of bidirectional composites to deform by trellis flow, while transverse flow results from the action of the diaphragms used to form the part.

The significance of inter-ply slip in CFRTP sheet forming provided the impetus to develop a finite element model for molten laminates, which treats each ply as a separate continuum. Contact between plies is modelled, with slip given a viscous response. Ply deformations are governed by a highly anisotropic elastic law, to handle the stiff fibres and as a first step towards a viscoelastic model of major intra-ply deformation modes.

The finite element model parameters were adjusted to fit the part shape and load response of unidirectional PLYTRON laminates in bending. However, a perfect fit is unobtainable due to local transverse flow occurring at the bend in the real laminate. Nevertheless, the bending of the remainder of the ply is well described by the elastic model, using a fibre direction stiffness 25000 times that in the transverse direction. With the present model, a somewhat less anisotropic set of parameters gives the best overall fit and has been applied in several thermoforming simulations.

As observed in experiments, matched-die bending simulations display ply buckling at high forming speeds. Hemispherical dome forming simulations exhibit out-of-plane buckling and
near-inextensible fibre behaviour, with trellis-like deformation predominant in cross-ply laminates. In simulated double-diaphragm forming of bends and hemispherical domes, tension superimposed on the laminate from the stretching diaphragms is shown to eliminate buckling. However, high forming pressures and excessive transverse flow are a problem with current, stiff diaphragms.

Final discussions look at improving contact modelling, reducing model sizes by adopting thin shell assumptions, and improving the ply model.
Completion of this work would not have been possible without the help of many people, and I take this chance to acknowledge their input.

Firstly, I would like to thank my joint supervisors, Associate Professor Debes Bhattacharyya and Professor Ian Collins for the knowledge and experience they have shared with me during this endeavour, and for the many times when their advice and encouragement helped me overcome obstacles to its completion. I wish to acknowledge equally the assistance of Associate Professor Peter Hunter both in the area of finite elements, and in proposing and helping with the application of surface fitting to Grid Strain Analysis.

I also acknowledge the technical assistance of numerous staff members and fellow students in the Department of Mechanical Engineering. I especially thank Simon Mander, Todd Martin and Russell Dykes for many helpful discussions and sharing of ideas in the area of thermoplastic composites. Thanks also to Ulf Hampel for digitising the grid strain components.

I owe many thanks to the German Academic Exchange Service (DAAD) for funding my year with the Institut für Verbundwerkstoffe in Kaiserslautern, Germany, where I had the freedom to begin the finite element component of this work. The friendly support of the IVW staff, in particular Professor Klaus Friedrich, has been greatly appreciated.

For their generous contributions towards the funding of this project I wish to thank the New Zealand Vice-Chancellors’ Committee, and Fisher and Paykel Ltd.

Finally, thankyou to my parents for their support throughout this lengthy study.
Contents

Abstract .................................................................................................................................. ii
Acknowledgments ................................................................................................................. iv
Contents ................................................................................................................................. v
List of Figures ....................................................................................................................... ix
List of Symbols .................................................................................................................... xii

Chapter 1: Introduction ................................................................................................. 1
  1.1 Continuous Fibre Reinforced Thermoplastics ............................................................... 1
    1.1.1 GF/PP PLYTRON® ...................................................................................................... 2
  1.2 Thermoforming of CFRTP Laminates ........................................................................... 4
    1.2.1 Ideal Flow Processes ................................................................................................. 4
    1.2.2 CFRTP Sheet Forming Processes ............................................................................... 6
  1.3 Thesis Outline ................................................................................................................ 7

Chapter 2: Buckling Stability in Homogeneous Flows of Bidirectional Composites .......... 10
  2.1 Introduction .................................................................................................................... 10
  2.2 Governing Equations ..................................................................................................... 12
    2.2.1 Kinematic Constraints ............................................................................................ 12
    2.2.2 Constitutive Equation ........................................................................................... 13
  2.3 Linear Stability Analysis ............................................................................................... 14
    2.3.1 Background Solution ............................................................................................. 14
    2.3.2 Perturbed Flow ...................................................................................................... 15
    2.3.3 Linearised Governing Equations .......................................................................... 17
    2.3.4 Determination of Stability Criteria ....................................................................... 20
  2.4 Discussion ...................................................................................................................... 22
    2.4.1 Influence of Wavelength ....................................................................................... 22
    2.4.2 Practical Consequences ......................................................................................... 25
Chapter 3: Strain Measurement in Sheet Forming

3.1 Grid Strain Analysis
3.1.1 Introduction
3.1.2 Current Techniques
3.1.3 Applications

3.2 Strain Analysis by Surface Fitting with the FEM
3.2.1 Elements for Representing Smooth Surfaces
3.2.2 Input and Preprocessing
3.2.3 Solution
3.2.4 Postprocessing

3.3 GSA Example: Composite Blister Fairing
3.3.1 \([0,90]_s\) Laminate
3.3.2 \([\pm45]_s\) Laminate

3.4 Further Uses

Chapter 4: A Finite Element Model of a Molten Laminate

4.1 Numerical Modelling of Thermoforming
4.1.1 The Ply as a Continuous Material
4.1.2 Literature Survey
4.1.3 SimForm Model and Program

4.2 The Finite Element Method - Non-Linear Overview
4.2.1 The Principle of Virtual Work
4.2.2 Discretization into Finite Elements
4.2.3 Solution of Non-Linear Finite Element Equations

4.3 Constitutive Equation for Elastic, Anisotropic Plies
4.3.1 Material Coordinates and Strains
4.3.2 Transverse Isotropy
4.3.3 Incompressible and Inextensible Formulation

4.4 Modelling Contact
4.4.1 Normal Contact Constraints
4.4.2 Friction Models
4.4.3 Managing Contacts

4.5 Finite Element Algorithm
References ........................................................................................................134

Appendix A: GSA User Guide ...........................................................................139

A.1 Overview ................................................................. 139
A.2 Data Input ............................................................ 140
A.3 Mesh Creation .......................................................... 142
A.4 Solution ................................................................. 145
A.5 Post-Processing .......................................................... 146
  A.5.1 Graphical Structures ............................................. 146
  A.5.2 Display Settings .................................................... 148
  A.5.3 Contour Plotting .................................................... 149
  A.5.4 Viewing Transformations ....................................... 149
  A.5.5 Plotting to File ....................................................... 150
  A.5.6 Outputting Strains ................................................ 151

Appendix B: SimForm Input File Format .........................................................152

Appendix C: SimForm Program Verification ...................................................159
# List of Figures

<table>
<thead>
<tr>
<th>Page</th>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>1.1</td>
<td>Micrograph cross sections of two PLYTRON GF/PP laminates.</td>
</tr>
<tr>
<td>5</td>
<td>1.2</td>
<td>Deformation mechanisms of molten continuous fibre-reinforced thermoplastic prepregs and their laminates.</td>
</tr>
<tr>
<td>6</td>
<td>1.3</td>
<td>Non-isothermal diaphragm forming.</td>
</tr>
<tr>
<td>7</td>
<td>1.4</td>
<td>Non-isothermal matched-die thermoforming.</td>
</tr>
<tr>
<td>10</td>
<td>2.1</td>
<td>Trellis flow of a bidirectional sheet showing coordinate system.</td>
</tr>
<tr>
<td>11</td>
<td>2.2</td>
<td>Instabilities occurring in forming flows of bidirectional composites.</td>
</tr>
<tr>
<td>20</td>
<td>2.3</td>
<td>Local coordinate system on perturbed sheet surface.</td>
</tr>
<tr>
<td>23</td>
<td>2.4</td>
<td>Stability plot for thickening trellis flows.</td>
</tr>
<tr>
<td>24</td>
<td>2.5</td>
<td>Stability plot for thinning trellis flows.</td>
</tr>
<tr>
<td>28</td>
<td>3.1</td>
<td>Deformation of a triangle described by three adjacent grid points.</td>
</tr>
<tr>
<td>28</td>
<td>3.2</td>
<td>Superimposed triangular grid elements from before and after deformation.</td>
</tr>
<tr>
<td>30</td>
<td>3.3</td>
<td>Principal strains as depicted in Arrow Diagrams.</td>
</tr>
<tr>
<td>32</td>
<td>3.4</td>
<td>Bicubic Hermite element showing the four nodal vector quantities.</td>
</tr>
<tr>
<td>33</td>
<td>3.5</td>
<td>One dimensional cubic Hermite basis functions.</td>
</tr>
<tr>
<td>35</td>
<td>3.6</td>
<td>Degenerate 3-noded bicubic Hermite element.</td>
</tr>
<tr>
<td>37</td>
<td>3.7</td>
<td>Finding the surface coordinate, $\xi_i$, of a data point in an element on the undeformed mesh.</td>
</tr>
<tr>
<td>42</td>
<td>3.8</td>
<td>Deformed over undeformed grid points for $[0,90]$, blister fairing.</td>
</tr>
<tr>
<td>43</td>
<td>3.9</td>
<td>Undeformed grid points and mesh for the $[0,90]$, blister fairing.</td>
</tr>
<tr>
<td>44</td>
<td>3.10</td>
<td>Elements on the fitted deformed mesh for the $[0,90]$, blister fairing.</td>
</tr>
<tr>
<td>45</td>
<td>3.11</td>
<td>Arrow diagram of principal Lagrangian strains for the $[0,90]$, blister fairing, plotted on the undeformed mesh.</td>
</tr>
<tr>
<td>46</td>
<td>3.12</td>
<td>Arrow diagram of principal Lagrangian strains on deformed surface for the $[0,90]$, fairing.</td>
</tr>
<tr>
<td>47</td>
<td>3.13</td>
<td>Contours of percentage thickness change for the $[0,90]$, fairing.</td>
</tr>
<tr>
<td>48</td>
<td>3.14</td>
<td>Principal strain plot for $[0,90]$, fairing example.</td>
</tr>
</tbody>
</table>
3.15 Arrow diagram of principal Lagrangian strains on the deformed surface of the [±45], blister fairing.

4.1 Micrograph of end of 8 x 0.5 mm PLYTRON bend specimen.

4.2 Continuous approximation to in-plane shear.

4.3 8-noded brick-type element.

4.4 Deformation of a fibre-reinforced sheet showing material coordinates.

4.5 Contact surface coordinate system.

4.6 6 x 6 contact points on bicubic element surface.

5.1 Schematic of Scherer’s ply-pull-out experiments.

5.2 Influence of velocity on inter-ply slip resistance, with linear best fit.

5.3 Load variation during simulated ply-pull-out at 0.25 mm/s.

5.4 Schematic of free-bending experiments of Martin et al.

5.5 Free bending loads at 169.5 °C, 500 mm/min, from Martin et al.

5.6 2-D simulated free bending for friction relation $\tau = 2 \times 10^7 v$.

5.7 Load curves for 2-D simulated free-bending under three friction conditions.

5.8 3-D simulated free bending of a [0] PLYTRON laminate.

5.9 Unstable behaviour in inextensible, incompressible free-bending model.

6.1 The SimView post-processor.

6.2 Set-up of matched-die, 90° bend forming simulations.

6.3 Matched-die, 90° bend forming example. Slow forming at 7 mm/s.

6.4 Matched-die, 90° bend forming example. Fast forming at 70 mm/s.

6.5 Fibre stresses early in 90° bend matched-die simulations.

6.6 Set-up of double diaphragm, 90° bend forming simulations.

6.7 2-D, double-diaphragm 90° bend forming of a unidirectional laminate.

6.8 Double-diaphragm forming without friction showing buckling at 0.039 s.

6.9 Matched-die forming a dome from a square, isotropic sheet.

6.10 Simulated matched-die dome forming of a [0₄] laminate.

6.11 Final fibre orientation in top ply of [0₄] matched-die dome.

Page Figure

6.12 Simulated matched-die dome forming of a [0,90]s laminate.

6.13 Final fibre orientations in [0,90]s matched-die dome.
6.14 Double-diaphragm forming a dome from a single, thick ply.

6.15 Fibre orientation in double-diaphragm formed dome at 1.506 s.

7.1 Bending of two isotropic sheets showing normal contact stresses.

8.1 Equipment for forming a cross-ply dome, using differential heating to encourage tension and limit transverse flow in surface plies.

A.1 Screen shot of GSA showing undeformed mesh and grid points for the blister fairing example.

A.2 Dialogue Box for creating .GSA files for data input.

A.3 Options|Display dialogue box for selecting Graphical Structures for display.

A.4 Options|Display Settings dialogue box.
List of Symbols

The following list defines all symbols used throughout this text, in the order they first appear. Since Chapter 2, Chapter 3 and Chapters 4 to 7 cover different methods of analysis, a slightly different set of symbols is employed in each to match those of the major references in their respective areas. To avoid confusion arising from the same symbol having a different meaning in different chapters, a separate symbol list is given for each of the three analyses.

By convention, all vector and matrix/tensor quantities are written in bold type, except where subscripted indices are used to refer to their individual components. Furthermore, repeated indices denote summation over the range of components involved, which is usually 3. For example, $F_{rS}X_S = F_{r1}X_1 + F_{r2}X_2 + F_{r3}X_3$.

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>$x_1$, $x_2$, $x_3$</td>
<td>Cartesian coordinate axes</td>
</tr>
<tr>
<td></td>
<td>$a$</td>
<td>unit vector in direction of first family of fibres</td>
</tr>
<tr>
<td></td>
<td>$b$</td>
<td>unit vector in direction of second family of fibres</td>
</tr>
<tr>
<td></td>
<td>$\phi$</td>
<td>angle of fibre orientation in $x_1$-$x_2$ plane, measured from $x_1$</td>
</tr>
<tr>
<td></td>
<td>$u$</td>
<td>velocity vector</td>
</tr>
<tr>
<td></td>
<td>$x$</td>
<td>Cartesian coordinate vector</td>
</tr>
<tr>
<td></td>
<td>$D$</td>
<td>rate of deformation tensor</td>
</tr>
<tr>
<td></td>
<td>$T$</td>
<td>total stress tensor</td>
</tr>
<tr>
<td></td>
<td>$R$</td>
<td>reaction stress due to kinematic constraints</td>
</tr>
<tr>
<td></td>
<td>$S$</td>
<td>deviatoric or extra stress, dependent on the deformation</td>
</tr>
<tr>
<td></td>
<td>$T_a$</td>
<td>tension reaction stress to inextensibility constraint in $a$-direction</td>
</tr>
<tr>
<td></td>
<td>$T_b$</td>
<td>tension reaction stress to inextensibility constraint in $b$-direction</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>hydrostatic pressure, reaction stress to incompressibility constraint</td>
</tr>
<tr>
<td></td>
<td>$\delta_{ij}$</td>
<td>Kronecker delta; 0 when $i \neq j$, 1 when $i = j$</td>
</tr>
<tr>
<td></td>
<td>$\mu$</td>
<td>isotropic shear viscosity</td>
</tr>
<tr>
<td></td>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>-----------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
<td></td>
</tr>
<tr>
<td>$h$</td>
<td>height of a fibre (in direction of axis $x_3$)</td>
<td></td>
</tr>
<tr>
<td>$h^0$</td>
<td>unperturbed fibre height</td>
<td></td>
</tr>
<tr>
<td>$h'$</td>
<td>form of the perturbation to the height of the fibre</td>
<td></td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>small, dimensionless perturbation parameter</td>
<td></td>
</tr>
<tr>
<td>$u^0, a^0$, etc.</td>
<td>unperturbed velocity, $a$-fibre direction, etc.</td>
<td></td>
</tr>
<tr>
<td>$u', a'$, etc.</td>
<td>perturbation added to velocity, $a$-fibre direction, etc.</td>
<td></td>
</tr>
<tr>
<td>$G$</td>
<td>convected growth rate</td>
<td></td>
</tr>
<tr>
<td>$h^<em>, \gamma^</em>$, etc.</td>
<td>initial amplitude of perturbation in fibre height, $a$-fibre direction, etc.</td>
<td></td>
</tr>
<tr>
<td>$i$</td>
<td>(when not used as indicial subscript) imaginary unit</td>
<td></td>
</tr>
<tr>
<td>$K$</td>
<td>wave number of the buckle</td>
<td></td>
</tr>
<tr>
<td>$\lambda$</td>
<td>buckle wavelength</td>
<td></td>
</tr>
<tr>
<td>$\theta, x_0$</td>
<td>buckle angle and direction</td>
<td></td>
</tr>
<tr>
<td>$\alpha, \gamma$</td>
<td>in-plane and out-of-plane perturbations to $a$-fibre direction</td>
<td></td>
</tr>
<tr>
<td>$\beta, \delta$</td>
<td>in-plane and out-of-plane perturbations to $b$-fibre direction</td>
<td></td>
</tr>
<tr>
<td>$T^*$</td>
<td>stress tensor on the free surface, in the surface coordinate system</td>
<td></td>
</tr>
<tr>
<td>$Q$</td>
<td>transformation matrix</td>
<td></td>
</tr>
<tr>
<td>$T_0$</td>
<td>fibre tension reaction stresses resolved in the buckle direction</td>
<td></td>
</tr>
<tr>
<td>$\kappa$</td>
<td>non-dimensional wave number squared</td>
<td></td>
</tr>
<tr>
<td>$\tau$</td>
<td>non-dimensional fibre tension factor</td>
<td></td>
</tr>
<tr>
<td>$x, y, z$</td>
<td>Cartesian coordinate axes</td>
<td></td>
</tr>
<tr>
<td>$X$</td>
<td>undeformed (reference, material) coordinates</td>
<td></td>
</tr>
<tr>
<td>$x$</td>
<td>deformed (spatial) coordinates</td>
<td></td>
</tr>
<tr>
<td>$F$</td>
<td>deformation gradient tensor</td>
<td></td>
</tr>
<tr>
<td>$C$</td>
<td>right Cauchy-Green deformation tensor</td>
<td></td>
</tr>
<tr>
<td>$\lambda_1, \lambda_2$</td>
<td>principal stretches in the plane of the sheet</td>
<td></td>
</tr>
<tr>
<td>$\phi_n$</td>
<td>basis functions for interpolating over the surface element</td>
<td></td>
</tr>
<tr>
<td>Chapter</td>
<td>Symbol</td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>---------</td>
<td></td>
</tr>
<tr>
<td>3 (cont.)</td>
<td>( \xi = (\xi_1, \xi_2) )</td>
<td>surface coordinate defined over the element</td>
</tr>
<tr>
<td></td>
<td>( u_n )</td>
<td>nodal vectors of position and slope, interpolated over the element</td>
</tr>
<tr>
<td></td>
<td>( S_i )</td>
<td>arc length in direction of ( \xi_i )</td>
</tr>
<tr>
<td></td>
<td>( \xi^{(d)} )</td>
<td>surface coordinate of grid point ( d )</td>
</tr>
<tr>
<td></td>
<td>( K )</td>
<td>global stiffness matrix</td>
</tr>
<tr>
<td></td>
<td>( f )</td>
<td>global force vector</td>
</tr>
<tr>
<td></td>
<td>( E )</td>
<td>weighted, squared sum of fitting errors</td>
</tr>
<tr>
<td></td>
<td>( D )</td>
<td>number of grid points</td>
</tr>
<tr>
<td></td>
<td>( w^{(d)} )</td>
<td>weighting factor on grid point ( d )</td>
</tr>
<tr>
<td></td>
<td>( \hat{u}(\xi^{(d)}) )</td>
<td>unknown interpolated deformed position of grid point ( d ), used interchangeably for ( x, y ) and ( z )</td>
</tr>
<tr>
<td></td>
<td>( u^{(d)} )</td>
<td>actual deformed position of grid point ( d ), used interchangeably for ( x, y ) and ( z ) at ( d )</td>
</tr>
<tr>
<td></td>
<td>( \bar{E} )</td>
<td>modified least squares sum</td>
</tr>
<tr>
<td></td>
<td>( \alpha )</td>
<td>factor for controlling in-plane stretching of the fitted surface</td>
</tr>
<tr>
<td></td>
<td>( \beta )</td>
<td>factor for controlling curvature of the fitted surface</td>
</tr>
<tr>
<td></td>
<td>( V )</td>
<td>matrix containing the eigenvectors of ( C )</td>
</tr>
<tr>
<td></td>
<td>( A )</td>
<td>matrix containing the eigenvalues of ( C )</td>
</tr>
<tr>
<td></td>
<td>( \varepsilon_i )</td>
<td>engineering strains</td>
</tr>
<tr>
<td></td>
<td>( e_i )</td>
<td>natural strains</td>
</tr>
<tr>
<td></td>
<td>( E_i )</td>
<td>Lagrangian or Green’s finite strains</td>
</tr>
<tr>
<td></td>
<td>( \lambda_3 )</td>
<td>through-thickness stretch</td>
</tr>
<tr>
<td></td>
<td>([0,90]_s)</td>
<td>denotes a symmetrical 4-ply laminate with 0° plies on the outer surfaces and two 90° oriented plies in the centre</td>
</tr>
<tr>
<td>4,5,6,7</td>
<td>( x )</td>
<td>spatial (current) Cartesian coordinates</td>
</tr>
<tr>
<td></td>
<td>( T )</td>
<td>Cauchy (true) stress tensor</td>
</tr>
<tr>
<td></td>
<td>( f^{(body)} )</td>
<td>body force vector per unit volume</td>
</tr>
<tr>
<td></td>
<td>( n^{(S)} )</td>
<td>unit surface normal on boundary</td>
</tr>
<tr>
<td></td>
<td>( t^{(S)} )</td>
<td>boundary traction (force) vector</td>
</tr>
<tr>
<td>Chapter</td>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>4,5,6,7</td>
<td>δu</td>
<td>virtual displacement</td>
</tr>
<tr>
<td>(cont.)</td>
<td>δW_{ext}</td>
<td>virtual work</td>
</tr>
<tr>
<td></td>
<td>S</td>
<td>current boundary of body</td>
</tr>
<tr>
<td></td>
<td>V</td>
<td>current volume occupied by body</td>
</tr>
<tr>
<td></td>
<td>ε</td>
<td>infinitesimal strain tensor</td>
</tr>
<tr>
<td></td>
<td>(\bar{T})</td>
<td>symmetric second Piola-Kirchoff stress tensor, later expressed as a six-component vector</td>
</tr>
<tr>
<td></td>
<td>E</td>
<td>symmetric Lagrangian finite strain tensor, later expressed as a six-component vector</td>
</tr>
<tr>
<td></td>
<td>(V_0)</td>
<td>reference (undeformed) volume originally occupied by body</td>
</tr>
<tr>
<td></td>
<td>X</td>
<td>material (reference) Cartesian coordinates</td>
</tr>
<tr>
<td></td>
<td>(\xi=(\xi_1,\xi_2,\xi_3))</td>
<td>parametric coordinates within an element</td>
</tr>
<tr>
<td></td>
<td>(a_j)</td>
<td>displacement or displacement slope vector at node (j)</td>
</tr>
<tr>
<td></td>
<td>(M)</td>
<td>number of nodes in the element, or in the whole model as necessary</td>
</tr>
<tr>
<td></td>
<td>(\psi^i(\xi))</td>
<td>basis functions for interpolating nodal quantities over the volume of the element</td>
</tr>
<tr>
<td></td>
<td>(F^{\text{adot}})</td>
<td>total displacement gradient tensor</td>
</tr>
<tr>
<td></td>
<td>(\delta a)</td>
<td>virtual nodal displacement vector</td>
</tr>
<tr>
<td></td>
<td>(\delta E)</td>
<td>virtual increment in Lagrangian strain tensor</td>
</tr>
<tr>
<td></td>
<td>(\bar{B}^i)</td>
<td>matrix for calculating incremental strains from nodal displacements</td>
</tr>
<tr>
<td></td>
<td>(B_0^i)</td>
<td>linear component of (\bar{B}^i)</td>
</tr>
<tr>
<td></td>
<td>(B_L^i)</td>
<td>large strain, non-linear component of (\bar{B}^i)</td>
</tr>
<tr>
<td></td>
<td>(\Psi(a))</td>
<td>residual vector</td>
</tr>
<tr>
<td></td>
<td>(f)</td>
<td>vector of generalised nodal forces</td>
</tr>
<tr>
<td></td>
<td>(K_T)</td>
<td>tangential stiffness matrix</td>
</tr>
<tr>
<td></td>
<td>(\Delta a)</td>
<td>increment of nodal displacements calculated in an iteration</td>
</tr>
<tr>
<td></td>
<td>(\bar{K})</td>
<td>large strain global stiffness matrix</td>
</tr>
<tr>
<td></td>
<td>(K_\sigma)</td>
<td>initial stress matrix</td>
</tr>
<tr>
<td></td>
<td>(K_f)</td>
<td>load-correction matrix</td>
</tr>
<tr>
<td>Chapter</td>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>4,5,6,7</td>
<td>$D$</td>
<td>incremental elasticity matrix</td>
</tr>
<tr>
<td>(cont.)</td>
<td>$\alpha$</td>
<td>a small number</td>
</tr>
<tr>
<td></td>
<td>$x_1, x_2, x_3$</td>
<td>Cartesian coordinate axes</td>
</tr>
<tr>
<td></td>
<td>$X_1, X_2, X_3$</td>
<td>Material (convected) coordinate axes</td>
</tr>
<tr>
<td></td>
<td>$a^0$</td>
<td>unit vector in direction of undeformed fibres</td>
</tr>
<tr>
<td></td>
<td>$dX$</td>
<td>differential line element in the undeformed state</td>
</tr>
<tr>
<td></td>
<td>$dx$</td>
<td>$dX$ transformed onto the deformed body</td>
</tr>
<tr>
<td></td>
<td>$F$</td>
<td>deformation gradient tensor</td>
</tr>
<tr>
<td></td>
<td>$C$</td>
<td>right Cauchy-Green deformation tensor</td>
</tr>
<tr>
<td></td>
<td>$W$</td>
<td>strain energy function (per unit volume) for hyperelastic material</td>
</tr>
<tr>
<td></td>
<td>$I_1, I_2, I_3, I_4, I_5$</td>
<td>invariants of tensors $C$ and $a^0 \otimes a^0$</td>
</tr>
<tr>
<td></td>
<td>$\delta_{ij}$</td>
<td>(when not used as a prefix for virtual quantities) Kronecker delta; 0 when $i \neq j$, 1 when $i = j$</td>
</tr>
<tr>
<td></td>
<td>$K_1, K_2, ... K_5$</td>
<td>material parameters in strain energy function</td>
</tr>
<tr>
<td></td>
<td>$E_f$</td>
<td>Young’s modulus in fibre direction</td>
</tr>
<tr>
<td></td>
<td>$E_T$</td>
<td>Young’s modulus in directions transverse to fibres</td>
</tr>
<tr>
<td></td>
<td>$G_L$</td>
<td>longitudinal shear modulus</td>
</tr>
<tr>
<td></td>
<td>$G_T$</td>
<td>transverse shear modulus</td>
</tr>
<tr>
<td></td>
<td>$\overline{W}$</td>
<td>modified strain energy function</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>hydrostatic pressure reaction stress to incompressibility constraint</td>
</tr>
<tr>
<td></td>
<td>$T$</td>
<td>tension reaction stress to inextensible fibre constraint</td>
</tr>
<tr>
<td></td>
<td>$b$</td>
<td>hydrostatic pressure at the node</td>
</tr>
<tr>
<td></td>
<td>$c$</td>
<td>tension at the node</td>
</tr>
<tr>
<td></td>
<td>$\hat{p}, \hat{T}$</td>
<td>interpolated pressure and tension, respectively</td>
</tr>
<tr>
<td></td>
<td>$\overline{N}_i$</td>
<td>basis functions for interpolating pressure and tension</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>point on surface “A” touching some other surface “B”</td>
</tr>
<tr>
<td></td>
<td>$p$</td>
<td>point on surface “B” that $k$ is in contact with</td>
</tr>
<tr>
<td></td>
<td>$n, t^1, t^2$</td>
<td>outward normal and tangent vectors to surface “B” at $p$</td>
</tr>
<tr>
<td>Chapter</td>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>---------</td>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>(d)</td>
<td>distance between points (k) and (p) along normal (n)</td>
</tr>
<tr>
<td></td>
<td>(S_n)</td>
<td>normal contact stiffness (spring stiffness per unit area)</td>
</tr>
<tr>
<td>4, 5, 6, 7</td>
<td>(\xi^{(k)}, \xi^{(p)})</td>
<td>surface coordinates of points (k) and (p)</td>
</tr>
<tr>
<td>(cont.)</td>
<td>(w)</td>
<td>integration/Gauss point weight</td>
</tr>
<tr>
<td></td>
<td>(A)</td>
<td>area associated with a contact point</td>
</tr>
<tr>
<td></td>
<td>(\mathbf{F}_n^{(k)})</td>
<td>normal force vector transferred onto point (k) by a contact spring</td>
</tr>
<tr>
<td></td>
<td>(\phi_i^{(A)}, \phi_i^{(B)})</td>
<td>basis functions defined over surfaces (A) and (B), respectively</td>
</tr>
<tr>
<td></td>
<td>(g_i^{(A)})</td>
<td>generalised nodal contact force vector on element surface (A)</td>
</tr>
<tr>
<td></td>
<td>(\kappa)</td>
<td>3 \times 3 contact stiffness matrix</td>
</tr>
<tr>
<td></td>
<td>(\Delta k, \Delta p)</td>
<td>displacement of points (k) and (p) during an iteration</td>
</tr>
<tr>
<td></td>
<td>(K^{(AB)}, \text{etc.})</td>
<td>contact stiffness terms</td>
</tr>
<tr>
<td></td>
<td>(C(a))</td>
<td>contact constraint equation</td>
</tr>
<tr>
<td></td>
<td>(\pi)</td>
<td>penalty number for applying contact constraints</td>
</tr>
<tr>
<td></td>
<td>(S_{11}, S_{12})</td>
<td>tangential contact stiffnesses</td>
</tr>
<tr>
<td></td>
<td>(\Delta t)</td>
<td>length of a time increment</td>
</tr>
<tr>
<td></td>
<td>(\eta)</td>
<td>viscosity (in Pa.s) of the fluid in the inter-ply region</td>
</tr>
<tr>
<td></td>
<td>(\theta)</td>
<td>estimated inter-layer thickness for contact friction calculations</td>
</tr>
<tr>
<td></td>
<td>(\tau)</td>
<td>shear stress applied as a result of inter-ply slip</td>
</tr>
<tr>
<td></td>
<td>(v)</td>
<td>relative velocity between two contacting surfaces</td>
</tr>
<tr>
<td></td>
<td>(c_f)</td>
<td>coefficient of sliding friction, such that (\tau = c_f v)</td>
</tr>
<tr>
<td></td>
<td>(\nu_f)</td>
<td>fibre volume fraction (ratio from 0.0 to 1.0)</td>
</tr>
<tr>
<td></td>
<td>(\rho_0)</td>
<td>density in reference (undeformed) state</td>
</tr>
<tr>
<td></td>
<td>(\ddot{u})</td>
<td>acceleration vector</td>
</tr>
<tr>
<td></td>
<td>(\dot{u})</td>
<td>velocity vector</td>
</tr>
</tbody>
</table>
Chapter 1: Introduction

1.1 Continuous Fibre Reinforced Thermoplastics

In the early Eighties, materials scientists first successfully impregnated highly collimated continuous fibres with thermoplastic polymer resins to create a new class of material called Continuous Fibre Reinforced Thermoplastics (CFRTPs). While possessing nearly all the advantages of existing thermoset matrix composites in providing optimised, designer material properties, CFRTPs promised several improvements in both performance and production efficiency due to the long, linear chain structure of the thermoplastic matrix. The greatest of these is the possibility of thermoforming, in which the material is heated to around the melting point of the matrix and formed into the desired shape. While not yet fully realised, this feature provides the potential to break from the labour-intensive, hand lay-up techniques and lengthy curing periods prevalent with thermoset matrix materials, and to move to efficient, mass production of composite structures.

To facilitate ease of lamination and part production, manufacturers supply CFRTP material in the form of thin, typically 0.125 to 0.5 mm thick preimpregnated tapes, commonly referred to as prepregs. Prepregs come in a variety of widths, depending on the manufacturing process they are to be used with; below 25 mm for filament winding, and at least 250 mm for sheet forming applications. While several other CFRTP product forms, including powder coated and commingled fibres are available, these drapable forms rely on impregnation subsequent to shaping and result in an inferior product when compared to prepregs [1]. In contrast to thermoset composites, CFRTPs possess the advantage of simplified and stable chemistry, leaving fabricators only the task of heating, forming and cooling. Prepreg products enhance these advantages further by providing a highly uniform, void-free and optimally aligned starting material from which most laminates and parts can be readily produced. Furthermore, the matrix serves to protect the fibres from physical and chemical attack in storage and handling.

The stable chemistry inherent with pre-polymerised thermoplastics is maintained in the reinforced material, where resistance to attack by water and numerous other solvents opens up many new applications for this class of composite material. The long, linear-chain structure of thermoplastic polymers imparts greater toughness to CFRTP structures when compared with thermoset matrix composites. While strong cross-linking bonds in thermosets give them a brittle mode of failure, weak Van der Waals forces between the entangled, long chain polymers in thermoplastics allows them to dissipate sudden stress concentrations caused by impacts through chain slippage, or flow.

A final potential advantage of CFRTPs over reinforced thermoset materials is in the area of recycling. Although neither product may be completely restored to its original form, CFRTP off-cuts and parts that have ended their service life could be reused in random chopped-fibre mat
material, or in long fibre injection moulding compounds, to increase the life-span of these often expensive materials. A future possibility could be chemical or other removal of the matrix for reuse in new prepreg product.

Despite the promise initially shown by CFRTPs, they represent only a small fraction of polymer matrix composites on the market today. Part of the blame for this may relate to the lack of widespread availability and experience with these materials. However, the real reason is probably the lack of a cost-effective, off-the-shelf solution for forming these materials. Despite several outstanding successes, CFRTP thermoforming is still plagued by difficulty and remains a topic of much research. This thesis hopes to expand the knowledge base in this crucial area, by concentrating on the problem of thermoforming parts from laminated sheets of CFRTP prepregs.

The early push in the development of thermoplastic composites was to produce high performance materials for aerospace applications. An example of such a material is “APC-2”, consisting of 61 vol% unidirectional carbon fibre reinforced polyetheretherketone (PEEK) [1]. More recently, the emphasis has shifted into developing CFRTPs for use in general industrial applications, where cost and production flexibility are of greater importance. A good example from this latter category is glass fibre reinforced polypropylene PLYTRON® [2,3]. In addition to its own areas of application, PLYTRON has found much use in research circles as a low cost test-bed for understanding the behaviour of the whole class of CFRTPs. As such, the deformation behaviour of PLYTRON will often feature in this study, and the following section describes this material in more depth. It must be noted, however, that most of the discussions and findings in this report are relevant not just to PLYTRON, or even CFRTP prepregs, but to sheet forming of any fibre-reinforced material, including thermoset-prepregs.

1.1.1 GF/PP PLYTRON®

PLYTRON is a commercial prepreg material consisting of 35 vol% unidirectional glass fibre reinforced polypropylene [2,3]. It was originally developed by ICI (UK) and is currently produced by Borealis (Norway) and Mitsui-Toatsu (Japan). PLYTRON was initially supplied in rolls of 0.5 mm thick, 250 mm wide tape, but is now also available in thinner, wider and black, UV-protected forms. In service the material has an axial tensile modulus of 25.5 GPa and strength of 650 MPa [3], making it competitive with some metals in structural sheet components.

The melting point of the polypropylene matrix is 165 °C, with the manufacturer recommending processing be carried out between 180 and 230 °C [3]. However, experience at the University of Auckland has shown that provided the material has been heated to above its melting point, the matrix remains pliable down to its recrystallisation temperature of around 125 °C. This phenomenal processing range is a special feature of PLYTRON, but successful sub-melting point forming has also been reported with other CFRTPs, including APC-2 [1]. One consequence of this is that the behaviour of the material during thermoforming cannot be simply characterised. At temperatures well above the melting point it may behave as a reinforced fluid, but at lower temperatures the still-formable material may more closely resemble a rubbery solid.
Much can be learned about the deformation behaviour of PLYTRON and other CFRTPs from their microstructure. The cross sections of PLYTRON laminates shown in Figure 1.1 are typical of this class of composites.

(a) \([0_6]\) Thickness = 2.4 mm.                             (b) \([0_2, 90]_s\) Thickness = 2.7 mm.

Figure 1.1 Micrograph cross sections of two PLYTRON GF/PP laminates.

The samples shown in Figure 1.1 were consolidated for 10 minutes at 200 °C, under a pressure of 750 kPa. Note that the laminates are thinner than the original 6 × 0.5 mm plies due to matrix being squeezed out through small gaps in the mould.

The main observation to be made from these images is that the fibres are very non-uniformly distributed, clustering into regions of nearly optimal fibre packing. Between these are zones of almost pure matrix, and as is most clearly evident from the cross-ply laminate, the matrix-rich zones occur on the boundaries where the six original plies have fused. In addition, sizeable matrix-rich zones are present on the outer laminate surfaces. PLYTRON prepregs possess generous, protective surface layers of almost pure matrix which persist in their laminates as so-called ‘matrix-rich inter-layers’. These provide the site for inter-ply slip to occur, as described in the following section. Such inter-layers occur in other fibre/matrix systems [4], and it is noted that even in laminates made from CFRTPs of high fibre volume fraction there will exist zones of low fibre concentration and weakness where the plies have fused together.
1.2 Thermoforming of CFRTP Laminates

1.2.1 Ideal Flow Processes

The most striking feature of molten continuous fibre reinforced thermoplastics during forming is the limited deformation that may occur in the direction of reinforcement in each ply. As a result, gross deformation of this class of materials takes place through a combination of the shear and transverse flow processes shown in Figure 1.2, and described here. Note that some of the processes are identical except for rotation of the specimen, but for the benefit of later discussions will be distinguished here.

*Resin Percolation* is the term given to flow of matrix along and between fibres in such a way as to destroy the continuity of the overall deformation. It occurs to some extent in all thermoforming and is especially noticeable at the end of unidirectional plies where resin is squeezed out from between the fibres. Resin percolation also covers the fusion process bonding plies together in the consolidation phase. In low viscosity, high fibre volume fraction thermoplastic composites, resin percolation is limited, and is frequently neglected in models of such materials.

![Diagram](image)

**Figure 1.2** Deformation mechanisms of molten continuous fibre-reinforced thermoplastic prepregs and their laminates.

In regions of the laminate subject to high normal compressive loads, *Transverse Flow* will occur, resulting in fibre spreading and often considerable laminate thinning. As this reduces the stiffness and strength of the laminate, transverse flow is largely seen as an undesirable process. Where lateral movement is restricted, the same conditions are likely to produce resin percolation along the fibres. On a microscopic scale, transverse flow in real CFRTPs is unlikely to mirror the simple behaviour shown in Figure 1.2. Rather, it will involve both resin percolation and complex interactions between twisted, misaligned and clustered fibres. *Transverse Intra-Ply Shear* may be considered identical to transverse flow, except for a rotation of 45°. However, it will occur under different conditions, during bending and when displacement of neighbouring plies causes shear...
stresses to be transferred, and need not involve any significant thickness change.

Two types of *Longitudinal Intra-Ply Shear* are depicted in Figure 1.2, both involving slip across inter-fibre spaces. While on the surface there appears to be no difference between these two processes, Section 4.1.1 describes a size effect that differentiates both how they occur and how best to model them.

*Inter-Ply Slip* is the term describing the effective net displacement of neighbouring plies as a result of massive shearing of the thin, matrix rich inter-ply layers described in Section 1.1.1 and clearly visible in Figure 1.1. Inter-layer thicknesses are highly variable and, frequently, fibres in neighbouring plies will touch or even merge if they are parallel. Nevertheless a weak point always exists on ply boundaries, allowing inter-ply slip to take place. For these reasons, inter-ply slip is often treated as a friction/lubrication phenomenon. Some researchers add *Inter-Ply Rotation* to this list to describe relative rotation between adjacent plies. Although the term is useful for describing certain deformations that can take place, from a process point of view it is merely another manifestation of inter-ply slip.

### 1.2.2 CFRTP Sheet Forming Processes

Many different sheet forming processes for CFRTPs have been devised and tested since the inception of these materials early last decade. Some of these, such as *matched-die forming* [1,5] and *hydroforming* [5] are adapted from successful sheet metal processes, while others, most notably *diaphragm forming* [1,5,6], were devised specifically for these materials, and make use of the low forces required to mould molten thermoplastics.

CFRTP forming processes differentiate themselves further by the way they maintain the temperature of the laminate during forming. The first option is for a heated, pre-consolidated laminate to be transferred into cold tools for rapid forming and cooling. The alternative is to hold the tooling at the same temperature as the laminate, possibly by placing the entire apparatus in an environmental chamber. This allows forming to be carried out at slower rates, but has the disadvantage of longer processing times, since tools must be heated and cooled once every cycle. Hence, for materials such as PLYTRON, intended for general industrial applications, non-isothermal, transfer processes will be favoured.

Two thermoforming processes of particular interest in this study are diaphragm forming and matched-die forming, described here.

![Figure 1.3 Non-isothermal diaphragm forming.](image)
Diaphragm forming involves first placing the plies in the desired lay-up between two thin diaphragms in a frame. This "sandwich" is then heated to the forming temperature with a vacuum drawn between the diaphragms to consolidate the laminate, at which point it must be placed in the tool. Pressure is then applied to one side of the molten sandwich, forcing the diaphragms and laminate to take the form of the porous tool. The pressure is maintained until the part has cooled sufficiently.

In this process the diaphragms provide not only a seal to enable forming with pressure, but act to tension the laminate during forming, reducing buckling defects in the part. Up till now most researchers have carried out diaphragm forming under isothermal conditions in an autoclave, using disposable superplastic aluminium [1,5,7] and polyimide [1,5,6] diaphragms. At the University of Auckland, good forming results have also been obtained using thin (0.75 mm) reusable silicone rubber diaphragms and rapid transfer of the heated specimen into cold tools [8].

**Figure 1.4** Non-isothermal matched-die thermoforming.

The matched-die thermoforming process depicted above has been extensively applied in producing parts from CFRTP materials [1,5,9-12]. Apart from its great simplicity, one of the benefits of this process is the close dimensional tolerance and good surface finish obtainable by having both sides of the material touching the tools. However, most parts produced using the simple process depicted in Figure 1.4 exhibit buckling or wrinkling defects (refer to Chapter 2), marring their appearance and adversely affecting their performance. As in sheet metal forming, applying edge-clamping to induce tension in the laminate has proved beneficial [5,12]. Diaphragms offer similar benefits, and have been used by O’Bradaigh [13] and Martin [14] in related punch indentation tests.

**1.3 Thesis Outline**

Research into CFRTP sheet forming is ultimately concerned with finding ways to improve and control the process so that optimised, defect-free parts can be efficiently and reliably produced with it. The first step towards this goal is to understand the deformation behaviour and load response of the material. Usually, this knowledge is enough to enable incremental improvements to be made to the process. However, as demands for greater part complexity and process optimisation increase, fabricators will turn to numerical and other analysis techniques to gain further insight into the process, and ultimately, to enable prediction of the conditions leading to
successful forming before any tooling has been made. Common to all such analysis tools is the need for a model to describe the behaviour of the material under study. Much of this thesis concerns the development of a molten CFRTP laminate model and its application in numerical analyses of sheet forming. Leading up to this are two preliminary studies into different aspects of forming. The first uses an existing molten composite model to look at the problem of gross-buckling, while the second presents a tool for analysing strains in sheet-formed parts.

One of the greatest problems in CFRTP sheet forming is the growth of out-of-plane buckling defects. Chapter 2 presents an analytical study into the causes of buckling in common homogeneous flows of bidirectional composites. It employs the Ideal Fibre-Reinforced Fluid model, consisting of inextensible fibres imbedded in an incompressible Newtonian fluid, which is frequently adopted in studies of molten thermoplastic composites. Linear stability analysis is used to determine the growth of small defects present in the composite into macroscopic out-of-plane buckles. This results in expressions relating the growth rate to the important flow parameters including stress, viscosity, flow rate and defect wavelength. The chapter finishes with a discussion on the practical implications of the results.

Detailed modelling of sheet forming processes is of limited use unless techniques are available to compare experimental results with model predictions. To this end, Chapter 3 describes a variation on existing Grid Strain Analysis techniques, which measure forming strains from the change in position of a grid printed on a part before it is formed. The new technique involves first laying out an undeformed surface mesh over the grid points in their original positions. A least-squares procedure is then used to determine the shape of the equivalent deformed mesh based on the final positions of the grid points. From these two surface representations strains and other indications of deformation kinematics may be determined at any point. The procedure was implemented in a program, GSA, which is used in Chapter 3 to analyse strains in a blister fairing formed from two CFRTP laminates. Appendix A presents a User Guide for the GSA program.

The occurrence of inter-ply slip in virtually all forming examples limits the applicability of any model that does not consider this effect. The remainder of this thesis is thus devoted to the development and verification of a discrete-ply model for molten CFRTP laminates. The model uses a highly anisotropic elastic material law to describe continuous deformations in the fibre-rich plies, while inter-ply slip is governed by a viscous contact/friction response over a negligible inter-ply gap. Chapter 4 describes the foundation of this model and its implementation into a non-linear, large strain finite element program, called SimForm. The program includes features such as tool contact and pressure loading to enable real forming conditions to be simulated. SimForm uses a flexible, custom input format, described in Appendix B, while Appendix C looks at the procedures used for program verification.

The following chapter sets out to verify the discrete-ply model used in SimForm by attempting to reproduce the forming behaviour and load response observed in documented experiments with PLYTRON material. The deviation of the model from the real material is pointed out, while a set of
best-fit parameters are chosen for use in subsequent forming simulations. Chapter 6 describes several two- and three-dimensional CFRTP thermoforming simulations. The examples include both matched-die and diaphragm forming, with laminate lay-ups and forming conditions varied to show interesting behaviour. The results are compared with others’ experimental observations, while several practical findings are noted.

Chapter 7 discusses the present limitations of both the SimForm program and the molten composite model it uses. Particular areas of discussion are the problems experienced with inter-element contact, as well as the enormous computing resources required by the program. Future improvements are suggested in these areas, as are changes to the elastic ply/viscous slip model to better describe the viscoelastic response observed in molten CFRTP laminates.

Finally, Chapter 8 summarises the major findings of this work, and presents an opinion on where development of CFRTP thermoforming should proceed in future.
Chapter 2: Buckling Stability in Homogeneous Flows of Bidirectional Composites

2.1 Introduction

Bidirectional fibre-reinforced thermoplastics can be produced in two ways. The first is to impregnate woven-fabric material, and if laminated to produce a thicker specimen, to ensure the fibre orientations match up between layers. The second method is to consolidate together preimpregnated unidirectional tapes so that only two directions of reinforcement are present in the final laminate. In either case extension in the direction of the two families of fibres is limited, causing gross deformation to occur through fibres rotating in a scissors-like action, as shown in Figure 2.1.

![Figure 2.1 Trellis flow of a bidirectional sheet showing coordinate system. (Thickness direction x\textsubscript{3} is out of the page).](image)

Due to its similarity in appearance and behaviour to that of a garden trellis, the deformation depicted in Figure 2.1 is commonly referred to as trellis flow, a term that will also be used here. In woven-fabric materials trellis flow is the only mechanism available for forming doubly-curved specimens. With bidirectional laminates of unidirectional prepregs, however, different flow mechanisms such as transverse flow and intra-ply shear (cf. Figure 1.2), occurring separately in each ply could bring about the same deformation. Experiments with such laminates have shown that they prefer to deform as one unit by trellis flow [15], an observation also noted from the examples of Section 3.3.

Two of the most common defects occurring during thermoforming of fibre-reinforced thermoplastic sheets into three-dimensional parts are out-of-plane buckling and in-plane wrinkling [5,6,15], as depicted in Figure 2.2 for bidirectional materials. Both defects can lead to a reduction in the mechanical performance of the part, and generally mar its appearance.
Out-of-plane buckling.

Figure 2.2 Instabilities occurring in forming flows of bidirectional composites.

(a) Out-of-plane buckling.
(b) In-plane wrinkling.

To combat such defects, manufacturers have adopted techniques from the sheet metal industry, such as clamping the edges of the blank in order to induce tension [5,12]. Nevertheless, it would be useful to obtain a theoretical expression for the conditions that lead to these defects, and this study aims to achieve this for the out-of-plane buckling phenomenon during homogeneous trellis flows.

Out-of-plane buckles are thought to grow from slight fibre misalignments present in the composite material. An accepted approach for analysing this type of growth problem is linear stability analysis [16,17], in which small perturbations are superimposed on the fibre orientation and on to the steady-state values of the other main variables influencing the flow. If the perturbations are given an exponential variation with time, the object is to find whether their growth rate is positive, meaning they will grow into macroscopic defects. Analyses of this type have been carried out previously for thermoforming of composites by Hull et al. [16,17], who determined the stability criteria in shear flows of unidirectional composites.

This chapter describes the application of linear stability analysis to the problem of out-of-plane buckling in homogeneous trellis flows of molten bidirectional composites. As is now common in such studies of molten composite forming, the material is treated as a Newtonian fluid constrained by inextensible fibres. It has been modified slightly from its original publication in Reference [18], also presented in abridged form as Reference [19].

2.2 Governing Equations

The molten bidirectional composite will be modelled as an Ideal Fibre-Reinforced Fluid [20,21] incorporating the kinematic constraints of incompressibility and inextensibility in each of the two directions of reinforcement. Extensive reference is made in this section to the work of Spencer, Rogers and co-workers [16,17,20-23] who introduced the theory of ideal fibre-reinforced materials.

The introduction of kinematic constraints limits the deformations that the material can undergo. The following subsection notes the extra
equations these relations introduce, while their effect on the state of stress in the material is looked at in Section 2.2.2.

2.2.1 Kinematic Constraints

As depicted in Figure 2.1, a bidirectional fibre-reinforced material can be described by the unit vectors $a$ and $b$ in the two directions of reinforcement. This figure also defines the coordinate system to be used throughout this analysis. The $x_1$ and $x_2$ axes are the bisectors of the angles between the two fibre direction vectors, so that $a = (\cos \phi, \sin \phi, 0)$ and $b = (\cos \phi, -\sin \phi, 0)$, which by reason of being unit vectors satisfy,

\begin{align}
    a_i a_i &= 1, \\
    b_i b_i &= 1,
\end{align}

where repeated indices denote summation over the values 1, 2 and 3.

The material is assumed to be incompressible, which introduces the kinematic constraint equation,

\[ \frac{\partial}{\partial x_i} u_{ii} = 0, \]

where $u_i$ are the components of velocity.

The second and third kinematic constraints follow from the assumption that the two families of fibres are inextensible, which can be expressed for the $a$-fibres by,

\[ a_i a_j D_{ij} = 0, \]

with a similar expression for the $b$-family, while the rate of deformation tensor is defined by,

\[ D_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \]

The fibres are assumed to convect with the flowing fluid, so that no matrix percolation can take place. Hence, the same element of fluid remains beside the same length of fibre throughout the deformation, which can be expressed by [22],

\begin{align}
    \dot{a}_i &= \partial a_i / \partial t + u_j \partial a_i / \partial x_j = a_j \partial u_i / \partial x_j, \\
    \dot{b}_i &= \partial b_i / \partial t + u_j \partial b_i / \partial x_j = b_j \partial u_i / \partial x_j,
\end{align}

where a point denotes the material derivative. It has however been shown that equation (2.4) can be derived from equations (2.1) and (2.6) [17] (and similarly in the $b$-direction). Hence, equations (2.1), (2.2), (2.3), (2.6) and (2.7) supply all the necessary kinematic constraints for the material.

2.2.2 Constitutive Equation

The total stress tensor in a constrained material is the sum of the reaction stress to the kinematic constraints and the deviatoric or extra stress [22]:

\[ T_{ij} = R_{ij} + S_{ij}, \]

The reaction stress is given in terms of reaction tensions $T_a$ and $T_b$ stemming from the inextensibility constraints in the $a$ and $b$ fibre directions,
respectively, and of the hydrostatic pressure $p$, the standard reaction stress to the constraint of incompressibility [21,22]:

$$R_{ij} = T_{ai} a_j + T_{bj} b_i - p\delta_{ij},$$

(2.9)

where in equation (2.9), $\delta_{ij}$ is the Kronecker delta.

Many different forms of the extra stress are possible for the type of constrained fluid studied here. A Newtonian fluid model will be chosen, and it is noted that the introduction of one or more families of fibres into the fluid will cause it to be anisotropic in the general case. References [20] and [21] present a number of constitutive equations for the simpler case of unidirectional reinforced Newtonian fluids which involve two viscosities, for shear along and transverse to the fibre direction. By analogy with the linear-elastic material of references [22] and [23], five viscosities are likely to be needed in the bidirectional case, dropping to four if the two families of fibres are indistinguishable except for their directions. However, noting that experimenters have not yet agreed on the relative values of the two viscosities for the unidirectional IFRF [1,17], it is unwise to continue with so many material parameters. Hence, the material is assumed to be a constrained isotropic Newtonian fluid, with the extra stress defined by,

$$S_{ij} = 2\mu D_{ij},$$

(2.10)

where $\mu$ is the shear viscosity of the fluid, while the rate of deformation tensor $D_{ij}$ was defined in equation (2.5).

Apart from the kinematic constraints and the constitutive equation, equilibrium internally and with external loads and inertial forces must be maintained throughout the material. The governing equations are thus closed by the three equations of motion,

$$\frac{\partial T_{ij}}{\partial x_j} = \rho \left( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right),$$

(2.11)

where $\rho$ is the density of the composite. It is often assumed that fibre-reinforced liquids undergo slow creeping flows, in which case inertia may be neglected by setting the density to zero.

### 2.3 Linear Stability Analysis

Perturbation analyses are common in the study of fluid flows, and were first applied to forming flows of fibre-reinforced fluids by Hull et al. [16,17]. Linear stability analysis proceeds first by finding the background or zeroth-order solution to the flow problem. A small perturbation is then superimposed onto the background solution, and perturbed components of the main variables equated to give the linearised governing equations from which stability criteria can be derived. In this section, each of these steps is carried out for the trellis flows under study.

#### 2.3.1 Background Solution

At a given instant in time, with fibre angle $\phi$, the homogenous trellis-flow deformation shown in Figure 2.1 is completely defined if just one of the normal terms in the rate of deformation tensor is known, since the $x_i$ axes are principal directions. For clarity, these principle rates of deformation will
be denoted $D_1^0$, $D_2^0$ and $D_3^0$. Given $D_1^0$, inextensibility in the fibres inclined at angles $\pm \phi$ to $x_1$ gives the relation,

$$ D_2^0 = -D_1^0 \cot^2 \phi. $$

(2.12)

Further, from the incompressibility assumption it is found that,

$$ D_3^0 = -D_1^0 (1 - \cot^2 \phi). $$

(2.13)

Note that only one of the $D_i^0$ may be kept constant with time in any deformation of this type, since the fibre angles also change with time.

The extra stress is determined from the rate of deformation tensor using equation (2.10). Also, given fibre tensions $T_a$ and $T_b$ and the hydrostatic pressure $p$, the reaction stresses are found from equation (2.9) to be,

$$ R_{11} = (T_a + T_b) \cos^2 \phi \quad R_{33} = -p $$

$$ R_{22} = (T_a + T_b) \sin^2 \phi \quad R_{12} = (T_a - T_b) \sin \phi \cos \phi $$

(2.14)

The composite is assumed to be a thin sheet with free surfaces on each face, so that it is in a state of plane stress. Hence, $T_{3i} = 0$ and $p = S_{33}$, while the in-plane total stress components are,

$$ T_{11} = (T_a + T_b) \cos^2 \phi + 2\mu D_1^0 \left(2 - \cot^2 \phi\right) $$

$$ T_{22} = (T_a + T_b) \sin^2 \phi + 2\mu D_1^0 \left(1 - 2 \cot^2 \phi\right) $$

$$ T_{12} = (T_a - T_b) \sin \phi \cos \phi $$

(2.15)

The background state is thus determined by four parameters, which can be chosen to be $\phi$, $D_1^0$, $T_a$ and $T_b$ or equivalently $\phi$, $T_{11}$, $T_{22}$ and $T_{12}$. The inverses of relations (2.15) are,

$$ D_1^0 = \left(T_{11} \sin^2 \phi - T_{22} \cos^2 \phi\right)/\mu \Theta $$

$$ T_a = +T_{12}/\sin 2\phi + \left(T_{11} \left(2 \cot^2 \phi - 1\right) + T_{22} \left(2 - \cot^2 \phi\right)\right)/\Theta $$

$$ T_b = -T_{12}/\sin 2\phi + \left(T_{11} \left(2 \cot^2 \phi - 1\right) + T_{22} \left(2 - \cot^2 \phi\right)\right)/\Theta $$

(2.16)

where,

$$ \Theta = 4 \left(\cot^2 \phi \cos^2 \phi - \cos 2\phi\right). $$

Spencer [22] made the important point that during this type of homogeneous deformation, only a finite contraction in the through-thickness direction can occur. The material is at its thinnest when the two families of fibres are mutually orthogonal, and any trellis-like deformation that shifts the fibre directions away from this state will make the material thicker.

### 2.3.2 Perturbed Flow

In order to ascertain its stability, a small perturbation will be superimposed onto each of the variables in the background flow described above. Since out-of-plane buckling is of primary interest, we imagine a fibre at height $h$, perturbed from its nominal position so that at time $t$, ...
where $h^0$ is the unperturbed height, $h'$ is the perturbation with a sinusoidal spatial variation and $\varepsilon$ is a small dimensionless perturbation parameter. In linear stability analysis all terms of order $\varepsilon^2$ and higher are ignored. Since the velocity of the matrix in the thickness direction varies linearly with $x_3$, the amplitude of the perturbation will change as the fibre convects with the matrix even if the background rates of deformation are unperturbed. The material derivative of the height is,

$$\dot{h} = \dot{h}^0(t) + \varepsilon \dot{h}'(x_1, x_2, t) + \ldots,$$

which, by continuity, must be equal to the 3-component of the velocity evaluated at $h$. In the background deformation this is,

$$u_3(h) = D_3^0 h^0 + \varepsilon D_3^0 h'(x_1, x_2, t) + \ldots,$$

so that the full velocity component is,

$$u_3 = D_3^0 h^0 + \varepsilon \left(D_3^0 h' + u'_3\right),$$

and thus,

$$\dot{h}' = D_3^0 h' + u'_3.$$

Consequently, as a result of the background deformation alone $h'$ grows at a rate of $D_3^0$, so that we define the extra convected growth rate $G$ to be such that,

$$\dot{h}' = \left(D_3^0 + G\right)h',$$

and so,

$$u'_3 = Gh'.$$

The system will be deemed unstable when $G > 0$, so that the perturbation is growing relative to the background homogeneous deformation.

The perturbed height will be assumed to vary sinusoidally with $x_1$ and $x_2$, so that at the generic instant ($t = 0$) it has the variation,

$$h'(x_1, x_2, 0) = h^* \exp\left(iK(x_1 \cos \theta + x_2 \sin \theta)\right),$$

where $h^*$ is the initial amplitude, $i$ is the imaginary unit, $K$ is the wave number so that $\lambda = 2\pi/K$ is the wavelength, and the ridges of the buckles are at an angle $\pi/2 + \theta$ to the $x_1$ axis, in the plane of the sheet. Axis $x_0$ at angle $\theta$ will be referred to as the *buckle direction*, since it is along this direction that the height of the fibres varies sinusoidally.

Even in the absence of a perturbed velocity field the wavelength and direction of these buckles will change as a result of the background deformation. Hence for $t > 0$, the perturbed height will be written in the form,

$$h'(x_1, x_2, t) = h^* \exp\left(\left(D_3^0 + G\right)t\right) \exp\left(iK\left(x_1 \cos \theta e^{-D_0 t} + x_2 \sin \theta e^{-D_2 t}\right)\right),$$

which reduces to (2.24) when $t = 0$, and as can be readily verified recovers (2.22) upon use of the standard expression for a material derivative:
\begin{align*}
\dot{h} &= \frac{\partial h'}{\partial t} + u_1^0 \frac{\partial h'}{\partial x_1} + u_2^0 \frac{\partial h'}{\partial x_2} \quad (2.26)
\end{align*}

A constructive proof of equation (2.25) is given in Reference [18].

### 2.3.3 Linearised Governing Equations

If the variables in the governing equations of Section 2 are perturbed in the same way as \( h \) in equation (2.17), then the terms of order \( \varepsilon \), equated, form the linearised governing equations. For example, equation (2.1) is modified as follows,

\[ a_i a_i = \left( a_i^0 + \varepsilon a_i' \right) \left( a_i^0 + \varepsilon a_i' \right) = a_i^0 a_i^0 + 2\varepsilon a_i^0 a_i' + O(\varepsilon^2). \quad (2.27)\]

Since this expression remains equal to unity, it follows that to order \( \varepsilon \),

\[ a_i^0 a_i' = 0. \quad (2.28)\]

The perturbation in the unit vector defining the fibre direction is seen to be orthogonal to the original fibre direction, so that the perturbation can be written,

\[ a' = (-\alpha \sin \phi, \alpha \cos \phi, \gamma) \quad (2.29)\]

for some values \( \alpha \) and \( \gamma \). Similarly, in the second fibre direction it is found that,

\[ b' = (\beta \sin \phi, \beta \cos \phi, \delta). \quad (2.30)\]

Note that the perturbations to the fibre directions are not themselves unit vectors.

The perturbed incompressibility condition (2.3) becomes,

\[ \frac{\partial u_i'}{\partial x_1} + \frac{\partial u_j'}{\partial x_2} + \frac{\partial u_k'}{\partial x_3} = 0 , \quad (2.31)\]

while substitution into (2.6) gives the perturbed convection equations for the first fibre direction as,

\begin{align*}
\frac{\partial a_i'}{\partial t} + u_1^0 \frac{\partial a_i'}{\partial x_1} + a_1^0 \frac{\partial u_i'}{\partial x_1} + a_2^0 \frac{\partial u_i'}{\partial x_2} &= a_i^0 D_i^0 + a_i^0 \frac{\partial u_j'}{\partial x_j} + a_2^0 \frac{\partial u_i'}{\partial x_2} \quad (2.32)\\
\frac{\partial a_j'}{\partial t} + u_1^0 \frac{\partial a_j'}{\partial x_1} + a_1^0 \frac{\partial u_j'}{\partial x_1} + a_2^0 \frac{\partial u_j'}{\partial x_2} &= a_j^0 D_j^0 + a_j^0 \frac{\partial u_i'}{\partial x_i} + a_2^0 \frac{\partial u_j'}{\partial x_2} \quad (2.33)\\
\frac{\partial a_k'}{\partial t} + u_1^0 \frac{\partial a_k'}{\partial x_1} + a_1^0 \frac{\partial u_k'}{\partial x_1} + a_2^0 \frac{\partial u_k'}{\partial x_2} &= a_k^0 D_k^0 + a_k^0 \frac{\partial u_i'}{\partial x_i} + a_2^0 \frac{\partial u_k'}{\partial x_2} \quad (2.34)
\end{align*}

Linearised convection equations for the second fibre family can be found by replacing \( a' \) with \( b' \) in the above expressions.

Up to this point the possibility of in-plane wrinkling occurring has not been explicitly eliminated. However, as the following examination will show, coupling between out-of-plane buckling and in-plane wrinkling can be neglected, enabling these two phenomena to be studied separately. The in-plane components of the perturbation to the fibre direction vector are assumed to have the same form as \( h' \) in equation (2.25), with amplitudes \( \alpha* \) and \( \beta* \), except that a total growth rate \( G_T \) will be used in place of \( D_3^0 + G \).

Substitution of these forms into the first two perturbed convection equations (2.32) and (2.33) yields,

\begin{align*}
-\alpha* \sin \phi \left( G_T - D_1^0 \right) - u_1^* i K \cos (\phi - 0) &= 0 , \quad (2.35)\\
\alpha* \cos \phi \left( G_T - D_2^0 \right) - u_2^* i K \cos (\phi - 0) &= 0 , \quad (2.36)
\end{align*}
and similarly for the second family of fibres,
\begin{align}
\beta^* \sin \phi (G_T - D_1^0) - u_1^* i K \cos (\phi + \theta) &= 0, \\
\beta^* \cos \phi (G_T - D_2^0) - u_2^* i K \cos (\phi + \theta) &= 0.
\end{align}
(2.37, 2.38)

Comparing the above relations yields,
\[\beta^* = \pm \alpha^* \frac{\cos (\phi + \theta)}{\cos (\phi - \theta)}.\]  
(2.39)

It is therefore clear that to have any in-plane wrinkling, then either,
\[\beta^* = 0 \text{ and } \cos (\phi + \theta) = 0 \quad \text{(ie. } \theta = \frac{\pi}{2} - \phi),\]  
(2.40)

or alternatively,
\[\alpha^* = 0 \text{ and } \cos (\phi - \theta) = 0 \quad \text{(ie. } \theta = \frac{\pi}{2} + \phi).\]  
(2.41)

Thus, the in-plane wrinkle must be orthogonal to either of the two families of fibres, so that one family merely slides along its axis as shown in Figure 2.2 (b).

Since the case of interest, pure out-of-plane buckling, has no such angle restriction, the in-plane components of the perturbation to the fibre direction vector, \(\alpha'\) (\(a_1'\) and \(a_2'\)) and \(\beta'\) (\(b_1'\) and \(b_2'\)) are set to zero in the linearised governing equations. The first two linearised convection equations (2.32) and (2.33) then give,
\[a_1^0 \frac{\partial u_1'}{\partial x_1} + a_2^0 \frac{\partial u_1'}{\partial x_2} = 0,\]
\[a_1^0 \frac{\partial u_2'}{\partial x_1} + a_2^0 \frac{\partial u_2'}{\partial x_2} = 0,\]
and similarly for the second family of fibres,
\[b_1^0 \frac{\partial u_1'}{\partial x_1} + b_2^0 \frac{\partial u_1'}{\partial x_2} = 0,\]
\[b_1^0 \frac{\partial u_2'}{\partial x_1} + b_2^0 \frac{\partial u_2'}{\partial x_2} = 0.\]

From the definitions of the fibre direction vectors it is apparent that all the differential terms in the above expressions are zero so that \(u_1'\) and \(u_2'\) can only depend on \(x_3\). Furthermore, from the linearised incompressibility condition (2.31) it can be deduced that,
\[\frac{\partial u_3'}{\partial x_3} = 0,\]  
(2.42)

so that \(u_3'\) depends only on \(x_1\) and \(x_2\). Thus, \(u_3'\), \(\gamma\) and \(\delta\) are related to \(x_1\) and \(x_2\) in the same way that \(h'\) is in equation (2.25) and their initial amplitudes, \(u_3^*\), \(\gamma^*\) and \(\delta^*\) will be constants. Substituting these perturbed forms into the third convection equation (2.34) gives,
\[\gamma^* G - u_3^* i K \cos (\phi - \theta) = 0,\]  
(2.43)

and similarly for the \(b\)-direction,
\[\delta^* G - u_3^* i K \cos (\phi + \theta) = 0.\]  
(2.44)

Although different in initial appearance, equations (2.43) and (2.44) are equivalent to equation (2.23). Here, instead of relating the perturbed velocity to the amplitude of the perturbation, the velocity is related to the out-of-plane components of the perturbed fibre direction vectors, \(\gamma\) and \(\delta\),
which to first order can be interpreted as the angle of inclination of each buckled fibre. The oscillation of this angle is out of phase with the amplitude by 90 degrees, hence the presence of $i$ in the above expressions. Comparing (2.43) and (2.44) yields the relation between the two angles of inclination of,

$$
\delta^* = \gamma^* \cos(\phi + \theta)/\cos(\phi - \theta),
$$

(2.45)

which can also be found simply from geometry.

The first order perturbations to the stress components are now found by substituting perturbed variables of the form in equation (2.17) into the constitutive equation (2.8-2.10), and taking only terms of order $\varepsilon$:

$$
\begin{align*}
T'_{11} &= T_{0}a_{1}a_{1} + T'_{0}b_{1}b_{1} - p' \\
T'_{22} &= T_{0}a_{2}a_{2} + T'_{0}b_{2}b_{2} - p' \\
T'_{33} &= -p', \\
T'_{12} &= T_{0}a_{1}a_{2} + T'_{0}b_{1}b_{2}, \\
T'_{13} &= T_{0}a_{1}a_{3} + T'_{0}b_{1}b_{3} + \mu(\partial u_{1}'/\partial x_{3} + \partial u_{1}'/\partial x_{1}), \\
T'_{23} &= T_{0}a_{2}a_{3} + T'_{0}b_{2}b_{3} + \mu(\partial u_{2}'/\partial x_{3} + \partial u_{2}'/\partial x_{2}).
\end{align*}
$$

(2.46)

The perturbed form of the equation of motion (2.11) is,

$$
\partial T'_{ij}/\partial x_{j} = \rho(\partial u_{i}'/\partial t + u_{i}' \partial u_{i}'/\partial x_{j} + u_{0}' \delta u_{i}'/\partial x_{j}),
$$

(2.47)

which upon substitution of (2.46) give,

$$
\begin{align*}
\partial T'_{0}/\partial x_{i}a_{1}a_{1} + \partial T'_{0}/\partial x_{i}b_{1}b_{1} - \partial p'/\partial x_{i} + \partial T'_{0}/\partial x_{2}a_{1}a_{2} + \partial T'_{0}/\partial x_{2}b_{1}b_{2} + \mu \partial^{2}u_{1}'/\partial x_{3}^{2} = \rho \big(\partial u_{1}'/\partial t + u_{1}'D_{1}^{0} + u_{0}' \delta u_{1}'/\partial x_{3}\big)
\end{align*}
$$

(2.48)

$$
\begin{align*}
\partial T'_{0}/\partial x_{2}a_{1}a_{2} + \partial T'_{0}/\partial x_{2}b_{1}b_{2} + \partial T'_{0}/\partial x_{2}a_{2}a_{2} + \partial T'_{0}/\partial x_{2}b_{2}b_{2} - \partial p'/\partial x_{2} + \mu \partial^{2}u_{2}'/\partial x_{3}^{2} = \rho \big(\partial u_{2}'/\partial t + u_{2}'D_{2}^{0} + u_{0}' \delta u_{2}'/\partial x_{3}\big)
\end{align*}
$$

(2.49)

$$
\begin{align*}
T_{0}a_{1} \partial a_{3}'/\partial x_{1} + T_{0}b_{1} \partial b_{3}'/\partial x_{1} + \mu \partial^{2}u_{3}'/\partial x_{1}^{2} + T_{0}a_{2} \partial a_{3}'/\partial x_{2} + T_{0}b_{2} \partial b_{3}'/\partial x_{2} + \mu \partial^{2}u_{3}'/\partial x_{2}^{2} - \partial p'/\partial x_{3} = \rho \big(\partial u_{3}'/\partial t + u_{3}'D_{3}^{0} + u_{0}' \delta u_{3}'/\partial x_{3} + u_{2}' \delta u_{3}'/\partial x_{2}\big)
\end{align*}
$$

(2.50)

2.3.4 Determination of Stability Criteria

The third linearised equation of motion (2.50) has only one variable in common with equations (2.48) and (2.49), the perturbed pressure. By comparison with the other variables in equation (2.50) it can be deduced that $\partial p'/\partial x_{3}$ is constant through the thickness. To find out its actual value, the stress boundary conditions on the top and bottom surfaces will now be investigated.
Since the surface of the material is in a state of plane stress, the stress tensor in the perturbed surface coordinate system shown in Figure 2.3 can be written as,

$$T^s_{ij} = \begin{pmatrix} T_{11}^s & T_{12}^s & 0 \\ T_{12}^s & T_{22}^s & 0 \\ 0 & 0 & 0 \end{pmatrix}$$ (2.51)

where the superscript $s$ stands for surface. This tensor can be transformed into the standard reference frame using the relation,

$$T = Q^T T^s Q,$$ (2.52)

where matrix $Q$ is defined as,

$$Q = \begin{bmatrix} \cos \omega \cos \theta & \cos \omega \sin \theta & \sin \omega \\ -\sin \theta & \cos \theta & 0 \\ -\sin \omega \cos \theta & -\sin \omega \sin \theta & \cos \omega \end{bmatrix}.$$ (2.53)

The rows of $Q$ are the basis vectors of the surface coordinate system given as component triples relative to the original reference frame. If this transformation is performed, it is found that,

$$T_{33}^s = T_{11}^s \sin^2 \omega,$$ (2.54)

which to first order in $\omega$ is zero. From equation (2.46), $p'=0$ on the surfaces and considering previous findings, is seen to be zero everywhere. Hence, the out-of-plane motion equation is found to be decoupled from in-plane behaviour as governed by perturbed equations of motion (2.48) and (2.49).

Expressions for $a_3'$, $b_3'$ and $u_3'$ in the form of (2.25) can now be substituted into (2.50) to give an equation in terms of variables $\gamma^*$, $\delta^*$ and $u_3^*$. However, equations (2.43) and (2.44) relate these remaining variables together, so they may be eliminated to give an expression for the convected growth rate,

$$G = -\left(\mu K^2 / 2\rho + D_3^0\right) \pm \sqrt{\left(\mu K^2 / 2\rho + D_3^0\right)^2 - K^2 T_\theta / \rho},$$ (2.55)

where:

$$T_\theta = T_a \cos^2 (\phi - \theta) + T_b \cos^2 (\phi + \theta)$$ (2.56)
is the fibre tension reaction stresses resolved in the buckle direction. If
inertia is neglected from the equations of motion, then stability criteria
(2.55) simplifies to
\[
G = -\frac{T_0}{\mu} .
\] (2.57)

Buckling instability occurs when the convected growth rate is greater
than zero. It can be seen from both (2.55) and (2.57) that the flow becomes
unstable when the fibre tensions resolved in the direction of the buckle
become negative. Note that equations (2.16) can be used to give the growth
rate in terms of the applied stresses.

Close inspection of equation (2.55) shows that \( G \) can also become
positive if:
\[
D_3^0 < -\frac{\mu K^2}{2\rho} ,
\] (2.58)
which corresponds to rapid thinning of the sheet.

2.4 Discussion

2.4.1 Influence of Wavelength

The above analysis determined the stability of a sinusoidal
perturbation of arbitrary wavelength superimposed onto the fibre position
and other flow variables. In reality, the slight fibre misalignments present in
composite sheets are of arbitrary form, perhaps influenced by the properties
of the fibres or the manufacturing process used to produce the sheet. This
does not nullify the analysis since any arbitrary shape can be described by a
Fourier series sum of sinusoidal terms, with wavelengths following some
statistical distribution. Assuming the flow is unstable, it is expected that
those wavelengths that are most represented and which have the highest
growth rates will dominate all others and appear as macroscopic buckles.

While the distribution of misalignment wavelengths is a function of
each particular material specimen, it remains to determine from theory
which wavelengths have the highest growth rates, assuming they are all
equally represented. To do this a non-dimensional growth rate will first be
defined by dividing equation (2.55) through by the modulus of the through-
thickness strain rate, \( |D_3^0| \). Cases when \( D_3^0 \) is positive and when \( D_3^0 \) is
negative must then be looked at separately.

If \( D_3^0 \) is positive (ie. for thickening flows in which angle \( \phi \) in Figure 2.1
is moving away from 45°), equation (2.55) can be written in the form,
\[
G/|D_3^0| = -(\kappa + 1) + \sqrt{(\kappa + 1)^2 - 2\kappa \tau} ,
\] (2.59)
where:
\[
\kappa = \frac{\mu K^2}{2\rho |D_3^0|} ,
\] (2.60)
will be referred to as the non-dimensional wave number squared, while,
\[
\tau = \frac{T_0}{\mu |D_3^0|} ,
\] (2.61)
is the non-dimensional fibre tension factor. Figure 2.4 plots the non-dimensional growth rate given by equation (2.59) against the squared non-dimensional wave number for a few values of the non-dimensional fibre tension factor. When $\tau$, and thus $T_0$ is negative, the flow is unstable, leading to buckle formation. Since high wave numbers are shown to have the highest growth rates, buckles with very small wavelengths are likely to develop.

![Figure 2.4 Stability plot for thickening trellis flows ($D_3^0 > 0$).](image)

When $D_3^0$ is negative (ie. during thinning flow in which the fibres rotate towards $\pm 45^\circ$), equation (2.55) can be written as,

$$G / |D_3^0| = (1 - \kappa) + \sqrt{(1 - \kappa)^2 - 2\kappa\tau},$$

(2.62)

where $\kappa$ and $\tau$ are defined as before. Figure 2.5 is the stability graph for the thinning flow regime. As before, negative fibre tensions cause unstable flow, but wavelength effects are more complex. When $\tau < -2$, small buckle wavelengths are expected to dominate, as for thickening flows. However,
under unstable flow conditions when \( \tau > -2 \), small wave numbers and consequently long buckle wavelengths are more likely to dominate. In the region \(-2 < \tau < 0\) this is likely to lead to gross buckling of the entire sheet.

![Figure 2.5](image)

**Figure 2.5** Stability plot for thinning trellis flows \( (D_3^0 < 0) \).

In those regions of Figure 2.5 in which inequality (2.58) is satisfied, positive growth rates exist despite the presence of tension in the fibres. However, it is noted in this case that wavelengths as large or larger than the part itself will dominate, meaning out-of-plane buckling will not be observed.

In some of the cases mentioned above, perturbations with very tiny wavelengths had the highest growth rates and were thus expected to grow into buckles. In real fibre-reinforced thermoplastics a limiting factor on how small buckles may become is the flexural stiffness of the fibres, neglected in this analysis.

### 2.4.2 Practical Consequences

The most important finding from this work is the dependence of buckling instability on the tension reaction stresses in the fibres. From
equation (2.56) a buckle is expected to grow at angle $\theta$ along which the reaction stress from the tensions is most negative. It is also possible for a second buckle to develop at right angles to the first so that together they appear as a blister on the sheet. The same stability criteria can be applied to the growth of this second buckle.

The ultimate goal of this study is to find ways to eliminate out-of-plane buckling defects in thermoformed parts. The results demonstrate that the only way to achieve this goal is to find some way to reduce or eliminate negative tensions in the fibres. The use of clamping, either directly on the edges of the blank [5,12] or around the diaphragms used to form the part [6], serves to superimpose tensile stresses on to the material, resulting in reduced buckling. Another successful approach to reducing buckling has been to minimise the size of unformed or unnecessary areas of the blank [6,15], which especially in deep-drawing operations can lead to considerable negative hoop stresses in flange areas. Mallon et al. [6] amongst others have demonstrated the benefit of using slower forming speeds. Since the stresses in a viscous fluid are proportional to the forming speed, all stresses, compressive or otherwise will be lowered. Not only are defect growth rates lower at slow forming speeds, in the real material the small degree of flexural rigidity in the fibres means they have some additional resistance to buckling not accounted for by this model.

The final points of discussion concern the validity of applying findings from both the ideal fibre-reinforced fluid model and the homogenous deformations studied here to actual forming operations on real materials. To the first point it is noted that in treating the matrix as a fluid, the model is more valid in the high temperature (or alternatively low forming speed) region of the processing windows of fibre-reinforced thermoplastics. Molten CFRTPs are viscoelastic materials, and at lower forming temperatures the elastic component of their response is expected to be more significant. This is likely to have a stabilising effect on their buckling behaviour.

In typical forming operations it is noted that pure, homogeneous trellis flows will seldom be observed. Either the deformation will not be homogeneous, or other flow processes such as inter-ply slip will take place simultaneously. However, there will exist areas on the sheet over which trellis flows are dominant and the deformation more-or-less homogeneous, meaning the results of this study are still applicable.
Chapter 3: Strain Measurement in Sheet Forming

3.1 Grid Strain Analysis

3.1.1 Introduction

In order to make rational decisions on how to improve sheet forming operations, fabricators need to visualise and understand the deformations taking place during them. Unfortunately this knowledge can seldom be gained merely by observing the shape of the formed part. Thickness variations are measurable and interesting, but alone do not describe the overall deformation since appreciable in-plane strains may take place without changing the sheet thickness, as is the case in pure drawing.

In forming fibre-reinforced materials the final fibre orientations give some clue to the deformations that have taken place. This is especially true with certain woven fabric composites for which such information is sufficient for determining overall forming strains. However, in the less constrained case of laminates made from unidirectional plies, the fibre orientations are much less useful since they give no measure of in-plane shear and transverse flow.

To gain a greater insight into sheet forming processes, the so-called Grid Strain Analysis (GSA) techniques [24-28] have been devised. These involve printing or etching a regular grid pattern onto the undeformed blank, forming it into its final shape, and measuring the deformed grid point positions. The undeformed and deformed grid points then describe the kinematics of the deformation, allowing forming strains to be determined in the part. An obvious requirement of the techniques is that the grid density be sufficient to describe the deformation of interest, otherwise details in the part will be lost.

GSA is principally used to determine in-plane strains on the surface of sheets. If the deformation is volume preserving, thickness changes in the sheet can be calculated from the in-plane strains. Similarly, knowledge of the curvature can also be used to determine the variation of strain through the thickness of the sheet in cases where through-thickness shear is negligible. With CFRTP laminates, GSA provides an understanding of the deformation undergone by the surface ply only, but will generally give an idea of how other plies influence its behaviour.

This chapter introduces a new approach to Grid Strain Analysis which involves fitting a surface mesh to the deformed grid points. First, however, a short review of the existing technique is given, followed by a brief discussion of some of the applications of GSA.
3.1.2 Current Techniques

The techniques for strain calculation presented in this section were first presented by Sowerby et al. [24]. References [25-27] outline and expand on the theory.

Figure 3.1 illustrates the displacement of three grid points from their initial positions $O, A$ and $B$ in the x-y plane to deformed positions $O', A'$ and $B'$, which could lie anywhere in space.

![Figure 3.1](image1)

**Figure 3.1** Deformation of a triangle described by three adjacent grid points.

It is assumed that curvatures are small and that strains are more-or-less homogeneous over the surface between points $O', A'$ and $B'$. Thus, although this surface can be curved, it is approximated by a plane indicated by the triangle in the above figure. The deformation of triangle $OAB$ into $O'A'B'$ is therefore considered planar and simple coordinate transformations are used to superimpose the two triangles on a single two dimensional coordinate system with both $O$ and $O'$ at the origin. Figure 3.2 shows this situation.

![Figure 3.2](image2)

**Figure 3.2** Superimposed triangular grid elements from before ($OAB$) and after ($O'A'B'$) deformation.

The coordinates of points $A$ and $B$ in Figure 3.2 are put into matrix form so that,

\[
X = \begin{pmatrix}
X_A & X_B \\
Y_A & Y_B
\end{pmatrix},
\]

(3.1)
where capital letters are used to emphasise that these are material coordinates. The deformed spatial coordinates of \( A' \) and \( B' \) are similarly placed into matrix,

\[
x = \begin{pmatrix} x_{A'} & x_{B'} \\ y_{A'} & y_{B'} \end{pmatrix}.
\] (3.2)

Since the deformation is assumed to be homogeneous over the triangle it follows that,

\[
x = FX,
\] (3.3)

where the unknown deformation gradient tensor is defined by [29],

\[
F = \begin{pmatrix} \frac{\partial x}{\partial X} & \frac{\partial x}{\partial Y} \\ \frac{\partial y}{\partial X} & \frac{\partial y}{\partial Y} \end{pmatrix}.
\] (3.4)

It is simple to rearrange equation (3.3) to obtain,

\[
F = xX^{-1}.
\] (3.5)

The deformation gradient tensor fully defines the deformation at a point. However, it is not easy to calculate strains directly from it since it also includes effects due to rigid body rotations. Instead, it is premultiplied by its transpose (denoted by the superscript \( t \)) to obtain the right Cauchy-Green deformation tensor [29],

\[
C = F'F.
\] (3.6)

Note that in this case \( C \) is a two-by-two matrix. Its eigenvalues are \( \lambda_1^2 \) and \( \lambda_2^2 \), the squares of the principal stretches in the plane of the material, and the corresponding eigenvectors are the principal directions of strain in the undeformed state. The desired principal strains can easily be calculated from these principal stretches, a topic left to Section 3.2.4.

The procedure outlined here can be used to calculate strains in any triangular region marked out by three grid points. It is clear that an ambiguity may arise as to which points should be used together in strain calculations, especially when the grid is non-regular. An algorithm that finds triangles that are as close as possible to equilateral, and using the nearest points could be devised to overcome such difficulties. In any case, unless some ingenious averaging scheme is employed, the method will provide measurements of strain at a finite number of locations on the part, usually assumed to be at the centre of the triangles.

A proven method for visualising surface strains are Arrow Diagrams. As shown in Figure 3.3, orthogonal pairs of two-headed arrows are drawn in the directions of principal strain, their lengths being proportional to the principal strain magnitudes. Tensile strains are drawn with arrowheads pointing away from where the strain is calculated, while compressive strains point inwards. Arrow diagrams can show the strain vectors on either the undeformed or the deformed
geometry. In the latter case it can be helpful to plot the strains on a backdrop of the deformed surface produced by connecting the grid points up to form a net. Section 3.3 presents several such arrow diagrams.

![Arrow Diagrams](image)

(a) Tension. (b) Biaxial tension. (c) Pure drawing.

**Figure 3.3** Principal strains as depicted in Arrow Diagrams.

Unfortunately, no single plot can display all information about the deformation. Arrow diagrams show strains only at discrete locations, which in this case is no disadvantage. Contour plots can be used to show single valued variables over the part surface, such as sheet thickness.

Much of the validity and usefulness of the results of Grid Strain Analysis derive from the assumption that the deformation is more-or-less monotonic, whereby the material strains smoothly into its deformed shape without undergoing complex load reversals. In general, this assumption is good for sheet forming, but in processes such as deep drawing large areas of the blank will not deform monotonically, especially in flange areas that undergo pure drawing followed by bending and unbending.

Zhang and Duncan [25] have recently made some improvement to the above technique by fitting splines between neighbouring points rather than linear sided triangles.

### 3.1.3 Applications

Applications of Grid Strain Analysis range from experimental forming studies to quality control on the production line. In any sheet forming problem an accurately generated arrow diagram will clearly show the deformations that have occurred allowing quantitative and qualitative comparison between parts formed under different conditions as well as with numerical models of the process.

Three dimensional sheet forming of CFRTP laminates proceeds through a number of flow processes, as described in Section 1.2.1, and with the possible exception of tight fibre cloth reinforced thermoplastics, any number of deformations and subsequent fibre arrangements could produce the same deformed shape. As the examples of CFRTP sheet forming presented in Section 3.3 demonstrate, Grid Strain Analysis can be used to decipher which flow processes dominate and the effects fibres in other plies in the laminate have on the deformation of the surface ply.
Grid Strain Analysis is even more useful in sheet metal forming. Given the assumption that the deformation is monotonic, GSA can aid the choice of modifications to tooling and clamping so as to reduce defects such as wrinkling and fracture. Knowledge of compressive strains in the vicinity of wrinkled areas could be applied to the placement of draw beads and other devices that induce tension in the sheet. Where fracture is a problem, it can aid the intelligent use of tool lubrication. In all cases the calculated strains can be plotted directly on the Forming Limit Diagram [30] for the material, to show how close the part nears the limit of formability of the sheet, possibly leading to the selection of a cheaper or more suitable metal for the part.

On the production line, if gridded blanks are regularly put through the forming machines and strain analysis performed, the quantitative strain data can be used as a continuous check on part quality and machine calibration.

3.2 Strain Analysis by Surface Fitting with the FEM

An alternative to the approach of Section 3.1.2 is to use the grid data to fit continuous, parametric representations of the undeformed and deformed surfaces. Strains could then be calculated at any point on the surface by finding the displacement gradient mapping the undeformed geometry to the deformed. An immediate improvement is gained with this approach if higher order surface patches are used. No longer are we limited to assuming homogeneous deformation between grid points since higher order functions are able to accurately describe a smoothly deformed sheet.

The approach starts by assuming a form for the undeformed surface. Since the undeformed grid points lie in a single plane, this just involves the placement of a planar finite element mesh over the points. The core of the process is in finding an equivalent deformed mesh, with identical numbers of elements and topology, in which the deformed data points occupy as nearly as possible the same locations relative to the elements as they did in the undeformed state. For this task a finite element least-squares surface fitting scheme is used.

The idea of using the surface fitting technique for grid strain analysis [31] has been developed by the author into a dedicated strain analysis package called GSA. The program, written for the Microsoft® Windows environment, allows interactive mesh design and post-processing, and was used to generate the figures in Section 3.3. Appendix A provides a guide to the use of GSA, while the following sections describe the surface fitting procedure it employs.
3.2.1 Elements for Representing Smooth Surfaces

To represent a surface accurately an element is required that can handle the curvature and continuity requirements while remaining computationally efficient and easy to use. Linear triangle and bilinear quadrilateral elements are therefore unsuitable, but several higher order elements are good candidates. Triangular and rectangular cubic elements seem especially good since they can represent very complex surfaces. A bicubic Lagrangian element can be visualised as a topologically rectangular patch passing through an array of 4-by-4 nodes in space. The drawback of using these elements is that when assembled into a large mesh only edge nodes are shared on neighbouring elements, resulting in a very large total number of nodes. As will be shown later, surface fitting can only be done if the number of data points exceeds the number of nodes in the mesh. Also, such elements display only \( C_0 \)-continuity of displacement across element boundaries, meaning sharp edges may result on the fitted surface.

A better choice, and one used exclusively in this work, is the bicubic Hermite element. This element, depicted in Figure 3.4, is topologically rectangular and consists of four corner nodes. To construct a full bicubic basis over the element, four vectors are stored at the nodes: the nodal position, the slopes of the two element sides leaving the node and the ‘twist’ vector controlling surface behaviour inside the element near the node.

![Figure 3.4 Bicubic Hermite element showing the four nodal vector quantities.](image-url)

Bicubic Hermite elements offer considerable advantages when joined to form a finite element mesh. Neighbouring elements share all the properties of their common nodes meaning that assemblies of such elements exhibit \( C_1 \)-continuity of displacement. This extra degree of surface smoothness means strains are continuous across element sides. That so many properties are shared by neighbouring elements leads to a second advantage, that the addition of extra elements adds far fewer extra degrees of freedom to the system than for bicubic Lagrangian elements. In fact, in an infinite 2-D array of such elements, the bicubic Hermite form needs only
four degrees of freedom per element per coordinate compared with double that number for the Lagrangian equivalent.

As shown in Figure 3.4, with four vector quantities at each node and four nodes per element, sixteen basis functions, $\phi_n$, are necessary for their interpolation so that the position $x$, at surface coordinate $(\xi_1, \xi_2)$, is given by,

$$ x = \sum_{n=1}^{16} u_n \phi_n(\xi_1, \xi_2), $$

(3.7)

where $u_n$ are the 16 nodal vectors. The basis functions for such surface elements are found by multiplying the one-dimensional cubic Hermite basis functions plotted in Figure 3.5 for each $\xi$-direction.

**Figure 3.5** One dimensional cubic Hermite basis functions.[32,33]

The basis functions plotted in Figure 3.5 describe a cubic curve such as any of the element sides in Figure 3.4. Functions $\phi_1$ and $\phi_3$ have zero slope and unit value at their respective curve ends and thus interpolate the positions of the end nodes. Functions $\phi_2$ and $\phi_4$ have zero values but unit slopes at their respective curve ends and are used to interpolate the nodal slope vectors mentioned above. The formulae for these 1-D basis functions are as follows [32,33]:

$$ \phi_1 = \left(\xi^3 - 3\xi^2 + 2\right)/4 $$

(3.8)

$$ \phi_2 = \left(\xi^3 - \xi^2 - \xi + 1\right)/4 $$

(3.9)

$$ \phi_3 = \left(-\xi^3 + 3\xi^2 + 2\right)/4 $$

(3.10)

$$ \phi_4 = \left(\xi^3 + \xi^2 - \xi - 1\right)/4 $$

(3.11)

Once nodal quantities for the surface element are known, equation (3.7) can be used to calculate the position of points on the surface. Moreover, several other useful surface quantities can be similarly computed, for example the derivatives of $x$ with respect to each of the surface coordinates, used in the calculation of strain in Section 3.2.4, are found using,

$$ \frac{\partial x}{\partial \xi_i} = \sum_{n=1}^{16} u_n \frac{\partial \phi_n}{\partial \xi_i}(\xi_1, \xi_2) $$

(3.12)

The element interpolation presented here is suitable for single elements, but may cause problems in element assemblies, especially when neighbouring elements are of different size. The reason is that the performance of Hermite elements is best when they are undistorted so that
derivatives from equation (3.12) are fairly equal over the entire element, and deteriorates the more this is violated. Neighbouring elements of different size will be quite distorted if they share the value $\partial \hat{x}/\partial \xi_i$ across the common boundary. This distortion may be alleviated by storing a different nodal slope, equal to the derivative of $x$ with respect to the arc length, $S$. Quantity $\partial \hat{x}/\partial S_i$ will initially be a unit vector, but when fitted to the deformed surface its length will change. This modification requires storage of the initial values of $\partial S_i/\partial \xi_i$ along the four element sides intersecting at each node. Interpolation equations (3.7) and (3.12) are still valid in each element, provided the vectors $u_i$ representing nodal slopes and twists are replaced by the actual nodal values ($\partial \hat{x}/\partial S_i$) multiplied by the appropriate $\partial S_i/\partial \xi_i$ factors for that element. This slight modification comes at the cost of some loss of continuity. Interelement slope continuity is preserved, however, strains calculated on either side of an element boundary will be subtly different unless the nodal constants $\partial S_i/\partial \xi_i$ are identical across the boundary.

A final note about bicubic Hermite elements concerns the range of geometries that assemblies of them may take up. A first guess of this range would be any distorted two dimensional array of $N$ by $P$ elements, with some elements possibly missing. However, more possibilities exist since a valid assembly requires only that surface coordinates $\xi_1$ and $\xi_2$ line up across neighbouring elements. Consideration of the more general assemblies this rule allows leads to the observation that often three-sided ‘holes’ will exist in the mesh which ordinary bicubic Hermite elements cannot fill. The solution is a degenerate form of the element in which two nodes are repeated, or rather, one side is reduced to a point. Figure 3.6 depicts such an element.

![Figure 3.6 Degenerate 3-noded bicubic Hermite element.](image)

Along reduced side 1-2 of the degenerate element the nodal quantities $\partial S_i/\partial \xi_1$ are zero. As a result, continuity across sides 2-3 and 4-1 is reduced, especially near the repeated node. Furthermore, since several of the degrees of freedom in the shared node are ignored by the degenerate element, the resulting finite element mesh will be singular unless the shared node is also used as a ‘normal’ corner in another element. In any case, no changes to the interpolation functions are necessary for these variants on the standard elements.
Despite the loss of continuity across the degenerate element, it is useful in allowing meshes of more arbitrary geometry to be defined. It is most effective for inserting extra rows of elements in the mesh where more degrees of freedom are required. In the long term, however, the surface fitting would benefit if a $C_1$-continuous triangular element could be found. That would remove any restrictions on the geometry and internal detail of the mesh.

### 3.2.2 Input and Preprocessing

The data requirements for Grid Strain Analysis by Surface Fitting differ little from those for the existing technique outlined in Section 3.1.2. Essentially, all that is required is a sufficient number of data points with $x, y$ locations before deformation and $x, y$ and generally $z$ locations afterwards. In both cases the use of a uniform grid will reduce the requirements to just the deformed positions. The new technique, however, requires no knowledge of the connectivity between the points; they may be entered in any order as long as the one-to-one correspondence between the points in the undeformed and deformed state is maintained. This is only a small advantage as a procedure could be devised to work out this connectivity automatically. In the surface fitting problem, the grid point locations are simply treated as data. Increasing their density at certain locations by no means influences the way the process works or how the results are displayed but may allow more elements to be placed there, thus giving a more accurate solution.

With either approach to Grid Strain Analysis, the major bottleneck still lies with the problem of accurately digitising the positions of hundreds of grid points from both the deformed and the undeformed geometry. To date, manual methods involving measurements from a three axis table [28] have provided good accuracy but at considerable operator expense. Attempts have been made to automate the process using photogrammetry [27,34], the image processing of two or more camera views of the deformed, gridded part.

After input of grid point data, preprocessing mainly involves laying out the undeformed element mesh over the undeformed grid points. This is in most cases a necessarily manual task, but with interactive computer graphics it can be made relatively untaxing. Key points to consider when laying out the undeformed mesh are as follows:

- To avoid a singular system matrix, the number of degrees of freedom in the mesh should not exceed those of the data points, either locally or over the entire mesh. In practise there should be a generous excess of data points to better define the surface and to reduce error.

- The outer edges of the mesh should lie just outside the range of data points, otherwise their deformed positions will be insufficiently defined by the data.
• Internal element sides should follow surface features such as bends. Higher element concentrations are also desirable in such places, otherwise the fitting process will smooth out the bend and errors will be large. An obvious rule is that the elements in the undeformed mesh should be able to take up the geometry of the deformed data points to the desired accuracy.

• As mentioned in Section 3.2.1, the undeformed elements should be as undistorted as possible. A real meshing example using some of these pointers is given in Section 3.3.1.

The need to manually provide an undeformed mesh is one of the drawbacks of this method when compared with the established grid strain analysis approach in Section 3.1.2. It can be partially automated using element subdivision and by getting the computer to calculate the nodal slopes that give a ‘relaxed’ mesh with elements as undistorted as possible. However, it is not a serious deficiency since a mesh need only be produced once for each blank/tool combination.

When the mesh is complete, the final preprocessing task involves finding the surface coordinate $\xi^{(d)}$ of each undeformed data point and the element they are inside. In the general case of a distorted element, as in Figure 3.7, this requires an iterative process.

![Figure 3.7 Finding the surface coordinate $\xi^{(d)}$ of data point $d$ in an element on the undeformed mesh.](image)

The rough description of the fitting process given at the start of Section 3.2 can now be made clearer. The aim of the fitting process is to find a deformed version of the initial mesh in which the locations referenced by the $\xi^{(d)}$ coordinates are as near as possible to the actual deformed data point positions.

3.2.3 Solution

It is now desirable to formulate the problem into a system of equations,

$$Ku = f,$$  \hspace{1cm} (3.13)
where $K$ is the global stiffness matrix, $f$ a force vector, and $u$ is a vector of the unknown deformed nodal parameters.

A weighted, squared sum of the error between the fitted surface approximation and the deformed data point coordinates is formulated,

$$E = \sum_{d=1}^{D} w^{(d)} \left( \hat{u}(\xi^{(d)}) - u^{(d)} \right)^2,$$

(3.14)

where $D$ is the number of data points and $w^{(d)}$ are optional weighting factors for each data point, a higher value placing more confidence in that measured coordinate so that the fitted surface approaches it more closely. In the above equation, $u^{(d)}$ stands for any of the deformed coordinates $x, y$ and $z$, which are fitted independently. The approximation to the deformed surface, $\hat{u} = (\hat{x}, \hat{y}, \hat{z})$, is the product of the basis functions and the unknown deformed nodal quantities as given by equation (3.7) so that (3.14) becomes,

$$E = \sum_{d=1}^{D} w^{(d)} \left( \phi_n(\xi^{(d)}) u_n - u^{(d)} \right)^2.$$

(3.15)

The least squares approximation is now found by differentiating (3.15) with respect to each $u_n$ and equating to zero, which gives,

$$\frac{\partial E}{\partial u_m} = 2 \sum_{d=1}^{D} w^{(d)} \left( \phi_n(\xi^{(d)}) u_n - u^{(d)} \right) \phi_m(\xi^{(d)}) = 0,$$

(3.16)

so that,

$$\left[ \sum_{d=1}^{D} w^{(d)} \phi_m(\xi^{(d)}) \phi_n(\xi^{(d)}) \right] u_n = \sum_{d=1}^{D} w^{(d)} u^{(d)} \phi_m(\xi^{(d)}).$$

(3.17)

The portion of equation (3.17) in square brackets is the global system matrix, the right hand side is the force vector which depends on the deformed coordinate data, and $u_n$ is the vector of unknown deformed surface parameters. Provided the weighting is identical for each of the coordinates $x, y$ and $z$, the same system matrix may be used to solve for all three, reducing computing time by almost two-thirds.

Solution of the system of equations results in a deformed surface representation which can be compared with the undeformed mesh to determine the strains at any point, a topic addressed in the following section. First, however, it is necessary to consider the errors involved in the fitting process.

The above procedure finds a solution that minimises the sum of the squared error, the error defined as the distance between the interpolated and actual deformed grid point positions. A slightly better fit may be obtained if the sum of the error norm rather than the squared error is
minimised. This, however, is no easy task and the improvement in fitting accuracy is not likely to warrant the extra cost. In any case the ‘true’ error can easily be calculated and visualised, and the weighting values or undeformed mesh adjusted on an iterative or trial-and-error basis to improve the fit.

Unfortunately, failure in correctly fitting the surface cannot merely be measured in terms of such error values at the data points. Visualising the deformed surface will occasionally show a surface that passes close to or through the data points but which exhibits severe oscillations. This occurs in areas where the mesh has a large number of degrees of freedom but relatively few data points so that there may not be sufficient information to tie down the solution. As mentioned in Section 3.2.2, a degree of freedom count is necessary to avoid a singular system matrix. Even when there are sufficient data points, small fluctuations in their positions can cause large oscillations to appear in the mesh. Decreasing the element density and increasing the number of data points both alleviate this problem, but another approach may also be used.

The problem comes down to a lack of data to adequately define a surface, just as three points cannot uniquely describe a cubic curve. A solution is to include additional terms into equation (3.14) that limit the amount of stretching and curvature that may occur, so that a new summed error $E$ is given by,

$$E = E + \int_A \left[ \alpha \left( \frac{\partial \hat{u}}{\partial S_1} \right)^2 + \left( \frac{\partial \hat{u}}{\partial S_2} \right)^2 \right] + \beta \left( \frac{\partial^2 \hat{u}}{\partial S_1 S_1} \right)^2 + 2 \left( \frac{\partial^2 \hat{u}}{\partial S_1 S_2} \right)^2 + \left( \frac{\partial^2 \hat{u}}{\partial S_2 S_2} \right)^2 \right] dA, \quad (3.18)$$

where integration is over the element area. A higher value of $\alpha$ will tend to limit stretching in the surface, whereas higher $\beta$ values limit the amount of curvature in the deformed surface. It is thus no coincidence that the bracketed quantity multiplied by $\beta$ in equation (3.18) is very similar in appearance to the governing equation for plate bending [35]. Generally, $\beta$ is more useful since the oscillations mentioned above involve high curvatures. In practice only small values of $\alpha$ and $\beta$ are needed to make a vast improvement in the appearance of the solution with little change in surface fitting error, since their influence is marked only in areas where there is currently little data to better define a surface.

### 3.2.4 Postprocessing

In the previous two sections, the procedures for producing continuous surface representations for the undeformed and deformed states have been given. Next it is necessary to calculate the forming strains at a point $\xi$ in some element on the surface.
Firstly, the two derivatives $\frac{\partial X_j}{\partial \xi_k}$ on the undeformed surface and the pair $\frac{\partial x_i}{\partial \xi_k}$ on the deformed surface are calculated at $\xi$ using equation (3.12). The two vectors $\frac{\partial X_j}{\partial \xi_k}$ can be placed into a 2-by-2 matrix and inverted to obtain $\frac{\partial \xi_k}{\partial X_j}$. Hence, the components of the 3-by-2 deformation gradient matrix at $\xi$ can be calculated from,

$$F_{ij} = \frac{\partial x_i}{\partial X_j} = \frac{\partial x_i}{\partial \xi_1} \frac{\partial \xi_1}{\partial X_j} + \frac{\partial x_i}{\partial \xi_2} \frac{\partial \xi_2}{\partial X_j}. \quad (3.19)$$

As was done in Section 3.1.2, the deformation gradient is premultiplied by its transpose to produce the 2-by-2 right Cauchy-Green deformation tensor,

$$C = F'F. \quad (3.20)$$

An eigenvalue analysis is then performed to find the strains. From the Principal Axes Theorem, matrix $C$ can be expanded to,

$$C = V' \Lambda V, \quad (3.21)$$

where,

$$\Lambda = \begin{bmatrix} \lambda_1^2 \\ \lambda_2^2 \end{bmatrix}, \quad (3.22)$$

is a matrix containing the eigenvalues of $C$ on the main diagonal, and $V$ is a square matrix containing the eigenvectors of $C$ as columns. The eigenvectors are the principal directions of strain. These vectors are orthogonal and point in directions on the undeformed surface along which purely extensional deformation and no shearing has taken place. The square roots of the above eigenvalues, $\lambda_1$ and $\lambda_2$, are the principal stretches and measure the ratios of the deformed to undeformed length of a differential element of the surface along the corresponding principal directions. From these stretches, several commonly used strains can be calculated:

- Engineering (small) strains are linearly related to the stretches according to,

$$\varepsilon_i = \lambda_i - 1. \quad (3.23)$$

- Natural strains are often used in sheet metal applications and are calculated using,

$$e_i = \ln(\lambda_i). \quad (3.24)$$

- Lagrangian or Green’s strains are frequently used in large strain elasticity problems, and are given in terms of the eigenvalues of $C$ by,

$$E_i = \frac{1}{2} \left( \lambda_i^2 - 1 \right). \quad (3.25)$$
In cases where the material is incompressible, the thickness strains can be calculated from the through-thickness stretch, $\lambda_3$, which is related to the principal in-plane stretches by the incompressibility condition,
\[ \lambda_1 \lambda_2 \lambda_3 = 1. \tag{3.26} \]

Note that up to now all principal directions have been relative to the undeformed surface. If the principal directions are written as vector sums of the two $\xi$-curves, it becomes a simple matter to convert them to equivalent directions on the deformed surface.

Data visualisation is central to postprocessing, and the use of interactive computer graphics is essential to get the most out of the analysis. Since it is now possible to calculate strains anywhere on the surface, much more freedom is available in the choice of how they are displayed. However, one is still faced with the problem that not all known quantities can be visualised simultaneously, and the arrow diagrams introduced in Section 3.1.2 are again the best approach. The following section includes several examples of arrow diagrams.

Since a complete mapping exists from every point on the undeformed surface to its deformed equivalent, there is vast scope for improving the understanding of the results, for example by viewing the deformed shape of what was a regular pattern on the undeformed surface. In Section 3.3 this idea is used to show the deformed positions of initially straight, parallel fibres.

### 3.3 GSA Example: Composite Blister Fairing

In this section strain analysis using the above theory is carried out on an illustrative example, that of an aerodynamic fairing thermoformed from a laminate of PLYTRON® continuous glass fibre reinforced polypropylene. The fairing was pressure formed using double diaphragms made of silicone rubber; specific details of its manufacture are given in Reference [8]. The flanged, half tear-drop shape of the part is evident from the pictures later in this section. Two examples are given, both 2 mm, 4-ply bidirectional laminates differing only in the fibre orientation relative to the axis of the tool.

A few practical difficulties present themselves when attempting GSA on fibre-reinforced thermoplastic laminates. Since the material is molten or semi-molten during forming, it is possible for the thin surface layer of matrix on which the grids are printed to be smeared by the diaphragms. This not only makes digitising difficult, but may mean the grid points no longer represent the bulk deformation of the surface ply. The specimens used in the following sections, however, did not display any evidence of smearing. Another temperature-related problem is the
danger of grid points convecting away from their original positions during the heating phase of the thermoforming process, which would add to the fitting error.

A further difficulty is related to the occurrence of discontinuous deformations in the sheet, a topic more fully discussed in Section 4.1.1. Tow splitting and discontinuous in-plane shearing are frequently observed in thermoforming of fibre-reinforced thermoplastics. On the deformed grid such behaviour is evidenced by sudden jumps or steps between grid points. If the mesh density is low in such areas, the surface fitting technique will tend to smooth out these discontinuities, giving a picture of the average deformation. Furthermore, these areas will be marked by high fitting errors. With a higher mesh density the discontinuities will become evident during the analysis from sudden strain gradients on the part.

3.3.1 \([0,90]_s\) Laminate

A first view of the blister fairing can be seen from Figure 3.8 which plots the positions of the deformed grid points suspended over the flat, regular grid measured off the blank from which it is formed.

![Figure 3.8](image)

**Figure 3.8** Deformed over undeformed grid points for \([0,90]_s\) blister fairing.

The first step in GSA is to lay out a planar mesh over the undeformed grid points. As the deformed points in Figure 3.8 illustrate, there are effectively three areas on the blank to consider when meshing: the flange area which remains approximately flat after deformation, the curved blister area, and the narrow bend region between them. Since the curvature in the bend region is high, smaller elements must be placed there, otherwise the sharp bend will be smoothed out on
the deformed surface. Experimentation with the data shows that the deformation in the flange and blister areas is relatively smooth, permitting the use of fewer elements in these regions. Figure 3.9 shows the undeformed element mesh used in this example. Note that the mesh incorporates one degenerate bicubic Hermite element to allow the element density to vary over the mesh. Also, it took some trial and error to position the band of elements that follow the bend.

Figure 3.9 Undeformed grid points and mesh for the [0,90], blister fairing.
Note from the above figure that only two thirds of the width of the fairing has been digitised and meshed. Nevertheless, the omitted left side should deform identically to the right due to the symmetry of both the part and the laminate lay-up.

Figure 3.10 plots the deformed mesh computed by the GSA program. The shape of the fairing can now be clearly seen. In this example the average error, or distance between the fitted and actual deformed grid points, is 0.253 mm, with a maximum of 0.877 mm. The initial grid spacing was 5.5 mm. The error seems rather large, but observation of its distribution over the part shows that it is high in just two or three places. One such place is on the front of the blister where tow splitting is indicated in Figure 3.8.

![Figure 3.10 Elements on the fitted deformed mesh for the [0,90], blister fairing.](image)

With the deformed surface fitted the results may be presented in several different ways, as Figures 3.11 to 3.14 demonstrate. In Figure 3.11, the principal strains are plotted on the undeformed surface between a grid formed by the surface fibres and the fibres in the ply immediately underneath. Arrows representing strains are scaled so that a 25 percent strain just touches the nearest fibre.

Figure 3.12 plots the same results on the deformed surface, reorienting the strain arrows of the previous diagram. The undeformed fibres are now projected onto the deformed surface with the solid curves being a fairly good representation of where the $0^\circ$ fibres have ended up. The dotted curves represent the fibres in the second ply, and should be interpreted with caution. While they are likely to be a good representation of the final $90^\circ$ ply fibre orientations, inter-ply slip and rotation are likely to have caused some displacement in the real part, not shown in these results. Note also that the tow splitting indicated in Figure 3.8 has resulted in oscillations in the strain values measured in that area.
Figure 3.11 Arrow diagram of principal Lagrangian strains for the [0,90], blister fairing, plotted on the undeformed mesh. Thin solid lines represent the $0^\circ$ surface fibre orientation. Dotted lines show the $90^\circ$ fibre orientation of the layer below.

In this example GSA is primarily a tool for understanding the deformations undergone by the laminate. From the arrow diagrams it is clear that two deformation processes dominate the
forming, each over a separate region of the part. The so-called trellis flow mode, described in Chapter 2, is the main process occurring in and around the bend area. In this process the whole laminate deforms as if it were an inextensible net, with extension in the direction bisecting the fibres and contraction in the transverse direction, so that what was initially a square is now a rhombus.

Figure 3.12 Arrow diagram of principal Lagrangian strains on deformed surface for the [0,90], fairing. Thin solid curves are deformed surface fibres. Dotted lines are probable positions of deformed fibres on the second ply.

Over the dome section of the fairing transverse flow is occurring, as indicated in the arrow diagrams by extension transverse to the surface fibres. Transverse flow causes undesirable ply thinning, also indicated in Figure 3.13 which plots contours of percentage thickness change over the ply. In this example, transverse flow is caused by a combination of the normal pressure applied by the diaphragms and friction against the diaphragms as they stretch to form the part. Laminate thinning has also been observed and measured over hemispherical domes produced under similar conditions [8,11].

Another way of displaying the results is to plot the principal strain pairs on an x-y chart. Figure 3.14 presents such a plot for the [0,90], blister fairing example. Note that it assumes \( E_1 > E_2 \), so all the points could equally well be reflected in the line \( E_1 = E_2 \). Although it is not especially clear from the diagram, there appear to be two clusters of strain points, those around \( E_2 \approx 0, E_1 > 0 \) corresponding to transverse flow, and those near the line \( E_2 = -E_1 \) corresponding to trellis flow.
Figure 3.13 Contours of percentage thickness change for the $[0,90]_s$ fairing.
If grid strain analysis is performed on sheet metal components the type of plot shown in Figure 3.14 is especially useful, since Forming Limit Diagrams [30] for various metals are of identical form. Plotting the results of the strain analysis directly on the FLD would be a great aid to tool design and material selection. For such applications the natural strains from Section 3.2.4 are generally used.

3.3.2 [±45], Laminate

The second forming example differs from that of the previous section only in the orientation of the laminate relative to the tool. Figure 3.15 shows an arrow diagram for this example plotted on the deformed surface. As before, surface fibres are shown as solid curves, and projected second ply fibres are dotted. In this example the same deformation processes, transverse flow and trellis flow again dominate, albeit in different positions and orientations relative to the axis of the fairing.
Further examples of grid strain analysis using surface fitting are given in Reference [36].

3.4 Further Uses

Using the surface fitting technique for performing grid strain analysis has already shown advantages in analysing smooth, but inhomogeneous deformations, as well as in post-processing potential. Some of its uses, however, go beyond the field of strain analysis. For example, Hunter [31] has used surface fitting of deformed grids measured from biaxial test specimens to obtain parameters for a constitutive law for heart tissue.

Having a full surface representation of the deformed part has further advantages. It could conceivably be input to subsequent finite element analyses to determine the behaviour of the structure under its intended loading. In anisotropic materials such as CFRTPs, the anisotropy of the component may be input directly with the deformed mesh - normally a complex procedure in such analyses.
Chapter 4: A Finite Element Model of a Molten Laminate

4.1 Numerical Modelling of Thermoforming

Numerical Solution Techniques such as the Finite Element Method provide an accepted and systematic approach to applying theories of material behaviour over continua of arbitrary geometry. As such, a numerical model of a molten composite could be developed simply by applying an appropriate constitutive equation, such as that of the Ideal Fibre-Reinforced Fluid in Chapter 2, along with suitable elements, to the geometry under study. The model should reproduce all the current analytical findings and allow more complex geometries and loading situations to be considered.

Up to this point no doubts have been raised on whether deformations of fibre-reinforced thermoplastics are to any extent continuous, a precondition for the use of most numerical methods. Considering a laminate is a stack of discrete plies, each containing thousands of discrete fibres, continuity is unlikely to be maintained throughout the material. The following section aims to answer this concern based on experimental observations of molten laminate behaviour. Following that, an overview is given of several numerical models of forming flows of CFRTP materials, taken from the literature. The final of these introductory sections describes the foundation of the molten laminate model developed in the course of this study.

4.1.1 The Ply as a Continuous Material

A significant feature of CFRTP prepregs is the thin surface layer of pure matrix which protects the fibres in storage, and during consolidation provides much of the “glue” which bonds the plies into a laminate. As described in Chapter 1, these matrix-rich inter-layers remain in the laminate and during thermoforming provide low viscosity zones over which shear flow can occur. From a macroscopic viewpoint this shearing is described as inter-ply slip, and in modelling the behaviour of the material the sudden discontinuity it introduces into the deformation should be accounted for.

In forming doubly-curved parts out of CFRTP laminates with more than 2 fibre orientations, inter-ply slip is an essential process, and must be included in any forming model. In more mundane forming examples where other deformation modes may be expected to dominate, the inter-ply slip mode invariably occurs either in addition to or in place of these other flows.
Figure 4.1 shows a micrograph of the end of a unidirectional bend specimen used to determine intra-ply shear properties. In this example, longitudinal through-thickness intra-ply shearing was expected, and would have resulted in a constant shear angle.

![Figure 4.1 Micrograph of end of 8 x 0.5 mm PLYTRON bend specimen [37].](image)

The above micrograph shows unequivocally the preference of the material to deform by inter-ply slip, and that intra-ply shear is almost negligible. Other researchers have noted similar behaviour [6,9,10,38], with both PLYTRON and other fibre/matrix systems. Note that the specimen shown in Figure 4.1 was formed at the relatively low temperature of 140°C. At temperatures above the melting point of the polypropylene matrix (165°C) some through-thickness intra-ply shear is observed, but inter-ply slip still dominates [37]. It is notable that an inextensible fibre model is unable to deform in the manner shown in Figure 4.1 unless some process such as buckling or transverse flow has occurred closer to the bend. These observations point to the need to develop a molten laminate model treating each ply as a separate continuum with an additional treatment to allow finite slippage between the layers.

PLYTRON and other CFRTP tapes are commonly made by impregnating groups of fibre bundles. With a single bundle through the thickness of a ply, more tenacious bonding between fibres is expected, resulting in a stiffer-than-average response. Perhaps more significantly, based on Figure 4.1 and other experimental observations, it can be concluded that deformations through the thickness of a ply are indeed continuous.
Unfortunately the same cannot be said for in-plane deformations. CFRTP prepregs are crossed by numerous zones of weakness and low fibre volume fraction stemming either from inter-bundle spaces or simply the random fibre distribution. The idealised in-plane longitudinal intra-ply shear process illustrated in Figure 1.2 of Chapter 1 is therefore characterised by sudden, discontinuous slip along these lines of weakness. At first glance identical to through-thickness longitudinal intra-ply shear, this process differs by a size effect completely analogous to that in the field of fracture mechanics which causes larger specimens to be weaker. In-plane shearing is expected to resemble the situation for the whole laminate in Figure 4.1, except that zones of weakness are unlikely to be spaced with such regularity. Krebs et al. [11] noted the occurrence of such discontinuous shear in thermoformed parts, and performed a series of experiments involving longitudinal shear across thin in-plane and through-thickness specimens of PLYTRON. The in-plane results exhibited two distinct viscosity levels, one approximately 50 times higher than the other and matching that found through the thickness, while the lower, presumably inter-bundle viscosity, was about the level of that for inter-ply slip.

In attempting to model in-plane behaviour there is virtually no alternative to treating the deformation as macroscopically continuous and assuming average shear properties for the ply. Figure 4.2 summarises this approximation. For models based on this assumption, it is now justifiable the use bend specimens like that in Figure 4.1 for the measurement of average intra-ply shear properties.

This discussion has avoided mention of the possibility of discontinuous deformation on the “sub-bundle” or “inter-fibre” scale. While such behaviour can and does occur, a macroscopic model cannot be expected to consider such microstructural events, again necessitating the type of averaging scheme described above.

To summarise, it is concluded that the ply is the most appropriate unit of continuous deformation to use in a model of a molten CFRTP laminate. Several studies in the following section use such a discrete-ply approach, and it forms the basis of the model developed as part of this study.
4.1.2 Literature Survey

A number of numerical studies of fibre-reinforced thermoplastic forming have been reported over the past decade. A variety of approaches and material assumptions have been adopted, but of late, thermoforming simulations using the Finite Element Method have tended to be the favoured approach.

Smiley [7,39] created a decoupled model of pressure forming of axisymmetric parts using superplastic aluminium alloy diaphragms. Decoupling was achieved by assuming forming geometry and loads depended only on the behaviour of the relatively stiff diaphragms. Between the diaphragms, transverse flow in each ply was independently computed on a series of two-dimensional planes, using a Newtonian fluid model, while a kinematic scheme was used to update fibre orientations. This model had several major drawbacks and lacked sufficient flexibility to be applied to more general forming situations. In particular, the decoupling between the diaphragms and laminate, and between the plies themselves is not physically realistic.

Tam and Gutowski [40] created an elegant, semi-analytical molten laminate model consisting of a number of elastic plies separated by thin layers of Newtonian fluid to simulate the matrix rich inter-layers. By assuming a strain field for the part, forming loads were predicted for several bending examples. One of the model’s assumptions was that the elastic layers had only extensional stiffness and no resistance to bending, making it most applicable to laminates containing very thin plies. Despite handling the coupling between laminae introduced by the viscous inter-layers, the model seems only suited to the simple, two-dimensional flows studied in that paper.

Scherer et al. [4,41-43] carried out a large number of experiments and finite element simulations to characterise the inter-ply slip phenomenon. Ply-pull-out tests were performed and smooth curves relating the applied force to the slipped distance and velocity were generated (see Section 5.2.1 for more details). Scherer then attempted to reproduce the ply-pull-out results in a finite element model, first by placing thin elements of fluid between elastic plies, and eventually by adopting a viscous-type contact/friction relation over a negligible inter-layer thickness. A further series of two ply, two-dimensional forming simulations investigated the influence of various slip conditions in practical forming situations. Since the study focussed on the inter-ply slip phenomenon, only a simple, orthotropic elastic ply model was employed. Despite this, Scherer’s use of a friction model for inter-ply slip is crucial in reducing the size and complexity of the problem, and has been widely adopted in other Discrete-Ply models.

The following studies all implement the Ideal Fibre-Reinforced Fluid model [16,17,20,21] in a finite element scheme. The particular model most widely chosen is of an
incompressible Newtonian fluid constrained by one direction of inextensibility. The two material parameters in this model are viscosities for shear flow in the directions longitudinal and transverse to the fibres.

O’Bradaigh et al. [44,45] studied plane stress flows of IFRFs and used a penalty method to impose the fibre inextensibility constraint with $Q^{9/4}$ biquadratic velocity/bilinear discontinuous tension elements. A linear, quasi-static scheme was used to calculate instantaneous velocities which were multiplied by the time step to displace the mesh. Such an approach involves the build-up of considerable error unless time steps are extraordinarily small. McEntee and O’Bradaigh [46] recently improved this model by developing a mesh updating scheme that incorporated finite incompressibility and inextensibility constraints. The new model also uses large displacement contact/friction elements to model tool contact and inter-ply slip between layers of IFRF, in plane strain.

A group at Engineering Systems International, France, has adapted that company’s PAM-CRASH metal stamping and crash simulation code to the problem of thermoforming [12,47]. Using this explicit, dynamic code, the group has been able to carry out some very large thermoforming simulations of multi-ply laminates. Shell elements are used across each ply, with a reinforced fluid model for intra-ply shear and a viscous contact/friction relation between the plies. Thermal effects have also been considered. The use of an explicit model involves the approximation of mass lumping and other matrix diagonalization, and its conditional stability requires the use of very small time steps. Nevertheless, this approach has produced some very impressive results and has enormous potential for industrial application.

4.1.3 SimForm Model and Program

Using a separate finite element mesh to describe deformations in each ply, with some additional treatment to handle inter-ply slip, was an appealing idea even at the beginning of this project. At that time, the work of Scherer (see above) represented the state-of-the-art for such models. However, Scherer’s forming simulations were primarily of two-ply laminates, and limited to two dimensions. Also, despite using a detailed inter-ply slip model, little emphasis was placed on accurately modelling ply deformations. The remainder of this project thus sets out to improve on Scherer’s work, through better modelling of intra-ply properties, and by expanding the model to handle more layers and eventually three-dimensions.

The first priority was to be able to solve models similar to Scherer’s but with more plies, and using an orthotropic, elastic ply model with as high a ratio of fibre direction to transverse stiffness as possible. The first such models were performed using standard 2-D continuum and 3-
D shell elements in the MARC general purpose finite element package. MARC has a contact procedure that was used to simulate inter-ply slip. The results were not promising. The standard elements performed strangely when ratios of anisotropy greater than about 10 were used, with quadratic elements locking into a linear-like behaviour in bending. Furthermore, the contact procedure did not appear to work between thin, deformable sheets, as was desired.

After these disappointing results, efforts were then put into developing a custom finite element code for solving this class of problems. Early experiments with linear and quadratic elements showed identical behaviour to those in MARC. However, cubic interpolation in the plane of the ply was found to perform well with anisotropy ratios as high as 1000. Further success with the contact algorithm of Section 4.4 meant that the remainder of this study has focussed on expanding this code. The program was initially written to solve two-dimensional, plane-strain problems and provided many interesting results for this class of deformations [48]. It has since been rewritten to solve three-dimensional problems and now includes a more systematic treatment of geometric and material non-linearity. The updated program, called SimForm, uses an implicit scheme for solving time-dependent behaviour, and employs bicubic Hermite interpolation to describe both the surfaces of the plies and undeformable tool surfaces used to shape the specimen. However, it still retains the central features of its predecessor, namely a viscous inter-ply slip response and a highly anisotropic elastic ply model. These and other aspects of the SimForm model need some justification.

In most of the discrete-ply models discussed in Section 4.1.2, inter-ply slip was treated as a viscous process. In models that reduce the matrix-rich inter-layers to just contact surfaces, this may be simulated by a velocity-dependent sliding friction relation. The same approach is taken in the SimForm model, and is believed to be a good approximation to reality. Scherer [4,42,43] noted some initial elastic response in his ply-pull-out tests, but this gave way to viscous behaviour as slip increased. In future it may become necessary to model inter-ply slip as a full, viscoelastic process. However, inter-element contact has proved to be a complex topic, and a more detailed treatment of inter-ply slip should only follow the successful implementation of the simpler model, as well as adequate modelling of other flow processes in the composite.

The model also neglects all thermal effects such as the change in material properties with temperature. This is appropriate for modelling isothermal forming processes, but is also thought to be adequate as a first step towards modelling the non-isothermal forming processes described in Chapter 1. These processes involve transfer of the blank from an oven or other heating device to the tool, and both transfer and forming must be rapid for the composite to remain within its forming window of temperature.
Modelling CFRTP plies at thermoforming temperatures as elastic materials is more difficult to justify. More generally, deformation by the major intra-ply flow processes shown in Figure 1.2 involves a viscoelastic response [37,38]. However, this type of viscoelasticity is characterised by an instantaneous, elastic response followed by time dependent flow as long chain polymers rearrange themselves to reduce stress. More significant than the elastic component of the polymer response is the effect of the elastic fibres themselves. Apart from their obvious axial and flexural stiffness, misaligned or twisted bundles of fibres will add reinforcement in directions other than the nominal fibre orientation. Evidence of this is given in Figure 4.1, and in many similar images in other studies. Since through-thickness intra-ply shear is seen to be negligible in these bending examples, one can only conclude that no inelastic deformation is occurring in these parts of the plies.

Despite its apparent flaws, the elastic ply approach is a necessary step in the development of a more complete model that considers the viscoelastic response of the material under transverse flow and in-plane intra-ply shear modes. In particular, the problem of obtaining the high degree of anisotropy present in the plies must first be adequately handled in the simpler model. Also, as the results in Chapter 5 will show, any divergence of the elastic model from the real material behaviour provides clues to how the model can be improved by incorporating flow effects.

The remainder of this chapter outlines the finite element theory implemented in the SimForm program. Section 4.2 looks at the formulation and solution of non-linear large-strain finite element problems. Following that is a description of the material model used in the plies, while Section 4.4 describes the contact procedure used to simulate inter-ply slip and interaction with tool surfaces. The final section of this chapter presents the main algorithm used in the program, showing how the various components for handling material properties, contact and iterative solution are pieced together. In addition to these theory sections, Appendix B outlines the flexible input format used in SimForm, which is tailored specifically for problems of laminate forming. Finally, Appendix C looks at some of the tests used to verify the calculations performed by the program.

4.2 The Finite Element Method - Non-Linear Overview

4.2.1 The Principle of Virtual Work

Finite element procedures for solving non-linear mechanics problems are concerned with satisfaction of the Principle of Virtual Work. This section provides an overview of the principle
as applied in the *SimForm* program. Extensive reference is given to the Continuum Mechanics text by Malvern [29], an excellent source for further reading in this topic.

In studying the deformation of materials one seeks to find a state at which the body, in the absence of inertial effects, satisfies the equilibrium partial differential equations,

\[
\frac{\partial T_{ji}}{\partial x_j} + f_{j}^{(\text{body})} = 0 ,
\]

throughout the interior of the body, and the boundary conditions,

\[
T_{ji} n_{j}^{(S)} = t_{i}^{(S)} ,
\]

on those parts of the boundary where tractions are applied. Here \(T\) is the symmetric Cauchy (true) stress tensor, \(f^{(\text{body})}\) is the body force vector per unit volume and \(n^{(S)}\) is the unit surface normal at the point on the boundary where traction \(t^{(S)}\) is applied. Where indicial notation for vectors and tensors is used, as in the above, repeated indices denote summation over the range from 1 to 3. If the above conditions are satisfied, the body is said to be subject to a *statically admissible* stress distribution.

If the body is given a set of infinitesimal, kinematically admissible virtual displacements \(\delta u\) from such an equilibrium state, the *virtual work* of the external surface tractions and body forces is given by [29]:

\[
\delta W_{\text{ext}} = \int_{S} T_{ji} n_{j} \delta u_{i} dS + \int_{V} f_{j} \delta u_{i} dV ,
\]

where integration is over the current, updated boundary, \(S\) and volume, \(V\) of the body. A *kinematically admissible* displacement distribution is one which satisfies prescribed boundary displacements where traction boundary conditions are not enforced, and possesses continuous first partial derivatives in the interior of the body [29]. After some manipulation with Green’s Theorem, the above identity becomes [29]:

\[
\delta W_{\text{ext}} = \int_{V} T_{ji} \delta \varepsilon_{ij} dV ,
\]

where \(\varepsilon\) is the infinitesimal strain tensor. The equality of equations (4.3) and (4.4) is the *Principle of Virtual Work*, also known as the *Principle of Virtual Displacements*. It is important to note that this is not an energy principle since the work calculated is purely fictitious. The principle is merely a mathematical identity and it can be demonstrated than it is no more than an alternative way of enforcing the equation of equilibrium (4.1) and traction boundary conditions (4.2) [29]. Consequently, it applies even in situations where mechanical energy is not conserved, such as in plastic deformation.
Another point to note is that although the principle mentions infinitesimal displacements, these are virtual displacements from any equilibrium state. No mention is made of the magnitude of strains up to that point, so finite strain measurements are equally applicable. The principle can therefore be presented in many forms depending on the conjugate pair of stresses and strains to be used. The form that is of particular relevance to this study is, ignoring body forces:

$$ \int_{V_0} \tilde{T} : \delta E dV_0 = \int_S \mathbf{t} : \delta \mathbf{u} dS, \quad (4.5) $$

where the first integral is over the reference (undeformed) volume, \( \tilde{T} \) is the symmetric second Piola-Kirchoff stress tensor, and \( E \) is the Lagrangian finite strain tensor. The components of the symmetric Lagrangian strain tensor are given by [29]:

$$ E_{ij} = \frac{1}{2} \left[ \frac{\partial \tilde{u}_i}{\partial X_j} + \frac{\partial \tilde{u}_j}{\partial X_i} + \frac{\partial \tilde{u}_k}{\partial X_i} \frac{\partial \tilde{u}_k}{\partial X_j} \right]. \quad (4.6) $$

The common convention to write undeformed, or material coordinates (see Section 4.3.1) as capitals will be followed wherever it is necessary to distinguish them from the current coordinates of the part, as in the above equation. Since the Lagrangian strain tensor refers strains back to a reference undeformed state, it is useful in cases involving grossly deforming elastic structures, as is assumed in this study. It is for this reason that the integral on the left of equation (4.5) is over the reference volume. Although it is not normal to mix reference states in expressions of the Virtual Work Principle, the right hand side of (4.5) is given in terms of the current, updated geometry since the forces acting on the body are most easily expressed in that state.

### 4.2.2 Discretization into Finite Elements

A fundamental step in the Finite Element Method is the approximation of a continuum by a finite number of discrete elements. While this step is covered in all texts on the subject, it is worthwhile to include a brief summary here to introduce the nomenclature to be used in the remainder of the theory.

Discretization breaks a complex geometry into a group of simple geometric forms such as the brick element shown in Figure 4.3.
Each element comprises a certain number of nodes, points marking what is usually the boundary of the element. Nodal variables of location, displacement and other vector, scalar or tensor quantities may be interpolated to return a value at any point in the volume represented by the element. For example, if vector $\mathbf{d}'$ represents the displacement of the $j^{th}$ of $M$ nodes in an element (where $M = 8$ for the element shown in Figure 4.3), then,

$$u(\xi) = \sum_{j=1}^{M} \psi_j(\xi) a_j,$$  \hspace{1cm} (4.7)

gives the value of the displacement at some interior point of the element represented by parametric coordinate $\xi = (\xi_1, \xi_2, \xi_3)$. The basis functions $\psi_j(\xi)$ are smooth, usually polynomial parametric curves interpolating the nodal quantities. For the brick-type element shown in Figure 4.3, the three $\xi$ coordinates describe a simple cubic space, which in SimForm ranges from $-1$ to $+1$ in each direction. A unique mapping from $\xi$-space to real $(x,y,z)$-space allows integration of various quantities over the real, likely distorted volume of the element to be carried out in the former, far simpler domain.

The derivatives of the basis functions with respect to the parametric coordinates $\xi_j$ may also be readily calculated. From these, equations similar to (4.7) can be used to calculate such quantities as the Jacobian matrix,

$$\frac{\partial x_i}{\partial \xi_j}(\xi) = \sum_{k=1}^{M} \frac{\partial \psi_k}{\partial \xi_j}(\xi).x_i^k,$$  \hspace{1cm} (4.8)

where $x_i^k$ may be either the undeformed or deformed nodal coordinates, as necessary. The determinant of the Jacobian matrix at a point equals the local ratio of material volume in real space to the equivalent volume in $\xi$-space, used as a scaling factor when integrating over the element volume. Also, the inverse of the Jacobian matrix can be used to convert derivatives with
respect to $\xi$-coordinates to be in terms of real $x$, $y$ and $z$ coordinates. Hence, the displacement gradient may be calculated from total nodal displacements using,

$$F_{ij}^{\text{tot}} = \frac{\partial u_i^{\text{tot}}}{\partial X_j} = \sum_{k=1}^{M} \frac{\partial u_k}{\partial X_j} a_j^{(k(\text{tot}))},$$  \hspace{1cm} (4.9)

where undeformed coordinates have been used.

One point not mentioned up to now is the fact that element geometry need not be described merely by a series of $(x,y,z)$ coordinates at nodal points. Many combinations of parameters, some less easy to visualise, could equally well be used to this effect. For example, *SimForm* uses elements with linear and quadratic through-thickness interpolation, but bicubic Hermite interpolation in the $\xi_1$ and $\xi_2$ directions. They have the same number of nodes as the element shown in Figure 4.3, only each node contains four vectors describing the shape of the element in the vicinity of the nodes. These are the location of the node, two vectors pointing in the direction of $\xi_1$ and $\xi_2$, and the “twist” vector describing the rate of change of vectors in the $\xi_1$ direction as one moves in the $\xi_2$ direction, and also the converse. Consequently, assemblages of these elements possess continuity of slope across their common boundaries and are said to be $C_1$-continuous. The surface fitting approach to grid strain analysis in Chapter 3 also utilised bicubic Hermite elements; Section 3.2.1 should be consulted for more details of this interpolation scheme. In particular, Figure 3.4 in that section shows a distorted bicubic Hermite surface patch, demonstrating the complexity of shapes such elements can smoothly describe.

The fundamental result of the discretization process is that the infinite number of variables otherwise needed to describe the continuum represented by each element are now approximated by a finite number of nodal variables. To represent the whole continuum of interest, a large number of elements are connected by sharing nodes on their boundaries, so that a very complex geometry may now be described by a finite number of variables. If the number of elements is sufficiently large then the real behaviour of the continuum may be accurately described by that list of nodal quantities.

It remains to find a discretized expression for the Principle of Virtual Work. First, to prevent repetition of calculations and to simplify the equations, equation (4.5) will be recast in a form that uses vectors to represent the stress and strain quantities. Hence,

$$\int_{V_0} \tilde{T} \cdot \delta EdV_0 - \int_S \mathbf{t} \cdot \delta udS = 0,$$  \hspace{1cm} (4.10)

where,

$$\mathbf{E}^t = \begin{bmatrix} E_{11} & E_{22} & E_{33} & 2E_{12} & 2E_{23} & 2E_{31} \end{bmatrix},$$  \hspace{1cm} (4.11)
and,
\[
\mathbf{T}^t = \begin{bmatrix}
\tilde{T}_{11} & \tilde{T}_{22} & \tilde{T}_{33} \\
\tilde{T}_{12} & \tilde{T}_{23} & \tilde{T}_{31}
\end{bmatrix},
\]  
(4.12)

(The superscript ‘t’ denotes a transpose in the above equations.) At a given current geometry the tractions \( t \) are known, and the Second Piola-Kirchoff stresses may be obtained from the current strain as is described in Section 4.3. Within each element virtual displacements and strains may be interpolated from a set of arbitrary virtual nodal displacements \( \delta a^i \), giving,
\[
\delta u(\xi) = \sum_{i=1}^{M} \psi'(\xi)\delta a^i, \quad (4.13)
\]
and,
\[
\delta E(\xi) = \sum_{i=1}^{M} \overline{B}'(\xi)\delta a^i, \quad (4.14)
\]
where \( \overline{B}' \) is the sum of first order (small strain) and second order (large strain) parts,
\[
\overline{B}' = B_0' + B_L', \quad (4.15)
\]
defined by [49],
\[
B'_i = \begin{bmatrix}
\frac{\partial \psi_i}{\partial x_1} \\
\frac{\partial \psi_i}{\partial x_2} \\
\frac{\partial \psi_i}{\partial x_3}
\end{bmatrix}, \quad (4.16)
\]
and [32],
The principle of virtual work can now be written in discretized form as,

\[
\sum_{i=1}^{M} \delta a^i \left[ \int_{V_0} (\mathbf{B}^T)^T \mathbf{T} dV_0 - \int_{S} \mathbf{t}^T dS \right] = 0, \tag{4.18}
\]

where, for generality, \( M \) may be considered the number of nodes in the entire model, since the distinction that the basis functions interpolate only over an element can be reinterpreted at any time as interpolating in a piecewise fashion over the entire body. Since equation (4.18) is valid for any arbitrary set of virtual nodal displacements, \( \delta a^i \), it follows that the \( M \) quantities inside the square brackets of equation (4.18) must be zero to obtain equilibrium internally and with forced boundary conditions in the finite element model. The following section looks at how to obtain such an equilibrium solution.

### 4.2.3 Solution of Non-Linear Finite Element Equations

The square-bracketed quantity in equation (4.18) is a measure of out-of-balance forces in the finite element model, and is termed the residual vector [32]:

\[
\Psi(a) = \int_{V_0} \mathbf{B}^T \mathbf{T} dV_0 - \int_{S} \mathbf{t}^T dS = 0. \tag{4.19}
\]

Here, all loads acting on the body have been included in a single vector of generalised nodal forces, \( \mathbf{f} \), while indices have been dropped for clarity. Also, the residual has been equated to zero in the above equation to stress the aim of the solution process.

Since equation (4.19) involves geometric, and probably also material non-linearity, its solution cannot be obtained in one step of linear algebra. Iteration is therefore required, and convergence is achieved when the residual vector is sufficiently close to zero. Zienkiewicz and Taylor [32] present a large number of procedures for solving such non-linear problems. This
section describes the standard Newton-Raphson method used in SimForm to solve equation (4.19), also from that reference.

The solution process at some increment of loading starts with the geometry at a certain position, initially the undeformed state but later some other state nearer to the eventual equilibrium. At this position the vector of nodal forces, \( f \), is computed, and the Load Norm, equal to the square root of the sum of squared quantities in this vector, is calculated. The vector of internal forces, the integral in equation (4.19), is then subtracted from the force to give the negative of the residual vector, the norm of which will be called the Residual Norm. In the first iteration there is no point in checking for convergence, but in later iterations the ratio of Residual Norm/Load Norm will be the measure of convergence, a value of say 0.001 to 0.01 being an acceptable solution [32]. If convergence has not been achieved, a Newton-Raphson iteration is performed.

The Newton-Raphson method requires equation (4.19) to be differentiated with respect to the nodal displacements to find the tangential stiffness matrix [32],

\[
K_T = \frac{\partial \Psi}{\partial a} = \int_{V_0} \mathbf{B}^T d\mathbf{T} dV_0 + \int_{V_0} d\mathbf{B}^T d\mathbf{T} dV_0 - \frac{\partial f}{\partial a},
\] (4.20)

where the variation of forces with displacement has been considered. In each iteration an increment in the displacements is calculated using the residual vector found earlier and the simple correction,

\[
K_T(a^i)\Delta a^i = -\Psi(a^i),
\] (4.21)

after which the total displacement of the body is incremented to:

\[
a^{i+1} = a^i + \Delta a^i,
\] (4.22)

and the process is repeated starting with the calculation of another residual vector. After a sufficient number of these linear steps, the geometry will have been updated to a point at which equation (4.19) is approximately satisfied. A solution for that load increment has thus been obtained. The load may be incremented any number of times if a complicated forming path is to be followed.

It remains to produce formulae for the tangential stiffness matrix suitable for inclusion in a finite element program. First, the stiffness is broken into the three parts listed in equation (4.20),

\[
K_T = \bar{K} + K_o + K_f.
\] (4.23)
The first part of the tangential stiffness is that present in linear finite element schemes plus those non-linear terms stemming from equation (4.17) [32]:

\[
\bar{K} = \int_{V_0} \bar{B}^i \bar{D} \bar{B} dV_0 ,
\]  
(4.24)

or in component form,

\[
\bar{K}_{ij}^{pq} = \int_{V_0} \bar{B}_{ki}^P D_{km} \bar{B}_{mj}^Q dV_0 ,
\]  
(4.25)

where \( P \) and \( Q \) range from 1 to \( M \), and \( i \) and \( j \) range from 1 to 3 for an element containing \( M \) nodes and three degrees of freedom per node. (For simplicity, with bicubic Hermite elements it is useful to consider each nodal vector quantity as a separate node). Gaussian quadrature will be used to integrate these equations over each element, while assembly into the global stiffness matrix proceeds as is described in any finite element text. The incremental elasticity matrix,

\[
D_{km} = \frac{\partial \bar{f}_k}{\partial \bar{E}_m}(E),
\]  
(4.26)

will be discussed in Section 4.3, since it is part of the constitutive equation.

The second stiffness in equation (4.23), \( K_\sigma \), is often called the initial stress matrix. It has a symmetric form similar to equation (4.24) but is less simple to write down succinctly. Reference [32] derives the initial stress matrix for a general large strain situation. The final stiffness to consider, the load-correction matrix, cannot be written down until specific loading cases are considered.

An alternative method for obtaining the tangent stiffness matrix is to numerically differentiate equation (4.19). This has advantages in that it is very simple to program and there is no danger that terms will be either incorrect or omitted altogether, a possibility when one considers the complexity of the terms involved. The tangential stiffness matrix is derived by noting that its \( i \)th row is just:

\[
K_{Ti} = \frac{\partial \Psi}{\partial a_i} \approx \frac{\Psi(a, a_i + \alpha) - \Psi(a, a_i - \alpha)}{2\alpha}.
\]  
(4.27)

Differentiation is thus performed numerically from first principles, sampling the residual vector at 2 points, one with a small quantity \( \alpha \) added on to \( a_i \), one with the same quantity subtracted. The major down-side to this approach is speed, being considerably slower than when direct stiffness formulae are used. Also, some error is introduced by the approximate differentials.
4.3 Constitutive Equation for Elastic, Anisotropic Plies

4.3.1 Material Coordinates and Strains

Figure 4.4 depicts a rectangular fibrous sheet in its undeformed state with sides parallel to the global $x_1$, $x_2$ and $x_3$ axes, and later when it has adopted some deformed state elsewhere in space. At any state a point in the body may be referred to by its current $(x_1,x_2,x_3)$ or spatial coordinates in the global reference frame, or by its original, undeformed coordinates denoted by capital letters $(X_1,X_2,X_3)$, called material coordinates since these axes convect with the material. As described below, various strain measurements can be defined by the difference between these two representations.

![Figure 4.4 Deformation of a fibre-reinforced sheet showing material coordinates.](image)

One measure of the deformation of the body at a point is the deformation gradient, a tensor which maps a differential line element $dX$ in the undeformed state to the same element $dx$ in the deformed state, so that,

$$dx = F dX,$$  \hspace{1cm} (4.28)

where, in component form,

$$F_{kM} = \frac{\partial x_k}{\partial X_M}.$$  \hspace{1cm} (4.29)

As in Chapter 3, we move from the generally unsymmetric deformation gradient to the more convenient deformation tensor,

$$C = F'F$$  \hspace{1cm} (4.30)

which in component form is,
The Lagrangian finite strain tensor of equation (4.6) may then be compactly written as,

$$2E = C - I.$$  \hfill (4.32)

In finite strain elasticity problems it is common to derive the constitutive equation from a strain energy function $W$ given in terms of the deformation tensor and possibly other tensors. Two options are available for describing the type of anisotropy brought about by reinforcement of an isotropic body by very stiff fibres. In the following section a transversely isotropic model is devised in which all strains contribute to the strain energy function, but high stiffnesses in the fibre direction and for volumetric strains limit deformations by these modes. The alternative is to apply constraint equations so that the material is incompressible and/or inextensible in the fibre directions. The latter approach is taken in Section 4.3.3.

4.3.2 Transverse Isotropy

Spencer [22,23] has determined that the strain energy in a transversely isotropic elastic material is a function of two tensors,

$$W = W(C, a^0 \otimes a^0),$$  \hfill (4.33)

where $a^0$ is a unit vector in the undeformed fibre direction as depicted in Figure 4.4, constant in a given ply. More specifically, $W$ can be expressed as a function of the following five invariants [22,23] in these two tensors:

$$I_1 = \text{tr} C, \quad I_2 = \frac{1}{2} \left\{ (\text{tr} C)^2 - \text{tr} C^2 \right\}, \quad I_3 = \text{det} C = \left( \rho_0 / \rho \right)^2,$$

$$I_4 = a^0 \cdot C \cdot a^0 = \lambda^2, \quad I_5 = a^0 \cdot C^2 \cdot a^0,$$  \hfill (4.34)

where $\lambda$ is the stretch in the fibre direction (ratio of deformed to undeformed length), while the trace of a tensor is given by,

$$\text{tr} C = C_{KK} = C_{11} + C_{22} + C_{33}.$$  \hfill (4.35)

Given a suitable form for the strain energy function, the symmetric Second Piola-Kirchoff Stress Tensor is determined using [23],

$$\tilde{T}_{RS} = \frac{\partial W}{\partial C_{RS}} + \frac{\partial W}{\partial C_{SR}} = \sum_{\alpha=1}^5 \frac{\partial W}{\partial I_{\alpha}} \left( \frac{\partial I_{\alpha}}{\partial C_{RS}} + \frac{\partial I_{\alpha}}{\partial C_{SR}} \right),$$  \hfill (4.36)

where [23],
\[
\frac{\partial I_1}{\partial C_{RS}} = \delta_{RS}, \quad \frac{\partial I_2}{\partial C_{RS}} = I_1 \delta_{RS} - C_{RS}, \quad \frac{\partial I_3}{\partial C_{RS}} = I_2 \delta_{RS} - I_1 C_{RS} + C_{RP} C_{PS}, \\
\frac{\partial I_4}{\partial C_{RS}} = \alpha^0_R \alpha^0_S, \quad \frac{\partial I_5}{\partial C_{RS}} = 2 \alpha^0_R \alpha^0_P C_{PS}.
\]

(4.37)

The technique described above is general enough to describe almost any transversely isotropic elastic behaviour, given enough higher order combinations of the invariants in equation (4.34). A limiting factor, however, is the number of material constants that need to be determined experimentally if a complicated function is chosen. For this reason most strain energy functions limit themselves to low order functions of these invariants. The transversely isotropic strain energy function implemented in \textit{SimForm} uses five material constants, effectively one for each invariant:

\[
W = K_1 \left( I_1 - 3 - \ln(I_3) \right) + K_2 \left( I_2 - 3 - 2 \ln(I_3) \right) + K_3 \left( I_3 - 1 \right)^2 + K_4 \left( I_4 - 1 \right)^2 + K_5 \left( I_5 - 1 - 2 \ln(I_4) \right).
\]

(4.38)

Equation (4.38) was chosen because it is one of the simplest forms that allows the initial “small strain” transversely isotropic material properties to be controlled. Hence, if \( K_3 \) and \( K_4 \) are given high values, corresponding to nearly incompressible behaviour and reinforcement by stiff fibres, respectively, initial elastic material properties are approximately given by,

Young’s modulus in fibre direction, \( E_f \approx 8K_4 \),

Transverse Young’s modulus, \( E_T \approx 8(K_1 + K_2) \),

Longitudinal shear modulus, \( G_L \approx 2(K_1 + K_2 + K_5) \),

Transverse shear modulus, \( G_T \approx 2(K_1 + K_2) \).

(4.39)

The natural logarithm terms in equation (4.38) are merely a device to ensure the material is unstressed in the undeformed state. By contrast, in the following section we will see that in a constrained material the unstressed state is obtained by giving arbitrary values to the pressure and tension reaction stresses.

Another reason for using equation (4.38) is that if \( K_4 = K_5 = 0 \), and \( K_3 \) is high, this model very closely approximates the behaviour of the incompressible, isotropic Mooney material [50]. For fully incompressible cases the two parameter Mooney model is frequently used to describe finite deformations of rubber materials and is often applicable to extensions of several hundred percent [50]. Hence, this approximate Mooney model will be applied to the behaviour of rubber diaphragms used in pressure forming examples described in later chapters of this report. Note
that the initial elastic properties of the Mooney material are a Young’s modulus of $6(K_1 + K_2)$ and a shear modulus of $2(K_1 + K_2)$.

Equations (4.36) and (4.38) combine to return the Second Piola-Kirchoff stress tensor at a given strain, which is entered into residual equation (4.19). Also, equation (4.36) can be differentiated again to give the incremental elasticity matrix of equation (4.26).

With strain energy equation (4.38), high values of $K_3$ and $K_4$ give the material nearly incompressible behaviour and high stiffness in the fibre direction $a^0$. Using cubic element interpolation, performance remains good for quite high values of these parameters. In particular, ratios of fibre direction to transverse stiffness in excess of 1000 are attainable. Much higher ratios tend to cause unstable behaviour, so an alternative approach to modelling stiff fibres is often used, involving the assumption of fibre inextensibility. This, and the related incompressible formulation are described in the following section.

### 4.3.3 Incompressible and Inextensible Formulation

Using the notation of Section 4.3.2, an incompressible material is one in which the constraint,

$$I_3 = 1,$$  \hspace{1cm} (4.40)

is satisfied at every point in the continuum. Similarly, fibre inextensibility is introduced by the constraint equation,

$$I_4 = 1.$$  \hspace{1cm} (4.41)

An incompressible, inextensible hyperelastic material has a modified strain energy function of the form \([22,23,50]\):

$$\mathcal{W} = W(I_1, I_2, I_3) - \frac{1}{2} p(I_3 - 1) + \frac{1}{2} T(I_4 - 1),$$  \hspace{1cm} (4.42)

where $p$ and $T$ are Lagrangian multipliers identifiable as the pressure and tension reaction stresses, respectively. Pressure and tension in the elements are then interpolated independently from displacement, using additional nodal variables $b$ and $c$ such that,

$$\hat{p} = \sum_{i=1}^{M} \overline{N}_i b_i ,$$  \hspace{1cm} (4.43)

and,

$$\hat{T} = \sum_{i=1}^{M} \overline{N}_i c_i ,$$  \hspace{1cm} (4.44)

where in SimForm, $\overline{N}_i$ are trilinear basis functions, and $M$ is therefore 8.
The system equations for the modified strain energy function are derived as usual by taking the variation of equation (4.42) in order to find a minimum. The first term of this equation is dealt with as for equation (4.38) and needs no further comment. The second and third terms produce very similar formulae, so only the latter will be considered in the following. Taking the variation of the volume integral of $\frac{1}{2} T(I_4 - 1)$ gives:

$$\frac{1}{2} \int \delta T_I dV_0 + \frac{1}{2} \int T \delta I dV_0 .$$

(4.45)

Substituting the interpolated variables from equations (4.14) and (4.44), the discretized form is given by,

$$\frac{1}{2} \delta \epsilon' \int \tilde{N}'(I_4 - 1)dV_0 + \frac{1}{2} \delta a' \int \tilde{B}' \frac{\partial I_4}{\partial E} \tilde{t} dV_0 .$$

(4.46)

The second integral term can be identified by comparison with the residual vector (equation 4.19) to which it is added. The product $\frac{1}{2}(\partial I_4 / \partial E) \tilde{t}$ is the tension reaction expressed as a second Piola-Kirchoff stress, allowing the residual to continue to represent out-of-equilibrium forces in the model. The first integral term is a measure of the lack of satisfaction of constraint equation (4.41), and acts as the force vector for the tangential inextensibility equation, described below. The values calculated from this term appear in the residual vector on additional rows set aside for the $c$ variables.

The new components of the tangential stiffness matrix are found by once again taking the variation of equation (4.46) with respect to the nodal parameters, which gives,

$$\frac{1}{2} \int \tilde{B}' \frac{\partial I_4}{\partial E} \tilde{t} dV_0 + \frac{1}{2} \int \tilde{B}' \frac{\partial^2 I_4}{\partial E^2} \tilde{B} \tilde{B} dV_0 + \frac{1}{2} \int \tilde{B}' \frac{\partial I_4}{\partial E} \tilde{N} dV_0 + \frac{1}{2} \int \tilde{N}' \left( \frac{\partial I_4}{\partial E} \right)' \tilde{B} dV_0 .$$

(4.47)

The first two terms in the above appear as coefficients in those rows and columns of the tangential stiffness matrix corresponding to displacement degrees of freedom. The first term is simply included with the initial stress matrix $K_\sigma$. The second term is zero in this case, but will be non-zero when pressure constraints are dealt with. By comparison with equation (4.24) it is seen that it can be included in the incremental elasticity matrix of equation (4.26).

The third term in equation (4.47) adds coefficients to the displacement rows and nodal tension columns of the tangential stiffness matrix, while the fourth does the converse. The fourth term is identified as the tangential inextensibility constraint, while the third provides coupling between the incremental tensions and the remainder of the tangential stiffness matrix.

Several points should be noted about the above equations:
• The tangential stiffness terms are symmetrical.

• Care must be taken when calculating such quantities as $\partial I_4/\partial E$, since a vector representation of the Lagrangian strains is used, as defined in equation (4.11). These vectors can be derived from equations (4.37).

• The above equations can be recast for the incompressibility constraint by replacing $I_4$ with $I_3$, and $c$ with $b$.

In spite of having spent some time implementing the incompressible and inextensible formulation no advantage has yet been found in using it. It has the major disadvantage of introducing extra degrees of freedom into the model, with associated increase in storage requirements and processing time. More significantly, its behaviour is almost identical to the model in Section 4.3.2 using high values of $K_3$ and $K_4$, with massive buckling and wrinkling in areas subject to even the slightest compressive stresses.

At this point it is appropriate to consider further details of how the model departs from real CFRTP behaviour. Although such high stiffness ratios are encountered in real composites, susceptibility to buckling is somewhat reduced by each fibre’s own flexural stiffness and the mutual support of closely bunched fibres. The SimForm model does not consider these effects, which explains its great sensitivity to buckling at high ratios of fibre to transverse stiffness. As a result, the model will give a more conservative prediction of where such instabilities are likely to occur in real parts.

4.4 Modelling Contact

Fundamental to modelling of sheet forming processes is the interaction between the blank and the dies used to form it. The model must provide for detection and loss of contact, and conformance of the specimen to the tool geometry, as well as between deformable elements. The inter-ply slip phenomenon occurring in thermoforming of fibre-reinforced thermoplastic laminates is a special type of deformable contact requiring the geometries of neighbouring plies to mate together even when curvatures are high and slippage is great. One of the strengths of the SimForm program is its use of higher order bicubic Hermite elements for describing such close-fitting surfaces.

The deformation of contacting regions in a model are clearly not independent. In his study of diaphragm forming of thermoplastic composites, Smiley [7,39] (see also Section 4.1.2) assumed that plies deformed independently, which may have produced acceptable results for his particular case of study. In general, however, dependence must be assumed and the deformations of all continua in the model solved for simultaneously. It will be shown in the following section
that the contact model couples together the deformation of touching bodies, ensuring no two bodies occupy the same volume of space, while transferring appropriate forces between them.

Implementation of the contact model was the major prerogative in the initial development of SimForm and has continued to absorb a large quantity of development time as improvements in stability and contact management were seen to be necessary. As will become clear when some of the examples in the following chapters are discussed, the procedures require a good deal of experience to use, since the interaction of contacting surfaces is a very complex issue.

4.4.1 Normal Contact Constraints

Suppose a point $k$ on element surface A is in contact with another surface B. Ideally, one would interpret this as meaning that $k$ lies exactly on the other surface. However, owing to the inexactness of mating together the discretized surfaces of a finite element model, the term “in contact” must be redefined. A point is therefore considered to be in contact with another surface if it has touched or penetrated that surface at some time in the past but has not yet met the requirements for separating from it. We can now say that $k$ is in contact with surface B at point $p$, chosen for convenience to be the closest point to $k$ along a line normal to surface B. As Figure 4.5 shows, a surface coordinate system is established at $p$ which consists of the outward normal $n$ and tangent $t^1$ (with a second tangent $t^2$ out of the page in three dimensions), each of unit length. Point $k$ lies some distance along $n$ from $p$, positive or negative.

![Figure 4.5 Contact surface coordinate system.](image)

Ignoring in this development any tangential friction forces, contact conditions seek to minimise the distance between $p$ and $k$ along $n$. Conceptually, a spring collinear with $n$ is set up which transfers compressive normal forces if the surfaces penetrate one-another, tensile if they pull apart and zero if the two surfaces are just touching. Calculating the overlap distance $d$ (negative if the surfaces are apart) using,

$$d = (p - k).n,$$  \hspace{1cm} (4.48)

the traction applied by the deformed spring on point $k$ of surface A is:
where $S_n$ is the stiffness per unit area of the spring. If surface B is deformable, the equal and opposite load acting on it must also be considered.

As in typical finite element surface integrals, quadrature will be employed to compute generalised nodal forces. That this involves determination of the above formulae at a finite number of points over each element surface is useful in that one can individually track which points are contacting and only apply contact conditions to them. A good choice for the locations of these “contact points” are the two dimensional Gauss points. Minimal $3 \times 3$ Gauss quadrature is sufficient for integrating uniform pressure over each bicubic element surface, especially in the limit as element size decreases. However, with partial contact and larger, grossly deforming elements, it will be desirable to use many more contact points. As contact points become more numerous, less advantage is gained by positioning them at Gauss Points. A simple alternative is to position them at even spacing over the element surface, as shown in Figure 4.6, which corresponds to quadrature using the trapezoidal rule.

![Figure 4.6 6 × 6 contact points on bicubic element surface.](image)

Each of the contact points now has an area of the surface associated with it. If $w$ is the quadrature weight of the contact point at $k$, with surface coordinate $\xi^{(k)} = (\xi_1, \xi_2)$ then this area is given by,

$$A = w \left| \frac{\partial x}{\partial \xi_1} (\xi^{(k)}) \times \frac{\partial x}{\partial \xi_2} (\xi^{(k)}) \right|.$$  

Equation (4.49) can now be rewritten to give the normal force vector transferred onto point $k$ by a finite spring:

$$F_n^{(k)} = (S_n A d) \mathbf{n}.$$  

(4.51)
Computing the surface basis functions $\phi_i^{(A)}$ at $\xi^{(k)}$, the contribution of the force at this contact point to the generalised nodal force vector for element surface $A$ may be written as,

$$g_i^{(A)} = \phi_i^{(A)} F_n^{(k)}.$$  

(4.52)

Equal and opposite generalised forces on surface $B$ are similarly found from the basis functions $\phi_i^{(B)}$ computed at $\xi^{(p)}$. Section 4.4.3 describes how the location of $p$ is determined at each stage of the simulation by an iterative scheme.

It remains to determine the components of the contact stiffness matrix that couple together the deformations of contacting regions, ensuring a non-singular global stiffness matrix. For simplicity, the following derivation assumes displacements and rotations are fairly small in each increment so that a constant surface normal $n$ may be adopted.

By differentiating equation (4.51) at each contact point, the change in the force at $k$ in terms of the relative displacements of $p$ and $k$ can be determined:

$$dF_n^{(k)} = \kappa (dp - dk),$$  

(4.53)

where,

$$\kappa_{qr} = S_n A n_q n_r,$$  

(4.54)

is the $3\times3$ stiffness matrix for this contact point. Given that $\Delta a_j$ is the vector of nodal displacements to be determined this iteration, displacements of points $k$ and $p$ are given by:

$$\Delta k = \sum \phi_j^{(A)} \Delta a_j^{(A)},$$  

(4.55)

and,

$$\Delta p = \sum \phi_j^{(B)} \Delta a_j^{(B)},$$  

(4.56)

Equation (4.53) is then readily extended to give increments of generalised nodal forces in terms of generalised nodal displacements as was done in equation (4.52). One additional wrinkle is that the global stiffness matrix and force vector are on opposite sides of the general finite element equation $Ka = f$, indicating that the stiffness calculated from the above must be subtracted from the global stiffness matrix. The contribution of each contact point to the global stiffness matrix contains four terms when two deformable surfaces are in contact. The first relates the forces on surface $A$ to the displacements of surface $A$ through point $k$:

$$K_{ij}^{(AA)} = \kappa \phi_i^{(A)} \phi_j^{(A)},$$  

(4.57)

the second gives the change in force on surface $A$ when point $p$ on surface $B$ moves:
\[ K_{ij}^{(AB)} = -\kappa \phi_i^{(A)} \phi_j^{(B)} , \]  

while the equal and opposite stiffnesses on surface B are similarly,

\[ K_{ij}^{(BA)} = -\kappa \phi_i^{(B)} \phi_j^{(A)} , \]  

and,

\[ K_{ij}^{(BB)} = \kappa \phi_i^{(B)} \phi_j^{(B)} . \]  

When surface B does not move in an iteration, as is the case for contact with Tool surfaces that are stationary or follow a predictable path, only equation (4.57) need, or indeed can be applied.

Needless to say, equation (4.52), its opposite and equations (4.57) to (4.60) are summed for every contacting point in the model and assembled into the global stiffness matrix in the usual manner for finite elements. It is now easy to see that equations (4.58) and (4.59) couple together the degrees of freedom in surfaces A and B, ensuring a non-singular system matrix. Note also that equations (4.57) to (4.60) maintain the symmetry of the global stiffness matrix.

The procedure outlined above is very similar to that given by Petersson [51]. That study, however, only considered the case where \( \phi_i^{(A)} \) and \( \phi_i^{(B)} \) are identical (ie. perfectly conforming elements) and only briefly considered problems of curved elements. The use of many discrete contact points over each element surface is one of the major improvements of the procedure presented here. Also, as the models in the following chapters verify, gross sliding contact behaviour is well handled by this approach.

One concern with this contact algorithm is the problem of selecting a suitable value for the normal contact stiffness, \( S_n \), of equation (4.49). Experience shows that a value similar to a typical Young’s Modulus for the material under study is a good choice. The higher the value, the more strictly are the contact constraints applied. However, in cases where perfect mating of surfaces is impossible, very high values must be avoided, as they could lead to spurious coupling effects. In general, a compromise value can be chosen that keeps overlap depths to an acceptable minimum.

Finally, it is possible to derive the formulae presented in this section using another approach. Writing the discrete constraint equation at a contact point as,

\[ C(a) = (dp - dk).n = 0 \]  

a penalty approach approximately satisfying this is found by adding the variation of \( \pi C(a)C(a) \) with respect to \( da \) to the discretized virtual work equation (4.18). Penalty number \( \pi \) is then seen to be proportional to the normal contact stiffness, \( S_n \).
4.4.2 Friction Models

Application of the Principle of Virtual Work (see Section 4.2.1) ensures satisfaction of the equations of equilibrium (or of motion if inertia forces are included) as well as any prescribed boundary tractions. The role of a friction model is to provide tangential (shear) tractions over contacting areas appropriate to the sliding conditions in effect.

In an approach analogous to that of the previous section, tangential springs at each contact point will serve to transfer forces resisting relative slip between contacting regions. In three dimensions a pair of orthogonal vectors tangential to surface B at point \( p \) in Figure 4.5 are chosen for the directions of the “springs” associated with the contact point at \( k \). Since any pair of tangent vectors, \( t^1 \) and \( t^2 \), could be used, SimForm aligns \( t^1 \) with the direction of surface coordinate \( \xi_1 \) at \( p \), with \( t^2 \) set to \( n \times t^1 \). Equation (4.54) is now modified to:

\[
\kappa_{qr} = n^q n_r S_{t1} t^1_q t^1_r + S_{t2} t^2_q t^2_r ,
\]

(4.62)

where \( S_{t1} \) and \( S_{t2} \) are the tangential spring stiffnesses. If these are zero, the frictionless conditions of the previous section remain in effect. Alternatively, very high tangential stiffnesses simulate no-slip conditions. These are by no means the only possibilities, since combinations of stiffness values in equation (4.62) and tangential forces can be applied to simulate any desired friction relationship, including non-linearity.

Remembering that the total deformation is broken up into a number of smaller increments, with several iterations needed to solve for non-linearities in each increment, one can see other possibilities for controlling friction. If a tangential spring is expected to return to its position at the start of deformation, then no-slip conditions are simulated. If on the other hand it is set to transfer zero force in its position at the start of each increment, then sliding friction may be simulated.

A particularly good model for inter-ply slip friction would be the viscous response of a thin layer of Newtonian fluid between the contacting surfaces. In a time increment of length \( \Delta t \), such “viscous” friction may be applied by setting the tangential stiffness to:

\[
S_{t} = \frac{\eta}{\Delta t \theta},
\]

(4.63)

where \( \eta \) is the viscosity in Pa.s of the fluid layer of thickness \( \theta \). Multiplying equation (4.63) by a unit slip distance in metres returns the shear stress in Pascals. Hence, in subsequent iterations the total distance slipped in the increment is used to calculate the shear forces to be accumulated into the system force vector. In cases where there is a finite inter-layer thickness, as the viscous
model intends to simulate, it will be useful to add a small gap between contacting surfaces to ensure shear strain rates in the inter-layer are realistic. A gap of constant thickness may be added on to $d$ in equation (4.48), and it must also be accounted for in the contact detection and separation algorithms described in the following section.

The frictionless and viscous (linear velocity-dependent) friction models will be used exclusively in the simulation examples presented in the following two chapters. Note that non-linear velocity-dependent friction, as well as Coulombic friction in which no-slip conditions are applied until a specified yield stress is reached, have also been tested in SimForm. However, convergence was slow or difficult to achieve in these cases, especially with Coulombic friction due to the non-smooth non-linearity present.

### 4.4.3 Managing Contacts

The development in the previous sections provided all the formulae necessary for coupling known contacting regions and applying suitable friction forces between them. In addition to these procedures, considerable overhead must be spent on managing contacts. Ignoring trivial procedures such as updating tool positions and calculating contact stresses, the three main management tasks that need to be performed are:

1. Detection of new contacts before each time increment, including at the start of the simulation.
2. After each iteration, updating the contact points to account for the slip that has occurred, to ensure the nearest points on the touching surfaces are held in contact.
3. Checking for loss of contact after each increment.

These three tasks are now discussed in more detail.

Contact Detection consists of checking whether each unattached contact point has penetrated another surface during the last increment. This is a complex task, and with large numbers of contact points and many surfaces it can be very slow to perform. For this reason, it is divided up into a series of tests which must all be passed for contact to be established. For efficiency, the most trivial rejection tests should be performed first. The first test should check whether the two surfaces are *allowed* to touch each other. As in the example input file given in Appendix B, the user should specify valid contact partners at the start of the program. The second test should check whether the contact point is anywhere near a given surface, and neglect the possibility of contact between them if it is not. Using the notation of Section 4.4.1, the steps used in SimForm to detect whether point $k$ has penetrated surface B during the last time increment are:

1. Ensure point $k$ is allowed to touch surface B.
2. Calculate the extents of the “trivial rejection box” containing surface B, with extra space to accommodate displacement of the surface and the contact point in the last increment. Reject the possibility of contact if \( k \) is outside this box.

3. Obtain an estimate for the surface coordinate \( \xi^{(p)} \) of \( p \), the closest point on surface B to \( k \).

   Iteratively find the exact \( \xi^{(p)} \) using the “nearest point” algorithm described below, and ensure \( p \) is indeed on surface B, not on a neighbouring surface patch.

4. Calculate the depth \( d \) of \( k \) into surface B and ensure it is positive (ie. beneath the surface).

5. Calculate the equivalent depth \( d_{\text{prev}} \) at the end of the previous time increment. Contact is detected if this is negative or if \( \xi^{(p, \text{prev})} \) was outside the range of surface B.

At the start of the simulation, initial contacts are detected by a slightly different procedure. As there are no previous time increments, step 5 is omitted. Step 4 proceeds as above, except that contact is detected merely if depth \( d \) is within a user-specified tolerance of surface B, positive or negative (see the \*CONTACTTOL command in Appendix B). This allows for contact with curved tools to be detected more easily in cases where tool displacement would otherwise delay first contact until after the first time increment.

During an iteration a given contact point \( k \) and its nearest point \( p \) are both expected to undergo displacement. In general, the movement will also result in \( \xi^{(p, \text{prev})} \) no longer representing the nearest point to the new location of \( k \). A procedure is therefore required to find the new nearest point - the same procedure being used to find \( p \) from an initial guess in the contact detection algorithm above. Due to the use of high order surface interpolation, an iterative approach must be used in the “nearest point” algorithm. Each iteration proceeds by finding the plane described by the \( dx/d\xi_1 \) and \( dx/d\xi_2 \) vectors at the current estimate of \( \xi^{(p)} \). The next estimate of \( \xi^{(p)} \) is then taken as the point on the plane closest to \( k \). Iteration proceeds until \( \xi^{(p)} \) is sufficiently unchanged in an iteration. If \( \xi^{(p)} \) leaves the area addressed by surface B during the algorithm, it continues to search on the adjoining surface, if it exists. Updating of the contact points is accompanied by calculation of the distance they have slipped along the other surface, used for friction calculations in subsequent iterations of the system solution process. The nearest point algorithm normally converges quite quickly. However, in cases where a contact point is some distance from a surface of high curvature, convergence can be slow. The algorithm is made simpler by the choice of \( C_1 \)-continuous bicubic Hermite surface interpolation in SimForm, since there will be no sudden changes in slope at element boundaries which would otherwise produce regions in which there is no surface normal passing through \( k \).
The final contact management task is to check whether contact between surfaces is lost. Contacts may be lost in one of two ways. If after the “nearest point” procedure $\xi^{(p)}$ is shifted beyond the edge of surface B, and there is no adjoining element or tool surface, then contact is lost by “slipping off”. Alternatively, contact may be lost if the normal stress calculated from the distance two surfaces are pulled apart exceeds a user-specified Separation Stress.

Some final comments concern the ordering of the above management tasks in the general finite element scheme. Firstly, contact loss should immediately precede detection routines since a point may lose contact with one surface and gain contact with another in the same increment. Secondly, it is apparent that contact loss and detection must be carried out between increments of time, after non-linearities are resolved by iteration of the global system of equations. Since changing contacts can bring about a vast change in the loading on the system, short time steps are necessary for accurate modelling. Other reasons for keeping time steps small are for improving friction calculations, maintaining the assumption of constant surface normal $n$ in Section 4.4.1 and generally reducing the number of Newton-Raphson iterations required each increment.

4.5 Finite Element Algorithm

The following is a summary of the main program in SimForm, containing the time increment and iteration loops needed to solve non-linear finite element problems involving large displacements and contact.

1. Read and check input file
2. Set initial tool positions
3. Detect initial contacts
4. Set time = 0
5. Output geometry and stresses
6. Start of increment loop
   1. Set length of this time increment and advance time
   2. Calculate time dependent simulation variables and loads
   3. Update tool positions
   4. Start of iteration loop
      1. Clear system force vector
      2. Add loading terms (incl. contact) to system force vector
      3. Get $\text{LoadNorm} = \text{norm of system force vector}$
      4. Subtract internal stress terms (cf. integral in equation (4.19))
      5. Get $\text{ResNorm} = \text{norm of system force vector (now the residual)}$
      6. If first iteration skip convergence check and go to 6.4.8.
      7. if $\text{ResNorm}/\text{LoadNorm} < 0.01$ (say) then converged, go to 6.5.
8. Clear system stiffness matrix
9. Add system stiffness coefficients
10. Add load stiffness terms (incl. contact stiffness)
11. Apply essential boundary conditions
12. Solve system of equations to obtain displacements in iteration
13. Update geometry of model
14. Update contact points
15. Go to start of iteration loop (6.4.)
5. Calculate and output tool loads
6. Output geometry and stresses
7. If total simulation time reached, go to End (7.)
8. Check for lost contacts
9. Detect new contacts
10. Go to start of increment loop (6.)
7. End
Chapter 5: Determining Model Parameters

5.1 Introduction

The material properties of a fibre-reinforced thermoplastic will vary considerably over its temperature forming window. At high thermoforming temperatures it would be reasonable to assume the fluid-like aspects of the material are dominant and to ignore any elastic effects apart from the axial stiffness of the fibres. At lower temperatures and higher forming speeds, however, the material response may have more parallels with that of a rubbery solid. The true response is viscoelastic, an oft-mentioned and broad term which in this case will be interpreted as meaning the molten CFRTP material exhibits an instantaneous elastic response with time-dependent molecular realignment (flow) taking place after some delay to relieve internal stresses. In other words, a Maxwell-type fluid model of a spring and dashpot in series would seem appropriate.

Even if a viscoelastic model of the material were available, the problem of obtaining values for material parameters governing elastic and viscous responses at several forming temperatures would remain. Experimental data from which model parameters may be extracted is scarce, especially when particular temperatures or fibre/matrix combinations are of interest. The problem is compounded by the choice of experimental approaches designed to produce results only useable with a given material model. An example is Wheeler and Jones’ [52] use of a linear oscillator to measure the anisotropic viscosity parameters of molten CFRTPs according to the Ideal Fibre-Reinforced Fluid model [20,21].

This chapter attempts to find appropriate SimForm model parameters for PLYTRON 0.5mm, 35 vol% continuous glass fibre reinforced polypropylene, introduced in Chapter 1, based on tests carried out with this material at elevated temperatures. Although the SimForm model is not strictly a viscoelastic model as discussed above, it does contain separate viscous and elastic components. The only mode of permanent deformation in the model is viscous inter-ply slip, which will be characterised separately from the other material parameters by comparison with ply-pull-out tests, as described in Section 5.2. Following that, Section 5.3 describes the trial-and-error approach used to find appropriate values for the elastic intra-ply properties by comparison with the loads and part shapes observed in the free bending tests of Martin et al. [38]. Since it is possible for the material to bend in several distinct modes, different assumptions of how the material deforms will lead to different material properties. Results for two such assumptions will be given, and comments will also be made on how the elastic ply model could best be modified to reproduce some of the viscous phenomena observed in reality.

Note that no attempt will be made to determine the effect of temperature on material properties nor to propose material properties valid over the entire forming window. The emphasis is merely on being able to understand and reproduce the most important aspects of molten composite behaviour in a finite element model.

5.2 Inter-Ply Slip/Friction Parameters

In SimForm, and indeed in most of the discrete-ply models described in Section 4.1.2, the extreme shearing undergone by the thin layers of pure matrix between plies is approximated by a friction/lubrication model across a negligible inter-ply gap. Friction conditions, as applied using the equations in Section 4.4.2, could range from frictionless to no-slip, with the capability to handle any non-linear yield or velocity-dependent relation. In spite of these possibilities a simple linear velocity-dependent relation was chosen, not just because of its computational simplicity, but according to evidence presented in the following section, it also appears to be quite realistic for describing inter-ply slip.
5.2.1 Ply-Pull-Out Experiments

Scherer and co-workers [4,41,42] used the ply-pull-out test depicted in Figure 5.1 to measure inter-ply slip resistance.

![Figure 5.1 Schematic of Scherer’s ply-pull-out experiments [4].](image)

In the tests a stack of three plies were held at certain temperatures and pressures between the plates of a heated press. With the outer plies held in place the centre ply was pulled out at constant velocity with loads and displacements measured. All major variables in the test were varied, and the following points summarise the findings:

1. Slip took place only after a certain yield shear stress was exceeded. Thereafter loads increased to a maximum before gradually reducing to zero as the ply was pulled out.
2. Increasing the temperature lowered the loads.
3. Normal pressure was found to have a minor influence with high pressures causing some increase in resistance. This suggested that inter-ply slip was a lubrication phenomenon.
4. The fibre orientations in the stack were found to have some influence on the loads. In particular, increasing the relative angle between fibres across the slip zone reduced resistance.
5. At a given temperature, the pull-out velocity appeared to have the most significant influence on the loads measured.

The material Scherer used for his tests was an experimental 0.2 mm ply, 41 Vol% unidirectional carbon fibre reinforced polypropylene, produced by ICI PLC. of England. The material differs from the 0.5 mm GF/PP PLYTRON studied here, but the results are thought indicative or even transferable to all reinforced polypropylene materials, or indeed to other fibre/matrix combinations. Figure 5.2 shows a typical velocity-dependence curve from Reference [4], for the 0°/0° case in which the fibres in all plies are aligned with the pull-out direction.
From Figure 5.2 it can be concluded that the yield shear stress is fairly low compared with the loads measured at these modest slip velocities. Further, despite Scherer’s use of the non-linear viscoplastic Herschel-Bulkley law for modelling the flow, it appears that the linear viscous relation shown by the dashed line in Figure 5.2 quite adequately describes the observed behaviour, and is not appreciably far from the experimental data either. As noted in Section 4.1.3, a linear law corresponds to the assumption that inter-ply slip is shear across constant thickness inter-layers of Newtonian fluid.

From a modelling viewpoint it is unwise at this stage of development to add any extra variables or non-linearity by choosing a more complex friction law, especially since considerable scope exists for studying the influence of slip resistance by varying the slope of the shear stress versus velocity curve. Furthermore, due to inherent variations in consolidation and inter-layer thicknesses, no single friction law can be considered “correct”.

5.2.2 Simulated Ply-Pull-Out

To verify the implementation of friction in *SimForm*, ply-pull-out tests were simulated with the program. The friction law followed the best-fit line in Figure 5.2, so that,

$$\tau = c_f \cdot \nu = 1 \times 10^7 \cdot \nu$$

(5.1)
gives the shear stress $\tau$, in Pascals, transferred when sliding at velocity $\nu$ in m/s.
Figure 5.3 Load variation during simulated ply-pull-out at 0.25 mm/s.

Figure 5.3 shows a typical load curve produced by the program. The load rises steeply to around 2500 Pa as expected from Figure 5.2 at 0.25 mm/s. As the ply is pulled out the resistance gradually reduces as observed by Scherer. Note that the decline in the load occurs in steps as discrete contact points, evenly spaced along the slip surface, periodically lose contact.

The objective of this section has been to justify the use of a linear velocity-dependent friction law for inter-ply slip, and to verify its implementation into SimForm. Apart from the single emulation of Scherer’s ply-pull-out results, no specific friction values have been deduced for the materials of interest to this study. However, comparing matrix viscosities and shear rates for different inter-layer thicknesses it is possible to obtain ball-park values for inter-ply slip resistance at different temperatures and with other matrix materials. Even so, the following section presents a method of fitting all model parameters simultaneously, including the single inter-ply slip parameter.

### 5.3 Intra-Ply Properties

Typically, tests to determine material properties are based around isolating particular modes of deformation within the confines of a universal testing machine. Given that predictable, homogeneous deformations will take place, analytical expressions are derived for a particular material model to return model parameters in terms of the measured loads and displacements. Several different tests would be needed to determine all the properties of a given molten continuous fibre reinforced thermoplastic. An example of one would be the ply-pull-out test from the previous section. However, as noted in Section 4.1.1, inter-ply slip always seems to occur to some extent when other deformation modes such as transverse flow and intra-ply shear are desired, making model verification and parameter determination by this process a non-trivial exercise.

With its ability to handle any arbitrary geometry or loading situation, the finite element method offers an alternative approach to determining model parameters. Having found a suitable model for the material based on considerations of its structure as well as experimental observations, this is implemented in a finite element program, as has been done in SimForm. Next, a reasonably simple forming experiment involving several or all major deformation modes is simulated in the program using an initial estimate of parameters. Simulations are then repeated using different model parameters on a trial-and-error basis until the loads, part shape and
deformations measured or observed in the actual experiment are reproduced to sufficient accuracy. An advantage of this approach is that different constitutive laws may be quantitatively and qualitatively compared with each other and the real material. Moreover, there is no extra step involved in taking the finite element model and using it to simulate other forming examples.

Something to avoid in designing forming tests for this purpose is friction or any other hard-to-quantify source of load resistance. A test well suited to this fitting process is the free bending experiment described in the following section.

5.3.1 Free Bending Experiments

Martin et al. [38] carried out a series of three-point bend tests on 8-ply unidirectional laminates of PLYTRON at thermoforming temperatures. The setup used, as well as specimen dimensions are shown in Figure 5.4.

![Figure 5.4 Schematic of free-bending experiments of Martin et al. [38].](image_url)

During the tests the punch was displaced a considerable distance, which would have resulted in sliding friction conditions if rollers had not been used as supports. By performing the tests inside an oven in a universal testing machine, loads could be measured under isothermal conditions. Figure 5.5 shows a typical load curve, in which the cross head was stopped after approximately 2.5 seconds.

Three distinct loading stages are evident from Figure 5.5. In the initial stages of forming, loads increase sharply as the laminate deforms elastically. After a time, viscous flow in the part causes loads to reach a plateau and even drop off as the forming geometry changes. Finally, after the crosshead is stopped, loads decrease in a manner typical of relaxation, converging to some residual elastic load.
During the first moments of forming the section between the rollers was observed to deform into a v-shape with outer sections of the laminate remaining horizontal, behaviour characteristic of shear-weak materials. Thereafter the outer sections started to rotate upwards, and this continued even after the crosshead was stopped as internal stresses were relieved through viscous flow. The ends of the specimens appeared steplike after deformation, as in Figure 4.1, indicating inter-ply slip had occurred.

Based on two alternative assumptions of how the material has deformed, the following sections aim to find SimForm parameters which in simulated free bending reproduce the above observations and the load curve in Figure 5.5.

5.3.2 2-D, Extensible Assumption

At temperatures around the melting point of polypropylene, 165 °C, the stiffness of glass fibres is not expected to differ appreciably from the room temperature value of around 70 GPa. Since the modulus of the molten matrix is negligible by comparison, the rule of mixtures gives the fibre-direction stiffness of molten PLYTRON, with $v_f = 0.35$ as,

$$E_f = 24.5 \text{ GPa}. \quad (5.2)$$

This places an upper limit on the stiffness that can be assumed in a continuum model of molten PLYTRON. The above value may only be exceeded if matrix-rich interlayers are of considerable thickness meaning a higher effective fibre volume fraction exists in the centre of the plies. A critical variable in all the models presented here and in the following section will be the anisotropy ratio, defined as the ratio of fibre direction to transverse stiffnesses in the ply.

Using strain energy function (4.38), and given that the fibre stiffness far exceeds that in other directions, equation (4.39) gives,

$$K_4 \leq 3 \times 10^9. \quad (5.3)$$

Also, since no evidence is available to differentiate $G_L$ and $G_T$ in (4.39), it will further be assumed that,

$$K_5 = 0. \quad (5.4)$$

In this section a 4-ply, two dimensional free bending simulation was used as a test-bed for different model parameters. As later figures will show, symmetry was also used to halve the model size. Accordingly, loads should be lower than those in Figure 5.5 by at least a factor of 4.
Note also that matrix-rich inter-layers were assumed to take up 20 percent of the thickness of each ply. The plies are thus 0.4 mm thick with 0.1 mm gaps separating them. These values are thought reasonable considering the micrographs of PLYTRON laminates in Figure 1.1.

From initial parameter tests with the 2-D model, the following points were noted:

- Taking the maximum value of $K_4$ from equation (5.3) with other parameters adjusted to give reasonable loads, the resulting high ratios of fibre direction to transverse and shear stiffnesses produced the v-shape deformation observed by Martin et al., but little inter-ply slip could take place. Slip friction parameters as high or higher than that in equation (5.1) had little influence on the load, making it impossible to produce the viscous peak shown in Figure 5.5.
- It was not possible to reproduce both the initial slope of Figure 5.5 and the final residual load using the same elastic ply parameters. The initial slope involves a far greater stiffness, indicating the later occurrence of viscous flow within the plies.
- At high anisotropy ratios out-of-plane buckling was prevalent in the plies under the punch.
- The tool loads were found to be relatively insensitive to the normal contact stiffness coefficient, $S_n$, from equation (4.49). However, unstable behaviour was noted at very high values, while with lower values too much penetration was encountered between touching surfaces. In the example described below, $S_n$ was chosen to be $2.5 \times 10^8$ between plies and against the punch. A lower value of $1 \times 10^8$ was employed for contact against the roller, since higher values caused noticeable jumps in the load curves as contacts were lost.

To reintroduce the expected influence of inter-ply slip, made negligible at high anisotropy ratios, the parameter search for the 2-D model assumed the effective stiffness of the ply was somewhat lower than the maximum of equation (5.2), perhaps due to some initial slack in the fibres, fibre migration or even microbuckling. Emphasis was placed on reproducing both the measured peak load and relaxation to the residual elastic load in Figure 5.5. From these guidelines the best-fit parameters for equation (4.38) were found to be,

$$K_1 = 5 \times 10^4, \ K_3 = 5 \times 10^6, \ K_4 = 1 \times 10^8, \ K_2 = K_5 = 0.$$  \hspace{1cm} (5.5)

Figure 5.6 shows the simulation at several points in time, while Figure 5.7 plots the load curve for several inter-ply slip friction coefficients, defined in equation (5.1).

![Figure 5.6](image)

**Figure 5.6** 2-D simulated free bending with friction coefficient $c_f = 2 \times 10^7$.

Punch stopped after 2.5 seconds.

Several points can be noted from the above figure:
• The model reproduced to some extent the shear-weak deformation observed by Martin et al., with laminate sections outside the rollers rotating downwards relative to inner laminate sections. By contrast, in an isotropic, elastic beam no curvature would be observed at the rollers. If anything, the bends under the punch and at the rollers are not as sharp as initially observed in the real material, indicating the actual ratio of anisotropy is higher than the value of 2000 used here.

• The final part shape at 5 seconds is close to that given by Martin et al. in reference [38]. It is interesting that the curvature of the laminate is somewhat less than that of the punch and the roller. The real laminate was the same - possibly due to the bending stiffness of the fibres. In the model it was found that as plies were made thinner, shear became less significant in bending. This suggests that even highly anisotropic materials approach thin shell behaviour in the limit as ply thickness is reduced.

• Convergence was slow in later time steps. To avoid excessive computation, time was advanced before complete convergence was obtained, resulting in the strange bulges in the load curves of Figure 5.7. The contact procedure appears to be at the root of this problem, and this is looked into further in Section 7.1.

**Figure 5.7** Load curves for 2-D simulated free-bending under three friction conditions. Punch moves at 500 mm/min for 2.5 seconds, then stops.

The load curves for this example show much similarity to Figure 5.5, especially during the latter part of forming and in the relaxation phase. However, at the start of forming the stiffness is much too low, being less than a quarter of the desired value. Furthermore, since the three friction coefficients could be interpreted as different forming rates, the results conflict with the experimental findings, which showed the same initial stiffness whether the punch speed was
50 or 500 mm/min [38]. However, the observed increase in peak load with higher forming rates is reproduced by the model.

The correct load response could be reproduced by a yield phenomenon in the inter-ply slip model, with viscous flow following. According to Scherer [4], and as noted in Section 5.2.1, this is likely to be true to some extent. However, in the following section a different and more plausible explanation of the deviation of the model from real material behaviour is presented.

5.3.3 3-D, Inextensible Assumption

In their free-bending experiments, Martin et al. observed that transverse spreading under the punch caused a local increase in strip width of around 10 percent [38]. This single observation leads to several conclusions about the material and how it deforms:

1. The fibre-direction stiffness of PLYTRON is as high as in (5.2), and approaches the condition of fibre inextensibility.
2. The true deformation is three dimensional so that rather than shearing along its entire length, the fibres in each ply take the easy option and spread out under the punch.
3. The spreading involves fibre buckling and wrinkling in such a way that through-thickness intra-ply shear is largely avoided. The occurrence of inter-ply slip and the resulting step-like laminate ends shown in Figure 5.6, and observed by Martin, are thus explained.
4. The complex spreading and wrinkling zone under the punch is created through inelastic transverse flow and to a lesser extent intra-ply shear. Bent fibres in this region are likely to be the main source of the elastic residual load indicated in Figure 5.5.

Considering these ideas, the SimForm model is well suited to describing the deformation of most of the laminate. It is only in the squeeze zone under the punch that viscous behaviour in the plies needs to be modelled. However, what happens in the squeeze zone influences the loads and boundary conditions for the remainder of the laminate, so that the SimForm model is likely to be invalid as soon as local spreading occurs. Nevertheless, since it appears from Figure 5.5 that loading is elastic in the first few moments of forming it is possible to obtain an estimate of the initial elastic properties of PLYTRON. Furthermore, it remains to be seen whether elastic spreading can occur under the punch in a 3-D model, causing the same drop-off in loads measured in the experiments.

A three dimensional simulation of the free-bending test was set up to look into the above issues. As before, only 4 layers are modelled, with symmetry taken advantage of in two directions. With $K_4$ given the maximum value from inequality (5.3), it was expected that the extreme anisotropy needed to fit the loads would produce such a shear-weak response that inter-ply slip friction would have no bearing on the initial loads. This proved to be the case, and meant also that the target load slope would be an eighth of that in Figure 5.5, or about 0.2 N in 0.1 seconds. The best fitting parameters for the initial loading, using strain energy function (4.38), were found to be,

$$K_1 \approx 1.2 \times 10^5, \ K_3 = K_4 = 3 \times 10^9, \ K_2 = K_5 = 0,$$

which has an anisotropy ratio $(E_f/E_T)$ of 25000. Note that from equations (4.39), only the sum $(K_1+K_2)$ need be around $1.2 \times 10^5$. Also, $K_3$ is merely given a “high” value, depending on how well incompressibility is to be approximated. Figure 5.8 shows two views of this model, which exhibits both the characteristic v-shape deformation of a shear-weak material, as well as spreading under the punch.
Despite the occurrence of spreading in the model, the load continued to rise at the initial rate, further backing up the conclusion that only viscous intra-ply flow, especially in the squeeze zone under the punch, could explain the drop-off in loads indicated in Figure 5.5. Unfortunately, shortly after the time step shown in Figure 5.8, the model became unstable, preventing the simulation from running the full 5 seconds. It has been found very difficult to use both the contact routines and such high degrees of anisotropy simultaneously. This problem is looked at in more detail in Section 7.1.

The incompressible, inextensible formulation described in Section 4.3.3 has also been applied to this problem. Using the same value for $K_1$ as in (5.6), the inextensible material was only just stiffer than the above model, indicating that using the ply stiffness value from equation (5.2) very nearly approaches inextensible behaviour. However, as shown in Figure 5.9, the inextensible model quickly displayed tiny buckles over elements in compression, and shortly afterwards became unstable. It is thought that this “instant buckling” stems in part from the use of linear interpolation for the inextensibility and incompressibility constraints, when displacement is described by a cubic. Using quadratic in-plane interpolation for pressure and tension variables seems to improve the results, but is impractical since it adds too many extra degrees of freedom to the model.
Figure 5.9 Unstable behaviour in inextensible, incompressible free-bending model.

To summarise this section, it appears that only a three-dimensional model can describe the deformations taking place in the free bending experiments of Martin et al. If the instability problems described above could be overcome, the SimForm model using the parameters given by (5.6) would be ideal for modelling the PLYTRON material, provided some extra treatment for the viscous transverse flow and intra-ply shear processes occurring at stress concentrations was added. It appears that the addition of a creep law, in which creep strain rates are related to total elastic strains and stresses, would make up for the deficiencies of the current model. As a first step, a single, isotropic creep coefficient (ie. viscosity) controlling these effects should be implemented. Extension to a more complicated non-linear viscoplastic law may eventually be deemed appropriate, as was already suggested for inter-ply slip.

5.4 Simulation Parameters

This chapter has highlighted several deficiencies in the SimForm model and pinpointed areas in which it could be changed to more realistically describe the behaviour of PLYTRON during thermoforming. In spite of this, it does describe two aspects of molten CFRTP laminates well: the extreme degree of anisotropy in each ply, and inter-ply slip. Furthermore, the program is capable of simulating the behaviour of the present model in any number of forming situations involving loading by moving dies or pressures acting on surfaces. In the following chapter, SimForm will be used to simulate several familiar forming cases, and the results compared with behaviour expected from experiments and common sense.

The material parameters in (5.5) will be used in all simulations presented in the following chapter. Although these values represent a material neither as stiff nor displaying the same degree of anisotropy as the real composite, they resulted in more reasonable overall loads than the more “correct” values in (5.6), and allowed the estimated inter-ply slip friction values from equation (5.1) and Figure 5.7 to have the expected influence on the loads. In any case, all such highly anisotropic elastic materials are likely to behave similarly. The final reason is that the stability problems experienced with very high anisotropy ratios, mentioned in Section 5.3.3, are avoided.

The use of bicubic Hermite in-plane, linear/quadratic through-thickness three-dimensional continuum elements in SimForm falls under the category of ‘Shells as a special case of three dimensional analysis’, the subject of chapter five of Zienkiewicz and Taylor’s The Finite Element Method, Volume 2 [32]. There, the authors warn of the error introduced by using linear interpolation through the thickness of the shell. Tests with SimForm have confirmed the effects
described: in isotropic and mildly anisotropic materials the bending stiffness of elements with 2 nodes through the thickness was found to increase with parameter $K_3$, but no such influence was noted when 3 thickness nodes were used. Despite more than doubling computational requirements per element, quadratic interpolation was a necessity for modelling isotropic materials such as the rubber diaphragms described below.

In contrast, since highly anisotropic fibre-reinforced materials deform mainly through shear, performance of the ply model was almost identical whether 2 or 3 nodes were used through the thickness. It is only in bending about the axis of a fibre that differences between linear and quadratic interpolation are noted for this transversely-isotropic material. For economy, 2 thickness nodes will be used universally for these materials.

Several of the examples in the following chapter simulate pressure forming using silicone rubber diaphragms, a process described in Section 1.2.2. In the simulations, the rubber diaphragms are also described by strain energy function (4.38), except that anisotropic terms are omitted. Based on unidirectional extension tests to 200 percent strain, material parameters for the diaphragms were chosen to be,

$$\begin{align*}
K_1 &= 1.1 \times 10^5, \\
K_2 &= 0.25 \times 10^5, \\
K_3 &= 5 \times 10^6, \\
K_4 &= K_5 = 0,
\end{align*}$$

(5.6)

with $K_1$ and $K_2$ corresponding to Mooney parameters [50].
6.1 Introduction

6.1.1 Simulation Capability of SimForm

The SimForm program was developed for two main purposes:
1. To provide a platform for testing models of molten CFRTCP laminates, and,
2. To enable thermoforming operations to be simulated using the material model thus obtained.

Based on the experimental observations of Section 4.1.1, the program was developed specifically to handle discrete-ply laminate behaviour, with the material model separately describing intra- and inter-ply properties. While in an expanded project the program framework could well be used with several material models, this study has concentrated purely on the highly anisotropic elastic ply / viscous inter-ply slip model, referred to simply as the SimForm model.

In Section 5.2 the use of a linear velocity-dependent shear stress relationship for inter-ply slip resistance was largely justified. However, the mere partial verification of the combined model later in that chapter calls into question its use in extensive thermoforming simulations, at least until the modifications suggested in Section 5.3.3 are implemented. Nevertheless, the behaviour of the current model in real forming situations is of interest, and if investigated would allow many of the features of SimForm, developed to fulfil purpose 2 above, to be fully utilised. In particular, interaction between moving tools and the material, as well as forming through pressure application on elastomer diaphragms are both areas of practical interest already handled by the program. The success of these modelling features can be assessed regardless of any flaws in the molten composite model.

In this chapter several thermoforming simulations will be performed, all using the model parameters from Sections 5.3.2 and 5.4. Normal contact stiffness parameters will be given for each simulation since it will occasionally be necessary to use different values, for reasons that will be explained at the time. Also, inter-ply slip friction coefficients will be adjusted or neglected as necessary to induce different behaviour in the composite.

An obvious point to note from the examples presented in this chapter is that they are somewhat trivial; the observed effects could equally be deduced from common sense. However, their simplicity, and the ability to assume correct behaviour from experience, makes them an ideal tool for program verification. There are two reasons why more complicated examples have not been attempted. The first is that larger models quickly fill the available computing resources, both memory and processing time. Hence, large models were impractical with the available workstations. The second reason is that even if larger computers were available - and they are - it makes no sense to do a large, detailed simulation using a flawed model, especially considering the lingering convergence problems noted in Section 5.3.2. Research time would be better spent developing an improved model.

Regarding the huge memory requirements mentioned above, a few simple calculations illustrate the size of the problem. Assuming each element contains 8 nodes, with 12 degrees of freedom per node (from the 4 nodal vectors in Figure 3.4), and double precision is used (8 bytes per real number), storage space for one element in the global stiffness matrix is $8 \times (8 \times 12)^2 = 72$ kilobytes. After assembly, some of this is shared with neighbouring elements, but memory usage rises during decomposition, and can as much as double depending on the matrix bandwidth. Considering a typical workstation has 64 megabytes of RAM, less that taken up by the system and other processes, not many elements may be used at all. While out-of-core storage of the global stiffness matrix would avoid these memory limitations, it does so at the great expense of speed. It is easy to see the advantage of using an explicit code, as ESI Ltd. did in their study (refer to Section 4.1.2 and references [12,47]), which does not require complete storage or...
decomposition of the global stiffness matrix. Section 7.2 looks at further ways of reducing the memory and processing requirements of the program.

In order to reduce the number of elements in the model, and thus computation time, all the examples presented in this chapter take advantage of any symmetry present. The models in Section 6.2 also assume the deformation is two-dimensional. While the computational benefits of these approximations are obvious, it is clear that they change the nature of the problem as well. For example, from the points made in Section 5.3.3 it is probably unrealistic to assume any practical forming problem involving CFRTP laminates is two dimensional. Furthermore, even when the tooling and initial laminate lay-up are symmetric, ply rotation and failure by in-plane wrinkling may occur in an unsymmetric fashion during forming. Finally, no matter how simple the die arrangement is, no symmetry exists in any sheet forming operation on laminates with more than 2 directions of reinforcement.

6.1.2 Post-Processing with SimView

Data visualisation is an essential part of numerical analysis, and to this end a graphical post-processor, SimView, was written by the author to display the results output by SimForm. SimView shares much code commonality with the GSA program of Chapter 3, since both were written for the Windows environment and involve interactive 3-D graphical display of the results of their respective analyses. Figure 6.1 shows a typical screen shot of the program in which stresses are plotted for the example from Section 6.3.1.

As input to SimView, SimForm post-processing files describe first the element topology and tool surfaces, then at selected time steps, tool positions, nodal geometry, stresses and contact loads (if required) are output. It is plainly impossible for a single image, or series of images over the duration of forming to display all the information in these files. In a multi-layer material as studied here, solid surface representation would obscure layers behind, so “wire-frame” graphics have been used throughout. Most of the figures in this chapter are HPGL (Hewlett-Packard...
Graphics Language) plots of wire-frame graphics generated by *SimView*. To enable greater control over how the results are visualised, *SimView* allows the following wire-frame graphical structures to be switched on or off:

- Element edges, in selected layers.
- Element surface detail in selected layers. Referring to Figures 3.4 and 4.3, these are curves of constant $\xi_1$ and $\xi_2$, at a chosen $\xi_3$.
- Element slice outlines on surfaces of constant $\xi_i$, in selected layers.
- Uniformly spaced fibres across surfaces of constant $\xi_3$, in selected layers.
- Contact stress lines (refer to Section 6.2.1).
- Tool patch outlines and surface detail.

In addition, contours of any of the 6 stress quantities (plus pressure and tension for the constrained model of Section 4.3.3) can be plotted in colour or monochrome patterns on the above element slices.

Viewing transformations controlled by dragging the mouse enable the images to be magnified and viewed from any angle with the program. Also, the triangular PLAY buttons in Figure 6.1 allow images at each time step to be cycled, providing an animated view of the forming process. Together, these post-processing features give an unparalleled view of the simulation results, and provide one of the greatest benefits of studying thermoforming in this way. In the limited space of this chapter, however, only a selection of the most informative images from the post-processing phase could be included.

6.2 2-D, 90° Bend Forming Examples

Producing a bend in sheet material is one of the simplest forming operations imaginable. It is, however, an important process since bends are present in the majority of sheet formed parts, and it provides the principal forming mechanism for the class of product including angle and channel sections, often produced by continuous processes such as roll forming.

When bending molten CFRTP laminates, it is difficult to avoid buckling in plies subject to compressive stresses in the fibre direction, inevitable on the inside of bends [4]. Chapter 2 also noted the negligible buckling resistance of CFRTP materials under compressive loading. Hence, with the aid of several simulations, bending of the most difficult case of unidirectional laminates is studied in this section. It is assumed that bending is over a very long specimen so that 2-D, plane strain conditions are applicable. As described under the *EQUATE command in Appendix B, such a 2-D model is produced in *SimForm* by constraining degrees of freedom in the width direction of the 3-D elements used.

6.2.1 Matched-Die Forming

Figure 6.2 illustrates the 90° bend matched-die tool and specimen geometry to be used in the simulations of this section. From symmetry, only one half of the part is modelled. As in Section 5.3.2, the examples use $4 \times 0.5$ mm ply, unidirectional laminates, less 0.1 mm inter-ply gaps. Normal contact stiffness factor $S_n$ (from equation 4.49) is given a value of $2.5 \times 10^8$ between plies and against tools, while inter-ply friction is applied with twice the resistance given by equation (5.1). Note that no clamping or other tensioning devices are employed in these examples.
In the first example, the punch was lowered at just over 7 mm/s, closing the tools to within 2 mm of each other in 5 seconds. Figure 6.3 shows the flawless forming achieved at this rate. Thin lines normal to the tools and laminate surface where they are touching show the magnitude of normal contact stresses, scaled so that 1 kPa = 1 mm in length, which can be compared against the ply thickness in the figure.

Unfortunately, forming at the slow rate shown in Figure 6.3 may cause problems with the non-isothermal thermoforming processes favoured in Section 1.2.2, since excess cooling will lead to parts of the laminate leaving the material’s temperature forming window. Without adding the complexity and reduced production efficiency introduced by heated tools or the use of an environmental chamber, forming at a higher speed would appear to reduce these problems, and improve the homogeneity of the material. Figure 6.4 shows the result of forming at a punch speed of just over 70 mm/s, 10 times the above rate.
Figure 6.3 Matched-die, 90° bend forming example. Slow forming at 7 mm/s.

Figure 6.4 Matched-die, 90° bend forming example. Fast forming at 70 mm/s.

Figure 6.5 compares the fibre stresses in the laminate region under the punch early in the forming examples shown in Figures 6.3 and 6.4. In the slow forming example inter-ply slip occurs easily, so that bending stresses are largely independent in each ply. By contrast, at high forming rates the laminate behaves more like a solid beam, with the ply on the inner bend radius under pure compression, leading to the buckling shown in Figure 6.4.
As Section 7.1 will discuss in greater depth, convergence becomes slower once buckling has occurred. The model may even be invalid at that stage since normal contact constraints between plies become unrealistic once the touching surfaces are no longer parallel. In any case, for parts in which any buckling is unacceptable, the simulation can be stopped once it occurs, and the forming deemed unsuccessful.

A further point to note is that since contact constraints are applied using a penalty method, or equivalently a series of lateral springs, the buckling phenomenon shown in Figure 6.4 corresponds to the case of ‘buckling of thin columns with an elastic foundation’. The real material has more like a viscous foundation due to the presence of matrix-rich inter-layers. Even though the type of out-of-plane kinking shown in Figure 6.4 does occur during thermoforming [9], the simulation would benefit if it was eliminated, for the following reasons:

• The ability of neighbouring plies to be oriented differently across the inter-ply zone represents a source of unnecessary degrees of freedom in the model.
• As noted above, contact equations become unrealistic once such buckling occurs. By eliminating these modes of instability, the model can run longer, while stability criteria such as equation (2.55) could still be applied at any time.
• In any case, in-plane wrinkling would still exist as an alternative mode of instability in any three-dimensional forming simulations.

Discussion Section 7.1 further investigates these ideas.

The matched-die forming described in this section is a process taken from the sheet metal industry and applied to CFRTP materials. While a bend is easy to form with sheet metals, more complex, 3-D parts often require tension-inducing edge clamping to eliminate wrinkling, and the same has been applied with success in the forming of thermoplastic composites [12]. However, applying clamping directly onto the laminate necessarily involves much wastage of these often expensive materials. Further, sticking with sheet metal processes is needless straight-jacketing when one considers the vastly lower loads required to form molten CFRTP laminates. This opens up the possibility of new and novel forming processes, one of which is looked at in the following section.

### 6.2.2 Double-Diaphragm Forming

The double-diaphragm pressure forming process has achieved much success in producing CFRTP parts [1,5,6,8,10,13-15]. Figure 1.3 of Section 1.2.2 illustrated the non-isothermal diaphragm forming process in which the heated laminate and diaphragms are transferred to cold tools for forming. There it was mentioned that reusable silicone rubber diaphragms have been used with some success, and it is this variant of the process that will be looked at here.

Figure 6.6 shows the set-up used in the simulations of double-diaphragm forming of a 90° bend. The same bend geometry as in the previous section is used, except that the bend radius is effectively reduced by the thickness of the lower diaphragm. Model parameters for the diaphragms are given in Section 5.4. The normal contact stiffness factor $S_n$ is given a value of
$1.0 \times 10^9$ for all contacts, the higher value necessary since greater loads are required to stretch and deform the rubber diaphragms.

**Figure 6.6** Set-up of double diaphragm, 90° bend forming simulations.

In the first example friction shear stresses of twice that in equation (5.1) are applied between all contacting surfaces. The upper diaphragm is pressurised from 0 to 100 kPa in 1 second, then held at that pressure. Figure 6.7 shows the part being formed at four different times. Lines representing normal contact stresses are scaled so that 10 kPa = 1 mm in length.
Figure 6.7 2-D, double-diaphragm 90° bend forming of a unidirectional laminate. Contours of fibre-direction stresses, in Pa, are plotted in the plies.
Figure 6.7 (cont.) 2-D, double-diaphragm 90° bend forming of a unidirectional laminate. Contours of fibre-direction stresses, in Pa, are plotted in the plies.

Note from the scale ranges in Figures 6.7 (a) and (b) that tensile stresses are predominant in the plies, especially in the lower ply where the degree of stretching is somewhat excessive. Hence, the part is deformed into its intended shape in (c) without buckling. However, soon after (c) the model developed the strange buckling shown in (d), which remained in the part. The buckling involves considerable penetration of the diaphragm into the tool, which a higher contact stiffness would have reduced. However, time prevented this from being attempted.
In the second example, shown in Figure 6.8, frictionless conditions were assumed between the diaphragms and the laminate. Here, build-up of compressive stresses has led to buckling in the top ply, and eventually through the laminate as the model broke down. This indicates it is the transfer of tension into the laminate by sliding friction against stretching diaphragms that is behind the success of diaphragm forming processes. Note that diaphragms could also provide the same benefits with the matched-die forming process, as long as the diaphragm thickness is accounted for in designing the tools. One of the main disadvantages of using matched-die forming with CFRTP laminates is this thickness inflexibility, since changing the laminate thickness or in some cases even the lay-up will require one of the dies to be replaced.

The rubber diaphragm forming process itself is not free of drawbacks. Firstly, since the current diaphragms are of substantial thickness, allowance has to be made for them in designing the tool. Secondly, from the simulations it is apparent that most of the pressure goes into stretching the diaphragms, not forming the material. This can lead to excessive squeeze flow in the laminate, and also means that substantial forming machinery is needed to handle the pressures.

A point not noted up to now is the special control problems involved in simulating pressure forming. As can be seen from the times in Figure 6.7, forming occurs rapidly at first, and only slows once sufficient membrane stresses are present in the diaphragms. For this reason, from an initial time step SimForm automatically chooses the duration of subsequent time steps so that displacements are not too large in any one increment (see the *DISPLCONTROL command in Appendix B for more details).

### 6.3 3-D, Dome Forming Examples

The hemispherical dome is among the simplest of doubly-curved structures, yet it presents certain forming difficulties for both CFRTP and metal sheets. The greatest of these is the build-up of compressive hoop stresses around the flange area, leading to the type of out-of-plane buckling shown in Figures 6.1 and 6.9, below. For this reason, dome forming has become one of the most common experiments carried out with CFRTP materials [5-11,13-15] and has also been the subject of several numerical studies [4,7,13,39,45].

Due to the interest in this forming geometry, several dome forming simulations have been performed with SimForm, and the following sections describe the results obtained. In all
the examples the lower, female tool consists of a hemisphere of radius 30 mm, with 10 mm radii around its edges. Where matched-die forming is simulated, the upper, male dome has a radius of 28 mm with 12 mm radii around its edges, to fit a 2 mm thick specimen between the closed tools. Symmetry has been taken advantage of to the maximum extent, so that only a quarter of the dome is modelled, and only laminate lay-ups that do not violate the symmetry are considered. Also, due to computing limitations, only very coarse meshes are used, limiting the detail of the solutions.

6.3.1 Buckling and Inertia

Figure 6.9 shows one of the earliest matched-die, dome forming simulations carried out with SimForm. In the example a 2 mm thick, 70 mm square of rubber diaphragm material, with isotropic parameters given in Section 5.4, is pushed into quarter of the female die.

At first it was not possible to run the model to completion, since buckling took place so rapidly that no amount of iteration could recover from the geometric non-linearity caused by the large lateral displacements involved. In such cases, SimForm repeats the increment using successively smaller time steps until displacements are within specified limits. However, since no viscous effects are present in this example, this proved ineffective. The problem was solved by including the inertia forces, neglected up to now, which introduce into the model the reality that no deformation can take place instantly.
Inertia forces per unit volume are equal to the product of the density of the material and the acceleration it is experiencing, and can be added into the body force term in equilibrium equation (4.1). It adds the following extra volume integral to the discretised virtual work equation (4.18):

$$- \int_{V_0} \psi \rho \ddot{u} dV_0 ,$$

where $V_0$ and $\rho_0$ are the initial volume and density, respectively, while $\ddot{u}$ is the acceleration, calculated below. The above integral is added into the system force vector.

Since inertia was added late in the program’s development, only a gross approximation has been used for the accelerations. As in Section 4.4.2, the velocity at the end of each time step is taken to be the displacement that increment divided by the duration of the time step, with zero velocity at time $t = 0$. From the velocities, components of acceleration are approximated by,
\[ \dot{u}_i^n = \frac{\dot{u}_i^n - \dot{u}_i^{n-1}}{\Delta t}, \]  
(6.2)

where \( \dot{u}_i^n \) is the velocity vector at the end of increment \( n \), and \( \Delta t \) is the time step. The above relation works adequately, but is not as elegant as that in the Newmark scheme described in reference [32]. Note that self weight can also be included by adding the acceleration due to gravity, \( g \), to \( \dot{u}_i^n \).

Based on the above equations, the inertial stiffness terms, added into the global stiffness matrix, are,

\[ \int_{V_0} \psi_m \frac{\rho_0}{\Delta t^2} \psi_n dV, \]  
(6.3)

from which it is easy to see how smaller time steps reduce displacements and provide added stability to the solution.

Returning to Figure 6.9, the model with inertia could now run to completion. As the dies closed together, the first 4 circumferential buckles became squashed, and took up higher frequency modes with 8 and then 12 waves, before these were finally flattened out. Note that in going from state (d) to (e) the centre buckle in the former figure shifts to the left in a non-symmetric fashion to make space for the new buckle. Figure 6.1 plots stress contours on the centre plane of the part, showing the compressive stresses in the flange responsible for the buckling. The stress component plotted in the figure is in the direction normal to the line from the centre of the dome to the far corner of the specimen (and all initially parallel directions), and is the nearest thing to a hoop stress in this example.

### 6.3.2 Matched-Die Unidirectional Dome Forming

The first practical example in this series involves matched-die forming a 2 mm thick, \([0_4]\) laminate into the tool described at the start of Section 6.3. Here, a 60 mm square quarter specimen was modelled using a uniform mesh of \( 6 \times 6 \) elements, while tool closure was in 2 seconds. The normal contact stiffness factor \( S_n \) was \( 2.5 \times 10^8 \) for all contacts, with inter-ply slip shear stresses taken from equation (5.1). Note that since each contact applies equal and opposite forces to the surface it is touching, contact and friction parameters are effectively twice the above values in cases where two deformable surfaces are contacting.

Figure 6.10 shows the part at 3 stages of the simulation. The fibre orientation in the plies is indicated in part (a) of this figure, and as the simulation proceeds their effect in increasing the bending and extensional stiffness of the composite in this direction becomes quite noticeable. At 1.3 seconds this high bending stiffness is largely responsible for a large buckle appearing in the \( 0^\circ \) direction, but this is later smoothly flattened out. In actual thermoformed unidirectional domes, small out-of-plane buckles transverse to the fibres often remain in this area, and it is expected that a finer mesh and a ply model involving some flow effects would reproduce this in a simulation. Note that little change was noticed in the part shape when it was held in the tool a further 2 seconds after (c).

While Figure 6.10 showed to some extent the near inextensible behaviour of the fibres, Figure 6.11 demonstrates most graphically the resulting pull-in effect in the \( 0^\circ \) direction. The curves shown in this figure represent fibres in the centre plane of the top ply, and were initially uniformly spaced. Note that in the transverse direction the material prefers to stretch to match the dome geometry, just as the isotropic model of Figure 6.9 did.
Figure 6.10 Simulated matched-die dome forming of a [0_4] laminate.
6.3.3 Matched-Die Cross-Ply Dome Forming

This example matches that in Section 6.3.2 in every respect except that a [0,90], laminate is used, so that fibres in the two inner plies are transverse to the two 0° outer plies. Note that Appendix B lists the SimForm input file for this example. As stages (a) to (c) of Figure 6.12 show, the part deforms almost symmetrically with large out-of-plane buckles appearing in both the 0° and 90° directions. After 1.5 seconds the top tool starts to push down on the buckles, leading to the unstable, higher order buckling modes shown in (d) and (e), before these are finally flattened out. Again, the smooth geometry after 2 seconds is unrealistic, since the real material is unlikely to recover from the sort of buckling that took place earlier in the simulation.

Figure 6.13 shows the fibre orientations in the top two plies at the end of the simulation. The symmetry of the deformation is very noticeable here, and it is remarkable to what extent transverse directions are pulled in with the fibres of neighbouring plies, resulting in the characteristic “trellis flow” deformation of Figure 2.1. Further evidence of this is seen along the 90° direction in Figures 6.12 (c)-(f), where there is virtually no inter-ply slip across 0/90 boundaries. As in the example of the previous section the simulation was allowed to run to 4 seconds. If inter-ply slip friction was negligible the plies would be expected to return to the shape shown in Figure 6.11 during this time. However, little change was observed, demonstrating how negligible transverse stiffnesses are in comparison to inter-ply shear forces.
Figure 6.12 Simulated matched-die dome forming of a [0,90]laminate.
Figure 6.12 (cont.) Simulated matched-die dome forming of a [0,90]_s laminate.
The results can be compared with the matched-die dome forming experiments of Martin et al. [15] on squares of APC-2 cross-ply laminates, which admittedly differed by the use of Upilex-R diaphragms. In the experiments, buckling was observed to initiate off-axis to the fibres, but this is likely to be influenced by the diaphragms. Despite this difference, the final part shape was very similar to that in Figure 6.13. Furthermore, one of the conclusions from that paper was that elastic effects in the laminate were responsible for the observed buckling behaviour.

6.3.4 Double-Diaphragm Unidirectional Dome Forming

As a final example, pressure forming with two 0.75 mm silicone rubber diaphragms is simulated. The diaphragms are 70 mm square in the quarter dome model, and use a uniform 7 × 7 element mesh, while the specimen uses the same mesh as the previous two examples. Due to computing limitations separate plies are not modelled, so the composite consists of a single, 2 mm thick block with fibres in the 0° direction. Thus, it is equivalent to the unidirectional laminate of Section 6.3.2, except that inter-ply slip is inhibited.

Diaphragm and specimen model parameters are given in Section 5.4. Normal contact stiffness factor $S_n$ is $1.0 \times 10^9$ between all contacts ($\times 2$ for contact between deformable layers), reflecting the increased loads needed to stretch the diaphragms, since they are constrained around their edges. Frictionless tool contact is used, but for slip between all other contacting bodies, shear stresses are given by twice equation (5.1).
Figure 6.14 Double-diaphragm forming a dome from a single, thick ply.
Figure 6.14 (cont.) Double-diaphragm forming a dome from a single, thick ply.

Figure 6.14 (a) shows the initial configuration. In the first 0.2 seconds pressure is applied equally to both diaphragms, increasing linearly from 0 to 10 kPa to simulate drawing a vacuum of that level between them. This causes the edges of the diaphragms to close together, as in (b). After 0.2 seconds only the pressure on the top diaphragm continues to increase, rising at the same rate to 100 kPa at 2 seconds, and is held at that level a further 2 seconds. The part forms
rapidly after 0.2 s, going from a flat sheet to half-way into the female tool at 0.546 seconds, as shown in Figure 6.14 (b). Forming proceeds without any sign of instability until it is fully formed in (c), after 1.5 seconds. It is interesting to note how much the specimen is pulled into the tool in the near inextensible fibre direction. This causes the diaphragms to stretch considerably in this direction, and in sliding past the part they superimpose tension just where it is needed. This effect continues long after the final part shape is obtained, as the diaphragm movement between (c) and (d) of Figure 6.14 indicates.

Figure 6.15 Fibre orientation in double-diaphragm formed dome at 1.506 s.

Figure 6.15 shows the fibres in the centre of the specimen after forming. Although the forming geometry differs slightly from the example in Section 6.3.2, some differences are evident when comparing this with Figure 6.11. The effect of the diaphragms and the normal pressure has been to cause some added squeezing and stretching in the transverse direction. This example has once again demonstrated the beneficial tension-inducing effects of diaphragms. However, as was noted in Section 6.2.2, concern must again be raised about the amount of pressure needed to deform the diaphragms, which is likely to cause excessive transverse flow in the real material, and will require heavily constructed forming equipment.
Chapter 7: Discussion On Numerical Modelling

This chapter looks at some of the difficulties encountered with the numerical modelling of the last three chapters and how the molten laminate model in SimForm may be improved. The major discussions on the results of this work have been included with Chapters 2 to 6.

7.1 Contact Stability and Limitations

Slow or non-convergence of a non-linear finite element procedure is a sign of either mathematical errors in its implementation, or simply inadequacies in the algorithm itself. While rigorous code-checking and program testing will hopefully sort out any of the first category of errors, a major rethink on how to attack the problem may be necessary if the algorithm is found to be at fault. The slow convergence and stability problems often encountered when using the contact procedures in SimForm, especially in conjunction with highly anisotropic material properties is thought to fall into the latter category, since lengthy program testing has failed to find any lingering errors in these routines. This section looks at what is going wrong with the contact algorithm.

It would appear that much of the difficulty experienced with the use of contact in SimForm stems from the problem of imperfect mating between deformable surfaces, each described by parametric functions. Despite using relatively high order bicubic Hermite interpolation, giving the outward appearance of excellent conformance, small gaps of varying thickness remain between the surfaces. This can result in the type of oscillatory normal contact stresses shown in Figure 7.1, especially when normal contact stiffnesses are high.

![Figure 7.1 Bending of two isotropic sheets showing normal contact stresses applied to the bottom layer. Tensile stresses point in to the layer, compressive away.](image)

These oscillatory forces have little effect on isotropic layers, due to their substantial shear and flexural stiffness. However, the same conditions applied between thin layers of shear-weak, highly anisotropic material can cause the entire laminate to buckle. This may have happened in the example shown in Figure 6.7(d). It should also be noted that once buckling occurs in one ply, as was the case in Figure 6.4, the same through-laminate buckling tends to follow, indicating a breakdown in the model.

To avoid the onset of such spurious behaviour, the models presented in Chapters 5 and 6 kept both levels of anisotropy and contact parameters within certain limits. This was particularly important with regard to the normal contact stiffness factor, $S_n$, of equation (4.49). As with any constraint applied through a penalty number, too high a value leads to satisfaction of the constraint at the expense of a physically realistic solution, as in the buckling cases described above. Even with these changes, slow convergence was frequently observed, especially once deformation and rotation of the body were large. In the model of Section 5.3.2, convergence to
the prescribed limits was approached but not obtained at later time steps, so time was simply incremented once a certain number of iterations had been performed. This lack of convergence is the likely cause of the “humps” in the load curves of Figure 5.7.

One plausible explanation is that excessive coupling at high values of $S_n$ is responsible for the lack of convergence, since from the contact equations very high contact stiffnesses in regions of high curvature may serve to lock adjoining layers together, permitting only small displacements in each iteration. To counter this, two different approaches have been tested, each involving a reduction in the contact stiffness by some means. The first was to use a much lower value for $S_n$, but to iteratively increase contact forces so that penetration was not excessive. This worked well in some cases, and often eliminated the oscillations in the contact stresses.

However, in larger problems it was simply unstable, and a common occurrence was for the layers in the model to literally fly apart. The second approach was to use non-linear contact springs, which from a moderate initial stiffness increased to a much larger value as layers penetrated one another. This worked well, but was not appreciably better than the simpler, linear approach. Moreover, it added further non-linearity to the problem, and failed to make any improvement to the rate of convergence.

A second possible explanation is that differences between the convergence characteristics of the contact and finite elasticity equations in neighbouring layers are causing an oscillating convergence. Several observations support this explanation. In cases where the model completely broke down, the body often oscillated with increasing amplitude as it diverged from the desired solution. Furthermore, in some of the slow converging cases the residual was found to oscillate between poorer and better converged values while the average approached zero.

With regard to the contact equations, Section 4.4 neglected to note that there are three possible ways to apply these constraints between deformable surfaces, as follows:

1. BOTH surfaces have contact points, are subject to contact forces depending on how close they approach the opposing surface AND each surface applies equal-and-opposite forces to the other.
2. As for case 1 except there is no transfer of equal-and-opposite loads, so each surface looks after its own loads.
3. One surface only has contact points and tries to stay in contact with the second, applying equal-and-opposite loads to it. In effect, one surface will be a “slave” to the other.

All of these approaches have been tested, with varying degrees of success. Stability problems with method 2 almost certainly result from the contact stiffness and forces acting on each surface differing slightly in magnitude and direction. Method 3 failed terribly in the diaphragm forming example of Figure 6.7. The reason was that as contact points on the upper diaphragm slid off the end of the laminate, the pressure caused the diaphragm to drop suddenly, causing a degree of geometric non-linearity that the solver was unable to recover from.

Method 1 is less susceptible to such problems, and is physically more realistic in that both surfaces support each other. For this reason, it has been used in all examples in this work. However, since it combines methods 2 and 3 above, it is subject to some of the problems experienced with them. Firstly, once contacting surfaces are no longer perfectly mating, differences in the contact constraints calculated on each surface will result in a certain fraction of the normal forces being applied in the tangential direction, and vice versa. This will tend to limit the tangential slip even when frictionless conditions are in effect. A solution could be to compute contact constraints on a third surface at an average position between the two layers, but this has not been attempted.

Of greater concern is the ability of the contact procedure to cope with large displacements as may result from sudden buckling, loss of contact and long time steps, especially when coupled with the finite elasticity model used in the plies. Use of a different solution scheme to the Newton-Raphson method employed here may be the answer, but for the
results presented in this work efforts were put into keeping displacements in each time increment to a manageable size.

Despite the problems encountered with the contact procedures, they perform adequately provided the anisotropy in the plies, normal contact stiffness factor $S_n$ and time increments are kept within reasonable limits. In fact very good performance is achieved with isotropic materials. Unfortunately, the problem at hand, that of modelling the deformation of fibre-reinforced materials, requires all values to be used at their limit of acceptable performance, and beyond. Moreover, the addition of one extra and artificial material parameter, namely contact stiffness $S_n$, and its effect on the loads and behaviour of the remainder of the model makes this a somewhat unsatisfactory solution.

It would be preferable to eliminate penalty numbers altogether by the use of Lagrange Multipliers to enforce the contact constraints. However, these will behave similarly to high contact stiffness values and interfere with the load response of the body, especially if a coarse mesh is in use. A much finer mesh would solve many of the contact woes, but at huge expense in processing terms. It would also be wasteful since ply deformations seem adequately described by coarser meshes. If one were prepared to develop a completely custom element for modelling CFRTP laminates, then considerable scope exists for both reducing the model size and solving the contact dilemma. It has been mentioned already that it is superfluous to store separate slope degrees-of-freedom for each ply in the laminate, since ideally they remain very nearly parallel throughout forming. It may be possible to express the coordinates of ply nodes in terms of some global slope and position variables for the laminate, so that plies are constrained to “tracks” in the plane of the laminate. Again, this suggestion is left to future study.

The size of time increments used in the analysis has an important bearing on contact. In early development of SimForm, attempts were made to handle loss and gain of contacts during the iterations of each time step. This proved quite unstable, with points often gaining and losing contact several times before either equilibrium is reached, or the model broke down. Similar behaviour was observed when a yield effect was tested with the friction model. Hence, contact gain and loss are now handled between time steps, requiring shorter time increments for accurate contact modelling. Friction calculations also benefit from shorter time steps since rotation effects are then reduced.

It should be clear from this discussion that contact modelling with the finite element method is a complex problem, especially for the inter-ply case studied here. Even more than the problem of model size, discussed in the following section, it represents the greatest obstacle to overcome before CFRTP sheet forming can be accurately simulated.

### 7.2 Optimising Elements and Procedures

Section 6.1.1 described the massive computing requirements of SimForm, which at present is the main reason why larger models may not be solved with the program. At fault here is the use of a 3-D continuum element in a role seemingly best suited to a shell. The decision to produce a three dimensional model was sensible considering virtually all CFRTP sheet forming operations fall into this category, a point also noted in that section. The use of bicubic Hermite interpolation in the plane of each ply is also not at fault, although eventually a triangular element should be sought since it would free the model from any geometric restrictions. It is the choice of linear or quadratic through-thickness interpolation, requiring 2 or 3 nodes (6 or 9 degrees of freedom) through the thickness of each ply, respectively, that is the source of the excessive degree of freedom count, which in turn leads to the inflated memory and processing requirements. Continuum elements were chosen initially due to their relative ease of implementation, especially compared to shells. A second advantage was that they did not require any assumptions to be made about the deformation of molten CFRTP prepregs, necessary if any of the various shell theories are to be adopted.
Thick shells approximate continuum elements by neglecting strain energy associated with normal through-thickness deformations, reducing the degrees of freedom through the sheet to 5. This is only a small improvement on the 6 used by the present continuum elements when linear interpolation is used with highly anisotropic materials. It is also uncertain how thick shells would handle the transverse flow mode of deformation from Figure 1.2. Thin shells go even further by eliminating through-thickness shear so that ‘plane, normal surfaces remain plane and normal’ throughout deformation, reducing the degrees of freedom through the ply to the minimum possible 3. However, referring back to Figure 1.2 in the Introduction, two modes of through-thickness intra-ply shear appear to be significant for molten CFRTP prepregs, making the change to thin shells even more uncertain.

It is however imperative that future finite element studies in this area use shell elements, even if it involves a considerable degree of approximation regarding material behaviour. Given the tendency to use more and ever thinner prepregs, as well as thin, polymeric diaphragms, studies will be forced to use thin shells to minimise the total degrees of freedom in the model. On the plus side, this will allow greater in-plane mesh refinement, permitting more detailed forming examples to be modelled.

The observations made in Section 5.3 provide some justification for using thin shells. Although a shear-weak, highly-anisotropic material model was found to fit the initial load response in Martin’s free bending experiments, subsequent transverse spreading under the punch meant significant through-thickness longitudinal intra-ply shear was avoided. This mode of shear is also not observed in Figure 4.1. It is therefore concluded that through-thickness longitudinal intra-ply shear is negligible in molten CFRTP prepregs. Furthermore, as mentioned in Section 5.3.2, thin shell behaviour is approached by the highly anisotropic model as plies are made thinner. Macroscopically, a thin shell would provide an adequate model of this bending process, as long as its bending response is programmed to match the real material. From present evidence it would be largely elastic, but in regions of high curvature it would be better characterised by a plastic or viscoelastic law. These observations were made with relatively thick, 0.5 mm PLYTRON plies. With thinner, higher fibre volume fraction prepregs, thin shell approximations are likely to be even more applicable.

A second process that can be ignored completely is transverse intra-ply shear, the third process depicted in Figure 1.2. Even though it is possible for this process to occur, it has a negligible impact on both the overall loads needed to form the material, and on other, more important flow processes. Together, inter-ply slip and transverse flow will make up for any added inflexibility in the material as a result of eliminating this shear process. Thin shell bending properties in the transverse direction, whether elastic or otherwise, need only approximate the real prepreg response.

With two sets of bending properties describing through-thickness behaviour, only in-plane properties need to be considered. Mid-plane fibre response would be adequately described by either an inextensible fibre or a high stiffness elastic model. This leaves only in-plane longitudinal intra-ply shear and transverse flow to take care of, the latter also affected by squeeze loads acting on the ply. A discussion on the best material models for these processes is left to the following section.

It appears that a thin shell provides sufficient degrees of freedom to model the important aspects of molten CFRTP prepreg deformations. However, the model will be complicated due to the additional overhead of storing and updating such variables as shell thickness and levels of inelastic strain throughout the ply. It must also be emphasised that the load response of each significant deformation process, including longitudinal ply bending, in-plane flows and especially inter-ply slip must be fairly accurate if the loads measured for the whole body are to be realistic. This is clearly a major aim in this exercise.

After optimised elements the greatest performance gains may be achieved by employing an explicit, dynamic solution scheme instead of the fully implicit scheme currently used in
SimForm. Explicit schemes [32] require no matrix inversion so iterations can be performed at very low cost, using much less memory. For stability, increments of time must be small, less than a critical value dependent on the minimum element size and the speed of sound in the material. As mentioned earlier, small time steps have obvious benefits for modelling contact, minimising rotation effects and changes of contact in each increment. Zienkiewicz and Taylor [32] also point out that as long as the governing equations are linear in velocity terms, iteration is unnecessary in each time step, so many non-linear problems can be solved with an explicit approach at no additional cost over a linear material. Whether this condition is met depends on the constitutive law for the material, discussed in the following section.

Implicit schemes should not be completely neglected, however. Their unconditional stability means that larger time steps may be used, assuming the contact procedure is designed to handle this. Overall computing time may therefore remain competitive. Furthermore, they involve no approximations such as mass lumping, which may prove to alter the behaviour of the model.

ESI Ltd. [12,47] (see also Section 4.1.2) have implemented both thin shells and an explicit code for the purpose of performing simulations of CFRTP thermoforming. Their program also considers thermal effects, and is capable of handling industrial-size problems. Due to commercial sensitivity, little information is available regarding the penalty method they used to enforce inter-ply contact constraints, nor their mixed fluid/fibre ply model. If their program can reproduce the load response of molten CFRTP laminates - an aspect of program verification not yet published - then they have already achieved the ultimate goal of a practical thermoforming simulation.

On a more critical note, it must be asked whether the effort put into such special purpose finite element codes, including SimForm, is really worth it. Eventually, general purpose finite element packages should manage to solve this type of problem. This would be a more appropriate solution considering the current low market penetration of these materials.

7.3 Material Model Issues

At the present stage of development it would be unwise to adopt too complex a constitutive law for governing ply deformations. The reason is that until improvements in inter-ply contact modelling and finer meshes eliminate any undesirable influence these factors have on the load response of the body, the subtleties of any such law will be masked. Nevertheless, good models for the most important flow processes can be suggested from current knowledge.

The previous section, along with Chapter 5, already summarised the main aspects of ply bending and through-thickness shear modes, and how a thin shell can be used to describe them. Within each ply, that leaves only transverse flow and longitudinal in-plane intra-ply shear to be characterised. Since it is likely that the same continuum theory will be used to govern both processes, only the latter will be looked at here.

In Section 4.1.1 it was noted that in-plane intra-ply shear could not be distinguished from through-thickness shear of a whole unidirectional laminate. Inter-ply slip is simply “inter-bundle” slip in this case. It was also concluded that a continuum model should be developed to describe this process macroscopically. The response of such a model could be deduced from our current knowledge of inter-ply slip and ply bending. We can also look directly at experimental results, most notably the vee-bend experiments of Martin et al. [37]. In their experiments, unidirectional PLYTRON laminates were bent around a central pivot by the action of two plates, which rotate from a horizontal position up to the 90° V-shape. Against the plates the material is forced to deform by simple shear. Figure 4.1, taken from that reference, shows that the main shear process occurring was inter-ply slip. The load curves showed a sharp initial rise to a fairly stable load during forming, dropping to a residual elastic load when the crosshead was stopped. At high temperatures (180 °C), some through-thickness intra-ply shear was observed, while the
residual load was small, corresponding to the bending resistance of all the individual fibres in the laminate. At lower temperatures, the final elastic load was considerably higher. The most significant result was that the viscous response was non-linear, with the apparent viscosity much reduced at higher strain rates. This phenomenon is referred to as shear thinning and is a common feature of thermoplastics. If a viscous model is to be applied to in-plane deformations, this non-linear behaviour will eventually need to be considered. An alternative and not dissimilar model that fits the observed loads even better would be an elastic body subject to a non-linear viscous-type creep law. In an advanced model the temperature dependence of all viscous effects must also be considered, and for this purpose reference [37] provides viscosity data for PLYTRON at several forming temperatures.

The inter-ply slip response must not be neglected from any list of CFRTP material properties. Indeed, if one includes in its definition slip against diaphragms and tool surfaces, it is the flow process in the laminate which has the greatest bearing on the overall forming loads. Considering the observations in the previous paragraph, it is not surprising that Scherer noted shear thinning behaviour in his ply-pull-out experiments (refer to Section 5.2.1 and references [4,41,42]). He also encountered a load threshold before flow took place, also noted by Cogswell [1]. In future models, it would be desirable to include these effects.

Where diaphragms are central to the forming process, they will generally contribute more resistance to the forming pressures than the material itself. This is especially true for the rubber diaphragms simulated in Chapter 6. Without an accurate diaphragm model, efforts at more refined modelling in the laminate will be wasted. In fact, with very stiff diaphragms relative to the laminate, such refinement may not be necessary at all. For instance, regardless of how transverse flow is modelled, where plies touch the diaphragms it will occur to almost the same extent as the lateral diaphragm stretching in that area. Ample evidence for this is given in the grid strain analysis example of Section 3.3.1. Eventually, experimenters may move to employing polymeric diaphragms with melting or pliability characteristics not too different from the material itself, in which case accurate viscous or viscoelastic modelling of their stretching will be necessary.

So far this discussion has centred on achieving a material model that reproduces to high accuracy the deformation and load response of molten CFRTP laminates in both simple tests and actual forming examples. To what extent such perfectionism is necessary depends on the desires of the modeller. Deformations of fibre-reinforced materials are so dominated by the kinematics of near inextensible fibres that almost any model that considers the degree of anisotropy present will produce realistic final fibre orientations and part geometry. Furthermore, even if the constitutive laws governing the various flow processes are only part-way correct, they may provide sufficient information for those performing the simulation to improve or predict forming success.
Chapter 8: Concluding Remarks

8.1 Main Findings

Out-of-plane buckling of molten bidirectional laminates during homogeneous “trellis” flows has been investigated using linear stability analysis. Treating the molten material as an incompressible Newtonian fluid reinforced by inextensible fibres in two directions, the growth of small out-of-plane imperfections in the laminate is found to relate to the tensions in the fibres, with viscosity, flow rate and defect wavelengths also influencing the rate of growth. Buckling occurs when the fibre tensions are negative, with small wavelengths favoured to grow in most cases. It is concluded that only by inducing tension in the composite can fabricators reduce or eliminate buckling defects.

Grid Strain Analysis techniques enable visualisation of the strains taking place during sheet forming operations. While current approaches calculate strains by comparing the position of a deformed grid digitised off the part with its original state, the new approach introduced here uses the grid point data to fit a continuous parametric surface to the deformed part. Comparison of the surface in its deformed and undeformed states then enables strains to be calculated at any point, permitting greater flexibility in post-processing. A further advantage of the new technique is that by employing high-order surface patches such as the bicubic Hermite elements used in this study, smooth, inhomogeneous deformations can be accurately analysed.

The new GSA technique has been tested on a blister fairing produced from cross-ply PLYTRON laminates. Arrow diagrams produced from the part demonstrate the preference of bidirectional laminates to deform by trellis flow. In addition, the occurrence of transverse flow over the top of the doubly-curved surface, due to squeezing and friction from the double diaphragms used to form the part, shows that a kinematic model is unable to predict the deformations of laminates made from unidirectional plies.

Inter-ply slip occurs with such ease in molten CFRTPs that in modelling thermoforming, the discontinuity it introduces into the through-thickness deformation of the laminate should be considered. Furthermore, intra-ply modes of deformation may also be discontinuous due to slip between individual or bundled fibres. However, in contrast to inter-ply slip, where slip distances may be large and between plies of different fibre orientation, intra-ply slip always occurs between parallel fibres, making it possible to model these processes as macroscopically continuous, using some average shear properties. Hence, in models of these materials the ply is the most sensible unit of continuous deformation.

A finite element program, SimForm, has been developed to simulate the deformations of such discrete-ply laminates. The program employs three-dimensional continuum elements to describe deformations in each ply, while a contact procedure couples the motion of neighbouring plies to simulate the inter-ply slip process. Inter-ply slip is given a linear velocity-dependent shear stress response, corresponding to shear of thin inter-layers of Newtonian fluid, which reproduces the main features of ply-pull-out tests. A transversely isotropic hyperelastic law is used to govern ply deformations, as a first step towards modelling the viscoelastic shear response noted by many researchers. However, added justification for the elastic ply model comes from the fact that through-thickness longitudinal intra-ply shear appears negligible in CFRTP sheet forming. Much effort has been put into handling the degree of anisotropy present in the plies due to fibre reinforcement of a molten matrix. In SimForm, the use of C1-continuous bicubic Hermite interpolation in the plane of the ply enables ratios of fibre direction to transverse stiffness in excess of 1000 to be comfortably handled.

To verify the model, material parameters were adjusted on a trial-and-error basis to simultaneously fit both the load response and part geometry of a documented three-point-bend
test on unidirectional PLYTRON laminates. However, the SimForm model is unable to simultaneously fit the initial stiffness, peak load and relaxation behaviour observed in the experiments, indicating the occurrence of inelastic intra-ply flow not accounted for in the model. The initial, elastic load response is reproduced using a fibre direction stiffness calculated from the rule of mixtures, and a transverse stiffness 25000 times lower. This approaches inextensible fibre behaviour, so that the ply undergoes shear rather than bending, with negligible inter-ply slip. Since the ends of the real laminate displayed the step-like deformation characteristic of inter-ply slip, it can only be concluded that in free-bending CFRTP laminates, local inelastic spreading and buckling at the load points occurs in preference to forcing through-thickness longitudinal intra-ply shear over the whole part. Hence, except in areas of high stress concentration such as point loads or regions of high curvature, the elastic ply bending model seems quite appropriate.

Since SimForm does not consider such local, intra-ply flow, the load for the near-inextensible model increases to well above that of the real laminate in three-point-bending. Hence, a somewhat less stiff model is selected as the “best-fit” to molten PLYTRON behaviour. With a ratio of fibre to transverse stiffness of 2000, this “extensible” model gives a similar peak load and relaxation behaviour to the real bend specimen, while its deformed geometry closely matches that observed in the experiments. Despite its deficiencies, this model gives much insight into the behaviour of CFRTPs, and has therefore been applied in several thermoforming simulations.

A series of two-dimensional, 90° bend tests were used to show the influence of forming speed on the behaviour of the model. As expected from experimental observations, the model exhibits successful forming at low speeds, but at high forming rates the build-up of compressive stresses in plies on the inside of the bend results in their buckling. From simulated double-diaphragm pressure forming of the same part it is concluded that diaphragms reduce buckling by inducing tension in the part as they stretch and slide past the outer plies. In these models, reusable rubber diaphragms used in practice were simulated using an isotropic hyperelastic material.

Simulations of hemispherical dome forming were carried out to investigate the behaviour of the SimForm model in three-dimensions. Early simulations broke down due to sudden out-of-plane buckling, requiring the addition of inertia into the model to reflect the fact that even elastic deformations do not take place instantly. In simulated matched-die dome forming of 4-ply unidirectional and cross-ply laminates, out-of-plane buckling is observed in the flange area. Furthermore, the stiff response of the plies in the fibre direction results in the characteristic “pull-in” in directions of reinforcement, with cross-ply laminates deforming symmetrically like the “trellis” deformation of an inextensible net. This indicates that even with an elastic ply model the inter-ply slip friction is sufficient to prevent transverse stretching in the plies. These observations are consistent with experimental results.

Double-diaphragm pressure forming of a unidirectional dome has also been simulated. Once again, tension transferred from the stretching diaphragms is shown to prevent buckling during forming. However, as in the 90° bend example, the bulk of the pressure goes into stretching the diaphragms. This produces excessive squeeze loads on the laminate, resulting in undesirable transverse flow.

The SimForm program is not without its limitations. The contact procedure often suffers slow convergence and stability problems when applied between highly anisotropic layers. It also produces some undesirable coupling effects that need to be overcome in future development. Further work is also necessary to reduce the enormous computing requirements of three-dimensional SimForm models. Moving to an explicit, dynamic solution scheme would reduce the calculations required. Efforts also need to be put into reducing the degrees of freedom in the model by using thin shells to model ply deformations. Since through-thickness intra-ply shear is
negligible in the longitudinal direction, and has negligible stiffness in the transverse direction, a
thin shell can model the salient features of CFRTP plies, provided its bending response is
programmed to approach that of the real material. As before, longitudinal bending should have
an elastic response, becoming inelastic where stresses are concentrated.

Of the major deformation processes involved in forming CFRTP laminates, only in-plane
intra-ply shear and transverse flow are not well described by the SimForm model. They would be
best served by a viscoelastic law, but if a simpler model is to be used, a viscous law would be
most appropriate, especially if the non-linear shear-thinning behaviour of the thermoplastic
matrix can be included. However, one must not forget how strongly the deformations of
continuous fibre reinforced thermoplastics are dominated by the kinematics of near-inextensible
fibres. Any laminate model that considers the level of anisotropy present and the inter-ply slip
process will produce not only realistic deformations but much useful information for overcoming
forming difficulties. The detail of the model should therefore match the needs of the modeller.

8.2 Improving CFRTP Thermoforming

So far the main discussions in this work have concentrated on improving the modelling
of continuous fibre reinforced thermoplastics. However, modelling is merely a means to an end,
in this case achieving successful sheet forming with this class of materials. The merits and
drawbacks of current thermoforming processes for CFRTPs have been analysed and discussed,
and this has inevitably led to practical ideas on how best to form these materials. The following
comments are mostly aimed at PLYTRON and other lower cost, lower temperature CFRTPs. For
high performance materials such as APC-2, their performance potential outweighs the costs of
investment in autoclaves and other process equipment suited to high quality, short production
run fabrication. Nevertheless, some of the following comments are pertinent to them too.

The use of double diaphragms, clamped at the edge of the tool to induce stretching and
tension in the laminate, should be central to any CFRTP thermoforming process. In comparison
to edge clamping applied directly to the laminate, much less material is wasted and high forming
loads and defects inherent with forming a larger-than-necessary specimen should be much
reduced. Although matched-die forming could be used with double diaphragms, the process
requires two tool surfaces of close tolerance, one of which would need replacement if any
change is made to the laminate thickness.

Using the diaphragms for pressure forming seems ideal, and its requirement for one
single, albeit porous, tool surface to form against reflects the fact that sheet-formed components
usually require good tolerance and surface finish on one side only. However, current double
diaphragm processes display a huge disparity between the forming pressures and the relatively
small loads required to form the soft, molten laminate itself. At fault is the choice of rubber or
polymer diaphragms too stiff for this task. The result is that, for safety, tooling must be enclosed
in an autoclave or be of sufficiently solid construction to resist the pressures, increasing the cost
outlay for potential users of these materials. Furthermore, high pressures cause excessive and
undesirable transverse flow in the material. The solution is to search for a thin, pliable, probably
one-use polymer diaphragm material that will enable forming to be achieved with vacuum rather
than positive pressure.

To be an efficient, low cost forming solution, such double-diaphragm vacuum forming
should utilise a separate heating system with transfer to the unheated tool for rapid forming and
subsequent cooling, as in Figure 1.3. However, thin diaphragms will cool quickly, and if a
certain elevated temperature is required for them, or slow forming is desired, outside heating will
be necessary. Since the vacuum forming apparatus can be open on one side, radiant heating can
be used on the diaphragms and laminate. As Figure 8.1 illustrates, to encourage desirable flow
processes such heating could be applied to selected parts of the specimen by masking other
regions from the heat source. Tinting the laminate or diaphragms could achieve a similar effect.
Figure 8.1 Equipment for forming a cross-ply dome, using differential heating to encourage tension and limit transverse flow in surface plies.

The “web” of the I-shaped mask in Figure 8.1(c) is designed to promote tension in the fibres of the surface ply, by causing the diaphragm to stretch apart predominantly in the middle. This will maximise diaphragm/ply slip in outer regions, superimposing tension on to the laminate. The “flanges” of the I encourage the diaphragms to stretch in the areas outside the laminate, reducing transverse flow caused by diaphragms sliding and stretching transverse to the fibres of the surface ply.

It is expected that even with an optimal forming process there will be limitations to the complexity of shapes that can be produced out of continuous fibre reinforced thermoplastics. Greater emphasis may need to be put on using fusion bonding processes to connect simpler parts. However, a solution that almost amounts to the same idea is simply to disregard the notion that fibres must be continuous across the whole part. Similar, and in some cases better performance will be achieved by custom placement of cut prepreg sections into the blank, either as local reinforcement or to enable the sheet to become drawable through large-scale inter-ply slip. This utilises the full potential of the concept of composite materials.

To date, PLYTRON and other general purpose CFRTP prepregs have not been widely used in commercial products. They must compete with a range of other materials for use in sheet components, and at present there is little to set them apart from their competition. However, advances in thermoforming could be the breakthrough for these materials, making them a real choice for rapid production of strong, lightweight structures.
References


Appendix A: GSA User Guide

A.1 Overview

The program GSA was written not only to perform the grid strain analyses presented in Chapter 3, but also to provide a useful tool for other sheet forming research carried out at the University of Auckland and beyond. Due to this wider interest, I have endeavoured to write the program under the common GUI standard of the Windows® environment, making it both convenient and easy to use. While its operation will be largely self-explanatory to those who understand the technique, this appendix provides a guide to the program and clarifies some of its technicalities.

The aim of Grid Strain Analysis is to produce an accurate picture of the deformations that occur during sheet forming. Chapter 3 described the main sequence for the surface fitting procedure, from which the main sections of this appendix are derived: Data Input, Mesh Creation, Solution and Post-Processing. To run GSA, just double-click its icon in the Windows Program Manager, and the window shown in Figure A.1 will appear, although the main screen will initially be blank.

![Figure A.1 Screen shot of GSA showing undeformed mesh and grid points for the blister fairing example from Section 3.3.1.](image)

A.2 Data Input

Input for GSA consists of grid point locations digitised from the part under study in its undeformed and deformed states. Points are listed in ASCII text files with one set of coordinates per line. The sequence and number of points read in for the two states must be identical. On each line of the input files, coordinates x, y (and z if necessary) are to be separated by space, comma
or [TAB] characters. All text after a quote mark (’) is ignored by the program, as are excess separation characters, so that:

1.000[TAB]2.056, 1.23 ’ Coordinates for point 10.

is converted to:

1.000 2.056 1.23.

If a line becomes empty through this cleaning process, it is treated as if it were never read in. Each number or word separated by a space in a cleaned input line has a ‘position’. In the above case, 1.000 is in position 1, 2.056 is in position 2, and so on. The program requires that each coordinate occupies the same position in every line of the input file. As will be described shortly, a format string is used to enable almost any input file to be read, provided these rules are adhered to.

The File menu in Figure A.1 contains several commands used to input the raw data for the grid strain analysis. This is slightly more involved than just selecting the input files to be processed. Since grid data comes from a variety of sources, and may need transformation before its use in the program, the input specification for the program is stored in a separate file with the extension .GSA. Selecting File|New presents the user with the dialogue box shown in Figure A.2, where the contents of the .GSA file may be edited.

![Figure A.2 Dialogue Box for creating .GSA files for data input.](image)

The contents of the .GSA file are as follows. The Title provides a place to store a description of the model. Internally, the program uses millimetres for all dimensions, so that if other units are used in the data files, mm per Unit must be set to a different value from 1.0. The Thickness value for the sheet is unused at present. The Beta Smoothing Factor is normally zero, but as described in Section A.4, making it a small number such as 0.0001 will help surface fitting in problem areas where too few points are available to describe the deformed surface.

Input fields for the Undeformed and Deformed data files are identical and thus described together. The name of the data file is shown in the first cell, and the [Browse...] button can be used to select the file from the disk. The Format field describes the position of the coordinates in each line of the input file: in the undeformed case X is followed by Y, separated by a space, and
Z is automatically zero. It will be explained later in this section how complex input formats can be read using different Format strings. As each point is read in, it is scaled by \textit{mm per Unit}, then the \textit{Origin} coordinates (in mm) are subtracted off so that the \textit{Origin} entered represents the new (0,0,0) position. The X- and Y- axes are used to rotate the input coordinates, and work as follows. Since the components of the X and Y axes may not already be orthogonal, vector cross products are used to find \(Z = X \otimes Y\) and then \(Y = Z \otimes X\). These three vectors are then normalised to unit length, thus creating the right-handed orthogonal axes along which the ‘X’, ‘Y’ and ‘Z’ coordinates are measured internally by the program. Note that if X and Y are parallel the program will report an error.

After clicking [OK] on the GSA Input Specification box, the point data are read into the program. Any errors in the input data, such as a different number of points in the undeformed and deformed files, will be reported. Otherwise, successful input results in the appearance of the undeformed data points in the centre of the window (unless the undeformed data points have been switched off using \textit{Options|Display}; See Section A.5.1). The .GSA file may be edited using \textit{File|Modify GSA File}, and saved using the \textit{Save} and \textit{Save As} options from the same menu. As a tip, keep all files used in the analysis in the same directory on the disk, so that they are easier to find. A saved .GSA file may be reopened using the \textit{File|Open} command, which automatically reads in the data points. Any .GSA file and data in memory may be discarded using \textit{File|No Strain Analysis}, useful if only the meshing functions are to be used.

Once the undeformed points are visible on-screen, check that the deformed points are also there, and that the pairing of each undeformed point to its equivalent deformed point is correct by viewing the ‘Data Point Joiners’. Undeformed Points, Deformed Points and lines joining them are each stored in separate ‘Graphical Structures’. Choosing \textit{Options|Display} (see Section A.5.1) allows the user to turn these on and off. Also, Viewing Transformations (see Section A.5.4) can be used to alter the 3-D view of these points and lines.

**Advanced Input Formats**

Different entries in the ‘Format’ fields of the GSA Input Specification in Figure A.2 can be used to allow more general and selective input from data files. It is even possible to generate a rectangular array of grid points automatically. These options are best explained by the following examples.

**Format String:** \(* \ * \ X \ Y \ Z\)

Commonly used for deformed grid data input when the same file is used for undeformed data. The first two numbers at the positions of the asterisks (*) are ignored, then the deformed X, Y and Z are read.

**Format String:** \(* \ * \ X \ -Y \ Z \ W \ 1\)

Two numbers per line are read and ignored, then the X coordinate is read. Y is set to minus the number read next, and followed by Z. The following number will be read in as the weighting factor for this point - normally set to 1.0 for all points. Giving a higher weight to a point, relative to those given for other points, indicates more confidence in the accuracy of its coordinates (see Section A.4). The last format character needs some explanation. Up till now it has been assumed that every data point read is useful. Occasionally data points are read in as place-holders, or it may be necessary to ignore highly erroneous points. The former commonly occurs when the undeformed state is generated as a rectangular grid larger than the blank size, and a flag on the deformed data points is used to mark the unneeded points off the specimen but within the rectangle. Putting any character in the format string apart from *, (-)X, (-)Y, (-)Z and W will cause a comparison to be made between that character and the character at that position in the input line. If they match, the point is deemed valid, if not, it is read in but will be ignored in all
further processing. In this example the data point is only valid if it has the character ‘1’ at the seventh position.

Format String: * * X -Y Z W !0

This example is similar to the previous one except that the logical operator ! (NOT) is used on the character ‘0’. Here, a data point is deemed valid if and only if the character at the seventh position in the line is not a ‘0’.

Note that there are no restrictions to the order of the formatting characters. Also, if weighting or valid point selection is set in both the undeformed and deformed data files, the latter overrides the former. Finally, the user should ensure that the coordinates as understood by the program are in a right-hand sense. This will ensure the view on screen matches the part from which the points are taken.

For evenly spaced rectangular arrays of grid points, an alternative form of input is to generate the points automatically, using the command:

\[
\text{GEN NumPointsX NumPointsY GridSpacingX [GridSpacingY]}
\]

in place of a format string. This generates an array of points emanating out from the Origin with NumPointsX points in the direction of the X-axis, and NumPointsY points in the direction of the Y-axis. Unlike before, the user-specified Y-axis is not forced to be orthogonal to X, but both are normalised to unit vectors and GridSpacingX and GridSpacingY are the spacing between adjacent points along the line of the X- and Y- axes, respectively. Note that if GridSpacingY is omitted, it is given the same value as GridSpacingX. The array of NumPointsX* NumPointsY points are ordered as follows. Points along the X-axis cycle most quickly, so that the first point is at the origin and the second is one GridSpacingX along the X-axis from the first. Point (NumPointsX+ 1) is one GridSpacingY along the Y-axis from the origin, and so on. Both the undeformed and deformed data may be generated in this way, enabling homogeneous deformations to be tested without using any data files.

A.3 Mesh Creation

Having verified the data points are correctly loaded into the program, the principle task in GSA is to create an undeformed element mesh to represent the surface in that state. As was described in Section 3.2.1, C$_1$-continuous bicubic Hermite elements, comprising four nodes linked in the form of a deformed rectangle, are used exclusively in GSA. In general the mesh creation will involve only placing nodes, connecting them to form elements, and moving the nodes to reach the desired shape. Limited control of nodal slopes is also possible. This section describes the mechanics of how to carry out these tasks. However, for guidance on what makes a suitable and non-singular mesh for fitting the deformed surface, consult Section 3.2.2.

The commands listed under ‘Mesh’ in the menu bar of Figure A.1 are used to open and save mesh files (extension .MSH) and to perform other less used mesh functions. Mesh creation and editing is carried out using interactive manipulation with the mouse. The first four modal buttons on the Toolbar in Figure A.1 control the effect of mouse actions, as described shortly.

Before editing the mesh ensure that Perspective is off and the top view Orthographic Projection is in use, as described in Section A.5.4. To work on finer details of the mesh it will often be necessary to use the zoom and translate functions, also discussed in that section.

**Node Mode Button**

With Node mode active, depressing the left mouse button in the editing window causes a node to appear at or near the mouse pointer. Until the left button is released the node, marked by a small square as in Figure A.1, may be dragged to its desired location, with its coordinates indicated in the top left of the window. The node will be restricted to a rectangular area around
the grid points, set when they are loaded. The Mesh|Extents function can be used to override this range, and changes will be stored in the .MSH file. Furthermore, nodes are initially “Snapped” to integer coordinates, in millimetres. The Mesh|Grid/Snap function can be used to switch off this feature, or to adjust the Snap Spacing.

Nodes are deleted in this mode by clicking inside their square with the right mouse button. Note that if a node is part of an element, it can not be deleted until that element is deleted.

Element Mode Button

Elements are created by clicking on 4 nodes, which become highlighted as they are selected. The new element appears as a quadrilateral, with a diamond-shaped selection box in its centre. The new element will be highlighted with local node numbers written next to its nodes. When adding elements to an existing mesh the program checks to see if the new element fits in with the other elements, and if not none will appear. In large meshes this checking process can be slow. If the new element borders other elements, the elements may become distorted as they try to maintain slope continuity at common nodes. Figure A.1 showed several such distorted elements.

Creating a new element or left-clicking inside the centre diamond of an element will “select” it for certain special functions. By similarly left-clicking an already selected element, the local node numbers of it and all neighbouring elements will be cycled so that node 2 becomes node 1, 3 becomes 2, 4 becomes 1, and 1 becomes 4. If the selected element is not connected with any others, it can be subdivided into an array of elements using the Mesh|Subdivide Element function. Elements are deselected by clicking outside their centre diamond or switching from element mode. Only one element can be selected at any one time.

Elements are deleted in this mode by right-clicking inside their diamond-shaped selection boxes.

By selecting the same node twice when creating a new element, a degenerate element (cf. Figure 3.6) can be created. Such elements can be used to increase element density in areas of the mesh, but do so at the expense of continuity near the repeated node. Note that the repeated node must also appear in an ordinary element otherwise a singular system matrix will result.

Edit Mode Button

In Edit mode, nodes may be moved by dragging their selection boxes with the left mouse button. As a node is moved its new coordinate is shown in the top left corner of the editing window. If the repositioned node is in an element the program will recalculate nodal slopes to minimise local mesh distortion, a process termed “relaxing” in the program. The user should try to place nodes so that the elements resemble as near as possible their nominal rectangular shape, since highly distorted elements reduce the accuracy of the fitting process. In particular, nodes must not be placed so that two elements occupy the same area.

With Edit mode active, right-clicking the mouse on any node contained in an element will highlight it and create 2 to 4 (depending on the number of elements the node is in) straight lines radiating from the node, each with boxes at the end. The lines point in the direction of the nodal slopes, and the lengths are proportional to the length of the slope vectors. On a one-element mesh, these nodal ‘slope boxes’ coincide with the adjacent nodes on each side of the element meeting at the highlighted node, thus making the element a perfect quadrilateral. When a node is so highlighted, the nodal slopes are changed by dragging these boxes to new positions with the right mouse button. The nodal slopes point in the directions of these lines, and their length controls the degree of element stretching in that corner. Experiment to see what can be done with these controls. It will usually not be necessary to adjust nodal slopes in this way.
However, the part shape may be more accurately represented if slopes are adjusted to align element sides with geometric features such as bends and the true outline of the undeformed blank. Note that moving the node as described above will remove any changes made to the slopes at that node. Similarly, invoking the Mesh Relax All Nodes function will set all nodal slopes in the mesh to their “relaxed” values.

Renumber Mode Button
Renumbering the mesh is not necessary to obtain a solution, but may speed it up. When Renumber mode is started, the current node and element numbers appear next to the node and element selection symbols. Nodes and elements are renumbered in the same way by left-clicking their respective selection symbols in the desired sequence. To help remember the current position, ‘Next Node: 1’ and ‘Next Elem: 1’ are written at the top left of the window, and incremented as each node or element is selected. Selecting a node or element with a number lower than the current ‘Next’ number gives an undefined result. Not all the nodes or elements need to be selected, since the current ordering of non-selected elements will be maintained. Simply stop when the desired numbering is achieved.

If most of the numbers are correct, time can be saved by right-clicking on the last correct node or element. New numbering will start from after that point. For example, if the node that should be 54 is currently 63, right-click on node 53, and the writing on the screen will read ‘Next Node: 54’, after which node 63 should be selected.

The most efficient mesh has the minimal bandwidth, which is proportional to the maximum range of node numbers contained in any element of the mesh. For a rectangular array of elements this is achieved by numbering the nodes across the side with the fewest elements first.

A.4 Solution
Performing the surface fit can take a little time, leaving one wondering whether the computer has crashed. In fact, if during the solution phase the computer runs out of memory, or the system matrix is singular, the program may actually crash. For these reasons, save your .GSA and mesh files before solving!

Solve Button
After pressing the Solve button several activities take place before control is passed back to the user:
1. The program compares the Degrees of Freedom (DOFs) in the mesh with that in the grid data. If the number of points is insufficient to fit the surface, either locally or over the whole mesh, an error will be reported and processing is stopped.
2. The system matrix for the least squares surface fit is assembled and solved to return the coordinates of the deformed surface nodes.
3. Graphical structures for post-processing are created using the current settings (see Section A.5). This is usually the slowest phase.

After solving, the accuracy and quality of fitting should be checked. First invoke the Results Solution Statistics command to check that the range of in-plane and through-thickness strains calculated on the fitted surface are reasonable. Through-thickness strains are calculated assuming volume constancy. The strain statistics presented are calculated at the centre of fibre grid squares and change with the fibre grid spacing (refer to Section A.5.2). Clicking [OK] at this point will display surface fit error statistics. The fitting error for a given data point equals the distance (in mm) between the actual deformed data point and the point on the fitted deformed
surface with the same surface coordinate as the undeformed grid point on the undeformed mesh. The smaller the average error length, the better the fitting. Large errors in one area of the surface could indicate a need to refine the mesh near that point. These fitting errors can also be viewed as lines on the screen by switching on the ‘Surface Fit Error Lines’ graphical structure (refer to section A.5.1).

One source of high fitting errors can be the presence of one or two erroneous grid points in the input data, either due to measurement error, or the points being out of sequence in one of the grid data files. In the latter case, displaying the ‘Data Point Joiners’ graphical structure should reveal the troublesome points, and the input files can be altered accordingly. If measurement errors have occurred, better fitting may be achieved by using the advanced input formats described in Section A.2 to eliminate or reduce the weighting on the erroneous points.

Section 3.2.3 mentioned the possibility that even when fitting errors are small, the deformed mesh may be unacceptable since it does not resemble the real deformed surface. A common manifestation of this problem is extreme deformations apparently measured at the edge of the sheet. The cause is the boundary of the undeformed mesh being too far from the undeformed grid points, causing wild extrapolation of the slightest tendencies in deformations at the edge of the sheet. When the local mesh density is too high for the number of grid points there, the deformed surface will sometimes appear to be buckled when no such buckling is observed on the formed part. Apart from reducing mesh density in such areas, the Beta Smoothing Factor mentioned in Section A.2 can be set to a small number, say 0.0001, to give the sheet a small flexural stiffness to smooth out these effects. Section 3.2.3 explained how this factor works. Experiment to see how different numbers modify the shape of the deformed mesh and influence the fitting error. The ‘right’ number should have little influence on the error, but gives a realistic final part shape.

Note that the deformed mesh may be discarded and the program returned to Mesh Editing mode by clicking on any of the mesh editing buttons described in Section A.3.

A.5 Post-Processing
A.5.1 Graphical Structures

GSA uses a simple technique to streamline graphical post-processing. Instead of recalculating the positions of points, lines, surfaces and other geometric constructs making up the image every time it is to be redrawn, they are pre-calculated and stored in large lists called Graphical Structures. Each separate entity has its own structure, for example, one structure stores the crosses marking the undeformed data point positions, another stores the lines making up the outline of the mesh, etc. The user is free to select which of these structures are to be viewed using the dialogue box shown in Figure A.3, invoked using the Options|Display command.
In addition to those Graphical Structures in the above figure, the following are available by scrolling the list:

- **Surface Data Points**
- **Positive Contour Boundaries**
- **Negative Contour Boundaries**
- **Positive Contour Shading**
- **Negative Contour Shading**
- **Contour Scale Trimmings**

To change the display status of a Graphical Structure in the dialogue box of Figure A.3, select it from the list and change its attributes. Each structure has a setting to say whether it is displayed when editing the mesh and when performing post-processing. A shortcut to changing these switches is to double-click the Graphical Structure text. This toggles the current viewing status depending on whether the model is being edited or has already been solved. In addition, each structure can be given a different Screen Colour for display, and Pen Number and Line Style for plot file output (refer to Section A.5.5).

The majority of Graphical Structures have both an undeformed and a deformed version, making it possible to superimposed views of the two states. The contents and purpose of each Graphical Structure is fairly obvious from its name, but they are nonetheless described here.

- **Data Point** locations in the two states are marked by crosses in their respective structures. Data Point Joiners connect the undeformed and deformed data points to aid checking of input data.

- **External Element Boundaries** form the outline of the mesh, and together with the Internal Element Boundaries mark out the positions of the elements. Element Isoparameter curves are usually displayed at the same time as these structures and show intra-element surface detail.

- The two Undefomed Fibres are straight, parallel lines on the undeformed mesh. Their origin and direction are specified in the Options|Display Settings dialogue box, described in the following section. The deformed fibres are the projections of these two lines onto the deformed surface.

- **Principal Strain** structures contain pairs of arrows representing the magnitude and orientation of the principal strains in the plane of the sheet. The length of each arrow is linearly proportional to the strain magnitude. Thickness Strains are calculated from in-plane strains, assuming a constant volume, and are displayed as arrows out of the plane of the sheet. All strains are calculated and shown in the centre of the square cells formed between the two sets of fibres.
Surface Data Points are the undeformed data points addressed in local element coordinates, and, assuming the search function has worked correctly, will coincide with the original data. If the same local coordinates are used to project these points on the deformed surface, they would represent the program’s ‘best fit’ for the actual deformed grid points, and a line connecting the two is a measure of the fitting error. The Surface Fit Error Lines structure stores just these lines, enabling fitting errors to be viewed on the deformed surface.

The final 5 Graphical Structures relate to contour plotting and should generally all be turned on or off simultaneously. Contour Boundaries are continuous curves marking off ranges of the plotted variable. Contour Shading uses vertical or horizontal hatching to visually distinguish the plot ranges. Contour Scale Trimmings stores lines marking the outline and range of the contour scale, if one is to be displayed. The contour plot feature is at present only used to plot percentage thickness change as calculated from the thickness strains. For this reason, positive and negative contours are coloured separately. Section A.5.3 describes how to change contour plot settings.

The user should experiment with combinations of these graphical structures to display the significant features of the deformation. The most useful plot for grid strain analysis, however, is the Arrow Diagram, many of which were presented in Chapter 3. This diagram can be displayed on the undeformed or the deformed state by switching on the appropriate External Element Boundaries, Principal Strains and both sets of fibres.

A.5.2 Display Settings

The Graphical Structures described in the previous section are generated automatically in the solution phase, using the current settings for fibres, curve approximation and other facets of these structures. These settings can be changed using the dialogue box shown in Figure A.4, called up by the Options|Display Settings command. Clicking [OK] from this screen causes all Graphical Structures to be recalculated. This can take some time, so make sure all changes are made at once.

![Figure A.4 Options|Display Settings dialogue box.](image)

The position, orientation and spacing of the Fibre Graphical Structures of the previous subsection are controlled by the first five input boxes in Figure A.4. The undeformed fibres emanate from the Origin (x, y) at the given Angles (in degrees), anticlockwise from the positive x-axis. The two sets of parallel fibres form a grid on the undeformed surface, the spacing of
which is controlled by the *Fibre Grid Spacing* entry. Avoid using too small a value for the *Fibre Grid Spacing* since it slows down processing considerably. In the Graphical Structures of the previous section, the undeformed fibres are trimmed to fit inside the Undeformed External Element Boundaries. The deformed fibres are found by projecting these lines onto the deformed mesh.

As the arrow diagrams of Chapter 3 showed, the *Principle Strain* Graphical Structures depict in-plane strains as sets of orthogonal arrows emanating from the centres of spaces in the fibre grid. The 100% *Strain Length* is used to scale these arrows, and equals the length of an arrow representing a 100 percent normal strain of the current *Strain Type*. The length of the arrow is measured from where the strain is calculated (at the centre of a space in the fibre grid) to its outer end. To improve the appearance of the arrow diagram, the 100% *Strain Length* should be adjusted relative to the *Fibre Grid Spacing* so that the strain vectors are contained within the spaces of the fibre grid.

*Element Curve Divisions* controls the number of line segments used to approximate an element side or an isoparameter curve. *Isoparameter Divisions* will be 1 greater than the number of isoparameter lines marking the element surface. The *Data Point Cross Size* is the length (in mm) of the arms of the crosses marking the position of Data Points.

The *Strain Type* is used to choose between *Linear* (Engineering), *Logarithmic* (Natural) and *Lagrange* (Finite or Green’s) strains as the strain measurement of choice. How these strains are calculated is described in Section 3.2.4.

Finally, the *Perspective Level* controls how much the image is distorted when Perspective is on (see Section A.5.4). The default value of 0.5 is a good choice, 0.0 is very distorted, while 10.0 is almost a parallel projection.

### A.5.3 Contour Plotting

When a solution has been obtained, contours of percentage thickness change may be plotted over the deformed or undeformed surface. The appearance of the contour plot is controlled by selecting **Options|Contour Plotting**. This displays a dialogue box in which several options can be chosen.

The first option is to select whether the contour plotting is to be performed on the undeformed or deformed surface. Shading, if turned on, consists of drawing many parallel lines close together. The *Minimum Screen Spacing* is the number of pixels between adjacent lines when shading is at its densest, the default of 1 giving a solid colour. The *Minimum Plot Spacing* is the number of HPGL (Hewlett-Packard Graphics Language) plotter units (1/40 mm) between the finest shading lines on the plotted output (see Section A.5.5). The remainder of the dialogue box is self-explanatory and controls the presence and appearance of the contour scale.

Remember to turn on the contour graphical structures to view the contour plot. Since shading is slow in comparison to the display of other graphical structures these should perhaps be switched on after the desired view has been found with the Viewing Transformations, described below. Also, when changing between contour plotting on the undeformed and deformed surfaces, expect a time delay, since the contour Graphical Structures must be recalculated.

### A.5.4 Viewing Transformations

At any stage during pre- and post-processing viewing transformations may be used to change the user’s view of the selected Graphical Structures. *GSA* allows the image to be viewed from any angle in three dimensions through interactive manipulation with the mouse, and both parallel and perspective projections are supported. The following describe how these features are used.
Transform Mode Button

Selecting this button invokes Transform mode, allowing the view to be changed through mouse movements in the editing window. Some of the controls may seem confusing at first, but by experimenting the user will soon be able to produce the desired view with minimal mouse movement.

Depressing the left mouse button and dragging the mouse pointer around the screen causes the view to be translated with it. The same point on the image will remain under the mouse pointer as it goes. Note that translation is limited by the range of the image, and is only possible once the user has zoomed-in on the picture. Zooming is performed by pressing and dragging the right mouse button in the editing window. Dragging down the screen enlarges the image, dragging up reduces it. The amount of enlargement and reduction is limited by the program to prevent the image from being lost from view.

By pressing and holding both the left and right buttons and dragging the mouse around the screen, the image can be rotated in three dimensions, rather like a trackball. This is particularly impressive with perspective on - in fact, the view is often difficult to interpret with it off. A problem with the 3-D rotation mode is that although the desired viewing angle can be quickly found, the image often needs rotation about an axis out of the screen to achieve comfortable viewing. GSA solves this problem by adding a fourth transformation method called ‘twist’. After holding down both mouse buttons, release the right button only and drag the mouse to rotate the image about its centre.

Orthographic Projection Button

After the image has been rotated and twisted in three dimensions, this button can be used to return to the original x-y (top view) orthographic projection. Mesh editing may be carried out in this view only. If an orthographic projection is currently in use, this button can be used to cycle between the top view and 4 other orthographic projections from the sides of the image.

Isometric Projection Button

This button switches between 4 preset isometric projections of the image, each looking down at the part along a 45 degree incline.

Perspective Button

The Perspective button toggles Perspective mode, and appears depressed when it is on. Perspective is essential for understanding three-dimensional images, and most plotted output will use it. However, it must be turned off when manipulating nodes in the mesh editing phase.

A.5.5 Plotting to File

Stored Graphical Structures are not only available for display on the screen, but can also be written to a plot file for reading into a word-processor, or for later sending to a compatible plotter. The program uses the common HGPL (Hewlett-Packard Graphics Language) standard for all plotting. The plotted image will be the same as the current image on the screen, so before invoking Plot mode, ensure the desired image is in view.

Plot Mode Button

When plot mode is active, depress the left mouse button and drag the mouse to mark out a box around the parts of the image to be plotted. At present the lines outside this box are not trimmed by the program, however, the area of the box is used for scaling. When the button is released a dialogue box appears listing the width and height of the plot box with a plot scale of...
1.0. Change the Plot Scale to scale the picture (eg. a Scale of 2.0 will double the dimensions in each direction). After confirming the scale, the user is prompted to choose a file name (with extension .HGL) to which the plot is sent. After clicking [OK] the file is sent and the display returns to normal.

At present only lines are output to the plot file, so the user will have to record and add contour scale text manually. Plot files can be read into Microsoft® Word and other word-processors if the HPGL graphics file filter is installed. The resulting line image can be edited, and text added if necessary. Note that if each Graphical Structure is given a different Pen Number (refer to Section A.5.1), their colours and attributes may be separately changed in the word-processor - otherwise they will all appear identical.

A.5.6 Outputting Strains

GSA can also output a file containing the magnitude and orientation of the principal strains calculated by the program. Selecting Results|Write Strains to File prompts the user to enter a filename (with extension .STR) into which the strains are written. The result is a file such as follows:

'GSA Output: Lagrange Strains
 x (mm)  y (mm)  Ang1(Deg)  E1      E2      E3
 5.00    5.00   45.00    0.0503 -0.2484  0.4027
 5.00   15.00   48.24   -0.0351 -0.2172  0.4507
 etc.

The first line describes the Strain Type in use, while the second describes the order of the numbers on all the remaining lines. The first two numbers in each line of data are the x and y coordinates of the point on the undeformed mesh at which the strain is calculated. These are the same points at which principal strains are plotted on-screen, in the centre of the spaces in the fibre grid. Next is the angle of orientation of the maximum in-plane principal strain ($E_1$) in the undeformed state, measured in degrees anticlockwise from the positive x-axis. Following that are the in-plane principal strains $E_1$ and $E_2$ ($E_2$ is oriented normal to $E_1$), and the through-thickness strain $E_3$, normal to the surface.

The strain data may be read into a spreadsheet for plotting. This is useful for working out the dominant modes of deformation that have occurred. For metals, an x-y plot of $E_1$ versus $E_2$ is the same format as a Forming Limit Diagram, used to present the region within which successful forming is expected to be achieved.
The SimForm program was designed from the outset to simulate forming of molten laminates of fibre-reinforced prepregs. As such, the input to the program employs a very flexible and efficient format that utilises the flat, multi-layered structure of the laminates. Defining the geometry and lay-up of the laminate is as simple as specifying the two dimensional mesh that marks the shape of the blank, and the thickness and fibre direction in each ply. The format is best introduced with an example, that of matched-die forming a dome from a cross-ply laminate given in Section 6.3.3. SimForm takes its input in the form of an ASCII text file (extension .sim). The file contains a series of commands each starting with an asterisk, and followed by parameters. Some commands expect further input on subsequent lines. Comments in the file are preceded by a quote mark ('). Following the example is a description of the main input file commands used in the program.

Example SimForm Input File

```
*TITLE Matched-die forming a 4 layer cross-ply dome
*INTEGRATION 4 4
*SURFINF 4 4 GAUSS
*MATERIALS
  'MatNo MatType K1 K2 K3 ...
    1 2 5.0E+4 0.0 5.0E+6 1.0E+8 0.0
*INERTIA 1 1480
*PATTERN 1 dome6.msh
*PATTERN 3 dome30b.msh
*PATTERN 4 dome30t.msh
*LAYERS 4
  'LayerNo PattNo TNodes   zbtm   ztop    MatNo FibreAngle
    1 1 2 0.00005 0.00045 1 0.0
    2 1 2 0.00055 0.00095 1 90.0
    3 1 2 0.00105 0.00145 1 90.0
    4 1 2 0.00155 0.00195 1 0.0
'Tool ToolNo PattNo OrigXVar OrigYVar OrigZVar RotVar
  *TOOL 1 3 1 1 1 1 1
  *TOOL 2 4 1 1 2 1 1
*GENERATEMODEL 2
*CONTACTDATA
  'MatNo1 MatNo2 StFact ZPFact CanSep? SepStress cf cn n
  OLDist OLFact Gap
    1 1 2.5E+8 0.0 n 1.0E+9 1.0E+7 0 0 1.0
1.0 0.00010
  'MatNo1 TOOL ToolNo StFact ZPFact CanSep? SepStress cf cn n
  OLDist OLFact Gap
    1 TOOL 1 2.5E+8 0.0 y 0.0 0.0 0 0 1.0
1.0 0.00005
    1 TOOL 2 2.5E+8 0.0 y 0.0 0.0 0 0 1.0
1.0 0.00005
*CONTACTTOL 0.00001
*ALLOWCONTACT
  'Layer1 Side1 Layer2 Side2 {format 1}
    1 1 1 2
    2 0 1 1
```
The following is a list of SimForm input file commands, including not only those used in the above example but also those used in other simulations in this report. The commands are listed in the order they should appear in the input file, since many commands require a mesh or other entity to be established before they can be invoked.

**TITLE** TitleText

Gives the model a title to be displayed during post-processing.

**INTEGRATION** NumGaussPts1 NumGaussPts2

Sets the number of quadrature points used when integrating element stiffness matrices and force vectors. The parameters specify the number of Gauss points in the direction of in-plane local coordinates $\xi_1$ and $\xi_2$, defined in Figure 4.3. Through the thickness of each element the same number of Gauss points are used as the number of through-thickness nodes (see **LAYERS**).

**SURFINT** NumSurfIntPts1 NumSurfIntPts2 [GAUSS]

Sets the number of contact points used in surface directions $\xi_1$ and $\xi_2$, also used as quadrature points for integrating pressure loads over element surfaces. By default, the points are evenly spaced for trapezoidal integration. If this command is followed by the word ‘GAUSS’, the points will be placed at the Gauss integration points instead.
*MATERIALS
MatNo  MatType  K₁  K₂  K₃ ... Kₙ

After the *MATERIALS command, subsequent lines specify the material parameters for each material number (MatNo). The MatType parameter is a value from 1 to 4, and is followed by up to 9 material parameters. In the example given, type 2 uses the strain energy function given in equation (4.38) of Chapter 4, which requires 5 material constants. Type 1 is an orthotropic linear elastic material with 9 material constants. Type 3 is the incompressible Mooney material, while Type 4 is an incompressible, inextensible model.

*INERTIA  MatNo  Density

Switches on inertia and self-weight calculations for material MatNo, using the entered Density, specified in kg/m³.

*PATTERN  PattNo  MeshFileName

List of elements to be excluded from pattern...

This command creates pattern PattNo from the specified mesh file. Meshes are stored in .msh files created using the GSA program described in Appendix A. Note that the millimetre units used in GSA are treated as metres in SimForm. Mesh patterns have two uses in SimForm. The first is to specify the two dimensional layout of elements used in the laminate. The z-coordinates in the mesh are ignored in this case. The second is to specify the shape of tools for forming the laminate (see command *TOOL, below). Listing mesh element numbers on subsequent lines will exclude them from any layers using this pattern. This feature has particular meaning with the *GENERATEMODEL command.

*LAYERS  NumLayers
LayerNo  PattNo  TNodes  zbtm  ztop  MatNo  FibreAngle

After specifying the number of layers in the laminate, subsequent lines set the mesh pattern, thickness and fibre orientation of each layer in the laminate, as well as the materials they are made of. Each layer is referenced by its LayerNo, contains a mesh of elements described by pattern PattNo, and its z coordinate ranges from zbtm to ztop. The number of nodes through the thickness of the layer is set by TNodes, which must be 3 for isotropic materials, while 2 is acceptable with highly anisotropic materials. Finally, each layer is given a MatNo from which its material properties are gained, and an initial FibreAngle, measured in degrees anticlockwise from the positive x-axis.

*TOOL  ToolNo  PattNo  OrigXVar  OrigYVar  OrigZVar  RotVar

Used to create and control tool surfaces. Tool ToolNo is in the shape of the mesh stored in pattern PattNo. The remaining parameters reference variables (see the *VARIABLE command, below) controlling the displacement and rotation (in degrees, about the -y axis) of the tool at its origin. In the example input file, Tool 1 is stationary, but Tool 2, as specified by Variable 2, is initially located 30 mm above its ‘closed’ position to which it is displaced at constant velocity during the first 2 seconds of forming.

*GENERATEMODEL  GenMode

Generates a finite element mesh from the entered layer information. GenMode can take a value of 1 or 2, and controls the node numbering sequence to be used in the mesh. Under mode 1 all the nodes are numbered in the first layer, followed by the second layer, and so forth. Mode 2 numbers nodes in the order of the pattern node numbers, such that all nodes through the laminate with the same pattern node number are consecutive. Under mode 2, the program requires that all patterns used by the layers have the same number of nodes and elements, but some elements (and
nodes) may be excluded (see *PATTERN). Mode 2 is preferred since it results in a faster, more
optimal mesh. Mode 1, however, has no restrictions on the patterns used in each layer.

**CONTACTDATA**

MatNo1  MatNo2  StFact  ZPFact  CanSep?  SepStress  cf  cn  n  OLDist  OLFact  Gap
OR
MatNo1  TOOL  ToolNo  StFact  ZPFact  CanSep?  SepStress  cf  cn  n  OLDist  OLFact  Gap

Specifies contact parameters to be used between materials MatNo1 and MatNo2, or between
MatNo1 and tool ToolNo. Parameter StFact is the normal contact stiffness factor $S_n$ in
equation (4.49). Entering ‘y’ at the CanSep? entry allows the contact pair to separate at the
specified SepStress. Parameter cf is the slip friction coefficient defined in equation (5.1), while a
constant Gap thickness is maintained between the contact partners. Parameters ZPFact, cn, n,
OLDist and OLFact are experimental parameters for non-linear contact and friction that are
unused at present, and should be set to zero (except for OLDist and OLFact, which must be 1.0).

**CONTACTTOL**  ContactTol

Allows the user to specify how close two nearby surfaces must be to be considered “in
contact” at the start of the program.

**ALLOWCONTACT**

LayerNo1  SideNo1  LayerNo2  SideNo2
OR
LayerNo1  SideNo1  TOOL  ToolNo

To speed up contact detection and to avoid physically impossible contacts, the user must
specify which sides (0 = bottom, 1 = top) of which layers may touch each other during the
simulation. Contact against tools can also be enabled with this command. In the example given,
contact between deformable surfaces is doubled-up, so that side 1 of layer 1 can touch side 0 of
layer 2, and side 0 of layer 2 can touch side 1 of layer 1. This has been found to be more stable
than one-way contact, but for reasons of flexibility it is not switched on automatically.

**VIEWCONTACT**

LayerNo  SideNo

Used to specify which contacts are output to the .pst file for viewing during post-
processing.

**VARIABLE**  VarNo  NumSteps

Step  Time  Value

Variables allow the user to control the load, boundary conditions and other parameters
precisely during the simulation. Each variable has a certain number of Steps, each specifying the
Value of a variable at a certain Time. The units of the value depend on what the variable is being
used for - most often displacement in metres. Between the steps the value at a given time is
calculated by interpolating between the values at the next lowest and next highest times in the
listed steps, and the last value is used after the time value of the last step. Thus variable 1 in the
example is zero throughout the simulation, while variable 2, controlling the vertical
displacement of the male dome tool, changes from 30 to 0 mm in 2 seconds, and is held
thereafter.

**SYMMETRY**  X|Y = Value  [WEAK|SUPER]

Used to enforce conditions of symmetry along the specified plane. All nodes along the
symmetry plane are subject to the specified displacement restriction. Since the nodes used in
bicubic Hermite elements possess both displacement and slope degrees of freedom, two different symmetry assumptions are possible. By following the command with the word WEAK, no slopes are affected on the specified symmetry plane, which allows for rotational symmetry. If WEAK is omitted, the appropriate slope and twist vectors are set to zero to simulate the more restrictive mirror symmetry. The SUPER parameter is used in conjunction with the *EQUATE command to constrain the model to two dimensions.

*SYMMETRY  Y = YLimit1  SUPER
*SYMMETRY  Y = YLimit2  SUPER
*EQUATE  0  2  3  5

Used together, these commands convert a three dimensional strip into a two dimensional model. The model must have one element only in the y (\(\xi_2\)) direction and these must be rectangular with two sides in the planes specified by the *SYMMETRY commands. The *EQUATE command forces those nodal displacements and slopes not set to zero by the SUPER symmetry constraints to be constant through the strip, thereby forcing a plane strain deformation. This set of commands was used in the examples of Sections 5.3.2 and 6.2.

*PRESSURES
LayerNo  SideNo  VarNo

Used to apply pressure loading on side SideNo (0 = bottom, 1 = top) of the specified LayerNo. Variable VarNo then sets the pressure (in Pa) to be applied to that surface over the course of the simulation.

*BDYCONDS  NumBCs
BCNo  LayerNo  PatNode  zNo  Vect  DOF  VarNo  [NATURAL]

With most forming simulations, sufficient boundary conditions will be set by the contact and symmetry routines to prevent rigid body motion. Nevertheless, it is occasionally necessary to set exact values of node degrees of freedom, or to apply point loads. Nodal degrees of freedom are specified with this command by the layer, node number in the pattern, and the through-thickness node in that layer (\(zNo = 1..TNodes\) of the *LAYERS commands). The Vect parameter ranges from 1 to 4, corresponding to the 4 nodal vectors shown in Figure 3.4. Each vector has 3 degrees of freedom, referenced by the parameter DOF. Variable VarNo controls the value of the specified degree of freedom throughout the simulation, and if Vect = 1, this will be a displacement. Adding NATURAL to the command line changes the boundary condition to apply a point load, in Newtons, to the specified nodal degree of freedom.

*TIME  TotalTime  NumStartTimeIncs  StartTimeIncs  MaxTimeInc  TimeError

Controls the length of initial time steps and total simulation time. The simulation ends as soon as TotalTime is reached. At the start of the simulation, the first NumStartTimeIncs time steps will be of duration StartTimeIncs. In the example, however, constant time increments are used for all time steps in the simulation. This is appropriate for controlling matched-die forming. The MaxTimeInc parameter sets the maximum time increment allowed in the analysis, while the TimeError is a small value used as a tolerance for comparing real numbers representing time. After NumStartTimeIncs time steps have passed, the program calculates time steps to keep maximum displacements nearly constant in each increment, as described under *DISPLCONTROL.

*ITERCONTROL  MinIters  MaxNewtonIters  MaxIters  AbortMaxIters  MaxResidualRatio
MinIterDispl
Controls the number of iterations and convergence of the non-linear solver. The number of iterations in each increment will be at least \texttt{MinIters} and at most \texttt{MaxIters}. If \texttt{MaxNewtonIters} is less than \texttt{MaxIters}, the solver will use a modified Newton-Raphson solution scheme after the specified number of Newton-Raphson iterations have been performed. Iterations are performed very rapidly with this scheme, but it takes many more to reach convergence, and it is often unstable. \texttt{AbortMaxIters} can be made less than \texttt{MaxIters} if it is desired to stop the program once that number of iterations have been performed in a given increment. The \texttt{MaxResidualRatio} is the ratio of Residual Norm/Load Norm mentioned in Section 4.2.3 as a measure of convergence. Lower values signify closer convergence; a value of 0.01 to 0.001 is usually acceptable. The program can also regard the scheme as having converged if the maximum displacement in any iteration is under \texttt{MinIterDispl}.

\textbf{*DISPLCONTROL TargetMaxDispl AbortMaxDispl DTRFraction \ [NORM]} \n
The \texttt{*TIME} command was used to specify the duration of the initial few time steps. When simulating diaphragm/pressure forming the initial forming rate is high, requiring initial time steps to be short. As membrane strains build up forming slows significantly, meaning longer time steps would be more appropriate. After the initial time steps, the program adjusts the time increments based on the current forming rate, so that the maximum displacement will be near to \texttt{TargetMaxDispl}. If NORM is specified, the maximum displacement uses the norm of the displacement vector, otherwise the greatest single component is used. The \texttt{DTRFraction} parameter should be given a value from 0.25 to 1. A value of 1 means that the displacement in the last increment is used exclusively to calculate the current forming rate. A lower value averages forming rates over several increments, smoothing out the effect of sudden changes in the magnitude of displacements in an increment. The \texttt{AbortMaxDispl} parameter should be set to a displacement value that is unacceptable in an iteration, signifying that the program should revert to the previous time and use a shorter time step.

\textbf{*POST PostFreqIncr PostInitIncr \ [ITER PostIterTime]} \n
This command tells the program to output the deformed mesh and calculated stresses to the post-processing file (extension .pst) every \texttt{PostFreqIncr} time increments, as well as after each of the first \texttt{PostInitIncr} increments. If \texttt{ITER} is entered on the line, output will be sent every iteration after the specified \texttt{PostIterTime}.

\textbf{*PIVOT PivotRange} \n
Controls the amount of implicit partial pivoting used in the LU decomposition. The \texttt{SimForm} system matrix is subdivided into ‘submatrices’ of $12 \times 12$ double precision real numbers, reflecting the degrees of freedom in each node. The \texttt{PivotRange} parameter tells the solver the number of submatrices below the main diagonal it should search to find the next pivot. A greater number increases the precision of the decomposition, but generally slows it down since pivoting can greatly increase the system bandwidth. The default \texttt{PivotRange} of 0 performs pivoting only in the submatrix on the main diagonal of the system matrix.

\textbf{*END} \n
Marks the end of the input file.
Appendix C: *SimForm* Program Verification

A major part of developing and maintaining a Finite Element code is to verify its calculations by comparison with analytical expressions and results produced by other programs. This appendix presents several example calculations verifying the implementation of the material models in *SimForm*. Following that is a brief summary of some of the tests used to check the contact procedures and other parts of the program.

The essence of these tests is to assess the performance of the main features of the program in isolation. The analysis of how *SimForm* handles the simultaneous use of finite strain and the highly anisotropic material laws, contact, and even inertia, forms the basis of Chapters 5 to 7.

**Small Strain Orthotropic Elasticity**

In addition to the large strain model of equation (4.38), the Mooney material and the incompressible, inextensible-fibre "Spencer" material, a small strain orthotropic material model has been implemented in *SimForm*. This uses nine material constants, comprising the three Young’s Moduli $E_1$, $E_2$ and $E_3$, the three Poisson’s ratios $\nu_{12}$, $\nu_{23}$, $\nu_{31}$, and three shear moduli, $G_{12}$, $G_{23}$ and $G_{31}$. With this model the incremental elasticity matrix of equation (4.26) is constant and equal to:

$$
D = \begin{bmatrix}
    cE_1 (1 - \nu_{23}\nu_{32}) & cE_1 (\nu_{21} + \nu_{23}\nu_{31}) & cE_1 (\nu_{31} + \nu_{32}\nu_{21}) & 0 & 0 & 0 \\
    cE_2 (\nu_{12} + \nu_{13}\nu_{32}) & cE_2 (1 - \nu_{31}\nu_{13}) & cE_2 (\nu_{12} + \nu_{13}\nu_{21}) & 0 & 0 & 0 \\
    cE_3 (\nu_{13} + \nu_{12}\nu_{23}) & cE_3 (\nu_{23} + \nu_{21}\nu_{13}) & cE_3 (1 - \nu_{12}\nu_{21}) & 0 & 0 & 0 \\
    0 & 0 & 0 & G_{12} & 0 & 0 \\
    0 & 0 & 0 & 0 & G_{23} & 0 \\
    0 & 0 & 0 & 0 & 0 & G_{31}
\end{bmatrix}
\quad (C.1)
$$

where, $\nu_{21} = \nu_{12}/E_1$, $\nu_{32} = \nu_{23}/E_2$, $\nu_{13} = \nu_{31}/E_3$, $c = 1/(1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{31}\nu_{13} - \nu_{12}\nu_{23}\nu_{31} - \nu_{21}\nu_{32}\nu_{13})$.

Isotropic linear elasticity is a special case of this model in which $E_1 = E_2 = E_3 = E$, $\nu_{12} = \nu_{23} = \nu_{31} = \nu$ and the shear moduli are given by $G_{12} = G_{23} = G_{31} = G/E/(2(1+\nu))$. This model has been introduced here since it is employed in the following comparisons between results calculated by *SimForm* and beam and plate theory.

**Simple Beam Theory**

This example is of a 10×10 mm square section, 200 mm long beam, simply supported and centrally loaded by a 200N force. The material is isotropic and linear elastic, with $E = 200$GPa and $\nu = 0.3$. Comparison will be made of central deflection and bending stresses.

Details on how to solve this problem using simple beam theory may be found in numerous mechanics texts, so only the results are given here. In Table C.1 the simple beam results are compared with two finite element models of different mesh density, both of which take advantage of symmetry to model half the beam, subject to half the load along the symmetry plane. Note that both finite element models use a single element with quadratic
interpolation through the thickness of the beam. Also, the stresses are extrapolated and smoothed from values calculated at the Gauss points, which are of greater accuracy.

Table C.1 Comparison of SimForm bending results with Simple Beam Theory.

<table>
<thead>
<tr>
<th>Model:</th>
<th>Simple Beam Theory</th>
<th>SimForm, 1 element</th>
<th>SimForm, 4 elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Central Deflection:</td>
<td>0.200</td>
<td>0.201</td>
<td>0.201</td>
</tr>
<tr>
<td>Maximum Bending Stress (MPa):</td>
<td>+60.0</td>
<td>+61.5</td>
<td>+59.6</td>
</tr>
<tr>
<td>Minimum Bending Stress (MPa):</td>
<td>-60.0</td>
<td>-61.1</td>
<td>-59.6</td>
</tr>
</tbody>
</table>

Plate Theory

The following plate examples are taken from Table 2.4 on page 96 of Zienkiewicz and Taylor’s The Finite Element Method, Volume 2 [32]. The examples are for a 10×10 square plate, simply supported around its edge and subject to a uniform pressure of 1 (using a consistent set of units). The material is isotropic with $E=10.92$ and $\nu=0.3$. Central displacements are calculated for ratios of side length to thickness ($a/t$) of 10 (thick plate) and 100 (thin plate). Table C.2 reprints their results for the soft support conditions most applicable to simple support.

Table C.2 Deflection at the centre of a square plate subject to uniform pressure, from Reference [32].

<table>
<thead>
<tr>
<th>Mesh density, $M$</th>
<th>Thick plate ($a/t$=10), $\times 10^{-1}$</th>
<th>Thin plate ($a/t$=100), $\times 10^{-4}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>4.3992</td>
<td>4.0903</td>
</tr>
<tr>
<td>4</td>
<td>4.4600</td>
<td>4.0737</td>
</tr>
<tr>
<td>8</td>
<td>4.5393</td>
<td>4.0719</td>
</tr>
<tr>
<td>16</td>
<td>4.5906</td>
<td>4.0756</td>
</tr>
</tbody>
</table>

Unfortunately, typographical errors in this table mean that the results and at least one of the parameters given are out by several orders of magnitude. The thin plate deflection should be around 1000 times that of the thick plate - not the reverse as printed. The most likely cause of this mistake was that $10^{-1}$ was typed in instead of $10^{-7}$, so the thin plate deflections are correct, but the thick plate values are $10^{6}$ times higher than they should be. However, thin plate theory (from the same reference) gives a centre displacement of $4.0623\times 10^{-4}$ (as listed in the same table) only if $E=10.92\times 10^{5}$.

To make matters more complicated it was assumed for the SimForm models that the $10^{-1}$ and $10^{-4}$ in Table C.2 should be swapped, so that $E=10.92\times 10^{5}$ gives results consistent with thin plate theory. As a consequence, the thin plate model involves large displacements, so its centre deflection converges to a much lower value after 8 Newton-Raphson iterations. However, as the results in Table C.3 show, the displacement in the first iteration matches that returned by thin plate theory, since it is inversely proportional to the Young’s modulus. Furthermore, the numerical values of displacement match those in Table C.2 well and differ only by the orders of magnitude expected from this discussion.
**Table C.3** Deflections at the centre of a square plate subject to uniform pressure, calculated by *SimForm*.

<table>
<thead>
<tr>
<th>Mesh (quarter plate)</th>
<th>Thick shell (a/t=10), $\times 10^{-4}$</th>
<th>Thin Shell (a/t=100), $\times 10^{-3}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Converged</td>
<td>First iteration</td>
</tr>
<tr>
<td>1 $\times$ 1</td>
<td>4.4981</td>
<td>4.1251</td>
</tr>
<tr>
<td>2 $\times$ 2</td>
<td>4.5811</td>
<td>4.0707</td>
</tr>
<tr>
<td>3 $\times$ 3</td>
<td>4.6282</td>
<td>4.0715</td>
</tr>
</tbody>
</table>

Note that a quarter of the plate was modelled for the above results, while quadratic through-thickness interpolation was used. The cubic elements used in *SimForm* perform very well in beam and plate problems since the analytically calculated deflected shape can usually be described by a high order polynomial, such as a quartic in the above thin plate case. It is also noteworthy how well the single element with a 50:1 aspect ratio has performed in the thin plate example.

**Finite Strain, Uniaxial Extension**

Expressions for the stresses in a block of incompressible, isotropic Mooney material subject to stretches $\lambda_1$, $\lambda_2$ and $\lambda_3$ in the three principal directions can be simply calculated by removing terms involving $I_4$ and $I_5$ from equation (4.42), and inserting this into equation (4.36). Principal Cauchy stresses are then found for this simple case by multiplying each $\tilde{T}_{jk}$ (not summed) by $\lambda_k^2$ to give,

$$\sigma_1 = -p + 2K_1\lambda_1^2 + 2K_2\left(\lambda_1^2\lambda_2^2 + \lambda_1^2\lambda_3^2\right),$$

(C.2)

$$\sigma_2 = -p + 2K_1\lambda_2^2 + 2K_2\left(\lambda_2^2\lambda_1^2 + \lambda_2^2\lambda_3^2\right),$$

(C.3)

$$\sigma_3 = -p + 2K_1\lambda_3^2 + 2K_2\left(\lambda_3^2\lambda_1^2 + \lambda_3^2\lambda_2^2\right).$$

(C.4)

The remaining equation governing such deformations is the incompressibility condition,

$$\lambda_1\lambda_2\lambda_3 = 1.$$  

(C.5)

For the case of uniaxial extension by a given $\lambda_1$, with transverse stresses $\sigma_2 = \sigma_3 = 0$, the other principal stretches are $\lambda_2 = \lambda_3 = 1/\sqrt[3]{\lambda_1}$, and,

$$\sigma_1 = 2K_1\left(\lambda_1^2 - 1/\lambda_1\right) + 2K_2\left(\lambda_1^2 - 1/\lambda_1^2\right),$$

(C.6)

$$p = 2K_1/\lambda_1 + 2K_2\left(\lambda_1 + 1/\lambda_1^2\right).$$

(C.7)

Table C.4 presents stresses and pressures calculated using the above formulae for several values of $\lambda_1$, taking $K_1=1.1\times10^9$ and $K_2=0.25\times10^9$. This is compared with three other cases calculated using a single element mesh in *SimForm*. Case 2 uses the same Mooney material with trilinear interpolation of pressure over the bicubic Hermite element with linear through-thickness variation. Case 3 uses strain energy function (4.38) with $K_1$ and $K_2$ as in the Mooney models, $K_3=5\times10^6$ and $K_4=K_5=0$. For interest, case 4 uses the small strain orthotropic model of equation (C.1), with $E=6(K_1+K_2)$, $G=2(K_1+K_2)$ and $\nu = 0.48$. The last two cases are compressible, so their volume changes are also listed in the table. Note that a constant volume was maintained in case 2.
Table C.4 Uniaxial extension of several material models. All stresses and pressures are in kPa.

<table>
<thead>
<tr>
<th>( \lambda_1 )</th>
<th>1. Equations (C.6,7)</th>
<th>2. Mooney, computed</th>
<th>3. Approx. Mooney</th>
<th>4. Small strain (C.1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_1 )</td>
<td>( p )</td>
<td>( \sigma_1 )</td>
<td>( p )</td>
<td>( V/V_0 )</td>
</tr>
<tr>
<td>1.00</td>
<td>0.0</td>
<td>320.0</td>
<td>0.0</td>
<td>320.0</td>
</tr>
<tr>
<td>1.01</td>
<td>8.1</td>
<td>317.3</td>
<td>8.1</td>
<td>317.3</td>
</tr>
<tr>
<td>1.10</td>
<td>79.9</td>
<td>296.3</td>
<td>79.9</td>
<td>296.3</td>
</tr>
<tr>
<td>1.25</td>
<td>198.3</td>
<td>270.5</td>
<td>198.3</td>
<td>270.5</td>
</tr>
<tr>
<td>1.50</td>
<td>401.1</td>
<td>243.9</td>
<td>401.1</td>
<td>243.9</td>
</tr>
<tr>
<td>2.00</td>
<td>857.5</td>
<td>222.5</td>
<td>857.5</td>
<td>222.5</td>
</tr>
</tbody>
</table>

Note that identical results have been produced using quadratic through-thickness interpolation and multi-element meshes.

The second finite strain example is of unit extension transverse to the fibre direction in an inextensible or near-inextensible model. Stresses in this case are still given by equations (C.2) to (C.4) at a given \( \lambda_1 \), by taking \( \lambda_2=1 \), so that \( \lambda_3=1/\lambda_1 \) by equation (C.5), while \( \sigma_3=0 \). Hence,

\[
\sigma_1 = 2\left(K_1 + K_2\right)\left(1 - 1/\lambda_1^2\right), \tag{C.8}
\]

\[
p = 2K_1/\lambda_1^2 + 2K_2\left(1 + 1/\lambda_1^2\right), \tag{C.9}
\]

\[
\sigma_2 = 2K_1\left(1 - 1/\lambda_1^2\right) + 2K_2\left(\lambda_1^2 - 1\right). \tag{C.10}
\]

Transverse stress \( \sigma_2 \) in the constrained Mooney material should be equal and opposite to the tension in the fibres of an inextensible material. Table C.5 compares the stresses calculated using the above formulae with those calculated by *SimForm* for the inextensible, incompressible “Spencer” material described by equations (4.40) to (4.42), and a third case using strain energy function (4.38). In all cases \( K_1=1.1\times10^5 \) and \( K_2=0.25\times10^5 \), while case 3 uses \( K_3=5\times10^6 \) and \( K_4=1\times10^8 \) to give near-incompressible and near-inextensible behaviour. \( K_5 \) is taken as zero in both cases 2 and 3.

Table C.5 Uniaxial extension of a transversely constrained block. All stresses and pressures are in kPa.

<table>
<thead>
<tr>
<th>( \lambda_1 )</th>
<th>1. Equations (C.8,10)</th>
<th>2. Spencer, computed</th>
<th>3. Approx. Spencer, computed</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \sigma_1 )</td>
<td>( \sigma_2 )</td>
<td>( p )</td>
<td>( T )</td>
</tr>
<tr>
<td>1.00</td>
<td>0.0</td>
<td>0.0</td>
<td>320.0</td>
</tr>
<tr>
<td>1.25</td>
<td>249.1</td>
<td>107.3</td>
<td>249.1</td>
</tr>
<tr>
<td>1.50</td>
<td>487.5</td>
<td>184.7</td>
<td>487.5</td>
</tr>
<tr>
<td>1.75</td>
<td>738.7</td>
<td>251.3</td>
<td>738.7</td>
</tr>
<tr>
<td>2.00</td>
<td>1012.5</td>
<td>315.0</td>
<td>1012.5</td>
</tr>
</tbody>
</table>
Other Tests

The numerical results of examples testing other aspects of SimForm are far less interesting than those presented above. In most cases it suffices to say that the results are as expected, or that the behaviour is logical. The following are some of the other significant verification tests carried out with SimForm.

- Provided pressure and contact loads are integrated over at least $3 \times 3$ Gauss points, uniform pressure over a flat plate against a flat surface gives a constant compressive stress in the material and constant normal contact stresses at the contact points, both equal in magnitude to the pressure applied.
- Several tests have confirmed that the contact friction calculations return shear stresses as would be predicted from equation (4.63), as well as the expected results in the ply-pull-out tests of Section 5.2.2.
- Pressure forming of single layers of diaphragm material (with parameters from equation (5.6)) into complex tool geometries works exactly as desired.
- Uniform extension starting in one axis and rotating to another produces the same net deformation and stresses as in the non-rotating case.
- Pressure is applied over a rectangular plate exactly covering an identical plate which is lying flat against a slightly larger rectangular tool surface. The tool is rotated about its edge in a series of steps until it is oriented at $90^\circ$ to its original position. At each step the same residual values and number of iterations are required to converge, while contact and internal stresses remain constant except for the direction change.
- As explained in Section 6.3.1, the inertia calculations are not as accurate as under the Newmark scheme. Nevertheless, they perform in a predictable and sensible manner. Simply supported beams and plates allowed to deflect under their own self-weight oscillate until they reach their static equilibrium position, equal to the deflection produced by a uniform pressure of the weight of the body per unit top surface area (assuming they are of constant thickness). Note that the oscillations diminish in magnitude not as a result of deliberate dissipative losses in the system, but due to inaccuracies in the inertia calculations.