Validation of material model for polypropylene (PP)

Kjetil Vange

Civil and Environmental Engineering
Submission date: June 2012
Supervisor: Arild Holm Clausen, KT
Co-supervisor: Marius Andersen, KT

Norwegian University of Science and Technology
Department of Structural Engineering
SUMMARY:
Polymers are the up and coming engineering material. The number of possible applications is increasing fast and so is the need for a good polymer material model. The scientists at SIMLab at NTNU have developed such a material model. In this thesis is the material model calibrated to represent an impact modified polypropylene used by Toyota.

The validation started with the calibration. A review of the theoretical background of the material model and a detailed explanation of the calibration procedure are given to describe the derivation of the calibrated material model. The model is then applied in FE-simulations of the material tests and the two validation tests. The first validation test is a quasi-static tension test using a specimen with a centric hole. This test resembles the material tests but the centric hole makes the load response slightly more complex. The other test is an impact test using a drop tower. This test validates the material models ability to represent dynamic problems and also tests the representation of the strain rate dependency.

The material model proves that it is capable of representing all the tests in this thesis quite accurately.
Validation of material model for polypropylene (PP)
(Validering av materialmodell for polypropylen (PP))

As a part of the development of the finite element method, significant effort has been devoted to propose new material models which are able to represent the material behaviour at different conditions. Relevant parameters involve strain level, strain rate, temperature, and the material at hand may also exhibit anisotropy, viscoelasticity etc. Today, material models for metals are in general more accurate than models for e.g. polymers. This is a general tendency for all non-linear finite element programs, including Abaqus and LS-DYNA. SIMLab is involved in the development of improved models for thermoplastics. A model applicable for ductile polymers was ready in 2010, and the research work continues with other classes of models.

All new models need to be validated against experimental benchmark tests for different types of polymers. Validation involves also calibration, requiring material tests on the material at hand. This master thesis is concerned with a PP material provided by Toyota. The selected benchmark tests are tension of a plate with a hole and impact tests on small plates. Subsequently, the test results are to be compared with numerical predictions applying the material model for ductile thermoplastics.

Some keywords for activities related to this master thesis project are:
- Literature: Polymers in general, relevant test results, material models.
- Experimental tests: Material and component tests. Presentation of test results.
- Calibration: Identify the coefficients of the material model.

The candidate may agree with the supervisors to pay particular attention to specific parts of the investigation, or include other aspects than those already mentioned.

The thesis is to be organized as a research report, recognising the guidelines provided by Department of Structural Engineering.

Supervisors: Arild H. Clausen, Marius Andersen and Anne Serine Ognedal

The report is to be handed in at Department of Structural Engineering not later than 11 June 2012.
Sammendrag


Resultatene viser at materialmodellen evner å representere testene svært tilfredsstilende.
Acknowledgements

I would like to express my gratitude to Professor Arild H. Clausen for his excellent guidance throughout the process of writing this thesis. I would also like to thank co-supervisor PhD student Marius Andersen and PhD student Anne Serine Ognedal for their contributions.

Tonje Hegni, Kristin Sælen and Torgrim Østen wrote similar theses. The meetings and discussions with this group have been decisive for the result.

A special thanks also goes to my fellow students for providing good input and feedback.

Trondheim, 7.6.2012
Contents

1 Introduction ........................................... 3

2 Theory .................................................. 5
   2.1 General chemical structure ......................... 5
   2.2 Mechanical properties ............................... 6
   2.3 Introduction to the SIMlab polymer material model ........ 9
   2.4 Polypropylene ........................................ 12

3 Material tests .......................................... 15
   3.1 Uniaxial tension tests ............................... 17
      3.1.1 Low strain rate, $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$ ... 18
      3.1.2 Average strain rate, $\dot{\varepsilon} = 10^{-2}$ s$^{-1}$ ... 19
      3.1.3 High strain rate, $\dot{\varepsilon} = 10^{-1}$ s$^{-1}$ ... 21
   3.2 Uniaxial compression tests .......................... 22
   3.3 Summary .............................................. 24

4 Calibration .............................................. 25
   4.1 The constitutive model ................................ 26
      4.1.1 Part A - Intermolecular resistance ............... 26
      4.1.2 Part B - Network resistance ...................... 30
   4.2 The calibration procedure ............................ 30
      4.2.1 Parameters based on transverse deformations ... 31
      4.2.2 Parameters based on the yield stresses .......... 33
      4.2.3 Parameters found from the stress-strain curve of the reference test .... 36
   4.3 Application of the calibrated material model .......... 40
      4.3.1 Uniaxial tension ................................. 40
      4.3.2 Uniaxial compression ............................. 45
      4.3.3 Comparison and discussion ....................... 48

5 Validation .............................................. 55
5.1 Plate with centric hole ........................................... 55
5.1.1 The material test ............................................. 55
5.1.2 The finite element model ...................................... 59
5.1.3 Results ......................................................... 60
5.1.4 Comparison and discussion ................................... 66
5.2 Impact loading on plate ........................................... 69
5.2.1 The material tests ............................................. 69
5.2.2 The finite element model ...................................... 70
5.2.3 Results ......................................................... 71
5.2.4 Comparison and discussion ................................... 77

6 Conclusion ......................................................... 79

A Material tests ....................................................... 3
A.1 Calibration ......................................................... 4
  A.1.1 T1-L1, $\dot{\epsilon} = 10^{-3} \text{s}^{-1}$ .......................... 5
  A.1.2 T2-L1, $\dot{\epsilon} = 10^{-3} \text{s}^{-1}$ .......................... 6
  A.1.3 T3-L1, $\dot{\epsilon} = 10^{-3} \text{s}^{-1}$ .......................... 7
  A.1.4 T4-L1, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ .......................... 8
  A.1.5 T5-L1, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ .......................... 9
  A.1.6 T6-L2, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ .......................... 10
  A.1.7 T7-L2, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ .......................... 11
  A.1.8 T8-L1, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ .......................... 12
  A.1.9 T9-T2, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ .......................... 13
  A.1.10 T10-T2, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ ........................ 14
  A.1.11 T11-L1, $\dot{\epsilon} = 10^{-1} \text{s}^{-1}$ ........................ 15
  A.1.12 T12-L1, $\dot{\epsilon} = 10^{-1} \text{s}^{-1}$ ........................ 16
  A.1.13 T13-T2, $\dot{\epsilon} = 10^{-1} \text{s}^{-1}$ ........................ 17
  A.1.14 T14-T2, $\dot{\epsilon} = 10^{-1} \text{s}^{-1}$ ........................ 18
  A.1.15 C1, $\dot{\epsilon} = 10^{-3} \text{s}^{-1}$ ............................. 19
  A.1.16 C2, $\dot{\epsilon} = 10^{-3} \text{s}^{-1}$ ............................. 20
  A.1.17 C3, $\dot{\epsilon} = 10^{-2} \text{s}^{-1}$ ............................. 21
A.2 Plate with centric hole .......................................... 22
  A.2.1 D1V1, $v=0.05 \text{mm/s}$ .................................... 23
  A.2.2 D2V1, $v=0.05 \text{mm/s}$ .................................... 24
  A.2.3 D1V2, $v=0.5 \text{mm/s}$ .................................... 25
  A.2.4 D2V2, $v=0.5 \text{mm/s}$ .................................... 26
A.3 Drop tower ......................................................... 27
  A.3.1 DT1, $E \sim 50 \text{J}$ .......................................... 28
  A.3.2 DT2, $E \sim 100 \text{J}$ .......................................... 29
  A.3.3 DT3, $E \sim 100 \text{J}$ .......................................... 30
A.3.4 DT4, E∼75 J ................................. 31
A.3.5 DT5, E∼60 J ................................. 32
A.3.6 DT6, E∼55 J ................................. 33

B Matlab scripts 35
B.1 Finding the critical cross-section of the material tests .......... 36
B.2 The true stress-strain curve for the compression tests .......... 40
B.3 Calculation of the yields stresses .......................... 43
B.4 Plate with centric hole .................................... 49
B.4.1 The transversal displacement in the gauges ............... 49
B.4.2 Identifying the correct deformation rate ................... 52
B.4.3 The longitudinal displacements ............................ 55

C LS-DYNA input files 57
C.1 Material cards .................................. 58
  C.1.1 ppFinal.k, E=400 MPa .......................... 58
  C.1.2 ppFinalE700.k, E=700 MPa ..................... 58
  C.1.3 ppFinalE750.k, E=750 MPa ..................... 59
  C.1.4 ppFinalDTOPT.k, E=700 MPa and C=0.07 .......... 59
  C.1.5 ppFinalDTOPT_075.k, E=700 MPa and C=0.070 .... 60
  C.1.6 ppFinalDTOPT_08.k, E=700 MPa and C=0.08 ....... 60
C.2 Finite element analyses of the calibration tests ............... 62
  C.2.1 Input file .................................. 62
  C.2.2 Excerpt from the geometry file geomHalf.k ........... 64
C.3 Finite element analyses of the plate with a centric hole ...... 68
  C.3.1 Input file .................................. 68
  C.3.2 Excerpt from the geometry file geomTFV2_4thick.k .... 69
C.4 Drop tower ...................................... 73
  C.4.1 Input file .................................. 73
  C.4.2 Excerpt from the geometry file geommdt.k ........... 77
List of Figures

2.1 Monomer to polymer ........................................ 6
2.2 Plastic deformation in semi-crystalline polymers ............... 9
2.3 Rheological model and resulting stress-strain diagram ....... 10
2.4 The monomer of polypropylene ................................ 12
2.5 Two of the possible tacticities of PP .......................... 13
2.6 Injection moulding machine .................................. 14
3.1 Painted specimen after testing ................................. 16
3.2 Tension specimen geometry and tension test rig ............... 17
3.3 Test results, \( \dot{\varepsilon} = 10^{-3} \text{ s}^{-1} \) ....................... 18
3.4 The external layer after fracture ............................... 19
3.5 Selected force-displacement curves, \( \dot{\varepsilon} = 10^{-2} \text{ s}^{-1} \) .... 20
3.6 Results from tests done with \( \dot{\varepsilon} = 10^{-1} \text{ s}^{-1} \) ............ 21
3.7 Compression test set-up and specimen geometry .............. 22
3.8 The results of the compression tests .......................... 23
4.1 Constitutive model and resulting stress-strain diagram ........ 25
4.2 Strain diagram for the reference test .......................... 31
4.3 Illustration of Considère’s criterion and the strain rate dependence . 34
4.4 The yield stresses and the linear fit used for extrapolation .... 35
4.5 The stress-strain curve of the material model .................. 40
4.6 The meshed geometries applied in the FEM analysis of the calibration tests .................................................. 41
4.7 Illustration of the Part A and Part B contribution to the stress strain curve ............................................. 42
4.8 Diagrams comparing the results of the model and the tests, \( \dot{\varepsilon} = 10^{-3} \text{ s}^{-1} \) .................................................. 43
4.9 Diagrams comparing the results of the model and the tests, \( \dot{\varepsilon} = 10^{-2} \text{ s}^{-1} \) .................................................. 44
4.10 Fringe plot of the deformed calibration model ................. 44
4.11 Diagrams comparing the results of the model and the tests, $\dot{\varepsilon} = 10^{-1}$ s$^{-1}$ .......................................................... 45
4.12 The finite element model of the compression specimen ............. 46
4.13 The results of the finite element model of the uniaxial compression test at $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$ .................................................. 47
4.14 The results of the finite element model of the uniaxial compression test at $\dot{\varepsilon} = 10^{-2}$ s$^{-1}$ .................................................. 48
4.15 Comparison of the strain-rate dependency of the finite element model and the material tests ........................................ 49
4.16 The fracture surface of test T5-L1 ....................................... 49
4.17 Comparison of the hardening at different strain rates ................. 50
4.18 Comparison of the response at different strain rates ................. 53
5.1 The specimens used in the first validation test .......................... 56
5.2 Deformed plates with centric hole (D1V1 and D1V2) .................. 58
5.3 The meshed geometry used in the finite element analyses (D1V2 and D2V2) ................................................................. 59
5.4 Calculation of the strain rate and the deformation rate of test D2V1 60
5.5 The results of the test and the FE-simulation of D1V1 ................. 61
5.6 The results of the test and the optimized FE-simulation of D1V1 .... 62
5.7 The results of the test and the FE-simulations of D2V1 ............... 63
5.8 The results of the test and the FE-simulations of D1V2 ............... 64
5.9 The results of the test and the FE-simulations of D2V2 ............... 65
5.10 The results of the test and the FE-simulations of D2V2 ............... 66
5.11 The strain-rate dependency illustrated by the results of the first validation tests ................................................................. 67
5.12 The test D1V1 and the model deformed approximately 10 mm in the longitudinal direction .............................................. 68
5.13 The meshed geometry of the drop tower model .......................... 70
5.14 The two drop tower specimens that absorbed the impact ............. 72
5.15 The results of test DT1 .................................................... 73
5.16 The results of the test and the analysis of DT1 ......................... 74
5.17 The results of the remaining drop tower tests and simulations ....... 76
5.18 The real and the modelled boundary clamp ............................ 77
B.1 The plot produced by the script shown in the appendix ............... 39
B.2 Illustration of the strain rate dependence ............................... 43
B.3 The transversal deformation in the gauges of test D2V1 ............... 49
B.4 Calculation of the strain and deformation rate of test D2V1 ........... 52
List of Tables

2.1 The eleven coefficients of the material model ................. 11
4.1 The parameters of the friction element ..................... 27
5.1 The specifications of the four validation tests ............... 56
5.2 The parameters used for comparison of the test and model of the first validation test .................................................. 57
5.3 The fracture point of DT2 and DT3 ............................... 75
5.4 The tests carried out using the drop tower ..................... 75
Chapter 1

Introduction

Polymers are the up and coming engineering material. The number of possible applications is increasing fast and so is the need for a good polymer material model. It is in the light of this that the Structural Impact Laboratory (SIMLab) at NTNU has developed a polymer material model for use in finite element simulations. This thesis will validate a calibrated version of this material model.

Polymers are versatile. This is because the polymer chains can be easily modified to achieve numerous properties. In addition is the mass density of polymers much lower than that of more classical engineering materials such as steel and aluminium. This makes polymers particularly interesting for the industry producing light weight structures such as cars and aeroplanes. Another reason for the interest from the automobile industry, is that polymers can be given good shock absorbing abilities. The material treated in this thesis is a version of such a polymer, namely a polypropylene (PP) optimized for use in car bumpers.

The objective of this thesis is to validate a calibrated version of the polymer material model. The material used for calibrating the material model is the PP mentioned above. The thesis starts with presenting some general theory regarding polymers. The next chapter then describes the material tests that make up the basis for the calibration. The next chapter then treats the calibration procedure in detail and derives the calibrated material model. The model is then applied to simulations of the material tests before being applied to the simulations of the validation tests. Two validation tests are carried out. An important point when choosing validation tests is to find tests that validate a wide spectre of properties. The first test chosen for this thesis is a tension test using a plate with a centric hole. This test is similar to the tests carried out when calibrating the material model, but the centric hole makes the load response a bit more complex. The
second test is an impact test using a machine known as drop tower. This is a test where a steel spear hits a plate with a given impact energy. The load response is dynamic and involves high strain rates. Through these two tests are both the quasi-static and the dynamic properties of the material model validated. All the analyses are carried out using the finite element code LS-DYNA.

The chapters are split into sections explaining the different parts of the process. In the chapters presenting results is the last section used to compare and discuss the presented result. The last chapter recapitulate the thesis in a conclusion and present suggestions for further work.
Chapter 2

Theory

The word polymer comes from Greek and means many particles [11]. And polymers are just that, chains of carbon based molecules packed tightly together. Polymers are becoming more and more important as an engineering material. It is a low-weight material that can be modified to achieve numerous properties. This thesis examines a version of the semi-crystalline thermoplast polypropylene, used by Toyota.

The three main categories of polymers are thermoplastics (amorphous and semi-crystalline), elastomers and duromers [12]. The main difference is the degree of cross-linking of the polymer chains, where the duromers have a high degree of cross linking, the elastomers a bit less and the thermoplasts none at all. The degree of cross-linking is important to the mechanical behaviour, which means that the three categories have quite different mechanical properties. Since the material treated in this thesis is a thermoplast, the chapter will only treat thermoplastics.

2.1 General chemical structure

The chemical structure of polymers differs quite a lot from the one found in many other engineering materials. The basic structure is an organic molecule, a monomer. A monomer can in its simplest form be an alkene or alkyne, where the double bond or triple bond is broken, making room for a connection to a neighbouring molecule on each side (see Figure 2.1). The process where these monomers connect and form a chain is called polymerization. When the monomer shown in Figure 2.1 is polymerized it becomes polyethylene, one of the most widely used polymers. This process is the process all polymers go through. By changing
the structure of the monomer or by mixing different monomers, a wide variety of characteristics can be achieved. A relevant example is to substitute one of the hydrogen atoms in the ethene monomer with a methyl group (CH$_3$). This results in the propylene monomer, which polymerizes to polypropylene (Figure 2.4).

The density of the polymer chains defines amorphous and crystalline regions. In crystalline regions the chains are more tightly packed than in amorphous regions. A short bond is stronger than a long bond, which means that the crystalline regions are stronger than the amorphous regions. This principle is especially important when dealing with semi-crystalline thermoplastic.

![Figure 2.1: Connection between an independent molecule and a monomer](image)

### 2.2 Mechanical properties

A certain combination of mechanical properties is often the objective when making a new or improving a polymer. The properties are easily modified, making it possible for a material to fit a certain application exactly. Thanks to this polymers have become preferable in many applications where more known engineering materials as steel and aluminium were used earlier.

The critical temperatures for polymers are often close to the service temperatures. The values of the critical temperatures also differ considerably from polymer to polymer. The critical temperature points are the glass temperature, $T_g$, and the melting temperature, $T_m$. None of them are exact temperatures, but rather a middle of a narrow range of temperatures. $T_g$ applies only to the amorphous regions of the material. As the temperature rises the internal energy and the specific volume increases. When passing $T_g$ the specific volume suddenly start to increase more than before. Making more room for the relative motion of the polymer chains. This affects the material stiffness, making it softer. Had it not been for the entangled structure of the polymer chains, the material would have lost all structure at this point. As the temperature closes $T_m$ the volume continues to increase and the material becomes softer and softer. At the melting temperature the energy of the covalent bonds in the polymer chains is reached, and the bonds break. Since the
chains are dissolved, the material now becomes uniformly amorphous and liquid like. And it has lost all of its stiffness and usability. Most polymers are, however, manufactured so that they do not operate close to $T_m$.

Used at service temperature, thermoplasts have a similar material response as more common engineering materials like steel and concrete. The response is divided into an elastic and a plastic part. The transition between the two is often hard to identify, because it is gradual and not well defined. Both the elastic and the plastic response is time dependent, i.e. the material is viscoelastic and viscoplastic. This is easily seen in a load response diagram, where it is observed that the initial stiffness and the peak load are proportional to the strain rate. The importance of other time effects such as relaxation is proportional with the loading time, i.e. relaxation effects are more significant when the load works over a longer time span.

The bonds between the monomers in a polymer chain are strong covalent bonds, which make the chains a stable and strong chemical structure. The chains are connected to each other by Van der Waals bonds, dipole bonds or hydrogen bonds. These are much weaker than the internal covalent bonds of the molecule chains. This means that more or less all initial elastic deformation takes place in these weaker intermolecular bonds. The elasticity is split into two mechanisms, energy and entropy elasticity. The energy elasticity describes the reaction in the bonds between the chain molecules, and applies to both the amorphous and the crystalline regions of the material. When the polymer is exposed to loading, the bonds are elongated. Upon unloading the bonds retract to their initial positions of a lower energy level, hence energy elasticity. By returning to these positions the material also returns to its initial configuration. This behaviour is dominating when the material operates below $T_g$. As the operating temperature gets closer to $T_g$, the entropy elasticity becomes more and more important. It should be noted that because the entropy elasticity only applies to the amorphous regions, the influence of this mechanism will depend on the degree of polymerization. But when it occurs it is a more complex process than the energy elasticity. In order to fully understand the entropy elasticity one have to understand the concept of entropy. It is not within the scope of this thesis to explain entropy to its full extent. A simplified model is, however, to consider the entanglement of the polymer chains (not to be confused with cross-linking). In its unloaded form the polymer chains are entangled in an intricate mesh. When subjected to loading, the chains are straightened between the entanglement points allowing the material to deform. When unloaded the straightened chains return to a more entangled configuration, through this increasing the level of disorder, which is known as a simple explanation of entropy.

When the deformation potential of the elastic mechanisms are fully exploited the
material starts to deform plastically. Since all the weak bonds are deformed to their maximum, the only possibilities are a fracture or plastic deformation. The outcome strongly depends on the temperature. At a temperature far below $T_g$ the material will fracture without deforming plastically. The closer the temperature gets to $T_g$ the more plastic deformation can be endured by the material. This plastic deformation starts with the breakage of the intermolecular bonds between the polymer chains. This allows the chains to slide relative to each other. The only thing holding the chains together now is the intricate pattern in which they are packed. In order to move, the polymer chain must be able to slide pass its neighbours without being obstructed. Very often the chains obstruct each other, thus increasing the energy needed for further deformation. When the chains are finally able to slide, the plastic deformation has started. This deformation produces heat. If this heat is not able to escape it will speed up the plastic deformation by increasing the volume between the chains. When the yield stress is reached a neck will occur. As the necking process goes further and further the chains in the neck become more and more oriented in the direction of the load. The closer the chains are oriented to the load direction, the larger is the share of the load taken by the covalent bonds in the polymer chains. When the load is fully transferred to the covalent bonds, the strength increases significantly. This is seen as an increase in stress, i.e. hardening, in the stress-strain diagram. The increase is so significant that the neck becomes stronger than its surrounding material. The result is a neck that propagates along the material. In principle is this process able to continue until the neck meets a change in geometry that makes the surrounding material stronger. But a fracture will often occur prior to this due to the accumulation of imperfections. The reduction of the cross-section is also making the neck more exposed for such a fracture.

The mechanisms explained in the paragraph above take place in the amorphous regions of the material. For a semi-crystalline polymer the general behaviour is similar but the deformation is split into the deformation of the amorphous regions and the deformation of the crystalline regions (see Figure 2.2). The amorphous regions are the first do deform (Figure 2.2b). Then the orientation of the crystalline regions is shifted in the direction of the load (Figure 2.2c). When the material deforms further, the crystalline blocks split into smaller blocks (Figure 2.2d and 2.2e). When the deformation has come to the stage shown in Figure 2.2d and 2.2e a neck has occurred. The semi-crystalline material follows the same necking mechanism as explained for the amorphous material. It is worth mentioning that semi-crystalline polymers are more affected by impurities and imperfections than amorphous polymers. This is because the impurities and imperfections are pushed out of the regions that crystallize, locating them right outside the crystalline regions. This makes the transition between the amorphous and the crystalline regions a
2.3. INTRODUCTION TO THE SIMLAB POLYMER MATERIAL MODEL

![Diagram of plastic deformation in semi-crystalline polymers]

Figure 2.2: Plastic deformation in semi-crystalline polymers, after [12]

The compression response of polymers is much less investigated than the tension response and this thesis will not try to reach any new conclusions on the matter. Some general considerations are that; as with tension, the response differs from polymer to polymer. Some polymers have a quite similar load response in tension and compression, distinguished mainly by a higher yield stress in compression than in tension. Others have a quite different response, e.g. softening in tension but not in compression. A higher yield stress in compression than in tension and also hardening effects are, however, seen in the load response of most polymers. A mechanism that is known to occur in compression, causing initially softening and later hardening, is the formation of shear bands. In this mechanism the response is determined by the polymer chain’s response to the shear forces set up by the axial loading [12].

2.3 Introduction to the SIMlab polymer material model

Material models for steel and aluminium have been developed and perfected for many years. The same has not been the case for polymers, and especially not for thermoplastics. The model applied in this thesis is part of a developing process
CHAPTER 2. THEORY

lead by the scientists at SIMLab. SIMLab is a Centre for Research-based Innovation (SFI) located at the Department for Structural Engineering at the Norwegian University of Science and Technology (NTNU). SIMLab is in the process of developing a comprehensive material model for thermoplastics. The objective is a material model that can satisfactorily describe the special properties of a wide range of thermoplastic polymers [1]. In addition the SIMlab group has focused on making the model attractive to the industry, i.e. making it fairly simple to use. This is mainly achieved by carefully examining each parameter of the material model. In other words; a property that has little influence on the response, but demands a complex calibration is left out. Next the chosen properties are incorporated into an easy-to-follow calibration procedure. In this chapter the model will just be briefly introduced, the main study is found in Chapter 4.

Figure 2.3: The rheological model and the resulting stress-strain diagram (after [2])

Figure 2.3 shows a graphical representation of the material model. It is split into two parts, Part A and Part B, which have the same deformation gradient $\mathbf{F} = \mathbf{F}_A = \mathbf{F}_B$. Part A describes the intermolecular resistance of the material, i.e. the stretching of the bonds in the molecules. First the weak and then the strong, as explained in the previous section. When the yield stress is reached Part A goes into a state of plastic flow, which is seen in Figure 2.3b as plastic deformation without increase of stress. The rheological model describes these mechanisms with a Neo-Hookean spring and a plastic flow element coupled in series. The plastic flow element consists of a friction element and a viscous damper coupled in parallel (see Figure 2.3a). Part B describes the intramolecular resistance, which is the
### 2.3. INTRODUCTION TO THE SIMLAB POLYMER MATERIAL MODEL

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$ Young’s modulus. Often given by manufacturer</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$ Poisson’s ratio. Determined from the transverse strains</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>$\sigma_T$ The yield stress at a fictitious “zero strain rate” (detailed explanation in Chapter 4)</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>$\sigma_s$ The saturation stress at a fictitious “zero strain rate” (detailed explanation in Chapter 4). A parameter used to describe the softening.</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>$H$ A rate independent hardening parameter</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>$\alpha$ The ratio between the yield stress in compression and tension</td>
<td></td>
</tr>
</tbody>
</table>
| 7 | $\beta$ A parameter controlling the plastic dilatation.                   | $\beta \geq 1$  
$\beta = 1.0$ = Isochoric (volume preserving) conditions |
| 8 | $\dot{\varepsilon}_{0A}$ The reference strain rate                        |                                                                      |
| 9 | $C$ A strain rate sensitivity parameter                                    |                                                                      |
| 10| $C_R'$ Initial elastic modulus                                            |                                                                      |
| 11| $\lambda_L$ Locking stretch                                               |                                                                      |

Table 2.1: The eleven coefficients of the material model. The colour coding shows the relation to the rheological model in Figure 2.3a.

Resistance set up by the adhesion and entanglement of the polymer chains, i.e. the mechanism of polymer chains sliding along each other and the mechanism in which polymer chains are stretched between entanglement points. This corresponds to the entropy driven mechanisms in the polymer (described in more detail in the previous section). The rheological model describes this with a hyperelastic spring which is controlled by an initial stiffness and a locking stretch. It should also be noticed that the two parts can be superpositioned, making the Cauchy stress tensor equal to:

$$\sigma = \sigma_A + \sigma_B \quad (2.1)$$

Experiments show that polymers dilate during plastic flow [9]. This means that it is possible to describe the plastic flow using a pressure-dependent yield criterion and an associated flow rule. A typical choice for a polymer material is Raghava’s yield criterion (details in Chapter 4). It is, however, shown that this assumption predicts unrealistically large volumetric strains (too much dilatation). A non-associative flow rule is thus needed. The solution is to apply a modified Raghava criterion where the parameter $\beta$ is introduced in order to control the dilatation (Table 2.1, row 7).
CHAPTER 2. THEORY

The material model is implemented as an user-defined material in LS-DYNA[6] and works for solid and shell elements. The model assumes isothermal and isotropic conditions and does not include any fracture criterion [1].

2.4 Polypropylene

The material treated in this thesis is a polypropylene used by Toyota. Toyota has given limited information about the chemical structure of the material. They have stated that it is injection moulded, is to be used for bumper applications, that the talc filling is 10-15% and that the elastomer content is unknown. On the basis of this limited knowledge, the considerations done in this section are quite general, and based on the chemical structure found in polypropylene versions with similar specifications.

Polypropylene (PP) is also known as polypropene. The latter name reveals that the base molecule, or monomer, of polypropylene is propene with chemical formula \( C_3H_6 \) (Figure 2.4). Propene is propane based and therefore part of the second most basic group of mers, only beaten by the ethane based mers.

![Figure 2.4: The monomer of polypropylene](image)

The methyl group seen in Figure 2.4 can be placed in different order around the backbone of the polymer chain. This is referred to as the tacticity of the polymer. PP is in most cases isotactic, meaning that the methyl group is placed on the same side of the backbone (see Figure 2.5a). Other possible tacticities for PP are syndiotactic (Figure 2.5b), where the side group alternates from side to side regularly, and atactic, where the side group is placed randomly (not shown). The tacticity is closely linked to the mechanical properties because it directly affects the polymer's ability to crystallize. In order to crystallize the material must be either iso- or syndiotactic. Since the PP in question is semi-crystalline, it is most probably isotactic.

Polypropylene is separated into three different categories: Homopolymer, consisting of only propylene based molecules. Random copolymer, a propylene-ethylene
2.4. POLYPROPYLENE

copolymer containing mainly propylene. And heterophasic copolymer, also referred to as high-impact PP or impact-modified PP. The latter is in most cases based on a homopolymer blended with an ethylene-propylene random copolymer (EPR) and block copolymers of different sequence lengths. For exterior use it is normal to add talc to stabilize the thermal and mechanical properties [3]. Given the information about the material (see the introduction to the section), it is reasonable to assume that the material at hand has a similar chemical structure as the heterophasic copolymer.

The material is injection moulded. The process is shown in Figure 2.6. First the plastic granulates are put into the cylinder. The granulates melt in the cylinder due to the heat produced by the heaters and the friction that occurs when the granulate is forced ahead by the screw. When a dose is ready for injection in the end of the cylinder, the melt is injected into the mould. The melt cools in the mould and is then ejected. Some of the advantages of injection moulding is low unit cost, high production rate and the ability to produce complex geometries. A main disadvantage is that the machine is very expensive [3].

The most critical part of the injection moulding process is called the holding pressure stage. In this stage the molten material is already injected into the mould. As it cools, the volume decreases (this especially applies to crystalline and semi-crystalline polymers). Since the mould is still pressurized the reduction is compensated by more melt from the cylinder, thus filling the mould completely. The applied holding pressure directly affects the properties of the finished product.
Figure 2.6: An injection moulding machine (Created by Brendan Rockey, University of Alberta Industrial Design, for a Wikipedia article about injection moulding)

The pressure must therefore be carefully adjusted so that the wanted properties are achieved and unwanted imperfections are avoided. In addition the material properties depend on how fast the melt is cooled. This again depends on the environment in the mould and the temperature of the melt that enters the mould.

The injection moulding process may leave some characteristics in the finished material. Whether this happens or not depends on the injected material, the mould geometry and the mould environment (as explained in the previous paragraph). The material at and closest to the surface is often subjected to very high shear rates and rapid cooling. This leads to a highly oriented and low-crystallinity microstructure which can be observed as a "skin" covering the material. This skin-like structure is easily observed on the PP treated in this thesis (see Figure 3.4). The shear effects that occur during the injection process also lead to an orientation of the material in the flow direction, i.e. the material has an anisotropic structure and will have a direction dependent load response [3].
Chapter 3

Material tests

A total of 17 calibration tests were carried out. 14 of them were tension tests and 3 were in compression. The results of these tests are used to find many central material parameters needed for the calibration such as Young’s modulus, the yield stress in tension and compression and the strain rate dependency. The machine used was a Dartec M1000 RK with a 20 kN load cell linked to an Instron controller. A camera was rigged in order to be able to use digital image correlation (DIC) in the post-processing. The respective instrument rigs will be commented in more detail in later sections. All tests were done at room temperature, which was somewhere between \(22 - 25^\circ C\).

The specimens were machined from two different rectangular plates into the geometry shown in Figure 3.2a. Of the 14 tension tests, 3 were taken from the transversal direction of the second plate. The rest were machined from the longitudinal direction and from both plates. Since the plates were only about 3.1 millimetres thick they were used with their original thickness. Before a specimen was cleared for testing the cross-section was measured on three different places along the gauge-length (the initial dimensions for each test are found in Appendix A) and then painted as shown in Figure 3.1. In order to eliminate any irregular specimens and minimize the scatter, each test was run twice. This meant that if two tests at the same configuration gave similar results, the results were declared valid. The tests were deformation-controlled, i.e. the test machine was set to a certain deformation speed, measured in mm/s. This speed had to be calculated so that the wanted strain rate was achieved in the specimen. The strain rates used in the calibration tests are \(\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}\), \(\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}\) and \(\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}\).

The test specimens had to be pre-treated in a special way. A prepared and tested specimen can be seen in Figure 3.1. The speckled pattern consists of an undercoat.
of white spray paint randomly speckled with black spray paint. To ensure that the paint would not become dry and crack up during the test, the paint was applied shortly before testing. The camera rig then photographed the specimens at a rate adapted to the strain rate. The rates used were 0.1 Hz for the slowest tests, 1 Hz for the middle rate and 10 Hz for the highest rate. The pattern created by the two layers of paint could then be read by an image correlation program, in this case a program named 7D. The program puts the pictures in succession, sorted by name. Then the user places a mesh on the images to mark the area from where information should be gathered. The program then uses the speckled pattern to correlate the physical deformation with the deformation shown in the pictures. By treating these correlated data the program can produce matrices containing different types of deformation data, e.g. deformations in x- or y-direction or maximum and minimum logarithmic strain. The latter matrices combined with the force data from the Instron controller were used to create the stress-strain diagrams shown in this chapter. In order to use the matrices from 7D they had to exported. This can be done in either text (.txt) format or in a Matlab format named .mat. Since Matlab was used for post-processing, the logarithmic strain matrices were exported in .mat format. The .mat-file contains a matrix for every picture taken by the camera. The dimension of the matrix is defined by the mesh that was placed by the user prior to the correlation process. In the meshes used in this chapter, the columns are placed along the transversal direction of the specimen and the rows along the longitudinal, i.e. a cross-section along the longitudinal axis of the specimen is described by a row. The strain was found to be constant over the cross-section. The mean strain value in each row can thus be used. Another reason for using mean values is that numerical noise that occur in the correlation process is reduced. By finding the row with the highest mean strain value, the cross-section of maximal deformation was located. The mean value of the longitudinal and transversal strain in this row was then gathered from each matrix. The final result being two vectors, one for the longitudinal strain and one for the transversal strain, showing the increase of strain through the deformation history (the MatLab scripts are found in Appendix B).
3.1 UNIAXIAL TENSION TESTS

The uniaxial tension tests were the first ones to be executed. The specimen geometry is shown in detail in Figure 3.2a. After the specimens had been prepared as described earlier they were placed in the rig as shown in Figure 3.2b. Because the strain rate dependency of polymers is much greater than for most metals the tests were run at three different strain rates, ranging from $\dot{\varepsilon} = 10^{-3} \text{s}^{-1}$ to $\dot{\varepsilon} = 10^{-1} \text{s}^{-1}$. The tests are labelled on the form "Tension (T)/Compression (C)""Test number" - "Direction of test""Plate number", i.e. the second tension test taken from the longitudinal direction of plate one is named "T2-L1".

The stress-strain relations shown in this section consist of true stress and logarithmic strain. To calculate the true stress the strain in all three material directions are needed. Earlier studies ([5], [7]) have, however, shown that the transversal strains are isotropic, i.e. if $\varepsilon_{\text{longitudinal}} = \varepsilon_x$ then $\varepsilon_y = \varepsilon_z$. This is also assumed in this thesis which yields the following expression for the stress:
\[ \sigma_{\text{True}} = \frac{F}{A} = \frac{F}{A_0 \cdot \exp(2\varepsilon_2)} \]  \hspace{1cm} (3.1)

where F is the force logged by the Instron controller, \( A_0 \) is the cross-sectional area at \( t=0 \) and \( \varepsilon_2 \) the transversal strain. As explained in the opening paragraphs, the strain was read from the cross-section of maximal strain, the stress is thus calculated in this cross-section.

### 3.1.1 Low strain rate, \( \dot{\varepsilon} = 10^{-3} \text{ s}^{-1} \)

With the gauge length of 33 mm the deformation speed was 0.033 mm/s. This is close to a quasi static loading, and the tests had a duration of about 45 minutes. This strain rate is chosen as the reference strain rate for the calibration.

![Figure 3.3: Test results, \( \dot{\varepsilon} = 10^{-3} \text{ s}^{-1} \)](image)

Figure 3.3 shows the results of the three tests made at this strain rate. The test T1-L1 failed the validity check and was rejected. Figure 3.3a shows that the polypropylene has a distinct peak load. Note also the low load capacity and high ductility compared to most other engineering materials.

Now looking at the stress-strain diagram in Figure 3.3b. It is seen that the material slightly softens before hardening is initiated. There is also a small difference in
the plastic response of the two tests. Test T3-L1 softens a bit more gradually than T2-L1 and also hardens somewhat less. Another difference between the two is that one of them developed a fracture and the other did not (T2-L1, see Figure 3.1). Exactly why this was the case is hard to point out exactly, but may be linked to the dispersion of imperfections in the specimens.

![Figure 3.4: The external layer after fracture](image)

In the fracture zone of test 3 (similar to the one in Figure 3.4) the external layer of the material is easily observed. It seems more elastic than the core material and deforms with the material until the core has completely fractured. Then it quickly breaks and folds, exposing the plastically deformed outer layer. This effect can also be observed at tests done at higher strain rates, but is most evident in these tests.

All in all the first test series did not bring many surprises. The validity check proved necessary through the rejection of test T1-L1. All the logging equipment also worked as expected, something it also did during the rest of the testing.

### 3.1.2 Average strain rate, $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$

These tests provide the first possibility to observe the strain rate dependency. Since these tests have a much shorter duration than the previous, this is the strain rate used in the majority of the tests. In this series specimens from both plates and from both the longitudinal and the transverse direction are tested.

In Figure 3.5 a selection of diagrams can be seen. The test T8-L1 is similar to the ones tested at $\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}$, only now with $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$. T7-L2 is oriented in the same direction as T8-L1, but is taken from plate two. The tests at this strain rate show a similar response as with $\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}$ (the response curves of all the tests are found in Appendix A). They have a distinct yield point and after some plastic
deformation they start to harden. The expected increase in Young’s modulus and the yield stress is observed. The last two tests, T9-T2 and T10-T2, are taken from the transverse direction of plate 2. As the figure shows, the results are quite dependent on the location. The yield stress and the peak load are 10-15% lower for the specimen from the transverse direction. The hardening also differs from the longitudinal specimens by being more evident in the transverse specimens. The directional dependent response is not unusual in injection moulded polymers and is most probably a result of the shear effects that occurred during the moulding (see [3] for further details). The non-uniform material structure is also observed in the somewhat special fracture surfaces and in the fact that some specimens break and others do not. Anisotropy and fracture mechanics are, however, not implemented in the material model and will not be represented in the numeric analyses.

Figure 3.5b shows the strain rate dependency. To better illustrate the phenomenon this plot also includes a curve from the tests where $\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}$. As the curve shows, the peak load increases with the strain rate. It is also seen that the increase from $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$ to $\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}$ is larger than the increase from $\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}$ to $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$. A similar behaviour is observed for the yield stress. The increase in load capacity with strain rate is a result of the non-Newtonian nature of polymers. This is included in the numerical model, and the implementation will be explained in Chapter 4.
3.1.3 High strain rate, $\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}$

At this strain rate the material starts to behave quite differently from the earlier tests. The peak load and yield stress increases and the total deformation is reduced. The fracture surface becomes more "cut-off", i.e. the outer layer is not stretched as much before fracture and becomes less visible. At this speed all tests end with fracture. Heat effects can also be of a magnitude worth considering, but these are not part of the material model and thus not reviewed in any extent in this thesis.

![Figure 3.6: Results from tests done with $\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}$](image)

The graphs in Figure 3.6 show some results from the testing. Two transversal specimens were tested also at this speed, the figures repeating the point of anisotropy. The initial response resembles that of the other strain rates, but after the yield point there are some differences. These tests do not harden as much as the others and they also fracture faster. These effects are most probably the result of the mentioned heat effects. The transverse specimens show a bit different behaviour. They have lower load levels and are able to deform more before fracturing. The reason for this is probably the lowered load level, which makes the heat effects less significant, thus allowing more deformation.
3.2 Uniaxial compression tests

The test rig used for the compression tests is shown in Figure 3.7a. The compression test specimens were only 3.2 millimetres thick and 6 millimetres in diameter and were machined using a belt puncher. Note that these were measured to 3.2 millimetre thickness instead of 3.1 millimetre even though they were taken from the same plate as the tension specimens. The exact reason for this is not known, it is probably a wrong measurement or some swelling mechanism caused by the machining with the belt puncher. The speckled pattern was applied onto the specimens and the camera was adapted to the small dimensions. To minimize the friction between the machine and the specimen, a 0.08 millimetre thick greased tape was applied (clearly visible at the contact surfaces shown in Figure 3.7a). Force was then applied until 1.5 millimetre (approximately 50% of the thickness) displacement was reached.

![Figure 3.7: Compression test set-up and specimen geometry](image)

(a) The test rig prepared for compression testing  
(b) The compression test specimen

In the post-processing it became obvious that the specimen was too small to be processed with 7D. This made it a bit more challenging to gather the needed information. Since the vertical displacement was logged by the Instron controller, the challenge consisted of finding the horizontal displacement, i.e. the elongation of the diameter. This displacement has to be known in order to calculate the area
3.2. UNIAXIAL COMPRESSION TESTS

of the deformed cross-section which again is needed to calculate the true strain. A Matlab script became the solution. The script read all the images taken by the camera, turned them into large matrices of ones and zeroes and then found the first and the last "one" in a row located at some cross-section along the thickness direction of the specimen (the script can be found in Appendix B). Since the greased tape removed much of the friction, the deformation was quite uniform and the deformation could be taken from the cross-section located at the mid-point of the specimen.

With all the needed deformation data at hand, the stress could be calculated quite easily:

\[ \sigma_{\text{True}} = \frac{F}{A} = \frac{F}{\left(\frac{\pi}{4}\right)d^2} \]  

(3.2)

Figure 3.8: The results of the compression tests

(a) Force displacement curves for the compression tests
(b) Force-displacement curves for tension and compression, \( \dot{\varepsilon} = 10^{-3} \text{ s}^{-1} \)

Figure 3.8: The results of the compression tests

Figure 3.8a shows the force-displacement diagrams of the three compression tests. It is seen that the strain rate dependency also applies to compression loading and that, unlike the tension tests, no unloading occurs. The main reason for doing the compression tests is to find the ratio between the yield stress in tension and compression (\( \alpha \)). This can be found from Figure 3.8b, which shows the stress-strain curve for both a tension and a compression specimen. The reason for the higher yield stress is the polymer chains reaction to compressive loading.
CHAPTER 3. MATERIAL TESTS

3.3 Summary

The test procedure described in this chapter provides the basis needed to fully calibrate the material model. The observations can be summed up in the following points:

• The results were not scattered. Only one of 17 tests was rejected.
• The material was found to be anisotropic
• The strain-rate dependency is obvious, being relatively near a log-linear pattern
• The yield stress in tension and compression are quite alike
• Softening is observed in tension but not in compression
• Hardening is observed both in tension and compression

In Chapter 4 it is shown how these parameters are analysed to define the different parameters of the material model.
Chapter 4

Calibration

This chapter explains and exemplifies the calibration of the material model. This process consists of identifying 11 non-zero coefficients that describe different parts of the material behaviour. The chapter starts with a review of the constitutive model. Then it is shown how the basic parameters, like stress-strain diagrams, yield stresses and forces are found from the results of Chapter 3. The next section will then show how to use this information to find the numbers, ratios and relations (the 11 coefficients) that can be understood by the LS-DYNA solver. The chapter is concluded with simulations of the material tests from Chapter 3, applying a calibrated version of the topical material model.

Figure 4.1: The constitutive model and the resulting stress-strain diagram (after [2] and [8])
CHAPTER 4. CALIBRATION

4.1 The constitutive model

Figure 4.1 (similar to Figure 2.3) shows an overview of the material model. In the following sections the derivation of the equations belonging to the different elements of the rheological model is shown. The theory is based on the following references [1], [9] and [2]. These documents and publications are produced by the team that created the material model and contains the theory behind it. This section contains only a representation of their work.

Polymers are both viscoelastic and viscoplastic. The material model, however, represents only the viscoplastic behaviour. This is because the model is designed for structural impact and crashworthiness simulations [9]. The material model also assumes constant temperature and an isotropic material structure [1].

The model is split into two main parts. This principle was introduced by Haward and Thackray in 1968 [4] and makes up the basis for most polymer material models today. The two parts describe; (i) the initial energy-elasticity followed by plastic flow and (ii) evolving entropy elasticity with increasing deformation. As mentioned in Section 2.3, these two parts have been named Part A and Part B in the material model. They have the same deformation gradient \( F = F_A = F_B \). This can be related to Figure 4.1a by imagining that the horizontal lines linking the Part A and Part B element will remain horizontal when the material deforms, thus deforming both parts equally. The change in volume is expressed by the Jacobian determinant of the deformation gradient. This means that Part A and Part B experience the same change of volume, since:

\[
J = \det F = J_A = J_B
\]  

(4.1)

The resistance contributions from the two parts can be summed. The Cauchy stress tensor is thus:

\[
\sigma = \sigma_A + \sigma_B
\]  

(4.2)

4.1.1 Part A - Intermolecular resistance

Part A describes the intermolecular resistance using an elastic spring, a friction element and a viscous damper (see Figure 4.1a). The deformation is described by the gradient \( F_A \) which is decomposed into an elastic and a plastic part, \( F_A = \ldots \)
4.1. THE CONSTITUTIVE MODEL

$F_A \cdot F_A^p$. This also applies to the Jacobian of the deformation gradient, and thus to the volume change.

The spring element is modelled as a Neo-Hookean spring. The spring is described by the well known elastic parameters Young’s modulus ($E_0$) and Poisson’s ratio ($\nu$). The Neo-Hookean formulation allows for large elastic deformations and yields the Kirchhoff stress tensor;

$$\tau_A = \lambda_0 \ln J_A^e I + \mu_0 (B_A^e - I)$$

where $\lambda_0$ and $\mu_0$ are the Lamé constants of linearised theory. These are linked to $E_0$ and $\nu$ through;

$$\mu_0 = G = \frac{E_0}{2(1 + \nu)}$$

$$\lambda_0 = \frac{E_0 \nu}{(1 + \nu)(1 - 2\nu)}$$

The viscoplastic behaviour of the material model is represented by the friction element and the viscous damper. The friction element is defined by the parameters in Table 4.1.

| $\sigma_s$ | The saturation stress | $\beta$ | The dilatation parameter |
| $\sigma_T$ | The yield stress in uniaxial tension | $H$ | Ramping parameter |
| $\alpha$ | The yield stress ratio |

Table 4.1: The parameters of the friction element

The parameters are implemented in the following yield criterion;

$$f_a = \sigma_A - \sigma_T - R$$

where $\sigma_T$ is the yield stress in uniaxial tension and R is an isotropic variable representing hardening or softening. If the material at hand had been structural steel or aluminium, the equivalent stress ($\bar{\sigma}_A$) would most probably have been calculated using the von Mises or Tresca criterion. This yield criterion is mean stress independent and assumes equal yield stress in tension and compression, properties which do not fit polymers very well. Another yield criterion is therefore assumed. It was first formulated by Raghava et al. [10]
(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 + 2(|\sigma_C| - |\sigma_T|)(\sigma_1 + \sigma_2 + \sigma_3) = 2|\sigma_C \sigma_T| \quad (4.7)

where \sigma_1, \sigma_2 and \sigma_3 are the principal stresses, \sigma_C and \sigma_T the yield stress in compression and tension, respectively. By introducing the first principal invariant \(I_1\) and the deviatoric stress invariant \(J_2\):

\[
I_1 = \sigma_1 + \sigma_2 + \sigma_3 
\]

\[
J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2] 
\]

plus the parameter \(\alpha\):

\[
\alpha = |\frac{\sigma_C}{\sigma_T}| \geq 0 
\]

the Raghava criterion can be formulated as [5]:

\[
f(I_1, J_2) = (\alpha - 1)\sigma_T I_1 + 3J_2 - \alpha \sigma_T^2 = 0 
\]

The uniaxial yield stress can be made separate in Equation (4.11) yielding:

\[
f(I_1, J_2) = \bar{\sigma} - \sigma_T 
\]

\(\bar{\sigma}\) is then the Raghava equivalent stress:

\[
\bar{\sigma} = \frac{(\alpha - 1)I_1 + \sqrt{(\alpha - 1)^2 I_1^2 + 12\alpha J_2}}{2\alpha} 
\]

If \(\sigma_C = \sigma_T\), i.e. \(\alpha = 1\), the pressure dependence is removed from Equation (4.13) and it becomes equal to the von Mises criterion:

\[
\alpha = 1 \Rightarrow \bar{\sigma} = \sqrt{3J_2} 
\]

Now the last term, namely \(R\), will be explained. This term is added to include hardening or softening after yield (illustrated as dotted blue lines in Figure 4.1b). \(R\) is a function of the accumulated plastic strain \(\bar{\varepsilon}_p\) and takes the form;
4.1. THE CONSTITUTIVE MODEL

\[ R(\varepsilon_A^p) = (\sigma_s - \sigma_T)[1 - \exp(-H\varepsilon_A^p)] \] \hfill (4.15)

where \( \sigma_s \) is the saturation stress, i.e. the stress at the end of the Part A contribution, and \( H \) is a ramping parameter used to optimize the fit of the stress-strain curve between the yield stress and the saturation stress. Introducing (4.15) into (4.6) yields:

\[ f_a = \bar{\sigma}_A - \sigma_T - (\sigma_s - \sigma_T)[1 - \exp(-H\varepsilon_A^p)] \] \hfill (4.16)

When put in the yield criterion (Equation (4.16)) and compared to Figure 4.1b it is seen that \( R \) represents hardening when positive, i.e. \( \sigma_s > \sigma_T \), and softening when negative (\( \sigma_s < \sigma_T \)).

The viscous damper in Figure 4.1a is described by the parameters \( C \) and \( \dot{\varepsilon}_{0A} \). \( C \) is found by using the relation between yield stresses at different strain rates in tension, as will be described in Section 4.2. \( \dot{\varepsilon}_{0A} \) is the reference strain rate chosen by the user, typically \( 10^{-4} - 10^{-3} \text{ s}^{-1} \). The next paragraphs show where these and the \( \beta \) parameter from the friction element belong in the material model.

Experiments have shown that an associated flow rule, i.e. a flow rule associated with the yield function (see Equation (4.17)), predicts too large volumetric plastic strains.

\[ \dot{\varepsilon}_A^p = \dot{\varepsilon}_A^p \frac{\partial f}{\partial \sigma_A} \] \hfill (4.17)

Associated flow rule

\[ \dot{\varepsilon}_A^p = \dot{\varepsilon}_A^p \frac{\partial g}{\partial \sigma_A} \] \hfill (4.18)

Non-associated flow rule

This is solved by introducing a new, non-associated relation for the plastic potential. In this case a Raghava-like plastic potential function \( g_A \):

\[ g_A = \frac{(\beta - 1)I_{1A} + \sqrt{(\beta - 1)^2I_{1A}^2 + 12\beta J_{2A}}}{2\beta} \geq 0 \] \hfill (4.19)

where the \( \beta \) parameter is introduced to control the volumetric plastic strains (confront row 7 of Table 2.1). The multiplier in the flow rule, \( \dot{\varepsilon}_A^p \), is strain rate dependent and described by the constitutive relation (see [9] for more details):
\[
\dot{\varepsilon}_A = \begin{cases} 
0 & \text{if } f_A \leq 0 \\
\dot{\varepsilon}_{0A} \exp \left[ \frac{1}{C} \left( \frac{\bar{\sigma}_A}{\sigma_T + R} \right) - 1 \right] - 1 & \text{if } f_A > 0 
\end{cases}
\] (4.20)

Note the location of the parameters C and \( \dot{\varepsilon}_{0A} \) in Equation (4.20).

### 4.1.2 Part B - Network resistance

Part B describes the network resistance with a hyperelastic spring. The spring needs the two input parameters \( C_R \) and \( \bar{\lambda}_L \). \( C_R \) represents the initial elastic modulus of Part B and \( \bar{\lambda}_L \) represents the locking stretch which is related to the maximum attainable deformation of the material [9]. They are both identified in the Part B contribution of the Cauchy stress tensor:

\[
\sigma_B = \frac{1}{J_B} \frac{C_R \bar{\lambda}_L}{3} L^{-1} \left( \frac{\bar{\lambda}}{\bar{\lambda}_L} \right) (B_B^* - \bar{\lambda}^2 I)
\] (4.21)

In Equation (4.21) \( J_B \) represents the Jacobian = \( \det (F) \) and \( L^{-1}(\phi) \) the inverse function of the Langevin function \( L(x) \equiv \coth x - \frac{1}{x} \). The kinematic variables \( \bar{\lambda} \) and \( B_B^* \) are defined as:

\[
\bar{\lambda} = \sqrt{\frac{1}{3} \text{tr}(B_B^*)}, \quad B_B^* = F_B^* \cdot (F_B^*)^T
\] (4.22)

They represent the distortional stretch and the distortional left Cauchy-Green deformation tensor, respectively. \( F_B^* = J_B^{-1/3} F_B \) is the distortional part of \( F_B \).

### 4.2 The calibration procedure

In this section it is shown how the data from the material tests are utilized to identify the parameters defining the material model. The parameters are determined from three categories of material data; the transverse deformation, the yield stresses and the stress-strain diagram of the baseline test. The representation of the procedure is mainly based on an unpublished guideline document for the material model written by Clausen et al. [1].

The first point of the calibration procedure is to decide which of the tests that should be the reference test. Based on the results found in Chapter 3, T3-L1
4.2. THE CALIBRATION PROCEDURE

was chosen as the reference test. In this case both the two slowest tests were good candidates, and the results of both tests were put through the calibration procedure. The calibration of T3-L1, however, gave the most optimal response curve and was thus chosen over the other option.

4.2.1 Parameters based on transverse deformations

The parameters identified from the transverse strains are the dilatation parameter $\beta$ and Poisson’s ratio $\nu_0$. Both are based on the reference test T3-L1. The strain relations are seen in Figure 4.2.

![Figure 4.2: Transverse vs. longitudinal strain and contraction ratio vs. longitudinal strain for the reference test $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$](image)

The left y-axis and the x-axis show the transverse strain, $\varepsilon_2$, as a function of the longitudinal strain, $\varepsilon_1$. Possion’s ratio is set to the slope of this curve for $\varepsilon_1 < \sim 0.1$. The value found is $\nu_0 = 0.24$, which is a rather low value compared to 0.4 in HDPE and 0.3 in PVC [5].

The contraction ratio $\rho$ indicates in which degree the volume changes. It is inversely proportional to the change of volume, i.e. a small $\rho$ means a large change in volume. The range of the ratio is $0 < \rho \leq 0.5$, where $\rho = 0.5$ corresponds to no change of volume. From Figure 4.2 it is seen that the topical contraction ratio
CHAPTER 4. CALIBRATION

is about 0.165, which is a low value. This means that the material dilates (the volume expands) considerably.

In order to derive the $\beta$ parameter the gradient of the plastic potential function have to be introduced. Equation (4.23a) is similar to the plastic potential function in Equation (4.18). The gradient of this function is shown in Equation (4.23b) [1].

$$\bar{\dot{L}}_A = \dot{\varepsilon}_A^p \frac{\partial g_A}{\partial \Sigma_A} \quad (4.23a)$$

$$\frac{\partial g_A}{\partial \Sigma_A} = f_1 \frac{\partial I_{1A}}{\partial \Sigma_A} + f_2 \frac{\partial J_{2A}}{\partial \Sigma_A} = f_1 \mathbb{I} + f_2 \partial \Sigma_{A}^{\text{dev}} \quad (4.23b)$$

where the functions $f_1$ and $f_2$ read

$$f_1 = \frac{\partial g_A}{\partial I_{1A}} = \frac{\beta - 1}{2\beta} + \frac{(\beta - 1)^2 I_{1A}}{2\beta \sqrt{(\beta - 1)^2 I_{1A}^2 + 12\beta J_{2A}}} \quad (4.24a)$$

$$f_2 = \frac{\partial g_A}{\partial I_{2A}} = \frac{3}{\sqrt{(\beta - 1)^2 I_{1A}^2 + 12\beta J_{2A}}} \quad (4.24b)$$

The dilatation yields that both $f_1$ and $f_2$ are non-zero. Since the stress state in the material tests is uniaxial, the first and second invariant of the Cauchy stress tensor read $I_{1A} = \sigma$ and $J_{2A} = (1/3)\sigma^2$. When these invariants are inserted into the non-zero $f_1$ and $f_2$ the result is

$$f_1 = \frac{\beta - 1}{2\beta} + \frac{(\beta - 1)^2 \sigma}{2\beta \sqrt{(\beta - 1)^2 \sigma^2 + 12\beta \sigma^2}} = \frac{\beta - 1}{\beta + 1} \quad (4.25a)$$

$$f_2 = \frac{3}{\sqrt{(\beta - 1)^2 \sigma^2 + 12\beta \sigma^2}} = \frac{3}{\beta + 1} \quad (4.25b)$$

The last step is now to formulate the rate of deformation for plastic flow, $\mathbf{D}_A^p$. It is defined by the current stress and strain state of the material. The current stress state is as mentioned uniaxial stress, the current strain state is defined by the assumptions of isotropic transverse strains, a constant $\rho$ value and proportional deformation.
4.2. THE CALIBRATION PROCEDURE

\[ D^p_A = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 0 \end{pmatrix} = \varepsilon^p_1 \begin{pmatrix} 1 & 0 & 0 \\ 0 & -\rho & 0 \\ 0 & 0 & -\rho \end{pmatrix} = \varepsilon^p_1 \frac{\partial g_A}{\partial \Sigma_A} \] (4.26)

The latter part of Equation (4.26) shows that this rate is equal to the rate of deformation shown in Equation (4.23a). By assuming \( \varepsilon^p_1 = \bar{\varepsilon}_p \), i.e. shortening the equivalent strain rate, and inserting the Equations in (4.25) into Equation (4.26) the following equations are derived:

\[
\begin{pmatrix} 1 & 0 & 0 \\ 0 & -\rho & 0 \\ 0 & 0 & -\rho \end{pmatrix} = \beta \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix} + \frac{3}{\beta + 1} \begin{pmatrix} (2/3)\sigma & 0 & 0 \\ 0 & -(1/3)\sigma & 0 \\ 0 & 0 & -(1/3)\sigma \end{pmatrix}
\] (4.27)

Solving the system of equations yields

\[ \beta = \frac{2 - \rho}{1 + \rho} \] (4.28)

\( \rho \) is set to 0.17. The \( \rho \) value is rounded up because the other possible reference test (T2-L1) had slightly higher \( \rho \) than the chosen reference test. The chosen value for \( \rho \) gives \( \beta = 1.56 \).

4.2.2 Parameters based on the yield stresses

In contrast to most metals, the yield stress for polymers is different in tension and compression, and it increases considerably with increasing strain rate. This behaviour is described using the ratios and relations explained in this section.

In order to use the relation between the yield stresses they have to be identified using the same principle. In this thesis the yield stresses were identified using the Considère criterion:

\[ \frac{d\sigma_t}{d\varepsilon_t} = \sigma_t \] (4.29)

where \( \sigma_t \) and \( \varepsilon_t \) are the true stress and the longitudinal logarithmic strain, respectively. Figure 4.3a shows how this relation appears graphically. The yield stress is defined as the point where the two curves intersect. The method was implemented
The yield stress calculation of test T3-L1

The relation between the yield stresses and the strain rate. (+) are tension tests and (×) are compression tests

Figure 4.3: The method used to identify the yield stresses and the yield stresses in relation to the strain rate in a Matlab code which calculated the yield stress for all the material tests (the script can be found in Appendix B.3).

The first and easiest parameter to identify from the yield stresses is $\alpha$. This is the ratio between the yield stress in compression and tension, i.e. $\alpha = \sigma_{yC}/\sigma_{yT}$.

As seen in Figure 4.3b is $\alpha$ not independent of the strain rate. The calibration procedure states that the two yield stresses are to be taken from a test done at the reference strain rate $\dot{\varepsilon}_0A = 10^{-3}$ s$^{-1}$. Even though the yield stress at the intermediate strain rate is much higher, it is the best solution to use the result of the reference test. The reason is that the test from the intermediate strain rate is not verified by a second test, and thus less valid. For this particular PP the $\alpha$ value is found to be $\alpha = 1.062$, which indicates that the yield stress is only slightly higher in compression than in tension.

The next parameters are central in describing the strain rate dependency. These are the yield stress in tension, $\sigma_T$, the reference strain rate, $\dot{\varepsilon}_0A$, and a slope ratio $C$. They are connected as shown in Equation (4.30):

$$\bar{\sigma}_A = \sigma_T \left(1 + C \ln \left(\frac{\dot{\varepsilon}_A}{\dot{\varepsilon}_0A} + 1\right)\right) \quad (4.30)$$
\( \varepsilon_0^A \) is the first to be identified. This is the reference strain rate and is defined as [1]; \( \ldots \) the local logarithmic strain rate \( \dot{\varepsilon} \) in the section experiencing the onset of necking and thereby the initial yielding.

For this particular PP this corresponds to a strain rate of \( \varepsilon_0^A = 10^{-3} \text{ s}^{-1} \). This parameter is incorporated into a variable called the relative strain rate. This is the ratio between the Part A plastic strain rate and the reference strain rate:

\[
\dot{\varepsilon}_{\text{rel}} = \frac{\dot{\varepsilon}_p^A}{\varepsilon_0^A} \quad (4.31)
\]

Figure 4.4: The yield stresses and the linear fit used for extrapolation

Now moving on to \( \sigma_T \) and C. A problem with \( \bar{\sigma}_A \) in Equation (4.30), i.e. the Raghava equivalent stress, is that it describes a point at the stress curve in which both Part A and Part B contribute. This means that if the yield stress from the reference test (\( \sigma_T \)) is applied as shown in Equation (4.30), the Part B contribution to the yield stress is left out. The result being that when the model sums the Part A and Part B contributions to the stress it ends up with \( \bar{\sigma}_A > \sigma_{\text{yield}} \). To overcome this transitional problem, a static yield stress \( \sigma_0 \) is introduced. Figure 4.4 illustrates the concept. The figure shows the yield stress at the three strain rates
applied in the material testing (see Chapter 3) as function of $ln(\dot{\varepsilon}_{rel} + 1)$. The dotted line represents a linear regression (Equation (4.32)) between these six points. This have to be done in order to identify both the newly introduced $\sigma_0$ and $C$.

$$ y = ax + b $$

$\sigma_0$ is shown as the X on the y-axis and thus corresponds to “b” in Equation (4.32). Since the x-axis shows a strain rate relation this placement indicates that $\sigma_0$ is the yield stress at a fictitious static or “zero” strain rate:

$$ \dot{\varepsilon}_{rel} = 0 \Rightarrow ln(\dot{\varepsilon}_{rel} + 1) = ln(1) = 0 \quad (4.33) $$

By doing this the Part B contribution is removed and the correct Part A equivalent stress can be formulated as:

$$ \bar{\sigma}_A = \sigma_0 + \sigma_0 C \cdot ln(\dot{\varepsilon}_{rel} + 1) \quad (4.34) $$

The C parameter can now be derived:

$$ C = \frac{\bar{\sigma}_A - \sigma_0}{\sigma_0 ln(\dot{\varepsilon}_{rel} + 1)} = \frac{a}{\sigma_0} \quad (4.35) $$

where “a” corresponds to a in Equation (4.32), i.e. representing the gradient of the linear regression curve. For the PP in this thesis $\sigma_0$ and $a$ are found to be $\sigma_0 = 19.4$ MPa (derived in the next section) and $a = 1.1$ which gives $C = 0.057$.

### 4.2.3 Parameters found from the stress-strain curve of the reference test

This section will identify the remaining Part A parameters $E_0$, $\sigma_T$, $\sigma_s$ and H and the two Part B parameters $C_R$ and $\lambda_L$. They are all found from a stress-strain diagram based on the chosen reference test, T3-L1.

Young’s modulus increases with increasing strain rate, i.e. polypropylene is viscoelastic. This is, as mentioned earlier, not incorporated in the material model and Young’s modulus, $E_0$, is to be found from the elastic response of the reference test. It is, however, well known that data from Digital Image Correlation (DIC) are subjected to noise for small strain values, thus making an Young’s modulus
4.2. THE CALIBRATION PROCEDURE

provided by the material manufacturer preferable. The manufacturer has, however, not provided any value for Young’s modulus and the modulus used in this thesis may therefore be a major source of error in the representation of the elastic behaviour. That said, it should be noted that the elastic behaviour has not been the primary focus in developing this material model and the elastic response makes up only a small part of the total response. Given the above conditions, $E_0$ has been identified using the stress-strain curves from the DIC. In the initial phases $E_0$ was found by fitting the DIC based stress-strain curve of the reference test exactly, resulting in $E_0 = 250$ MPa. This is a very low value and numerical simulations indicated that the correct value was more in the range of 400-500 MPa, which resulted in choosing $E_0 = 400$ MPa. This value also represents a mean of Young’s modulus over the three strain rates applied. This deviates a bit from the guidelines [1], but it has proved a reasonable estimation in the numerical simulations.

The parameters not yet derived depend on the dilatation and whether the material softens or hardens. The topical PP shows a significant dilation and a slight softening. The softening means that $R(\dot{\varepsilon}_p^A)$ in Equation (4.6) is negative and that the stress-strain curve has a local maximum quite early in the response. It is assumed that the material behaviour is elastic up to this maximum point, i.e. the yield stress $\sigma_T$ is equal to the stress value at this point. This assumption is more correct the more distinct the softening is, i.e. a material experiencing a softening of low gradient may have a more gradual transition to plastic behaviour than assumed. The assumption, however, does not exempt $\sigma_T$ from being adjusted down to $\sigma_0$ as explained earlier, rendering:

$$\sigma_T = \sigma_0 = 19.4 \text{ MPa} \quad (4.36)$$

Now to the saturation stress, $\sigma_s$. This parameter describes the stress in the material at the end of the transition described by the R parameter. For the PP this means that $\sigma_s$ is the stress at the point where the softening ends, i.e. lower than $\sigma_T$. It has proved more challenging to find this stress value for materials that softens, than for the ones that do not. This is because this and the three other remaining parameters, $H$, $C_R$ and $\bar{\lambda}_L$, are not independent of each other. Making a direct addition of the individual curve fits inaccurate. The solution is to incorporate all four parameters into a function describing the entire stress-strain curve.

Some additional information is required in order to derive the function describing the complete stress-strain curve. The main objective is to describe the total stress through the contributions of Part A and Part B:
\[ \sigma_{\text{target}} = \sigma_A + \sigma_B = \sigma_{\text{total}} \]  \hspace{1cm} (4.37)

Equation (4.21) shows the stress contribution of Part B, \( \sigma_B \). The latter part of this equation is

\[ (B_B^* - \bar{\lambda}^2) \]  \hspace{1cm} (4.38)

where \( B_B^* \) is

\[ B_B^* = J^{-2/3} F_B (F_B)^T = J^{-2/3} \begin{pmatrix} \lambda_1^2 & 0 & 0 \\ 0 & \lambda_2^3 & 0 \\ 0 & 0 & \lambda_3^2 \end{pmatrix} \]  \hspace{1cm} (4.39)

and the parameter \( \bar{\lambda} \), also shown in Equation 4.22 on page 30, is equal to

\[ \bar{\lambda} = \sqrt{\frac{1}{3} \text{tr}(B_B^*)} = \sqrt{\frac{1}{3} J^{-2/3} (\lambda_1^2 + 2 \lambda_2^2)} \]  \hspace{1cm} (4.40)

where the volume change is \( J = J_B = \text{det}(F) = \lambda_1 \lambda_2 \lambda_3 \). By putting Equation (4.39) and (4.40) into Equation 4.21 on page 30 the Part B stress contribution is derived:

\[ \sigma_B = \begin{pmatrix} \sigma_{B1} & 0 & 0 \\ 0 & \sigma_{B2} & 0 \\ 0 & 0 & \sigma_{B3} \end{pmatrix} = J^{-5/3} \frac{C_R}{3} \frac{\bar{\lambda}}{\lambda} L^{-1} \left( \bar{\lambda} \right) \begin{pmatrix} \frac{2}{3} (\lambda_1^2 - \lambda_2^2) & 0 & 0 \\ 0 & \frac{1}{3} (\lambda_2^2 - \lambda_1^2) & 0 \\ 0 & 0 & \frac{1}{3} (\lambda_3^2 - \lambda_1^2) \end{pmatrix} \]  \hspace{1cm} (4.41)

From Equation (4.41) it is seen that \( \sigma_B \) represents a deviatoric stress state. This means that in order to make the total stress (cf. Equation (4.37)) define an uniaxial stress state the stress component \( \sigma_{A2} (= \sigma_{A3}) \) must fulfil:

\[ \sigma_{A2} = -\sigma_{B2} = \frac{1}{2} \sigma_{B1} \]  \hspace{1cm} (4.42)

Equation (4.41) shows that \( \sigma_{B1} \) contains the inverse of the Langevin function. The nature of this function makes it complicated to use in regression. This is solved by using a Padé approximation to \( L^{-1}(\phi) \) [1]. This yields the following expression for \( \sigma_{B1} \):
4.2. THE CALIBRATION PROCEDURE

\[ \sigma_{B1} = \frac{2}{9} C_R J^{-5/3} \frac{3 \lambda_L^2 - \bar{\lambda}^2}{\lambda_L^2 - \bar{\lambda}^2} (\lambda_1^2 - \lambda_2^2) \]  

(4.43)

The next step is to use the Raghava yield criterion to formulate an expression for the stress in Part A. Based on the above assumptions \( I_{1A} \) and \( J_{2A} \) read:

\[ \begin{align*}
I_{1A} &= (\sigma_{A1} + 2\sigma_{A2}) \\
J_{2A} &= \frac{1}{3}(\sigma_{A1} - \sigma_{A2})
\end{align*} \]

(4.44a)

(4.44b)

By introducing this into the Raghava yield criterion (cf. Equation (4.19)) and then applying some algebra, the stress contribution of Part A can be formulated as:

\[ \sigma_{A1} = \frac{1}{k_1} \sigma_{A,ua} + \frac{1}{2 k_1} \sigma_{B1} \]  

(4.45)

where \( k_1(\alpha) \) and \( k_2(\alpha) \) are constants derived through the algebra, extracting \( \alpha \) (the yield stress ratio) into constants. \( \sigma_{A,ua} \) in the above equation describes the uniaxial Part A stress. Since the majority of the response is plastic \( \sigma_{A,ua} \) is set equal to

\[ \sigma_{A,ua} = \sigma_T + (\sigma_s - \sigma_T)[1 - \exp(-H \bar{\varepsilon}_p)] \]  

(4.46)

Equation (4.46) and (4.43) are introduced into Equation (4.37) and the result is:

\[ \begin{align*}
\sigma_{target} &= \sigma_{A1} + \sigma_{B1} = \left( \frac{1}{k_1} \sigma_{A,ua} + \frac{1}{2 k_1} \sigma_{B1} \right) + \sigma_{B1} \\
\sigma_{target} &= \frac{1}{k_1} \left( \sigma_T + (\sigma_s - \sigma_T)[1 - \exp(-H \bar{\varepsilon}_p)] \right) + \left( \frac{1}{2 k_1} \right) \frac{2}{9} C_R J^{-5/3} \frac{3 \lambda_L^2 - \bar{\lambda}^2}{\lambda_L^2 - \bar{\lambda}^2} (\lambda_1^2 - \lambda_2^2)
\end{align*} \]

(4.47a)

(4.47b)

This equation has been implemented into a worksheet in Microsoft Excel by Clausen et al. The regression gave the following values; \( \sigma_s = 12.5 \) MPa, \( H = 2.19 \), \( C_R = 3.79 \) MPa and \( \bar{\lambda}_L = 50.43 \).

The parameters are summarized in Table 3.2. The fit of the calibrated stress-strain curve to the one of T3-L1 is shown in Figure 4.5.
Table 3.2: The eleven calibrated parameters (colour coded with Figure 2.3a on page 10)

<table>
<thead>
<tr>
<th></th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$E_0$</td>
<td>400 MPa</td>
</tr>
<tr>
<td>2</td>
<td>$\nu_0$</td>
<td>0.24</td>
</tr>
<tr>
<td>3</td>
<td>$\sigma_T$</td>
<td>19.4 MPa</td>
</tr>
<tr>
<td>4</td>
<td>$\sigma_s$</td>
<td>12.5 MPa</td>
</tr>
<tr>
<td>5</td>
<td>$H$</td>
<td>2.19</td>
</tr>
<tr>
<td>6</td>
<td>$\alpha$</td>
<td>1.062</td>
</tr>
<tr>
<td>7</td>
<td>$\beta$</td>
<td>1.56</td>
</tr>
<tr>
<td>8</td>
<td>$\dot{\varepsilon}_{DA}$</td>
<td>0.001 s$^{-1}$</td>
</tr>
<tr>
<td>9</td>
<td>$C$</td>
<td>0.05716</td>
</tr>
<tr>
<td>10</td>
<td>$C_R$</td>
<td>3.79 MPa</td>
</tr>
<tr>
<td>11</td>
<td>$\lambda_L$</td>
<td>50.43</td>
</tr>
</tbody>
</table>

Figure 4.5: Stress-strain curves for the reference test and the calibrated material model

4.3 Application of the calibrated material model

In this section the material model is applied in the finite element code LS-DYNA [6]. The uniaxial tension and uniaxial compression tests will be treated in this chapter. Each subsection starts with a short review of the modelling process. This is followed by a presentation of the results, split into a subsection for each strain rate. The results shown in these first subsections are all from simulations with a reference material model, i.e. a material model where the parameters are strictly based on the calibration derived in this chapter. At the end of the section the results are compared and discussed and some parameters are treated in a brief parameter study. The material cards and the input files can be found in Appendix C.

4.3.1 Uniaxial tension

The specimen was modelled both as full-size and as only the gauge length. In order to get the best representation of the strain rate, the full-size model was used. For the medium and low strain rate tests the full-sized model became very time consuming. A version of the full-size model with symmetry about the longitudinal axis was therefore used to reduce the time of the calculations. Since the specimen was only three millimetres thick, a thickness symmetry was considered unnecessary.
4.3. APPLICATION OF THE CALIBRATED MATERIAL MODEL

The applied geometries are shown in Figure 4.6. Fully integrated solid elements were used (element formulation 2 in LS-DYNA [6]). In the gauge length the elements measure approximately $1\text{mm} \times 1\text{mm} \times 1\text{mm}$, which correspond to 33 elements along the longitudinal axis of the gauge, 12 along the transversal and 3 in the thickness direction. Except from the simulation of the low-strain-rate test, this mesh has proved the most optimal with regard to accuracy and computational expense. For the lowest strain rate a finer mesh had to be applied in the gauge length to avoid numerical error. The refinement was done by doubling the number of elements in the longitudinal direction (see Figure 4.6c), thus making the elements more and more cubic as they deform.

The first five longitudinal elements at each end were defined in node sets representing the clamps of the test machine. The node set in one end was defined as fixed and the set in the other end was prescribed a constant deformation in mm/s.

Strains and stresses were collected from the elements defining the mid-point of the modelled specimen, i.e. in the mid-point of the gauge. This is also the section that experiences the initial yielding thus corresponding to the section used for collecting data from the material tests.

Mass scaling of $10^5$ was applied. The mass density of the material was found to
be approximately $\rho = 1000$ kg/m$^3$. With the applied units and the mass scaling of $10^5$ the effective weight in the analysis was $\rho = 1.00 \cdot 10^{-5}$ tonne/mm$^3$. As long as the problem is close to static the mass scaling can be applied without affecting the results in any extent. Other levels of mass scaling have been applied but they have been found less optimal than the one chosen.

Low strain rate, $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$

This is the reference strain rate. Three material tests were done at this speed, where of one was rejected. Of the two remaining, the test T3-L1 was chosen as the reference model and is thus the basis of the applied material model.

Figure 4.7: The contribution of the two parts separate and combined

Figure 4.7 shows the results of a finite element simulation where $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$. The results are split into Part A and Part B and compared with the same parts of the calibrated response curve, see Figure 4.5. In Figure 4.7a it is seen that Part A of the simulation and the calibration agree quite well. The opposite is seen in Figure 4.7b. The initial response is similar, but the deviation increases with increasing strain. The sum of the two, however, is not too bad. This indicates that part of the error in the Part B contribution is counterbalanced by Part A. This error allocation can indicate that the degree of agreement between the material tests and the simulations lies in how well Part B fits to the response of the topical material test.

Figure 4.8 shows the results of the simulation of the tests with the lowest strain rate. The force-displacement diagram in Figure 4.8a shows that the model matches quite well with the force-displacement curves of the material tests. But on the other hand shows Figure 4.8b that the model deviates quite a bit from the stress-strain response of the same tests. This is because the two diagrams display a
4.3. APPLICATION OF THE CALIBRATED MATERIAL MODEL

(a) Force-displacement diagram
(b) Stress-strain diagram

Figure 4.8: Diagrams comparing the results of the model and the tests

global response and a local response, respectively, and it appears that the global response is well represented by the model, but the local is not. This is also seen when comparing the deformed shape of the test specimens and the model. The necking is less evident in the specimens than in the model. The diffuse necking might indicate that the resistance mechanisms are a bit more complex than the ones represented by the model.

Intermediate strain rate, $\dot{\varepsilon} = 10^{-2}$ s$^{-1}$

At this strain rate the parameters describing the strain rate dependency are tested for the first time. It seems that the result of the response is satisfactory.

In contrast to the low-strain-rate modelling, the finite element model fits nearly perfect to the tests at this rate. The maximum force fits very well and the response continues to agree all the way until the force stabilizes. In the stress-strain diagram the assumed $E_0$ of 400 MPa seems to fit very well and so does the yield stress and the plastic material behaviour. It is challenging to find an exact reason for why this simulation agrees better with the test results than the low-strain-rate tests. One point contributing to the agreement is that this is the strain rate defining the mid-point of the linear strain rate regression line (cf. Figure 4.4), something which calls for a good estimate of the yield stress. The assumptions regarding the elastic behaviour also agree very well.
CHAPTER 4. CALIBRATION

(a) Force-displacement diagram  
(b) Stress-strain diagram

Figure 4.9: Diagrams comparing the model and the tests

Figure 4.10: The deformed finite element model applied a fringe plot showing the maximum principle stress ($\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$)

Figure 4.10 shows the deformed mesh applied a fringe plot of the maximum principle stress. The red colour indicates maximum values and blue minimum. In each end of the gauge (in the transition from orange to yellow) can the end points of the propagating neck be observed. It is seen that the entire neck is exposed to high stresses. The gradual transition from high to low stresses at the end of the gauges indicates that the transitions from the gauge to the clamping areas are well represented by the applied mesh.

High strain rate, $\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}$

The high-strain-rate tests were over in seconds. At strain rates this high there might be significant heat effects that are not represented by the numerical model. Figure 4.11 shows the response curves for the high-strain-rate tests. In Figure 4.11a
a deviation in both peak load and unloading is observed. The peak load of the finite element model is a bit low. The gradient of the unloading is also too low compared to the material tests. Another difference is that the material test fractured before a stable load level was reached (seen for the intermediate strain rate in Figure 4.9a). This is not captured in the finite element analysis because the material model does not contain any fracture criterion. Figure 4.11b shows the stress-strain diagram. For these high-strain-rate tests the stress-strain curve is closer than the force-displacement curve. But there are some differences around the yield point; first the yield stress is a little low, then there is a slight difference in the softening, where the model predicts a too low gradient. The assumed Young’s modulus, however, fits rather well, implying that the viscoelastic effects are more present when going from $\dot{\varepsilon} = 10^{-3} \text{s}^{-1}$ to $\dot{\varepsilon} = 10^{-2} \text{s}^{-1}$ than from $\dot{\varepsilon} = 10^{-2} \text{s}^{-1}$ to $\dot{\varepsilon} = 10^{-1} \text{s}^{-1}$. The flat plastic response shown by the material tests in Figure 4.11b indicates that there might be heat effects impeding the hardening of the material.

4.3.2 Uniaxial compression

Three material tests were carried out in compression. Of these were two made at the lowest strain rate and one at the intermediate. The main objective of the compression testing was to find the $\alpha$ parameter. The mechanisms that decide the response to compression loading have been subject to less research than the
tension response. To define and compare these mechanisms is outside the scope of this thesis and the compression tests have thus been given less attention than the tension tests.

Figure 4.12: The symmetric part used in the simulations. And the same part mirrored to show the full geometry (no analysis was carried out using the geometry shown in Figure 4.12c)

The full-sized specimen had the dimensions; diameter, $D = 6$ mm and thickness, $t = 3.2$ mm (see Figure 3.7b). The finite element analyses were carried out using the geometry shown in Figure 4.12a. It has a boundary condition along both the surfaces facing inwards. This is defined to allow movement in the in-plane directions and no movement in the direction perpendicular to the surface. Any rotation at these surfaces was also prohibited. The top and bottom surfaces are defined in separate node sets. The top is defined to move with constant speed in mm/s. The bottom is locked in the thickness direction and allows movement in the radial direction. With these boundary conditions the near-frictionless situation from the material tests is captured and the specimen deforms uniformly

The mesh is made with approximately 4 elements per millimetre in both the radial and thickness direction. The strains and stresses are read from the same location as in the material test, i.e. a cross-section at the midpoint of the specimen.

**Low strain rate, $\dot{\varepsilon} = 10^{-1}$ s$^{-1}$**

Figure 4.13 shows the results of the low-strain-rate compression simulation. The force displacement diagram is closest to the response of the test. The transition from elastic to plastic behaviour happens for a load approximately 50 N higher in the finite element analysis than in the test. After this point the model represents the plastic behaviour quite well before it starts to deviate with a significant
4.3. APPLICATION OF THE CALIBRATED MATERIAL MODEL

Figure 4.13: The results of the finite element model of the uniaxial compression test at $\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}$

load increase. The stress-strain diagram will not be correctly represented for this material. This is because the numerical model represents the same stress-strain behaviour in tension and compression, i.e. a yield point followed by softening and then hardening. The only difference is the $\alpha$ value that defines how much higher the yield stress is in compression than in tension. When the material has a compression response with a gradual transition to plastic behaviour and then hardening directly afterwards (as shown in Figure 4.13b), the material model will have problems representing it.

**Intermediate strain rate, $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$**

The finite element model describes more or less the same at this strain rate as for the previous. The main difference is that the increased strain rate makes the yield stress a little higher.

It is seen from Figure 4.14 that the response deviates quite a bit. The global force-displacement response (seen in Figure 4.14a) is not completely off. It misses the transition from elastic to plastic behaviour and ends up on a higher maximum load. The stress-strain curve on the other hand is much less accurate. The reason for this is as explained for the low strain rate. The elastic behaviour, however, is
almost correct at this strain rate. This is because the applied Young’s modulus is more correct for this intermediate strain rate than for the low strain rate (also seen for the uniaxial tension tests).

### 4.3.3 Comparison and discussion

In general, the material model manages to represent the topical polypropylene well. There are, however, some major and minor deviations. In this section the results are compared and discussed.

The strain rate dependency is challenging to represent. This is especially true for this material because it has a larger difference in the yield stress between the highest and the intermediate strain rate than between the intermediate and the low. This makes the strain-rate relation less log-linear (cf. Figure 4.4). In addition the shape of the curves vary with the strain rate.

The strain rate dependency is shown in Figure 4.15. An obvious deviation is the viscoelastic behaviour. On the other hand shows Figure 4.15a that the strain-rate dependency of the yield stress is quite well represented by the simulations. When examining the yield points in Figure 4.15a it is seen that it is approximately 2
4.3. APPLICATION OF THE CALIBRATED MATERIAL MODEL

Figure 4.15: Comparison of the strain-rate dependency of the finite element model and the material tests

MPa difference in the yield stress from strain rate to strain rate. Examination of Figure 4.15b reveals that the difference is approximately 1.5 from the low to the intermediate strain rate and 2.5 from the intermediate to the high strain rate, i.e. the mean difference over the spans is the same.

Figure 4.16: The fracture surface of test T5-L1

The graphs in Figure 4.15 also show that the representation of the softening does not fit perfectly, especially not for the lowest strain rate. The material tests in Figure 4.15b show a linear elastic behaviour almost all the way up to the yield point. But close to this point the behaviour becomes less linear. Figure 4.15a shows that this gradual transition is not represented by the material model. The
especially gradual softening seen in the slowest material test in Figure 4.15b is partly linked to the outer layer of the material (see Figure 4.16). This layer is, as mentioned in Chapter 2, the part of the material that has been in direct contact with the mould surface during the cooling phase of the manufacturing. It has therefore been cooled more rapidly than the rest of the material and may possess other mechanical properties. The skin on the topical PP shows a more ductile behaviour than the core material and is often intact even though the core material has fractured. The question is then: how is the adhesion between this layer and the core? If there is a distinct transition between the core and the outer skin, it might lead to a material behaviour that resembles the behaviour of a composite material. Such a transition may also mislead the Digital Image Correlation software into giving only the strains of the outer skin and not of the core material. It has also been observed that the outer skin becomes less distinct with increasing strain rate. Which might be part of the explanation to why the curves fit better with increasing strain rate.

![Comparison of hardening](image.png)

Figure 4.17: Comparison of hardening. T12-L1 is a high-strain-rate test. The two other curves are FE models with \( \beta \) as in the calibration (\( \beta = 1.56 \)) and with \( \beta \) based on the high-strain-rate tests (\( \beta = 1.71 \)).

The material model predicts more strain hardening than the material tests show. The main reason for this is assumed to be the heat effects that occur during high-strain-rate plastic deformation. These are, as mentioned earlier, not incorporated.
4.3. APPLICATION OF THE CALIBRATED MATERIAL MODEL

in the material model. Another source of error linked to this is the dilatation parameter $\beta$. In the topical PP this parameter also varies quite a bit with the strain rate. This variation is not represented by the material model.

A short recapitulation of the effect of $\beta$ (see Section 4.1 for details); $\beta$ is directly dependent on the contraction ratio $\rho$ (not the mass density) found from the material tests. The smaller the $\rho$ value, the larger the value of $\beta$. A higher $\beta$ value means that the dilation (increase of volume) is larger. When the volume increase is high the neck becomes less evident. The difference in $\beta$ between the baseline test and the fastest tests is close to 0.15, which corresponds to a difference in $\rho$ of about 0.07. Since the range of $\rho$ is only $0 < \rho < 0.5$ this difference could have a significant effect. An assumption regarding the effect of $\beta$ is that; due to the increase in volume the polymer chains are less oriented in the direction of the load, hence less hardening effect. Figure 4.17 shows the effect of $\beta$ on the stress-strain relation. The lower curve is the FE model with $\beta = 1.71$, which is the $\beta$ value found in the fastest material tests. The middle line represents the baseline model ($\beta = 1.56$) and the upper line is one of the high-strain-rate tests, T12-L1. The hardening in the lower curve ($\beta = 1.71$) is significantly less than in the baseline model and fits better with the hardening of the material test. On the other hand is the softening more correct for the baseline test. It is not certain that the effect shown in Figure 4.17 is solely because of $\rho$, but the variation of this parameter certainly has some effect.

The material is anisotropic (cf. Chapter 3). Since anisotropy is not included in the material model, the material calibration is dependent on the direction of the loading. For the topical PP the difference between the longitudinal and transversal response seems to correspond quite nearly to the difference between two strain rates in the longitudinal direction. This relation is most probably a coincidence, but still worth noticing. The graphical representation is seen in Figure 4.18. The agreement is quite good, especially for the tests done at the high strain rate and the finite element simulation at the intermediate strain rate. Here the maximum load, the yield stress and the strain hardening agrees nearly perfect. Some unevenness is observed in the unloading of the material test in Figure 4.18a and none of the two tests reach a stable load before fracture. For the comparison of the intermediate material tests and the slow finite element model, the accordance is slightly less. The strain hardening seen in the material tests in Figure 4.18d is the most prominent strain hardening of all the 14 tests. If it had not been this prominent, the agreement with the FE simulation would have been even better.

There are some sources of error related to the calibration procedure. The first is the input data from the DIC. If for example the camera frequency is set too low, the result can become too coarse, i.e. contain too few data points to correctly
describe the deformation. Some additional uncertainty is connected to the speckled pattern painted on the specimens (more in Chapter 3). The next thing is the decision regarding the baseline test. There might be properties that could be described better by choosing one of the other material tests as basis. Even though the calibration guidelines state that the baseline test should be a test from the lowest strain rate, this is not an absolute demand. In this thesis a test from the intermediate material tests was considered, but the calibration did not turn out more optimal than the calibration based on the low-strain-rate test. After choosing the baseline model and starting the calibration the next uncertainty is, as mentioned in earlier sections, Young’s modulus. Here the DIC makes up a mediocre foundation for a certain decision, and the best thing is to find it from tests designed for deciding this parameter. When evaluating the yield stresses the main uncertainty is connected to describing the strain rate dependency. As will be seen in the numerical validation of the material model, the strain-rate dependency is not necessarily log-linear for large strain rates. This might be solved by adding a high strain rate test to the linear regression, thus adjusting the fit to this kind of response. The cost of this being reduced accuracy in modelling of the low strain rate response. The macroscopic approach to the microstructure of the material [9] can also be a source of error. The microstructure varies quite a bit from polymer to polymer, and some structures might be easier to fit to the material model than others. And as closure, some human error might be present in treating the experimental data. This source of error is of course endeavoured minimized by the author.
4.3. APPLICATION OF THE CALIBRATED MATERIAL MODEL

Figure 4.18: Comparison of the response of the high strain rate and intermediate strain rate transversal material tests and the intermediate and slow strain rate FE simulations, respectively
Chapter 5

Validation

Two different validation tests were carried out. The first is a tension test using a plate with a centric hole and the other is a high-strain-rate test using a machine known as drop tower. The main objective of the validation is to test a wide range of the properties of the calibrated material model. The chapter is divided into one section for each test. Each section is divided into the following four parts; the test set up, the numerical discretization, the results and the conclusion. In the conclusion, the results are compared and discussed.

5.1 Plate with centric hole

This test was chosen because of its pure and simple loading. In spite of the simple loading the load response is more complex and gives good information about how the material model manages to represent more complex deformations than those found in the calibration tests.

5.1.1 The material test

The specimen is a 3.1 millimetre thick rectangular plate with a centric hole. The force is applied in the longitudinal direction, which is oriented along the axis parallel to the side that measures 160 millimetres (see Figure 5.1). The point of making the hole is that it creates two similar gauges, one at each side of the hole. By varying the strain rate it is possible to evaluate how the model manages to represent the strain rate dependency for more complex load responses than in the
CHAPTER 5. VALIDATION

Figure 5.1: The specimens used. The large hole to the left and the small to the right

calibration tests. Two hole diameters at two different deformation velocities were tested. The tests are summarized in Table 5.1.

<table>
<thead>
<tr>
<th>Test number</th>
<th>Test name</th>
<th>Deformation speed [mm/s]</th>
<th>Hole diameter [mm]</th>
<th>Gauge length [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>D1V1</td>
<td>0.05</td>
<td>30</td>
<td>15.39</td>
</tr>
<tr>
<td>2</td>
<td>D2V1</td>
<td>0.05</td>
<td>20</td>
<td>20.09</td>
</tr>
<tr>
<td>3</td>
<td>D1V2</td>
<td>0.5</td>
<td>30</td>
<td>15.07</td>
</tr>
<tr>
<td>4</td>
<td>D2V2</td>
<td>0.5</td>
<td>20</td>
<td>20.54</td>
</tr>
</tbody>
</table>

Table 5.1: The specifications of the four validation tests

The gauge lengths in Table 5.1 were supposed to be equal (see Figure 5.1) but a slight difference in the placement of the hole was detected. This deviation results in a slightly unsymmetrical response in some of the tests.

The tests were named as shown in Table 5.1. The principle is that D”number” gives the diameter, where D=30 mm is named 1 and D=20 mm is named 2. The V”number” gives whether the deformation speed is 1 = 0.05 mm/s or 2 = 0.5 mm/s.

The specimens were tested in the same machine as the material tests, namely a Dartec M1000 RK with a 20 kN load cell connected to an Instron controller. The clamps were mounted approximately 30 millimetres from the edge of the plate making the effective specimen length 100 millimetres. Digital image correlation was used to capture the deformation. Two cameras were applied; one was fixed at the surface with the hole, the other was fixed at the surface perpendicular to the surface with the hole, i.e. monitoring the thickness. To be able to process the pictures with 7D, the specimens were painted with the same speckled pattern as
the material tests (see Figure 5.2).

The speed was set in millimetres per second. The velocities of 0.05 mm/s and 0.5 mm/s were set to give strain rates in the gauge areas of approximately $0.0025 \text{ s}^{-1}$ and $0.025 \text{ s}^{-1}$, respectively.

| $U_{l,25}$ | The longitudinal displacement found 25 millimetres above the hole. Shown as the upper red dot in the figure to the right |
| $U_{l,C}$ | The longitudinal displacement at the “top” of the circle. Seen as the lower red dot in Figure 5.3a |
| $U_{t,1}$ | The transversal deformation of gauge 1. Shown as $U_1$ in the figure to the right |
| $U_{t,2}$ | The transversal deformation of gauge 2. Shown as $U_2$ in the figure to the right |
| $F$ | The force needed to deform the specimen. It is the top clamp that moves, making this the point where the load is applied |

![Diagram showing plate with centric hole and displacement markers](image)

Table 5.2: The parameters that are to be compared

The post-processing was mainly carried out using 7D and Matlab. The main interest in these tests was to see how the longitudinal and transverse deformation and the force-displacement relation were represented by the finite element model. On this basis a collection of parameters were chosen for the comparison of the tests and the finite element analyses. The parameters and their location are shown in Table 5.2. In order to compare these displacements, they had to be taken from the same location in both the test specimen and the finite element model. The process of collecting the deformation data started with 7D. The pictures taken during the test were processed in the same way as the material tests in Chapter 3. The difference was that the displacements were added to the output. The matrices constructed by 7D containing the strain and displacement data were loaded into Matlab. A Matlab script was then used to define the points from where the deformation was to be gathered. The same script also output the needed
deformation vectors. An additional script was made to find $U_{t,1}$ and $U_{t,2}$. The script was similar to the one used for treating the test data from the compression tests. It used the pictures directly from the camera, made them into black and white and found the first and last white point at each gauge. The difference between these points in each gauge was stored for every picture, resulting in two displacement vectors describing the displacement $U_{t,1}$ and $U_{t,2}$ (the scripts can be found in Appendix B). $U_{t,1}$ and $U_{t,2}$ were stored until a fracture occurred in the gauge, which in some cases was a bit earlier than the end of the test.

The load response was much as expected for the four tests. The load level was inversely proportional to the hole size, the peak load increased with increasing strain rate and the fast tests fractured faster than the slow ones. Yet again the fracture surfaces were subject to some extra examination. These were very different in the slow and the fast tests. The two specimens with D=30 mm are shown in Figure 5.2. In Figure 5.2a it is seen that the external skin becomes clearly visible in the tests at the lowest speed. Since the strain data from the gauge is only used indirectly in these analyses, there should be minimal effect of the layer on the load response. Apart from this it is seen that the cross-section of the slow test is significantly reduced in the gauges. At the highest speed the fracture was much more sudden, something that is also shown in Figure 5.2b. The gauge cross-sections are only reduced very locally. Much of the deformation is no longer visible because the material retracted a bit after fracturing. All the results can be found in Appendix A.2.
5.1.2 The finite element model

This test was chosen for its simple yet complicated nature. The complicated part of the problem became clearer in the process of creating a good finite element model.

Figure 5.3: The meshed geometry used in the finite element analyses (D1V2 and D2V2)

Figure 5.3 shows two of the applied meshed geometries. Symmetry about the mid-transversal axis was applied resulting in a model representing 40 mm of the total 160 mm. Since the holes were not placed entirely symmetric, there was no symmetry about the longitudinal axis. This asymmetry also meant that four separate models (one for each test) had to be made in order to attain the wanted level of accuracy. The mesh is graded so that it is finest near the gauge and coarser further away from the gauge. Four elements were used in the thickness direction. In spite of looking a bit untidy, the mesh performed well with the fully integrated element formulation “2” of LS-DYNA [6]. Part of the reason for the untidy impression is the unsymmetrical placement of the hole. This makes the refinement of the mesh close to the gauge more challenging.

A boundary condition restraining the longitudinal displacement and allowing transversal displacements was applied in the gauge. The deformation was applied to the model using a boundary condition defining a prescribed motion. This was applied along the edge opposite the hole. Other mesh geometries and boundary conditions were tried, but were all less optimal than the solution presented above.

The correct strain rate was found from the DIC data. First the critical cross-section had to be found. This was done using a similar script as for the material tests, i.e. a script using the mean values of the longitudinal strain to locate the
critical cross-section. The script found that the critical cross-section was located at the mid-point of the gauge. Then a strain-time diagram (seen in Figure 5.4a) was plotted and a linear regression was fitted to the linear part of the resulting diagram (the red line in Figure 5.4a). As a validation of this, the same was done for the longitudinal deformation. The longitudinal displacements were output from the wanted locations as defined in 7D. Both operations were carried out using Matlab and the scripts are added in Appendix B.4.

The analyses went well after preparing the model as described above. When they were finished the results were gathered from points in the model corresponding to those defined on the specimen.

### 5.1.3 Results

In this section the results of both the material tests and the finite element analyses are presented. The finite element analyses will consequently be shown as full lines in the plots, the model consequently with dashed lines. The presentation will start with the tests done with $v=0.05$ mm/s and then move on to the ones where $v=0.5$ mm/s. There will be a short discussion for each test and a finishing comparison and discussion at the end.
5.1. PLATE WITH CENTRIC HOLE

D1V1

(a) The displacements. The upper curves represents the longitudinal displacements \((U_{l,25}, U_{l,C})\). The lower curves represent the transversal displacements \((U_{t,1}, U_{t,2})\)

(b) The force plotted against the displacement

Figure 5.5: The results of the test and the FE-simulation of D1V1

Figure 5.5 shows the results of the test and analysis of D1V1, i.e. the test where \(D=30 \text{ mm}\) and \(\nu=0.05 \text{ mm/s}\). It is seen from Figure 5.5a that the displacements are quite well represented. A slight difference in the gradients is observed. This difference occurs because the deformation speed in the physical test is not constant. It is a bit lower in the beginning and then it stabilizes at a speed similar to that of the analysis. The analysis, however, is defined to be constant from the start until the end.

The force displacement diagram in Figure 5.5b agrees less with the material tests than Figure 5.5a. It is seen that the modelled peak load and plastic response is more or less correct. This means that the main deviation is found in the stiffness, which is too low. This can be corrected by adjusting Young’s modulus. The cost of this being that the peak load becomes slightly higher, and thus less correct.

Figure 5.6 shows the force-displacement response for a higher value of Young’s modulus. The correction makes the stiffness nearly correct and yields only a minor increase in the peak load. In this test the strain rate in the gauges is approximately 0.0025 s\(^{-1}\). If the calibration results are taken as reference, Young’s modulus should be somewhere between 300 and 400 MPa for this strain rate.
most correct analysis result is, however, attained with a modulus as high as 750 MPa. The exact reason for this is not clear. The increase in Young’s modulus causes a negligible decrease in the deformations (seen in Figure 5.6a).

D2V1

The results of this test is in many ways similar to the first. An important difference is the load level. Since this hole is only 2/3 the size of the largest, the forces are significantly larger.

Figure 5.7a shows that the deformation fits even better for this test than the previous. The variation in the longitudinal displacement is captured by the model and is seen as the second curve from the top. This curve represents \( U_{l,C} \), i.e. the displacement at the top of the circle. This variation is, however, not captured by the model in the displacement 25 millimetres above the circle, \( U_{l,25} \), which has a constant gradient. The gauge displacements fit very well, and represent the displacement more or less exact.

A similar behaviour as for D1V1 is observed in Figure 5.7b. The stiffness is too low, but the peak load and the plastic behaviour fit better with the test. The curve representing the test where \( E=750 \) MPa is also added to Figure 5.7. The
5.1. PLATE WITH CENTRIC HOLE

Figure 5.7: The results of the test and the FE-simulations of D2V1

(a) The displacements. The upper curve represents the longitudinal displacement $U_{l,25}$. The curve parallel to this is $U_{l,C}$. The lower curves represent the transversal displacements ($U_{t,1}$, $U_{t,2}$).

(b) The force plotted against the displacement $U_{l,25}$.

agreement becomes significantly better, again reducing the accuracy of the peak load. The difference in displacements is not shown because it was so small that it could be neglected.

D1V2

This was the first test at the high deformation speed. The main difference between the velocities was the gauge deformation, which was much more local for this and the next test than for the two previous. In addition some differences were observed in the plastic response and the longitudinal displacements.

Figure 5.8 sums up the results. The difference in timespan should be noted. These tests were through in about 20 seconds, while the slower ones took between 5 and 10 minutes. The displacements are described quite accurately. Figure 5.8a shows that the gradient of the longitudinal deformation in the material test (the upper dotted line in Figure 5.8a) is varying. It has an initial gradient slightly lower than that of the model, then it gradually approaches the model, before it ends up with a higher gradient than the model. The longitudinal displacements $U_{l,25}$ and $U_{l,C}$ from the model are different from each other in this test too. $U_{l,C}$ (the second full
(a) The displacements. The upper curve represents the longitudinal displacement $U_{l,25}$. The curve parallel to this is $U_{l,C}$. The lower curves represent the transversal displacements ($U_{t,1}$, $U_{t,2}$)

(b) The force plotted against the displacement $U_{l,25}$. The peak force agrees almost perfectly for $E=400$ MPa and is only slightly off for the adjusted Young’s modulus. The transition from elastic to plastic behaviour is a bit different. The model where $E=700$ MPa has a more sudden transition. The test and the simulation where $E=400$ MPa has a more gradual transition. The difference in the load level at the end of the test indicates that the gradient of the model’s plastic behaviour is a little low.

Figure 5.8: The results of the test and the FE-simulations of D1V2

D2V2

This specimen was the one with the highest difference in gauge size. Gauge 1 is 0.74 millimetres wider than gauge 2. This results in a slightly unsymmetrical response. Because of the small hole and the high deformation rate, this is also the test where the highest load level occurs. The results of this test are shown in four instead of two plots, this to avoid plots containing too many curves.
5.1. PLATE WITH CENTRIC HOLE

(a) The displacements. The upper curve represents the longitudinal displacement $U_{l,25}$. The curve parallel to this is $U_{l,C}$. The lower curves represent the transversal displacements ($U_{t,1}, U_{t,2}$)

Figure 5.9: The results of the test and the FE-simulations of D2V2

(b) The force plotted against the displacement $U_{l,25}$. The curve parallel to this is $U_{l,C}$. The lower curves represent the transversal displacements ($U_{t,1}, U_{t,2}$)

Figure 5.9 shows the results of D2V2. The longitudinal displacements of the model are more scattered in this test than in the previous tests. The test result is now located between the two curves instead of corresponding to one of them. It is still only $U_{l,C}$ from the model that has a changing gradient. A slight scattering is also seen in the test’s transversal gauge displacements. The corresponding displacements from the finite element model deviate a bit but end up at approximately the same displacement.

The force-displacement diagram shows similar results as the previous. The differences are a higher and more correct peak load and a slightly better agreement in the stiffness. If the stiffness is adjusted the results become as shown in Figure 5.10.

The adjustment of Young’s modulus results in changes in both the displacement and the force-displacement diagram. The longitudinal displacement $U_{l,C}$ of the model now corresponds better to the longitudinal displacement of the test. In addition, the transversal deformations have moved down a notch, which results in a better fit with the transversal deformations of the test.

The force-displacement relation is very well represented. In this case both the stiffness and the peak load are correct. The deviation is now located at the transition from elastic to plastic material behaviour, where the test turns and becomes just slightly steeper than the simulation.
(a) The displacements. The upper curve represents the longitudinal displacement $U_{l,25}$. The $U_{l,25}$ curve parallel to this is $U_{l,C}$. The lower curves represent the transversal displacements ($U_{t,1}$, $U_{t,2}$)

(b) The force plotted against the displacement $U_{l,25}$

Figure 5.10: The results of the test and the FE-simulations of D2V2

5.1.4 Comparison and discussion

The four tests have proved that the material model is capable of describing quite complicated deformations.

Figure 5.11 shows the strain-rate dependency of the tests and the finite element simulations. The tests show a minimal increase in Young’s modulus with increasing strain rate, indicating that modelling of the viscoelastic behaviour can be left out without losing much accuracy. When examining the strain rate dependence, it is seen that this is very well represented by the model. The transition from elastic to plastic behaviour is quite gradual in the tests. In the simulations this gradual transition is only captured by the geometry with the largest hole. The geometry with $D=20$ mm has a much more distinct transition. For the plastic deformation of the tests it is seen that the slowest tests almost stabilize at a constant load level before the fracture occurs. This is not the case for the faster tests, which descend constantly until the fracture occurs at about 8 millimetres of longitudinal displacement. The simulations vary a bit in their representation of this plastic behaviour. The two geometries where $D=30$ mm display the same stabilizing
5.1. PLATE WITH CENTRIC HOLE

(a) The strain-rate dependency observed in the tests

(b) The strain-rate dependency observed in the FE-simulations

Figure 5.11: The strain-rate dependency
trend as the tests for both deformation rates. The tests with the smaller hole has a behaviour resembling that of the fastest tests, i.e. a constant decrease of capacity. The slowest of these two (D2V1) stabilizes gradually, but much slower than the test.

These tests bring new information regarding Young’s modulus. Since a major part of the deformation was elastic, the response was quite dependent on this parameter. In the calibration tests the fastest tests deformed with \( E \approx 600 \) MPa and the slowest with \( E \approx 350 \) MPa. In these validation tests the lowest strain rate was only slightly higher than the lowest strain rate applied in the calibration testing, but still \( E = 750 \) MPa gave good results. It is also a bit strange that Young’s modulus had to be slightly lowered in order to attain the best fit with the fastest test. The logic assumption would be the opposite. This indicates that Young’s modulus may vary a bit, not only with the strain rate but also with other factors linked to the geometry and the load application.

The sources of uncertainty are always present. An obvious source of error introduced quite early in this testing is the unsymmetrical placement of the holes. Digital image correlation also has its known weaknesses. The process of defining the points from where the deformations were to be found is also a significant source of inaccuracy. This was done with the measurement tools in 7D, which resulted in approximative distances. The method used to find the transversal deformation of the gauges utilizes pictures with a wide spectre of grey scales. These were con-
verted into pictures containing only black or white by defining a limit for where white ended and black started. The number of white pixels will vary with where this limit is set, thus introducing an inaccuracy in the measuring of the gauge widths.

![Image](image.jpg)

Figure 5.12: The test D1V1 and the model deformed approximately 10 mm in the longitudinal direction

Despite these inaccuracies, the finite element simulations managed to represent the problem quite well. This is exemplified in Figure 5.12, which shows the deformed specimen and model. It is seen that the shape of the specimen and the model is very similar. The load response became close to perfect after the slight adjustment of the elastic parameters. The plastic behaviour was well represented through the correct peak loads and the more or less correct plastic responses.
5.2 Impact loading on plate

This test is basically a steel spear impacting a plate of the material with a user-defined kinetic energy. The main application of the material model is, as mentioned earlier, high-strain-rate problems like impact and crashworthiness simulations [9]. The response to the high strain rates achieved in this test is thus of interest. The section will have a similar structure as the previous.

5.2.1 The material tests

The specimens used in these tests were very simple. They were quadratic plates with 10 millimetre edges and a 3 mm thickness.

The test machine was a CEAST 9350 drop tower. The basic components of the test machine are a mechanism to control the speed, weight and drop height for the steel spear and a mounting mechanism for the test specimens. The spear can be equipped with different end pieces. The one used in these tests was a half sphere with a diameter of 20 mm. Before mounting the specimen the wanted weight was placed on the spear. The plate was then mounted in the machine simply by putting it on a ramp. After a safety door was closed, a clamp was lowered to lock the plate in place. The clamp held the specimen down using a pressure of 5 bar. The next point was to configure the wanted impact energy. After this was configured the test rig was ready, and the “play” button could be pushed.

Table 5.4 on page 75 shows the specifications of tests carried out in the drop tower. They are named successively. The mass column gives the weight of the falling spear and the energy is calculated according to Equation (5.1).

\[ E = \frac{1}{2}mv^2 \]  

(5.1)

The impact energy was adjusted to the capacity of the plate. Meaning that if a test fractured, the energy was adjusted downwards, if it did not fracture, the energy level was adjusted upwards. This approach is illustrated in Table 5.4. The table also shows that the capacity of the specimens used was somewhere between 55 and 60 J. In addition the energy was varied by changing either the mass or the impact velocity.

The data used for comparison with the finite element simulations were logged by the drop tower, through a sensor in the spear. The comparison was done with the force-time and the force-displacement relations.
5.2.2 The finite element model

This finite element model has to represent a response which is quite different from the previous validation test. It is an impact problem with short response time and significant inertia forces. The objective is to test how the material model manages to represent a problem involving such high strain rates.

An axisymmetric approach was used to model the problem. This means that only a radian of the full geometry was modelled. Figure 5.13 shows the modelled section. The y-axis at x=0 is the symmetry axis defined by LS-DYNA. This means that the model had to be constructed with this axis as a centre. The placement of the y-axis can be seen in Figure 5.13, it corresponds to the longitudinal axis of the spear. The axisymmetric definition was given through the use of element definition 15 in LS-DYNA [6]. The optimal number of Gauss points varied with the impact energy. In the simulations of the tests where the plate eventually fractured, four Gauss-points were used, in order to eliminate the hourglass energy that would else have occurred when the material was deformed to its limits. In the tests that absorbed the fracture, a reduced integration scheme involving two Gauss-points was successfully used.

The model was split into three parts. First the plate, the yellow part in Figure 5.13, was modelled. This is only a rectangle measuring 35 mm × 3.1 mm. The mesh was
found by starting with a relatively coarse mesh and then refining. After some trial
and error the mesh shown in Figure 5.13 was found as the most optimal. The mesh
is graded so that the elements are smallest closest to where the spear impacts.

The modelling of the spear also called for some trial and error. A major difference
between the previous finite element models and this, is that this is an impact
problem, i.e. the inertia forces are important. To represent the correct physical
problem the mass of this radian of the spear had to be exact. It was not an
option to model the spear and the weight rig in full size. The solution became
a two-component solution, as seen in Figure 5.13. The upper part (the red part)
represents the majority of the weight of the spear. This part is not in contact
with the plate and acts as a sort of point mass. The wanted mass is achieved by
adjusting the mass density. To get the correct mass of 5.045 kg used in some of
the tests, the mass density of this part had to be set as high as $1.064 \cdot 10^8$ kg/m$^3$.
A problem with density values this high is that it makes the representation of
the forces inaccurate. In order to fix this problem, the blue part was introduced.
This is modelled as hardened steel. The steel was modelled with its real mass
density of 7850 kg/m$^3$, which resulted in a correct representation of the forces.
A cross-section in the straight area of the blue part was chosen as the reference
cross-section. The spear can be considered almost rigid compared to the plate,
meaning that a relatively coarse mesh could be applied in the spear without loss
of accuracy.

The next thing was the contact definition. An edge-to-edge contact situation in-
volving shell elements is not an optimal. This is mainly because shell elements
in most cases are applied using edge-to-surface or surface-to-surface contact. Af-
fer some more trial and error the choice fell on the contact formulation CON-
TACT_2D_AUTOMATIC_SINGLE_SURFACE [6]. This is an “automatic” contact
definition, meaning that it defines the master and slave surface on its own.
It is also especially suited for 2D problems. The friction parameters were set to
$\mu_{\text{static}} = 0.25$ and $\mu_{\text{dynamic}} = 0.9 \cdot \mu_{\text{static}} = 0.225$. The penalty parameter was set
to 1.5.

The boundary conditions were less complex. The right end was defined as rigid
and the left edge was constrained through the axisymmetric element formulation.
The spear was given an initial velocity in units mm/s according to column three
of Table 5.4.
5.2.3 Results

In this section the results of the tests and simulations of the drop tower will be presented, mainly using force-time and force-displacement diagrams.

(a) The specimen used in the test DT1. The plate absorbed the impact energy of 50 J by deforming as shown.

(b) The specimen used in the test DT6. The energy in this test was 55 J. The 5 J energy increase from DT1 resulted in a noticeable difference in the deformed shape.

Figure 5.14: The two specimens that absorbed the impact

Fracture occurred in all the specimens where the energy was higher than 55 J. The fracture appeared to be caused by a plug-like mechanism, i.e. a shear fracture around the area of impact causing the impact area to be pushed out in one piece. The two specimens that did not fracture are shown in Figure 5.14. DT1, the test with 50 J impact energy, is shown in Figure 5.14a. It is seen that the plate has absorbed the energy by deforming. The deformation is quite smooth, and decays gradually away from the point of impact. A more local deformation is seen in Figure 5.14b. This is the deformed shape of the specimen that was used in the test DT6. In this test, the impact energy was 55 J. The deformed shape is quite different from that of DT1, even though the energy is only 5 J higher. This is probably because this energy level is close to the maximum capacity of the plate. If only 5 more joules were added, as was done in DT5, the specimen fractured.
5.2. IMPACT LOADING ON PLATE

In the previous validation tests the strain rates were in the range of $10^{-3} \text{ s}^{-1}$ to $10^{-2} \text{ s}^{-1}$. In these tests the strain rates are in the range of $10^2 \text{ s}^{-1}$ to $10^3 \text{ s}^{-1}$. Since polymers are non-newtonian, i.e. the load capacity increases with increasing strain rate, the load levels observed in this section should be significantly higher than in the previous tests.

![Force-time diagram](a) The force-time diagram
![Force-displacement diagram](b) The force-displacement diagram

Figure 5.15: The results of test DT1

Figure 5.15 shows the results of the test DT1. Even though this is one of the “slowest” drop tower tests, the peak load is almost 1 kN higher than the highest peak load observed in the previous validation tests. The expected load increase is thus confirmed. When looking at the force-time diagram in Figure 5.15a, three phases can be observed. First load application and material absorption, then the peak load is reached and the absorption stops. The third phase is unloading. For this test, which did not fracture, are all the phases described, and the curve is smooth from start till end. If a fracture had occurred, the curve would have fallen to zero load at some point.

The result of the finite element simulation is seen as the dotted line in Figure 5.15. It is observed that both the stiffness and the peak load are a little low. The curves are relatively correct in shape, but a bit shifted to the right compared to the test results.

The deviations observed in Figure 5.15 are similar for all the tests. The deviations
can be reduced by minor alterations to the material model. An adjustment of Young’s modulus reduces the deviation in the initial stiffness. The value found to give the best results is 700 MPa, which is the same value that was applied to some of the analyses of the previous validation tests. The other parameter to be altered is linked to describing the strain rate dependency. Recall the strain rate dependency of the material model (cf. Figure 4.4). The relation is modelled as a linear regression between values of the yield stress at two or more strain rates. These reference strain rates are in most cases between $10^{-3} \text{s}^{-1}$ and $10^{-1} \text{s}^{-1}$, and the linear approximation produces good results in this strain-rate regime. The current strain rates are, as mentioned, significantly higher. And it is not unlikely that the strain-rate dependency becomes less log-linear with increasing strain rates. The above argument is stated to justify a slight increase in the parameter controlling the gradient of the strain-rate regression, namely the parameter $C$. This slight increase will yield a significant change in the predicted yield stress for the high strain rates seen in the drop tower tests. This naturally yields an increase in the peak load too.

Figure 5.16: The results of the test and the analysis of DT1. Here $E=700 \text{ MPa}$ and $C=0.07$

Figure 5.16 shows the results of the above adjustments. $E$ is adjusted to 700 MPa and $C$ is set equal to 0.07. Most of the deviation in the initial phase of the response is now gone due to the increase of $E$. In addition the increase of the $C$ parameter
5.2. IMPACT LOADING ON PLATE

has brought the peak load up to a near perfect match. The remaining deviation is in the unloading. This part of the response happens with another rate than the initial load response, thus yielding another modulus than the initial modulus. The representation of a different modulus in unloading than in loading can not be represented without applying viscoelastic effects, which are not included in the material model.

Figure 5.17 presents the force-time relation of the five remaining tests. The modified material model with E=700 MPa is applied in all the analyses. In addition, the parameter C is adjusted so that each analysis attain a good match with the peak load. Table 5.4 gives the specifications of each test.

<table>
<thead>
<tr>
<th>Test</th>
<th>Time [ms]</th>
<th>Displacement [mm]</th>
<th>Force [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>DT2</td>
<td>4.82</td>
<td>25</td>
<td>3785</td>
</tr>
<tr>
<td>DT3</td>
<td>6.1</td>
<td>21</td>
<td>3762</td>
</tr>
</tbody>
</table>

Table 5.3: The fracture point of DT2 and DT3

<table>
<thead>
<tr>
<th>Name</th>
<th>Mass [kg]</th>
<th>Velocity [m/s]</th>
<th>Drop [m]</th>
<th>Approx. energy [J]</th>
<th>Fracture?</th>
</tr>
</thead>
<tbody>
<tr>
<td>DT1</td>
<td>5.045</td>
<td>4.43</td>
<td>1</td>
<td>50</td>
<td>No</td>
</tr>
<tr>
<td>DT2</td>
<td>5.045</td>
<td>6.26</td>
<td>2</td>
<td>100</td>
<td>Yes</td>
</tr>
<tr>
<td>DT3</td>
<td>10.045</td>
<td>4.43</td>
<td>1</td>
<td>100</td>
<td>Yes</td>
</tr>
<tr>
<td>DT4</td>
<td>10.045</td>
<td>3.864</td>
<td>0.761</td>
<td>75</td>
<td>Yes</td>
</tr>
<tr>
<td>DT5</td>
<td>5.045</td>
<td>4.877</td>
<td>1.213</td>
<td>60</td>
<td>Yes</td>
</tr>
<tr>
<td>DT6</td>
<td>5.045</td>
<td>4.67</td>
<td>1.112</td>
<td>55</td>
<td>No</td>
</tr>
</tbody>
</table>

Table 5.4: The tests carried out using the drop tower
Figure 5.17: The results of the remaining drop tower tests and simulations
5.2.4 Comparison and discussion

This test has the most complex load response of the three tests in this thesis. In spite of these complexities, the material model has managed to represent the response quite accurately.

In an impact response, the strain rate in the specimen is closely linked to the impact velocity. This is also seen in Figure 5.17. In the tests DT3 to DT5, the peak load is approximately the same (3750 N). These tests also have relatively equal impact velocities, but varying mass. DT2 confirms the theory by having the “lowest” weight, the highest impact velocity and the highest peak load.

The friction is given little attention in the previous sections. This is a quite important parameter in this problem. The friction between the spear and the specimen is significant and it involves both dynamic and static friction. No data about the actual friction coefficients between the polypropylene in question and various surfaces has been found, so the value chosen for use in the analyses is based on trial and error and general assumptions on what it usually is for polymers. A typical assumption is that $\mu_{static} = 0.2$ for polymer-metal contact and that the dynamic friction is 90% of this static friction. On this basis, values from 0.1 up to 0.5 were tried as only variable in a FE-analysis. The peak load and the numerical noise increased with increased values of $\mu_{static}$. For smaller values the trend was opposite. The chosen value of $\mu_{static} = 0.25$ turned out be a good midpoint between load representation and numerical noise.

![The real and the modelled boundary clamp](image)

**Figure 5.18: The real and the modelled boundary clamp**

The boundary conditions had to resemble the clamp seen in Figure 5.18a. The upper part seen in the figure is lowered down and presses the specimen down with a chosen pressure (5 bar in these tests). Three main concepts were tried. The first was to model the clamp but not the clamping pressure, i.e. a model like in Figure 5.18b but with no extra pressure from the upper part. The plate is defined
as rigid at the nodes along the right edge of Figure 5.18b. With this model the plate has a smoothed transition from the clamped to the free state. The second method was a version of the same principle. Now the upper plate was applied a downward force corresponding to the 5 bar pressure applied in the machine. The rigid definition of the nodes in the plate was removed leaving the plate without any explicit boundary condition. The last option that was tried was simply to define the right end of the plate as rigid without modelling the clamp (see Figure 5.13). The main difference between these principles was the smoothing provided by the clamped model. And this smoothing did not affect the results to any extent. The most basic model was thus chosen.

The complexity of the problem means that there are many sources of error. The simple geometries of the specimens and the relatively foolproof mounting of the specimen in the machine make the probability of error in these phases of the process quite small. In modelling, the situation is quite different. Here the sources of error can be found in almost all elements of the model. First the question of element formulation. Could modelling the whole problem with cubic elements have given other results? It is hard to tell exactly. In principle, there should not be any difference. On the other hand could the cubic elements have made it easier to calculate the exact mass of the spear. And there might also be other effects that are not represented by the current model. One thing is for sure, a model of cubic elements would have become very computationally expensive. An advantage of the current model is that it is very effective and can be run with a lot of different parameters in a short period of time. Such operations had not been possible with the cubic model. The next uncertainty is linked to defining what parameters that are involved in the different phases of the simulation. In Section 5.2.3, good results were obtained by altering only two parameters. These two alone do most probably not represent the full response, they are only central parameters that affect the total response. When doing a parameter study the different parameters are tested one by one, making it difficult to see synergy effects. Even better results might have been achieved by further examining such synergies. The problem with this process is of course the complexity that arises.

The drop tower test involves a quite complex material response. This response is modelled using a quite simplified material model. In spite of this, the material model has proved that it is able to represent the problem in a satisfactory manner
Chapter 6

Conclusion

The previous chapters have shown the calibration and validation of the SIMlab polymer material model. The material model has been calibrated to resemble an impact modified polypropylene. This material has proved to be a bit challenging to exactly represent, but the material model has overcome most of the problems and managed the representation quite accurately.

The process started with the unaxial tension and compression tests. Through the tension tests it became clear that the material deformation was a bit unusual. It dilated significantly and if it fractured it fractured in the top of the gauge length. The fracture seemed to occur in the core and then propagate to the surface, where a more ductile external skin was stretched before the specimen finally fractured completely. The stress-strain curves showed a mixture of earlier observed behaviours for materials like PVC and HDPE [5]. The curves showed softening, but only slightly, thus being somewhere between the PVC, which softens significantly, and HDPE, which does not soften at all. Then the stress was constant for a while, through this resembling the behaviour of other versions of polypropylene, before it slightly hardened. From the tension tests it was also found that the load level was significantly lower in the specimens from the transversal direction than in those from the longitudinal direction, which means that the material was anisotropic. The compression response was found to be different from the tension response. Here the material did not soften and hardening was initiated immediately after yield.

The next step was to calibrate the material model to fit the response of the chosen baseline test. The unusual load response of the material made it a bit challenging to identify some of the material parameters and some deviations between the response curves had to be accepted. The deviations were located in the elastic
response and in the transition from the elastic to the plastic response.

The calibrated material model was now ready to be tested in finite element simulations. The first tests to be simulated were the material tests, i.e. the uniaxial tension and compression test. The representation of these tests was challenging and significant deviations were observed in the simulation of the slowest tests. The calibration was redone in order to check for errors. But the examination came up with the same result. The conclusion was that due to the significant dilatation and the irregular deformation were the material tests hard to represent exactly.

After making sure that the calibration was correct, the validation could begin. This process consisted of two different tests, which tested different properties of the material model. The first test was the plate with a centric hole. The main difference between this test and the material tests was that this specimen had a more advanced geometry and a slightly more complex load response. The other validation test was the drop tower. This test represented a dynamic problem involving high strain rates and a complex load response, it thus validated other properties of the material model than the first validation test. Through the validation tests it became clear that Young’s modulus was more in the range of 700 - 750 MPa than 400 MPa. The drop tower tests showed that the strain rate dependency deviates from a log-linear behaviour for high strain rates, thus yielding a slight increase of the parameter C. By adjusting these two parameters the material model managed to represent both tests very well.

As a conclusion it can be stated that the material model has successfully managed to represent the polypropylene treated in this thesis. Even though the calibration procedure is fairly simple it has not stopped the material model from representing complex problems. A further enhancement would be to include the viscoelastic behaviour and a fracture criterion. By adding these properties can the material model represent an even wider range of problems. They should, however, be implemented in a way that does not complicate the calibration in any extent, thus conserving the simple yet complex nature of this material model.
Bibliography


81


Appendices

June 7, 2012
Appendix A

Material tests

This part contains a sheet for every material test that was carried out. It starts with the calibration tests, then the plate-with-centric-hole (PWCH) tests are presented and at the end the drop tower tests are presented.

The calibration tests are presented with pictures of the tested specimen, three diagrams, a table containing key information and a short description. The diagrams are a force-displacement diagram, a stress-strain diagram and a strain diagram. The strain diagram contains the contraction parameter $\rho$ and the transversal strain plotted against the longitudinal strain. The transversal strain curve is cut before its final value to better show the part of the curve that is of interest ($\varepsilon_l \leq 0.1$). The compression tests are also shown. The post-test measures of the compression tests are a bit too high compared to the final strain value shown in the stress-strain diagram. This is because the specimens swelled after the load was removed.

The PWCH and drop tower tests are presented in a similar manner as the calibration tests. The plots presented for the PWCH tests are the force-displacement diagram and plots showing the deformation at the different points on the specimen. For the drop tower tests the force-time, velocity-time and displacement-time relation are shown.
A.1 Calibration
A.1.1 T1-L1, $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>240 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>20 MPa</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td></td>
</tr>
<tr>
<td>b [mm] t [mm] A [mm$^2$]</td>
<td></td>
</tr>
<tr>
<td>12.0 3.1 37.2</td>
<td></td>
</tr>
<tr>
<td>After test</td>
<td></td>
</tr>
<tr>
<td>11.6 2.9 33.6</td>
<td></td>
</tr>
</tbody>
</table>

The test was not valid and is therefore not included in any further processes of this thesis. The purpose of including it here is to show what level of scatter that was accepted.
A.1.2 T2-L1, $\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>250 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>20 MPa</td>
</tr>
<tr>
<td>$b$ [mm]</td>
<td>$t$ [mm]</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>12.0 3.1 37.2</td>
</tr>
<tr>
<td>After test</td>
<td>9.0 1.9 17.1</td>
</tr>
</tbody>
</table>

The test deformed without developing a distinct neck and did not fracture. The material has a distinct yield point and hardens approximately 5 MPa.
A.1.3  T3-L1, $\dot{\varepsilon} = 10^{-3} \text{ s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Yield stress</td>
</tr>
<tr>
<td>b [mm]</td>
</tr>
<tr>
<td>Initial dimensions</td>
</tr>
<tr>
<td>After test</td>
</tr>
</tbody>
</table>

The test developed a neck and fractured. This test was chosen as the baseline test for use in the calibration. The external layer is clearly visible in the pictures.
A.1.4 T4-L1, $\dot{\varepsilon} = 10^{-2}$ s$^{-1}$

The first test at a higher strain rate. The test developed a neck and fractured as show. The yield point is more distinct than at the previous strain rate. The yield stress and Young’s modulus have also increased.
A.1.5 T5-L1, $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>410 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>22 MPa</td>
</tr>
<tr>
<td>$b$ [mm]</td>
<td>11.9</td>
</tr>
<tr>
<td>$t$ [mm]</td>
<td>3.1</td>
</tr>
<tr>
<td>$A$ [mm$^2$]</td>
<td>36.9</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>11.9</td>
</tr>
<tr>
<td>After test</td>
<td>10.1</td>
</tr>
</tbody>
</table>

A neck occurs and propagates. The fracture is gradual, with the core fracturing first and then outer skin. The fractured core material is seen as the tip of the shortest part in the pictures.
**A.1.6 T6-L2, $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$**

![Specimen Images]

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>410 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>22 MPa</td>
</tr>
<tr>
<td>$b$ [mm] $t$ [mm] $A$ [mm$^2$]</td>
<td></td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>11.9  3.1  36.9</td>
</tr>
<tr>
<td>After test</td>
<td>11.4  2.8  31.9</td>
</tr>
</tbody>
</table>

A specimen from the second plate from which specimens were machined. Almost no necking and a very local fracture. It hardens less than the specimens from plate 1.
A.1.7  T7-L2, $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>400 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>22 MPa</td>
</tr>
<tr>
<td>$b$ [mm]</td>
<td>$t$ [mm]</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>12.0</td>
</tr>
<tr>
<td>After test</td>
<td>11.1</td>
</tr>
<tr>
<td>$A$ [mm$^2$]</td>
<td>3.2</td>
</tr>
<tr>
<td></td>
<td>38.4</td>
</tr>
</tbody>
</table>

A slight necking is observed, and then a similar fracture as in T6-L2. The stress-strain curve has a distinct yield point and a slight hardening.
A.1.8 T8-L1, $\dot{\varepsilon} = 10^{-2}$ s$^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>420 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>22 MPa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>b [mm]</th>
<th>t [mm]</th>
<th>A [mm$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial dimensions</td>
<td>12.0</td>
<td>3.1</td>
</tr>
<tr>
<td>After test</td>
<td>8.7</td>
<td>2.3</td>
</tr>
</tbody>
</table>

A last specimen from the first plate at this strain rate. Has a similar response as T2-L1. Withstands large strains and does not fracture.
A.1.9  T9-T2, $\dot{\varepsilon} = 10^{-2} \text{ s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Yield stress</td>
</tr>
<tr>
<td>$b$ [mm]</td>
</tr>
<tr>
<td>Initial dimensions</td>
</tr>
<tr>
<td>After test</td>
</tr>
</tbody>
</table>

A specimen from the transversal direction of the second plate. Uniform deformation. Distinct yield point and more hardening than the longitudinal specimens.
A.1.10  T10-T2, $\dot{\varepsilon} = 10^{-2}$ s$^{-1}$

A specimen from the transversal direction of the second plate. Uniform deformation. Distinct yield point and more hardening than the longitudinal specimens and the previous transversal specimen T9-T2.

<table>
<thead>
<tr>
<th>Material data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Yield stress</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$b$ [mm]</th>
<th>$t$ [mm]</th>
<th>$A$ [mm$^2$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial dimensions</td>
<td>12.0</td>
<td>3.1</td>
</tr>
<tr>
<td>After test</td>
<td>9.0</td>
<td>2.3</td>
</tr>
</tbody>
</table>

Graphs showing force vs. displacement, true stress vs. logarithmic strain, and contraction ratio vs. longitudinal logarithmic strain.
A.1.11 T11-L1, $\dot{\varepsilon} = 10^{-1} \text{ s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>360 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>25 MPa</td>
</tr>
<tr>
<td>$b$ [mm]</td>
<td>$t$ [mm]</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>12.0</td>
</tr>
<tr>
<td>After test</td>
<td>11.6</td>
</tr>
</tbody>
</table>

The first specimen from the highest strain rate. The fracture occurred after 16 seconds. Necking tendency near the fracture. The stress-strain diagram shows a distinct yield point and almost no hardening. Low $\rho$ value
Material data

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus</td>
<td>480 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>25 MPa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>b [mm]</th>
<th>t [mm]</th>
<th>A [mm²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial dimensions</td>
<td>12.0</td>
<td>3.1</td>
</tr>
<tr>
<td>After test</td>
<td>11.7</td>
<td>3.0</td>
</tr>
</tbody>
</table>

Necking tendency near the fracture. The stress-strain diagram shows a distinct yield point and no hardening.
A.1.13 T13-T2, $\dot{\varepsilon} = 10^{-1}$ s$^{-1}$

The specimen is from the transversal direction of plate two. Local necking occurs close to the fracture. The tip in the gauge area of the lower part indicates that the core has fractured a bit earlier than the outer layer. The stress-strain diagram shows a distinct yield point and hardening. More hardening in this than in the two from the longitudinal direction.
A.1.14 T14-T2, $\dot{\varepsilon} = 10^{-1} \text{s}^{-1}$

The specimen is from the transversal direction of plate two. Almost no necking. The stress-strain diagram shows a distinct yield point and a slight hardening. More hardening in this than in the two from the longitudinal direction.
A.1.15  C1, $\dot{\varepsilon} = 10^{-3}$ s$^{-1}$

Material data

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>380 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>21 MPa</td>
</tr>
<tr>
<td>$D$ [mm]  $t$ [mm]</td>
<td>$A = \pi r^2$ [mm$^2$]</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>6.1 3.2 29.2</td>
</tr>
<tr>
<td>After test</td>
<td>7.3 2.2 41.9</td>
</tr>
</tbody>
</table>

The compression tests were stopped at approximately $\varepsilon = 0.5$. The specimen deforms without barreling. There is no distinct yield point and only a slight hardening.
A.1.16  \( C2, \dot{\varepsilon} = 10^{-3} \text{ s}^{-1} \)

The compression tests were stopped at approximately \( \varepsilon = 0.5 \). The specimen deforms without barreling. There is no distinct yield point and only a slight hardening.

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>380 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>21 MPa</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>( D \text{ [mm]} )</td>
<td>( t \text{ [mm]} )</td>
</tr>
<tr>
<td>Initial dimensions</td>
<td>6.0</td>
</tr>
<tr>
<td>After test</td>
<td>7.3</td>
</tr>
</tbody>
</table>
A.1.17  C3, $\dot{\varepsilon} = 10^{-2} \text{s}^{-1}$

<table>
<thead>
<tr>
<th>Material data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus</td>
<td>470 MPa</td>
</tr>
<tr>
<td>Yield stress</td>
<td>25 MPa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Initial dimensions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$D$ [mm]</td>
<td>6.0</td>
</tr>
<tr>
<td>$t$ [mm]</td>
<td>3.2</td>
</tr>
<tr>
<td>$A = \pi r^2$ [mm$^2$]</td>
<td></td>
</tr>
<tr>
<td>After test</td>
<td></td>
</tr>
<tr>
<td>$D$ [mm]</td>
<td>7.3</td>
</tr>
<tr>
<td>$t$ [mm]</td>
<td>2.1</td>
</tr>
<tr>
<td>$A = \pi r^2$ [mm$^2$]</td>
<td></td>
</tr>
</tbody>
</table>

This test has significantly higher yield stress than the two previous test. The difference in yield stress is larger than the expected difference due to the increased strain rate. The stress-strain curve shows no softening and a 2-4 MPa hardening.
A.2 Plate with centric hole
A.2.1 D1V1, v=0.05 mm/s

The majority of the deformation took place in the gauges. The deformation in each gauge is a bit different, indicating some effect of the asymmetric placement of the hole. The specimen did not fully fracture and the outer skin was still intact some places when the specimen was removed from the test machine.

<table>
<thead>
<tr>
<th>Test specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Approx. $\varepsilon_{\text{gauge}}[\text{s}^{-1}]$</td>
</tr>
<tr>
<td>$b_{t,1}$ [mm]</td>
</tr>
<tr>
<td>Initial dimensions</td>
</tr>
</tbody>
</table>
A.2.2 D2V1, v=0.05 mm/s

The majority of the deformation took place in the gauges. The deformation in each gauge is a bit different, indicating some effect of the asymmetric placement of the hole. The specimen did not fully fracture and the outer skin was still intact some places when the specimen was removed from the test machine. It the tips on the gauges of the top piece indicates that the core has fractured before the outer skin.
A.2.3  D1V2, v=0.5 mm/s

<table>
<thead>
<tr>
<th>Test specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Approx. $\dot{\varepsilon}_{gauge}$ [s$^{-1}$]</td>
</tr>
<tr>
<td>$b_{1,1}$ [mm]</td>
</tr>
<tr>
<td>15.1</td>
</tr>
</tbody>
</table>

Very local deformation and fracture. The gauges deform only 1.25 mm. Except for the fracture in the gauge area, is the hole more or less unaffected. The core and the outer skin have fractured simultaneously. The transversal deformation in the gauges is quite equal.
A.2.4 D2V2, \(v=0.5\) mm/s

Highest peak load. A local fracture occurs. The core and the outer skin have fractured simultaneously. The transversal deformation in the gauges is different confirming the asymmetrical placement of the hole(cf. the table).

<table>
<thead>
<tr>
<th>Test specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Approx. (\varepsilon_{\text{gauge}}) [s(^{-1})]</td>
</tr>
<tr>
<td>Initial dimensions</td>
</tr>
<tr>
<td>20.5</td>
</tr>
</tbody>
</table>
A.3 Drop tower
A.3.1 DT1, E~50 J

<table>
<thead>
<tr>
<th>Test specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spear mass [kg]</td>
</tr>
<tr>
<td>Impact velocity [m/s]</td>
</tr>
<tr>
<td>Drop height [m]</td>
</tr>
</tbody>
</table>

The specimen absorbed the impact and deformed as shown in the pictures. The ring where the plate was clamped is shown in the middle picture. The spear bounced back a bit after the impact thus the negative speed at the end of the velocity-time plot.
A.3.2 DT2, E~100 J

The specimen fractured and the impact area was nearly parted from the rest of the specimen by a plug-like fracture mechanism.

<table>
<thead>
<tr>
<th>Test specifications</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Spear mass [kg]</td>
<td>5.045</td>
</tr>
<tr>
<td>Impact velocity [m/s]</td>
<td>6.26</td>
</tr>
<tr>
<td>Drop height [m]</td>
<td>2</td>
</tr>
</tbody>
</table>
A.3.3 DT3, E~100 J

The specimen fractured and the impact area was nearly parted from the rest of the specimen by a plug-like fracture mechanism.

<table>
<thead>
<tr>
<th>Test specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spear mass</td>
</tr>
<tr>
<td>Impact velocity</td>
</tr>
<tr>
<td>Drop height</td>
</tr>
</tbody>
</table>
A.3.4 DT4, E~75 J

The specimen fractured and the impact area was nearly parted from the rest of the specimen by a plug-like fracture mechanism. It should be noted that the velocity ends up with a negative value at the end. This is a little strange since the specimen fractures. The reason is not known.

<table>
<thead>
<tr>
<th>Test specifications</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Spear mass [kg]</td>
<td>10.045</td>
</tr>
<tr>
<td>Impact velocity [m/s]</td>
<td>3.86</td>
</tr>
<tr>
<td>Drop height [m]</td>
<td>0.76</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Deformation [mm]</th>
<th>0</th>
<th>4</th>
<th>8</th>
<th>12</th>
<th>16</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time [ms]</td>
<td>0</td>
<td>4</td>
<td>8</td>
<td>12</td>
<td>16</td>
<td>20</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Force [N]</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time [ms]</td>
<td>0</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Deformation [mm]</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time [ms]</td>
<td>0</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>4</td>
<td>5</td>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Velocity [m/s]</th>
<th>-0.5</th>
<th>0</th>
<th>0.5</th>
<th>1</th>
<th>1.5</th>
<th>2</th>
<th>2.5</th>
<th>3</th>
<th>3.5</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time [ms]</td>
<td>-0.5</td>
<td>0</td>
<td>0.5</td>
<td>1</td>
<td>1.5</td>
<td>2</td>
<td>2.5</td>
<td>3</td>
<td>3.5</td>
<td>4</td>
</tr>
</tbody>
</table>
A.3.5 DT5, E~60 J

The specimen fractured and the impact area was nearly parted from the rest of the specimen by a plug-like fracture mechanism.
A.3.6 DT6, E~55 J

The specimen absorbs the impact and recoils the spear

<table>
<thead>
<tr>
<th>Test specifications</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Spear mass [kg]</td>
<td>5.045</td>
</tr>
<tr>
<td>Impact velocity [m/s]</td>
<td>4.67</td>
</tr>
<tr>
<td>Drop height [m]</td>
<td>1.11</td>
</tr>
</tbody>
</table>

The specimen absorbs the impact and recoils the spear
Appendix B

Matlab scripts

The Matlab scripts are added in the appendix in order to show the derivation of some of the results mentioned in the thesis. The scripts are described so that it should be possible to follow them without knowing the code beforehand. Some basic knowledge is, however, needed in order to understand them.

The scripts are arranged by the appearance in the thesis.
B.1 Finding the critical cross-section of the material tests

This script finds the cross-section with the highest strain values along the specimen length. It uses the strain matrices from 7D as input and returns the longitudinal and transversal strain in separate vectors.
clear all           % Clearing the memory
clc                 % and the command window

file = 'T2';        % Defines the test that is to be analyzed
testnumber = 2;     % Defines which Excel-sheet the data should be
                     % written to.

load('-mat',file)   % Loads the ".mat" file with name "file".

names = load('-mat',file); % Lists the names and the dimension
                          % of all the matrices in file.mat into the
                          % vector "names"

MatNum = fieldnames(names, '-full'); % Gets the names of the matrices
                                      % from "names"

NumFrame = (length(MatNum))/2;       % Finds the number of matrice-names
                                      % in MatNum and divides it by 2.  
                                      % This is because half the matrices
                                      % are the longitudinal strain the
                                      % other half the transversal strain

dim = size(elnmaxi001);            % Finds the dimension of the
                                    % matrices.

% The following for loops read an "elnmaxiXXX/elnminiXXX" matrix and
% register it as the matrix Ml/Mt. The next line then finds the mean of
% each row of Ml and puts it into the matrix meanRVl. meanRVl has the same
% amount of columns as there are elnmaxi matrices. The next line
% then constructs a vector containing the mean of each row of meanRVl and
% thus a second mean over all the rows of Ml.

for i = 1:NumFrame
    for j = 1:dim(1,1)
        Ml = eval(['elnmaxi%.3d',i]);
        meanRVl(j,i) = mean(Ml(j,:));
        maxMeanl(j,1) = nanmean(meanRVl(j,:));
    end
end

for m = 1:NumFrame
    for n = 1:dim(1,1)
        Mt = eval(['elnmini%.3d',m]);
        meanRVt(n,m) = abs(mean(Mt(n,:))); 
        maxMeant(n,1) = abs(nanmean(meanRVt(n,:))); 
        counter(n) = n;
    end
end

[row col] = find(meanRVl==max(max(meanRVl))));  % Finds the row and column
                                                 % containing the maximum
                                                 % mean strain value of
                                                 % all the means in meanRVl
                                                 % 37
\[ r = \text{find}(\text{maxMean}_1 = \text{max}(\text{max}(\text{maxMean}_1))); \quad \% \text{Finds the cell containing the maximum mean strain value} \]
\[ [\text{rowt colt}] = \text{find}(\text{meanRV}_t = \text{max}(\text{max}(\text{meanRV}_t))); \quad \% \text{The same as above but for the transversal direction.} \]
\[ rt = \text{find}(\text{maxMeant} = \text{max}(\text{max}(\text{maxMeant}))); \]
\[ \text{dimmRV} = \text{size}(\text{meanRV}_l); \quad \% \text{Finds the dimension of meanRV}_l \]
\[ \text{for } k = 1:\text{dimmRV}(1,2), \quad \% \text{Reads the mean longitudinal strain along the row rt, which is the row with the highest mean strain.} \]
\[ \quad \text{epsmax}(k) = \text{meanRV}_l(r,k); \]
\[ \quad \text{epsmin}(k) = \text{meanRV}_t(rt,k); \]
\[ \text{end} \]
\[ h1 = \text{figure}; \quad \% \text{Plots the curves shown in the figure below} \]
\[ \text{subplot}(2,1,1) \]
\[ \text{title}('\text{strains}') \]
\[ \text{plot}(\text{loops}, \text{epsmax}, 'b-') \]
\[ \text{hold on} \]
\[ \text{plot}(\text{loops}, \text{epsmin}, 'r-') \]
\[ \text{axis}([0 300 0 1.3]) \]
\[ \text{xlabel}('\text{Loops}') \]
\[ \text{ylabel}('\text{Strain}') \]
\[ \text{subplot}(2,1,2) \]
\[ \text{title}('\text{Critical Cross-Section}') \]
\[ \text{plot}(\text{maxMean}_1, -\text{counter}, 'b-') \]
\[ \text{hold on} \]
\[ \text{plot}(\text{maxMeant}, j-\text{counter}, 'r-') \]
\[ \text{axis}([0 1.1*\text{max}(\text{maxMean}_1) -\text{dimmRV}(1,1) 0]) \]
\[ \text{xlabel}('\text{Mean strains}') \]
\[ \text{ylabel}('(-)\text{Critical row}') \]
\[ \% \text{The following lines write the vectors epsmax and epsmin to the Excel sheet "DIC strain.xlsx"} \]
\[ \text{filename} = '\text{DIC Strain.xlsx}'; \]
\[ \text{xlswrite}(\text{filename}, \text{epsmax}', \text{testnumber}, 'A2') \]
\[ \text{xlswrite}(\text{filename}, \text{epsmin}', \text{testnumber}, 'C2') \]
Figure B.1: The plot produced by the script above. The critical row is row 29 for the test analysed in the code.
B.2 The true stress-strain curve for the compression tests

The following script was applied to calculate the logarithmic strains and the true stress in the compression tests. The code uses the pictures taken by the rig camera to define the geometry of the specimen throughout the test. At the end is the result written to a Excel file so that it can be easily applied in other analyses. The script has to be located in the same folder as the pictures in order to work.
clear all % Clearing the memory
clc % and the command window

% Reading all the picture names into the matrix NumBmp
NumBmp = dir('PP-1C-T15_2012_02_28_15h35m01s_Cam1_Frame_*.bmp');
% delFrame defines the frame in which the deformation starts
delFrame = 0;
% A difference needed for the for loops
NetNumFrame = length(NumBmp) - delFrame;
% The next two lines defines which sheet of the Excel file with the name
% filename the data should be written to. This allows for efficient
% processing of multiple tests.
SheetNumber = 1;
filename = 'CompData.xlsx';

% Predefining the matrices
fWProw = zeros(1,NetNumFrame);
lWProw = zeros(1,NetNumFrame);
mWProw = zeros(1,NetNumFrame);
fWPcol = zeros(1,NetNumFrame);
lWPcol = zeros(1,NetNumFrame);
Hdef = zeros(1,NetNumFrame);
Ddef = zeros(1,NetNumFrame);
t = zeros(1,NetNumFrame);

% The following for loop loads one of the pictures taken by the camera in
% the test rig. It then turns it into a picture containing only black and
% white based on a level defining which colors that are to be white and
% which that are to be black. A picture in black and white is the same as a
% matrix containing only ones (white) and zeroes (black). The transitions
% from black to white are used to define the geometry of the specimen
% at any point in time.

for i = 1:NetNumFrame
    level = graythresh(imread(NumBmp(i).name)); % Defining the point on the
    bw = im2bw(imread(NumBmp(i).name),level); % greyscale where black and
    % white should be parted
    wb = 1-bw; % Defines an version of the picture where black and white
    % are switched
    % The next two lines find the rows that define the top and bottom of
    % the specimen
    fWProw(i) = find(bw(:,2448),1,'first');
lWProw(i) = find(bw(:,2448),1,'last');
mWProw(i) = round((fWProw(i)+lWProw(i))/2); % Defines the midpoint of
    % the specimen
    % These lines find the vertical sides of the specimen
    fWPcol(i) = find(wb(mWProw(i),:),1,'first');
lWPcol(i) = find(wb(mWProw(i),:),1,'last');
These lines define the difference between the rows and columns marking the transitions from black to white. This difference gives the height and diameter of the specimen throughout the course of the test.

\[
\begin{align*}
H_{\text{def}}(i) &= \text{lWProw}(i) - \text{fWProw}(i); \\
D_{\text{def}}(i) &= \text{lWPcol}(i) - \text{fWPcol}(i);
\end{align*}
\]

\[d_0 = 6.09;\]

% Uses the diameter of specimen to define the ratio of millimetres/pixel.

\[
\text{mmPRpix} = \frac{d_0}{D_{\text{def}}(1)};
\]

\[
\begin{align*}
H_{\text{defmm}} &= H_{\text{def}} \times \text{mmPRpix}; \\
D_{\text{defmm}} &= D_{\text{def}} \times \text{mmPRpix};
\end{align*}
\]

% Defining the Excel file that the force and displacement data are to be read from.

\[
\text{file} = '{...\BMP\Comp.xls}';
\]

% Defining the area and height at t=0

\[
A_0 = \frac{(\pi() \times d_0^2)}{4}; \\
L_0 = 3.2;
\]

% Defining the displacement and force vectors

\[
U = \text{xlsread(file,SheetNumber,'B2:B3000')};
\]

\[
\text{Ut} = D_{\text{defmm}} - D_{\text{defmm}}(1);
\]

\[
F = 1000 \times \text{xlsread(file,SheetNumber,'C2:C3000')};
\]

% Predefining vectors for increased performance

\[
\begin{align*}
A &= \text{zeros}(\text{NetNumFrame},1); \\
\text{stress} &= \text{zeros}(\text{NetNumFrame},1); \\
\text{epsPIC} &= \text{zeros}(\text{NetNumFrame},1);
\end{align*}
\]

% Calculating the longitudinal and transversal logarithmic strains and the true stress

\[
\begin{align*}
\text{for} \ j = 1:\text{NetNumFrame} \\
\quad \text{epsLOG}(j) &= \text{abs}(\log(1 - (U(j)/L_0))); \\
\quad \text{epsTRAN}(j) &= \text{abs}(\log(1 - (\text{Ut}(j)/d_0))); \\
\quad A(j) &= (\pi() \times D_{\text{defmm}}(j)^2)/4; \\
\quad \text{stress}(j) &= \text{abs}(F(j)/A(j));
\end{align*}
\]

% Writing the vectors to the Excel file named "filename" (defined earlier)

\[
\text{xlswrite(filename,epsLOG',SheetNumber,'A2')} \\
\text{xlswrite(filename,eps,1,'C2')} \\
\text{xlswrite(filename,stress,SheetNumber,'E2')}
\]
B.3 Calculation of the yields stresses

The yield stresses were calculated using Considère’s criterion. This is stated as:

\[
\frac{d\sigma_{\text{true}}}{d\varepsilon_{\text{log}}} = \sigma_{\text{true}} \tag{B.1}
\]

A graphical representation is shown in Figure B.2a.

(a) The yield stress calculation of test T3-L1
(b) The relation between the yield stresses and the strain rate. (+) are tension tests and (×) are compression tests

Figure B.2: The method used to identify the yield stresses and the yield stresses in relation to the strain rate

The Matlab script calculates the yield stresses for a single test. The needed inputs are the tension test number and how much of the stress-strain curve that is to be used in the calculation. The latter argument is defined by defining the last cell of the strain and stress vector that are to be used in the calculations. To find the coordinates of the crossing curves, a function made by the German Matlab user Sebastian Hölz is applied. This function is named “Curve Intersect 2” and is applied as a user defined function in Matlab. The function is shown at the end of this section.

The code was modified to work with the compression tests and with series of tests. These codes are not shown here.
clear all  % Clearing the memory
clc  % and the command window

s=1; c=20;  % s equals the testnumber and c is the endcell of the
            % strain vector and the stress vector.

% The two next lines reads the longitudinal logarithmic strain and the true
% stress from an Excel-file
epsRead = epsL(s);
stressRead = StressCalc(s);

% These lines adjust the length of the stress and strain vector
eps = epsRead(1:min(length(stressRead),length(epsRead)));
stress = stressRead(1:min(length(stressRead),length(epsRead)));

rows = find(eps<0.75,1,'last');  % Defines an upper limit for the strain
                                      % values that are to be used.

% Predefining the vectors for the for loop
consid = zeros(rows,1);
deps = zeros(rows,1);
dstress = zeros(rows,1);

% The following loop calculates the vector representing \( \frac{d(\epsilon)}{d(\sigma)} \)
for i = 2:min(length(consid))
    deps(i-1) = mean(eps(i))-mean(eps(i-1));
    dstress(i-1) = mean(stress(i))-mean(stress(i-1));
    consid(i-1) = dstress(i-1)/deps(i-1);
end

dercell = c;  % Defining c to be the endcell

% The following line applies the function Curve Intersect 2
[x y] = curveintersect(eps,stress,eps(1:endcell),consid(1:endcell));

% Puts the result of the function curveintersect into the vector coord
coord = [x y];

% Sets sigmaY equal to the y-coordinate found from curveintersect
sigmaY = coord(1,2);
function [x,y]=curveintersect(varargin)
% Curve Intersections.
% [X,Y]=CURVEINTERSECT(H1,H2) or [X,Y]=CURVEINTERSECT([H1 H2]) finds the
% intersection points of the two curves on the X-Y plane identified
% by the line or lineseries object handles H1 and H2.
% [X,Y]=CURVEINTERSECT(X1,Y1,X2,Y2) finds the intersection points of the
% two curves described by the vector data pairs (X1,Y1) and (X2,Y2).
% X and Y are empty if no intersection exists.
% Example
% ------
% x1=rand(10,1); y1=rand(10,1); x2=rand(10,1); y2=rand(10,1);
% [x,y]=curveintersect(x1,y1,x2,y2);
% plot(x1,y1,'k',x2,y2,'b',x,y,'ro')
% (%Original Version (-> curveintersect_local)
% %---------------------------------------
% % D.C. Hanselman, University of Maine, Orono, ME 04469
% % Mastering MATLAB 7
% % 2005-01-06
% % (%Improved Version (-> this function)
% % -----------------------------------
% % S. Hölz, TU Berlin, Germany
% % v 1.0: October 2005
% % v 1.1: April 2006 Fixed some minor bugs in function 'mminvinterp'

x=[]; y=[];
[x1,y1,x2,y2]=local_parseinputs(varargin{:});
ind_x1=sign(diff(x1)); ind_x2=sign(diff(x2));

ind1=1;
while ind1<length(x1)
    ind_max = ind1+min(find(ind_x1(ind1:end)-=ind_x1(ind1)))-1;
    if isempty(ind_max) | ind_max==ind1; ind_max=length(x1); end
    ind1=ind1:ind_max;
end

ind2=1;
while ind2<length(x2)
    ind_max = ind2+min(find(ind_x2(ind2:end)-=ind_x2(ind2)))-1;
    if isempty(ind_max) | ind_max==ind2; ind_max=length(x2); end
    ind2=ind2:ind_max;

    % Fallunterscheidung
    if ind_x1(ind1(1))==0 & ind_x2(ind2(1))~=0
        x_loc=x1(ind1(1));
        y_loc=interp1(x2(ind2),y2(ind2),x_loc);
        if ~((y_loc>=min(y1(ind1)) & y_loc<=max(y1(ind1)))); y_loc=[];
    end
    elseif ind_x2(ind2(1))==0 & ind_x1(ind1(1))~=0
        x_loc=x2(ind2(1));
        y_loc=interp1(x1(ind1),y1(ind1),x_loc);
        if ~((y_loc>=min(y2(ind2)) & y_loc<=max(y2(ind2)))); y_loc=[];
    end

    % ----------------------------------------------
    function [x,y]=curveintersect_local(x1,y1,x2,y2)
    if ~isequal(x1,x2)
        xx=unique([x1 x2]); % get unique data points
        xx=xx(xx>=max(min(x1),min(x2)) & xx<=min(max(x1),max(x2)));
        if numel(xx)<2
            x=[];
            y=[];
            return
        end
        x=[x; xx(:)];
        y=[y; yy(:)];
        ind2=ind2(end);
    end
    ind1=ind1(end);
end

% ----------------------------------------------
yy = interp1(x1, y1, xx) - interp1(x2, y2, xx);

else
    xx = x1;
    yy = y1 - y2;
end

x = mminvinterp(xx, yy, 0); % find zero crossings of difference
if ~isempty(x)
    y = interp1(x1, y1, x);
else
    x = [];
    y = [];
end

%-----------------------------------------
%-----------------------------------------
function [xo, yo] = mminvinterp(x, y, yo)
% MMINVINTERP 1-D Inverse Interpolation. From the text "Mastering MATLAB 7"
% [Xo, Yo] = MMINVINTERP(X, Y, Yo) linearly interpolates the vector Y to find
% the scalar value Yo and returns all corresponding values Xo interpolated
% from the X vector. Xo is empty if no crossings are found. For
% convenience, the output Yo is simply the scalar input Yo replicated so
% that size(Xo) = size(Yo).
% If Y maps uniquely into X, use INTERP1(Y, X, Yo) instead.
% See also INTERP1.

if nargin ~= 3
    error('Three Input Arguments Required."
end

n = numel(y);
if ~isequal(n, numel(x))
    error('X and Y Must have the Same Number of Elements.')
end
if ~isscalar(yo)
    error('Yo Must be a Scalar.')
end

x = x(:); % stretch input vectors into column vectors
y = y(:);

if yo < min(y) || yo > max(y) % quick exit if no values exist
    xo = [];
    yo = [];
else % find the desired points
    below = y < yo; % True where below yo
    above = y > yo; % True where above yo
    on = y == yo; % True where on yo
    kth = (below(1:n-1) & above(2:n)) | (above(1:n-1) & below(2:n)); % point k
    kp1 = [false; kth]; % point k+1
    xo = [];
end

% distance between x(k+1) and x(k)
if any(kth);
    alpha = (yo - y(kth))./(y(kp1)-y(kth));
    xo = alpha.*(x(kp1)-x(kth)) + x(kth);
end
xo = sort([xo; x(on)]);
% add points, which are directly on line
yo = repmat(yo,size(xo));
% duplicate yo to match xo points found
end
%---------------------------------------------------------------------
function [x1,y1,x2,y2]=local_parseinputs(varargin)
if nargin==1 % [X,Y]=CURVEINTERSECT([H1 H2])
    arg=varargin{1};
    if numel(arg)==2 && ...
        all(ishandle(arg)) && all(strcmp(get(arg,'type'),'line'))
        data=get(arg,{'XData','YData'});
        [x1,x2,y1,y2]=deal(data{:});
    else
        error('Input Must Contain Two Handles to Line Objects.');
    end
elseif nargin==2 % [X,Y]=CURVEINTERSECT(H1,H2)
    arg1=varargin{1};
    arg2=varargin{2};
    if numel(arg1)==1 && ishandle(arg1) && strcmp(get(arg1,'type'),'line')... && numel(arg2)==1 && ishandle(arg2) && strcmp(get(arg2,'type'),'line')
        data=get([arg1;arg2],{'XData','YData'});
        [x1,x2,y1,y2]=deal(data{:});
    else
        error('Input Must Contain Two Handles to Line Objects.');
    end
elseif nargin==4
    [x1,y1,x2,y2]=deal(varargin{:});
    if ~isequal(numel(x1),numel(y1))
        error('X1 and Y1 Must Contain the Same Number of Elements.');
    elseif ~isequal(numel(x2),numel(y2))
        error('X2 and Y2 Must Contain the Same Number of Elements.');
    end
    x1=reshape(x1,1,[]); % make data into rows
    x2=reshape(x2,1,[]);
    y1=reshape(y1,1,[]);
    y2=reshape(y2,1,[]);
else
    error('Incorrect Number of Input Arguments.');
end
if min(x1)>max(x2) | min(x2)>max(x1) | min(y2)>max(y1) | min(y1)>max(y2) % Polygons can not have intersections
    x1=[]; y1=[]; x2=[]; y2=[]; return
end
if numel(x1)<2 || numel(x2)<2 || numel(y1)<2 || numel(y2)<2
    error('At Least Two Data Points are Required for Each Curve.');
end
B.4 Plate with centric hole

B.4.1 The transversal displacement in the gauges

This script calculates the transversal deformation of the gauges by using a similar approach as for the compression tests. The script reads the pictures from the test-rig camera and turns them into ones and zeroes (black and white). The transitions between black and white will change from picture to picture. By storing the position of this transition from each picture the displacements can be represented.

Due to noise in the pictures taken during the tests are the deformations represented by exponential regression functions. This introduces a small deviation in the displacements but it also makes them much easier to compare.

The script has to be located in the same folder as the pictures in order to work.

Figure B.3: The plot produced by the script. The plot shows the transversal deformation of the test D2V1 (the uneven curve) and the curve fitted to the data (the smooth curve).
clear all % Clear the memory
clc       % Clear the command window

% These five lines define the input and output files used in the script.
% It starts with defining the sheet of the Excel file "filename.xlsx" that
% the data should be written to. Then is the name of the pictures defined
% in the vector NumBmp. The last two lines define the applied time vector.
SheetNumber = 2;
filename = 'dx.xlsx';
NumBmp = dir('PPV-02_2012_03_19_13h07m37s_Cam1_Frame_*.bmp');
timeread = dlmread('PPV-02.txt','	',1,0);
time = timeread(:,1);
delFrame = 100; % Defines how many pictures that are to removed at the end
                % of the picture series
StartFrame = 1; % Defines the starting picture
EndFrame = length(NumBmp)-delFrame; % Defines the difference between start % and end

mmPp = 19.76/(1481-1100); % The mm to pixel ratio. It is calculated with
        % respect to the diameter of the hole.
        % -> diameter in mm/number of pixels across the diameter

% The if loop adjusts the number of pictures to be processed based on how
% many pictures that are removed in the beginning and the end.
if StartFrame > 1
    FrameDiff = EndFrame - StartFrame;
else
    FrameDiff = EndFrame;
end

level = 0.25; % Defines the greyscale limit

% The next two lines define the size of the pictures by converting a
% picture into a matrix and finding the dimension of the matrix
dimBW = im2bw(imread(NumBmp(50+(StartFrame-1)).name),level);
dim = size(dimBW);

% StartRow and EndRow define the rows that are located more or less at the
% top and at the bottom of the gauges in the picture, i.e. they give the
% location of the gauges in the picture.
StartRow = 630;
EndRow = 1500;

RowDiff = EndRow-StartRow; % Difference used in the for loop

for i = StartFrame:FrameDiff
    % Reads the picture
    bw = im2bw(imread(NumBmp(i+(StartFrame-1)).name),level);
    for j = 1:1:RowDiff
        % The next four lines of code define the transitions from black to
        % white in the gauge areas. The two upper lines define one
        % gauge, the two below define the other.
        fWPcolII = find(bw(StartRow+j,670:(dim(2)/2)),1,'first');
lWPcolII = find(bw(StartRow+j,670:(dim(2)/2)),1,'last');

fWPcolVI = find(bw(StartRow+j,(dim(2)/2):1840),1,'first');
lWPcolVI = find(bw(StartRow+j,(dim(2)/2):1840),1,'last');

% The difference between the transition points is found
coldiffII(j) = lWPcolII - fWPcolII;
coldiffII(~coldiffII) = inf;

coldiffVI(j) = lWPcolVI - fWPcolVI;
coldiffVI(~coldiffVI) = inf;

end
% The deformation is stored as the minimum distance between transitions
% for every picture.
dxIIpix(i-(StartFrame-1))= min(coldiffII);
dxVIpix(i-(StartFrame-1))= min(coldiffVI);

end

% Defines the length of the time vector.
t = time(1:length(dxIIpix));
dxII = mmPp*(dxIIpix(1)-dxIIpix); % Converts the deformations from pixels
dxVI = mmPp*(dxVIpix(1)-dxVIpix); % to millimetres

tdcII = polyfit(t,dxII',2); % These four lines find an exponential function
tdcVI = polyfit(t,dxVI',2); % that fits the deformation curves

dxIIfit = polyval(tdcII,t); %
dxVIfit = polyval(tdcVI,t);

% Plots a figure of the deformations and the fitted curve
h1 = figure;
hold on
grid on
plot(t,dxII)
plot(t,dxIIfit)
plot(t,dxVI,'r-')
plot(t,dxVIfit,'r-')

% These last three lines write the resulting vectors to the defined Excel file
xlswrite(filename,t,SheetNumber,'A3')
xlswrite(filename,dxIIfit,SheetNumber,'C3')
xlswrite(filename,dxVIfit,SheetNumber,'E3')
B.4.2 Identifying the correct deformation rate

This script finds the correct deformation rate based on strain data from the DIC software. The input is longitudinal strain matrices and the output is an approximated strain and deformation rate for the gauge of the analysed test.

Figure B.4: Calculation of the strain and deformation rate of test D2V1

Figure B.4 shows the curves produced by the script. The strain-time plot is used to define the range where the linear regression should be made. This means that a range including the strains from t=0 till t=end will yield a near straight line from the origin to approximately $\varepsilon = 1.2$, t=450 s. Such a curve will not represent the correct strain rate. The curve shown in Figure B.4a is adjusted so that the strain rate is more or less correct.
clear all  % Clear the memory
clc        % Clear the command window

file = 'PPV 1';
testnumber = 1;

mmPp = 29.85/(1507-935);  % The mm to pixel ratio. It is calculated with
                         % respect to the diameter of the hole.
                         % -> diameter in mm/number of pixels across the diameter
load('-mat',file)  % Loads the ".mat" file with name "file"
names = load('-mat',file);  % Lists the names and the dimension
                           % of all the matrices in file.mat into the
                           % vector "names"

MatNum = fieldnames(names,'-full');  % Gets the names of the matrices
                                        % from "names"

NumFrame = (length(MatNum))/4;  % Finds the number of matrice-names
                                 % in MatNum and divides it by 2.
                                 % This is because half the matrices
                                 % are the longitudinal strain the
                                 % other half the transversal strain

dim = size(dy001);  % Finds the dimension of the
                    % matrices.

% The for loop below finds the longitudinal strain in the cross-section
% with the highest strains. This cross-section is identified by finding the
% mean of each row of elnmaxiXXX matrix and putting this into the matrix
% meanRVl. The next step is to find the means of all the rows in meanRV1
% and putting them into the vector maxMeanl. The cell having the highest
% value corresponds to the row located at the same place as the critical
% cross-section.
for i = 1:NumFrame
    for j = 1:dim(1,1)
        Ml = eval(sprintf('elnmaxi%.3d',i));  % Reads the matrix
        meanRVl(j,i) = nanmean(Ml(j,:));  % Finds the mean of each row
        maxMeanl(j,1) = nanmean(meanRVl(j,:));  % Finds the mean of each
    end  % row in meanRV1
    [r c] = find(maxMeanl==max(max(maxMeanl)));  % Finds the cell containing
    epsL(i) = nanmean(Ml(r,:));  % Defines the longitudinal strain vector
end

% Does a similar operation as the script above but for the longitudinal
% displacements.
for m = 1:NumFrame
    for n = 1:dim(1,1)
        My = eval(sprintf('dy%.3d',m));
        meanRVy(n,m) = mean(My(n,:));
        maxMeany(n,1) = nanmean(meanRVy(n,:));
    end
dyI(m) = mmPp*mean(My(1,:));  % Defines the longitudinal displacement vector
end

% These two lines define a time vector based on "DIC FvsU.xlsx"
tread = xlsread('DIC FvsU.xlsx',testnumber,'A3:A600');
t = tread(1:length(epsL));

% The next six lines find the linear part of the strain-time/displacement-time curve. The last line in each set of three defines the strain/displacement vector in the relevant range.
fcell = find(epsL>0.01,1);
lcell = find(t>245,1);
interereps = epsL(fcell:lcell);

cell = find(dyI>0,1);
lcelly = find(dyI==max(dyI),1);
interedyI = dyI(fcelly:lcelly);

% These four lines find a linear fit(y=ax+b) to the strain-time/displacement-time curve
fitter = fit(t(fcell:lcell),intereps,'poly1');  % Strains
fittery = fit(t(fcelly:lcelly),interdyI,'poly1');  % Displacements

epscoeff = coeffvalues(fitter);  % Puts the coefficients of the linear regression curve into a vector.  
y=ax+b -> [a b]
dycoeff = coeffvalues(fittery);

% These lines get the gradient of the interpolated curve
veps = epscoeff(1);  % [a]
v = dycoeff(1);  % [a]

% Since half the gauge is modelled the gradients are scaled to fit the model
scaled_veps100 = veps*0.5
scaled_v100 = 0.5*v

% Plots the strain-time and the displacement-time curves. The linear range is found by interpolation using these plots.
h1 = figure;
hold on
grid on
plot(t,epsL,'m:');
plot(t(fcell:lcell),intereps,'r-');
plot(t(fcell:lcell),fitter(t(fcell:lcell)))

h2 = figure;
hold on
grid on
plot(t,dyI,'m:');
plot(t(fcelly:lcelly),interdyI,'r-');
plot(t(fcelly:lcelly),fittery(t(fcelly:lcelly)))
B.4.3 The longitudinal displacements

The script input is the longitudinal displacement matrix. The user then have to define two points in the matrix from where the deformation should be stored. The two points chosen in this analysis are:

- 25 millimetre above the circle
- At the “top” of the circle

The points are defined by finding the corresponding coordinates in the matrix “dy001” using the “data cursor” tool.
clear all % Clear the memory
clc       % Clear the command window

file = 'PPV 1'; % Defines the test that is to be analyzed
testnumber = 1; % Defines which Excel-sheet the data should be
                % written to. This makes it easy process mutiple tests with
                % the same script

mmPp = 29.85/(1507-935); % The mm to pixel ratio. It is calculated with
                         % respect to the diameter of the hole.
                         % -> diameter in mm/number of pixels across the diameter

load('-mat',file) % Loads the ".mat" file with name "file".

names = load('-mat',file); % Lists the names and the dimension
                          % of all the matrices in file.mat into the
                          % vector "names"

MatNum = fieldnames(names, '-full'); % Gets the names of the matrices
                                       % from "names"

NumFrame = (length(MatNum))/4; % Finds the number of matrice-names
                                % in MatNum and divides it by 4.
                                % This is because there are four
                                % different categories of matrices
                                % defined in the .mat file.

dim = size(dy001); % Finds the dimension of the
                  % matrices.

imagesc(dy001) % Displays the first matrix so that the points from
               % where the displacement is to be found can be found

% The for loop below loads a longitudinal displacement matrix and then two
% locations in this matrix is defined. These locations agree with the two
% points from where the deformation was to be found.
for m = 1:NumFrame
  for n = 1:dim(1,1)
    My = eval([sprintf('dy%.3d',m)]); % Loads the deformation matrix
  end
  BotCirc(m) = mmPp*mean(My(47,53:63)); % Stores the deformation at the
                                         % top of the circle

  dyI(m) = mmPp*mean(My(1,55:60)); % Stores the deformation 25 mm
                                    % above the circle
  end

% These two lines write the vectors to the Excel sheet "testnumber" in the
% Excel document dy.xlsx
% xlswrite('dy.xlsx',dyI',testnumber,'A2')
% xlswrite('dy.xlsx',BotCirc',testnumber,'C2')
Appendix C

LS-DYNA input files

This appendix contains input files for LS-DYNA. One representative input file is presented for each of the problems that were modelled. All the applied material cards are shown. The part of the geometry file defining the sets and sections are given for the respective analyses.
C.1 Material cards

C.1.1 ppFinal.k, E=400 MPa

$MATERIAL MODEL DEFINITIONS$

*MAT_USER_DEFINED_MATERIAL_MODELS

*MID rho User mat # LMC # Hist var IBULK IG

1 1.000E-04 49 16 50 0 15 16

$IVECT IFAIL ITHERM IHYPER IEOS

0 0 0 0 1 0

$E Poisson eps0 C sigma_T Cr N (lamda_L) alfa

400 0.24 0.001 0.057163 19.3671 3.788897 50.426718 1.062

*$END

C.1.2 ppFinalE700.k, E=700 MPa

$MATERIAL MODEL DEFINITIONS$

*MAT_USER_DEFINED_MATERIAL_MODELS

*MID rho User mat # LMC # Hist var IBULK IG

1 1.000E-04 49 16 50 0 15 16

$IVECT IFAIL ITHERM IHYPER IEOS

0 0 0 0 1 0

$E Poisson eps0 C sigma_T Cr N (lamda_L) alfa

700 0.24 0.001 0.057163 19.3671 3.788897 50.426718 1.062

*$END

58
C.1.3 ppFinalE750.k, E=750 MPa

$ MATERIAL MODEL DEFINITIONS $

*MAT_USER_DEFINED_MATERIAL_MODELS
$ MID rho User mat # LMC # Hist var IBULK IG
  1 1.000E-04 49 16 50 0 15 16
$ IVECT IFAIL IHERM IHYPER IEOS
  0 0 0 1 0
$ E Poisson eps0 C sigma_T Cr N (lamda_L) alfa
  750 0.24 0.001 0.057163 19.3671 3.788897 50.426718 1.062
$ beta kappa sigma_ss H K G
  1.56410 0.00 12.48980 2.19287 480.77 302.4194
*END

C.1.4 ppFinalDTOPT.k, E=700 MPa and C=0.07

$ MATERIAL MODEL DEFINITIONS $

*MAT_USER_DEFINED_MATERIAL_MODELS
$ MID rho User mat # LMC # Hist var IBULK IG
  1 1.000E-09 49 16 50 0 15 16
$ IVECT IFAIL IHERM IHYPER IEOS
  0 0 0 1 0
$ E Poisson eps0 C sigma_T Cr N (lamda_L) alfa
  700 0.24 0.001 0.070000 19.3671 3.788897 50.426718 1.062
$ beta kappa sigma_ss H K G
  1.56410 0.00 12.48980 2.19287 480.77 302.4194
*END
C.1.5  ppFinalDTOPT_075.k, E=700 MPa and C=0.070

MATERIAL MODEL DEFINITIONS

*MAT_USER_DEFINED_MATERIAL_MODELS
$ MID rho User mat # LMC # Hist var IBULK IG
1 1.000E-09 49 16 50 0 15 16
$ IVECT IFAIL ITHERM IHYPER IEOS
0 0 0 0 1 0
$ E Poisson eps0 C sigma_T Cr N (lamda_L) alfa
700 0.24 0.001 0.075000 19.3671 3.788897 50.426718 1.062
$ beta kappa sigma_ss H K G
1.56410 0.00 12.48980 2.19287 448.7179 282.25806
$END

C.1.6  ppFinalDTOPT_08.k, E=700 MPa and C=0.08

MATERIAL MODEL DEFINITIONS

*MAT_USER_DEFINED_MATERIAL_MODELS
$ MID rho User mat # LMC # Hist var IBULK IG
1 1.000E-09 49 16 50 0 15 16
$ IVECT IFAIL ITHERM IHYPER IEOS
0 0 0 0 1 0
$ E Poisson eps0 C sigma_T Cr N (lamda_L) alfa
700 0.24 0.001 0.080000 19.3671 3.788897 50.426718 1.062
$ beta kappa sigma_ss H K G
1.56410 0.00 12.48980 2.19287 448.7179 282.25806
$END
<table>
<thead>
<tr>
<th>$\beta$</th>
<th>$\kappa$</th>
<th>$\sigma_{ss}$</th>
<th>$H$</th>
<th>$K$</th>
<th>$G$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.56410</td>
<td>0.00</td>
<td>12.48980</td>
<td>2.19287</td>
<td>448.7179</td>
<td>282.25806</td>
</tr>
</tbody>
</table>

$\text{---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8}$

*END*
C.2 Finite element analyses of the calibration tests

C.2.1 Input file

```
$# LS-DYNA Keyword file created by LS-PrePost 3.2 - 16Jan2012(01:28)
$# Created on Mar-22-2012 (12:50:14)
*KEYWORD
*TITLE
$# title
LS-DYNA keyword deck by LS-PrePost
*INCLUDE
ppFinal.k
*INCLUDE
geomHalf.k
*CONTROL_ENERGY
  2  2  1  1
*CONTROL_TERMINATION
  200.00000  0  0.000  0.000  0.000
*CONTROL_TIMESTEP
  0.000  0.500000  0  0.000  0.000  0  0  0
  0.000  0  0
*DATABASE_BINARY_D3PLOT
  4.000000
*DATABASE_EXTENT_BINARY
  59  0  3  1  0  0  0  0
  0  0  4  0  0  0  0
$--------1--------2--------3--------4--------5--------6--------7--------8
*DATABASE_GLSTAT
  0.400000  0  0  0
*DATABASE_MATSUM
  0.400000  0  0  0
*DATABASE_NODOUT
  0.400000  0  0  0  0.000  0
*DATABASE_ELOUT
  0.400000  0  0  0  0.000  0
*DATABASE_SECFORC
  0.400000  0  0  0
*DATABASE_SPCFORC
  0.400000  0  0  0
*DATABASE_HISTORY_SOLID_SET
  1  0  0  0  0  0  0  0
```

62
*DATABASE_CROSS_SECTION_SET_ID
  iReadSet
    3  1  0  0  0  0  0  0
*DATABASE_HISTORY_NODE_SET
  4  0  0  0  0  0  0  0
*BOUNDARY_PRESCRIBED_MOTION_SET_ID
  OMoving
    2  1  0  1  0.330000  01.0000E+28  0.000
*BOUNDARY_SPC_SET_BIRTH_DEATH
  1  0  1  1  1  0  0  0
    0.0001.0000E+20
*PART
  
    2  1  1  0  0  0  0  0
*SECTION_SOLID_TITLE
  Section ONE
  1  2  0
*DEFINE_CURVE_SMOOTH_TITLE
  Smoooooooooth
  1  0  500.00000  0.000  0.000  1.0000E-4  1.000000
*END
C.2.2 Excerpt from the geometry file geomHalf.k

```
$# LS-DYNA Keyword file created by LS-PrePost 3.2 - 16Jan2012(01:28)
$# Created on May-19-2012 (14:18:37)
*KEYWORD
*TITLE
$# title
LS-DYNA keyword deck by LS-PrePost
*BOUNDARY_SPC_SET_BIRTH_DEATH
  5 0 0 1 0 1 1 1
  0.0001.0000E+20
*SET_NODE_LIST_TITLE
Rigid
$# sid da1 da2 da3 da4 solver
  1 0.000 0.000 0.000 0.000MECH
$# nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
  44 45 46 47 48 49 51 52
  53 54 55 56 58 59 60 61
  62 63 65 66 67 68 69 70
  72 73 74 75 76 77 79 80
  81 82 83 84 86 87 88 89
  90 91 1036 1038 1040 1042 1043 1044
  1045 1046 1047 1048 1049 1050 1051 1052
  1053 1054 1055 1056 1057 1059 1061 1063
  1064 1065 1066 1067 1068 1069 1071 1071
  1072 1073 1074 1075 1076 1077 1078 1080
  1082 1084 1085 1086 1087 1088 1089 1090
  1091 1092 1093 1094 1095 1096 1097 1098
  1099 1101 1103 1105 1106 1107 1108 1109
  1110 1111 1112 1113 1114 1115 1116 1117
  1118 1119 1120 1122 1124 1126 1127 1128
  1129 1130 1131 1132 1133 1134 1135 1136
  1137 1138 1139 1140 1141 1143 1145 1147
  1148 1149 1150 1151 1152 1153 1154 1155
  1156 1157 1158 1159 1160 1161 1162 1164
  1166 1168 1169 1170 1171 1172 1173 1174
  1175 1176 1177 1178 1179 1180 1181 1182
*SET_NODE_LIST_TITLE
Move
$# sid da1 da2 da3 da4 solver
  2 0.000 0.000 0.000 0.000MECH
$# nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
  134 135 136 137 138 139 141 142
```
### RefNodes

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nid1</th>
<th>nid2</th>
<th>nid3</th>
<th>nid4</th>
<th>nid5</th>
<th>nid6</th>
<th>nid7</th>
<th>nid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>689</td>
<td>723</td>
<td>757</td>
<td>791</td>
<td>825</td>
<td>859</td>
<td>893</td>
<td>2857</td>
</tr>
<tr>
<td>2858</td>
<td>2859</td>
<td>2953</td>
<td>2954</td>
<td>2955</td>
<td>3049</td>
<td>3050</td>
<td>3051</td>
</tr>
<tr>
<td>3145</td>
<td>3146</td>
<td>3147</td>
<td>3241</td>
<td>3242</td>
<td>3243</td>
<td>3337</td>
<td>3338</td>
</tr>
<tr>
<td>3339</td>
<td>3433</td>
<td>3434</td>
<td>3435</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### Tracker

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nid1</th>
<th>nid2</th>
<th>nid3</th>
<th>nid4</th>
<th>nid5</th>
<th>nid6</th>
<th>nid7</th>
<th>nid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1310</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### SymNodes

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nid1</th>
<th>nid2</th>
<th>nid3</th>
<th>nid4</th>
<th>nid5</th>
<th>nid6</th>
<th>nid7</th>
<th>nid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>43</td>
<td>44</td>
<td>45</td>
<td>46</td>
<td>47</td>
<td>48</td>
<td>49</td>
<td>134</td>
</tr>
<tr>
<td>135</td>
<td>136</td>
<td>137</td>
<td>138</td>
<td>139</td>
<td>140</td>
<td>250</td>
<td>251</td>
</tr>
<tr>
<td>252</td>
<td>253</td>
<td>254</td>
<td>255</td>
<td>256</td>
<td>257</td>
<td>258</td>
<td>259</td>
</tr>
</tbody>
</table>
392 393 394 395 396 397 398 399
400 401 674 675 676 677 678 679
680 681 682 683 684 685 686 687
688 689 690 691 692 693 694 695
696 697 698 699 700 701 702 703
704 705 1036 1037 1038 1039 1040 1041
1042 1043 1044 1045 1046 1047 1048 1049
1050 1051 1052 1053 1054 1055 1056 1057
1310 1311 1312 1313 1314 1315 1316 1317
1318 1319 1320 1321 1322 1323 1324 1325
1326 1327 1328 1329 1636 1637 1638 1639
1640 1641 1642 1643 1644 1645 1646 1647
1648 1649 1650 1651 1652 1653 1654 1655
1656 1657 1658 1659 1660 1661 1662 1663
1664 1665 2026 2027 2028 2029 2030 2031
2032 2033 2034 2035 2036 2037 2038 2039
2040 2041 2042 2043 2044 2045 2046 2047
2048 2049 2050 2051 2052 2053 2054 2055
2812 2813 2814 2815 2816 2817 2818 2819
2820 2821 2822 2823 2824 2825 2826 2827
2828 2829 2830 2831 2832 2833 2834 2835
2836 2837 2838 2839 2840 2841 2842 2843
2844 2845 2846 2847 2848 2849 2850 2851
2852 2853 2854 2855 2856 2857 2858 2859
2860 2861 2862 2863 2864 2865 2866 2867
2868 2869 2870 2871 2872 2873 2874 2875
2876 2877 2878 2879 2880 2881 2882 2883
2884 2885 2886 2887 2888 2889 2890 2891
2892 2893 2894 2895 2896 2897 2898 2899
2900 2901 2902 2903 2904 2905 2906 2907

*SET_SOLID_TITLE
RefEl
$# sid solver
1MECH
$# k1 k2 k3 k4 k5 k6 k7 k8
2575 2576 2577 2674 2675 2676 2773 2774
2775 2872 2873 2874 2971 2972 2973 3070
3071 3072 0 0 0 0 0 0

*ELEMENT_SOLID
$# eid pid n1 n2 n3 n4 n5 n6 n7 n8
889 2 43 44 51 50 1037 1036 1057 1058
890 2 1037 1036 1057 1058 1039 1038 1059 1060
891 2 1039 1038 1059 1060 1041 1040 1061 1062
C.3 Finite element analyses of the plate with a centric hole

C.3.1 Input file

$# LS-DYNA Keyword file created by LS-PrePost 3.2 - 16Jan2012(01:28)
$# Created on Mar-22-2012 (12:50:14)
*KEYWORD
*TITLE
$# title
LS-DYNA keyword deck by LS-PrePost
*INCLUDE
ppFinal.k
*INCLUDE
geomTFV2_4thick.k
*CONTROL_ENERGY
2 2 1 1
*CONTROL_TERMINATION
25.000000 0 0.000 0.000 0.000
*CONTROL_TIMESTEP
0.000 0.500000 0 0.000 0.000 0 0 0
0.000 0 0
*DATABASE_BINARY_D3PLOT
0.25
*DATABASE_EXTENT_BINARY
59 0 3 1 0 0 0 0
0 0 4 0 0 0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*DATABASE_GLSTAT
0.050000 0 0 0
*DATABASE_MATSUM
0.050000 0 0 0
*DATABASE_NODOUT
0.050000 0 0 0 0.000 0
*DATABASE_ELOUT
0.050000 0 0 0 0.000 0
*DATABASE_SECFORC
0.050000 0 0 0
*DATABASE_SPCFORC
0.050000 0 0 0
*DATABASE_CROSS_SECTION_SET_ID
1ReadSet
C.3.2 Excerpt from the geometry file geomTFV2_4thick.k

```plaintext
C.3.2 Excerpt from the geometry file geomTFV2_4thick.k

#$ LS-DYNA Keyword file created by LS-PrePost 3.2 - 16Jan2012(01:28)
#$ Created on May-24-2012 (16:30:36)
*KEYWORD
*SET_NODE_LIST_TITLE
Move
#$ sid da1 da2 da3 da4 solver
  1  0.000  0.000  0.000  0.000 0.000MECH
#$ nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8
  1  2  4  7  11  16  22  29
  37  46  56  67  78  89 100 111
  122 133 144 155 166 177 188 199
  210 569 572 573 576 577 580 581
  584 610 612 614 616 650 652 654
```
| 656 | 686 | 688 | 690 | 692 | 726 | 728 | 730 |
| 732 | 774 | 776 | 778 | 780 | 862 | 864 | 866 |
| 868 | 938 | 940 | 942 | 944 | 1018 | 1020 | 1022 |
| 1024 | 1102 | 1104 | 1106 | 1108 | 1194 | 1196 | 1198 |
| 1200 | 1290 | 1292 | 1294 | 1296 | 1386 | 1388 | 1390 |
| 1392 | 1474 | 1476 | 1478 | 1480 | 1558 | 1560 | 1562 |
| 1564 | 1634 | 1636 | 1638 | 1640 | 1718 | 1720 | 1722 |
| 1724 | 1790 | 1792 | 1794 | 1796 | 1882 | 1884 | 1886 |
| 2076 | 2158 | 2160 | 2162 | 2164 | 2242 | 2244 | 2246 |
| 2248 | 2330 | 2332 | 2334 | 2336 | 0 | 0 | 0 |

*SET_NODE_LIST_TITLE
SymZ

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000MECH</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nid1</th>
<th>nid2</th>
<th>nid3</th>
<th>nid4</th>
<th>nid5</th>
<th>nid6</th>
<th>nid7</th>
<th>nid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1375</td>
<td>1376</td>
<td>1394</td>
<td>1470</td>
<td>1486</td>
<td>1554</td>
<td>1626</td>
<td>1667</td>
</tr>
<tr>
<td>1668</td>
<td>1678</td>
<td>1710</td>
<td>1738</td>
<td>1750</td>
<td>2087</td>
<td>2088</td>
<td>2166</td>
</tr>
<tr>
<td>2238</td>
<td>2326</td>
<td>2422</td>
<td>2471</td>
<td>2472</td>
<td>2498</td>
<td>2530</td>
<td>2554</td>
</tr>
<tr>
<td>2570</td>
<td>2578</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

*SET_NODE_LIST_TITLE
SymY

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000MECH</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nid1</th>
<th>nid2</th>
<th>nid3</th>
<th>nid4</th>
<th>nid5</th>
<th>nid6</th>
<th>nid7</th>
<th>nid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1377</td>
<td>1378</td>
<td>1379</td>
<td>1380</td>
<td>1395</td>
<td>1396</td>
<td>1471</td>
<td>1472</td>
</tr>
<tr>
<td>1487</td>
<td>1488</td>
<td>1555</td>
<td>1556</td>
<td>1627</td>
<td>1628</td>
<td>1669</td>
<td>1670</td>
</tr>
<tr>
<td>1671</td>
<td>1672</td>
<td>1679</td>
<td>1680</td>
<td>1711</td>
<td>1712</td>
<td>1739</td>
<td>1740</td>
</tr>
<tr>
<td>1751</td>
<td>1752</td>
<td>2089</td>
<td>2090</td>
<td>2091</td>
<td>2092</td>
<td>2167</td>
<td>2168</td>
</tr>
<tr>
<td>2239</td>
<td>2240</td>
<td>2327</td>
<td>2328</td>
<td>2423</td>
<td>2424</td>
<td>2473</td>
<td>2474</td>
</tr>
<tr>
<td>2475</td>
<td>2476</td>
<td>2499</td>
<td>2500</td>
<td>2531</td>
<td>2532</td>
<td>2555</td>
<td>2556</td>
</tr>
<tr>
<td>2571</td>
<td>2572</td>
<td>2579</td>
<td>2580</td>
<td>359</td>
<td>369</td>
<td>371</td>
<td>380</td>
</tr>
<tr>
<td>381</td>
<td>389</td>
<td>395</td>
<td>396</td>
<td>400</td>
<td>401</td>
<td>404</td>
<td>406</td>
</tr>
<tr>
<td>407</td>
<td>456</td>
<td>467</td>
<td>478</td>
<td>490</td>
<td>505</td>
<td>518</td>
<td>519</td>
</tr>
<tr>
<td>524</td>
<td>529</td>
<td>531</td>
<td>535</td>
<td>538</td>
<td>540</td>
<td>1373</td>
<td>1374</td>
</tr>
<tr>
<td>1393</td>
<td>1469</td>
<td>1485</td>
<td>1553</td>
<td>1625</td>
<td>1665</td>
<td>1666</td>
<td>1677</td>
</tr>
<tr>
<td>1709</td>
<td>1737</td>
<td>1749</td>
<td>2085</td>
<td>2086</td>
<td>2165</td>
<td>2237</td>
<td>2325</td>
</tr>
<tr>
<td>2421</td>
<td>2469</td>
<td>2470</td>
<td>2497</td>
<td>2529</td>
<td>2553</td>
<td>2569</td>
<td>2577</td>
</tr>
</tbody>
</table>

*SET_NODE_LIST_TITLE
RefNodes

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000MECH</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>nid1</th>
<th>nid2</th>
<th>nid3</th>
<th>nid4</th>
<th>nid5</th>
<th>nid6</th>
<th>nid7</th>
<th>nid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>70</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>sid</td>
<td>da1</td>
<td>da2</td>
<td>da3</td>
<td>da4</td>
<td>solver</td>
<td></td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>------</td>
<td>------</td>
<td>------</td>
<td>------</td>
<td>--------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
| *SET_NODE_LIST_TITLE*
| BotCirc $# nid1 | nid2 | nid3 | nid4 | nid5 | nid6 | nid7 | nid8 |
| 871  | 0    | 0    | 0    | 0    | 0      | 0    | 0    |

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
</tr>
</tbody>
</table>
| *SET_NODE_LIST_TITLE*
| Tracker $# nid1 | nid2 | nid3 | nid4 | nid5 | nid6 | nid7 | nid8 |
| 1400 | 0    | 0    | 0    | 0    | 0      | 0    | 0    |

<table>
<thead>
<tr>
<th>sid</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>MECH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>#</th>
<th>k1</th>
<th>k2</th>
<th>k3</th>
<th>k4</th>
<th>k5</th>
<th>k6</th>
<th>k7</th>
<th>k8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1082</td>
<td>1083</td>
<td>1084</td>
<td>1085</td>
<td>1094</td>
<td>1095</td>
<td>1096</td>
<td>1097</td>
<td></td>
</tr>
<tr>
<td>1170</td>
<td>1171</td>
<td>1172</td>
<td>1173</td>
<td>1182</td>
<td>1183</td>
<td>1184</td>
<td>1185</td>
<td></td>
</tr>
<tr>
<td>1250</td>
<td>1251</td>
<td>1252</td>
<td>1253</td>
<td>1318</td>
<td>1319</td>
<td>1320</td>
<td>1321</td>
<td></td>
</tr>
<tr>
<td>1354</td>
<td>1355</td>
<td>1356</td>
<td>1357</td>
<td>1362</td>
<td>1363</td>
<td>1364</td>
<td>1365</td>
<td></td>
</tr>
<tr>
<td>1366</td>
<td>1367</td>
<td>1368</td>
<td>1369</td>
<td>1398</td>
<td>1399</td>
<td>1400</td>
<td>1401</td>
<td></td>
</tr>
<tr>
<td>1422</td>
<td>1423</td>
<td>1424</td>
<td>1425</td>
<td>1434</td>
<td>1435</td>
<td>1436</td>
<td>1437</td>
<td></td>
</tr>
<tr>
<td>1726</td>
<td>1727</td>
<td>1728</td>
<td>1729</td>
<td>1802</td>
<td>1803</td>
<td>1804</td>
<td>1805</td>
<td></td>
</tr>
<tr>
<td>1874</td>
<td>1875</td>
<td>1876</td>
<td>1877</td>
<td>1966</td>
<td>1967</td>
<td>1968</td>
<td>1969</td>
<td></td>
</tr>
<tr>
<td>2066</td>
<td>2067</td>
<td>2068</td>
<td>2069</td>
<td>2118</td>
<td>2119</td>
<td>2120</td>
<td>2121</td>
<td></td>
</tr>
<tr>
<td>*ELEMENT_SOLID</td>
<td>$#</td>
<td>eid</td>
<td>pid</td>
<td>n1</td>
<td>n2</td>
<td>n3</td>
<td>n4</td>
<td>n5</td>
</tr>
<tr>
<td>----------------</td>
<td>----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
<td>-----</td>
</tr>
<tr>
<td>458</td>
<td>2</td>
<td>250</td>
<td>257</td>
<td>262</td>
<td>256</td>
<td>541</td>
<td>542</td>
<td>543</td>
</tr>
<tr>
<td>459</td>
<td>2</td>
<td>541</td>
<td>542</td>
<td>543</td>
<td>544</td>
<td>545</td>
<td>546</td>
<td>547</td>
</tr>
<tr>
<td>460</td>
<td>2</td>
<td>545</td>
<td>546</td>
<td>547</td>
<td>548</td>
<td>549</td>
<td>550</td>
<td>551</td>
</tr>
<tr>
<td>461</td>
<td>2</td>
<td>549</td>
<td>550</td>
<td>551</td>
<td>552</td>
<td>553</td>
<td>554</td>
<td>555</td>
</tr>
<tr>
<td>462</td>
<td>2</td>
<td>251</td>
<td>258</td>
<td>263</td>
<td>257</td>
<td>557</td>
<td>558</td>
<td>559</td>
</tr>
<tr>
<td>463</td>
<td>2</td>
<td>557</td>
<td>558</td>
<td>559</td>
<td>542</td>
<td>560</td>
<td>561</td>
<td>562</td>
</tr>
</tbody>
</table>
C.4 Drop tower

C.4.1 Input file

$# Created on Apr-24-2012 (16:50:16)
*KEYWORD
*TITLE
Droptower. High steel spear, instead of rigid material.
*INCLUDE
gemo.png
*INCLUDE
ppFinalDTOPT.k
*CONTROL_TERMINATION
$# endtim endcyc dtmin endeng endmas
0.014400 0 0.000 0.000 0.000
*CONTROL_TIMESTEP
$# dtinit tssfac isdo tslimt dt2ms lctm erode ms1st
0.000 0.500000 0 0.000 0.000 0 0 0
$# dt2msf dt2mslc imscl
0.000 0 0
*CONTROL_SHELL
0.0000000 0 0 0 0 0 0
*CONTROL_DAMPING
0 0.0000000 0.0000000 0.0000000 0.0000000 0 0.0000000 0
*CONTROL_CONTACT
0.0000000 0.0000000 0 0 0 0 0 0
*CONTROL_OUTPUT
0 0 0 0 0.0000000 0 0 0
*CONTROL_ENERGY
2 2 2 1
*CONTROL_CPU
0.0000000
*DATABASE_ELOUT
$# dt binary lcur ioopt
0.1800E-4 0 0 1
*DATABASE_GLSTAT
$# dt binary lcur ioopt
0.1800E-4 0 0 1
*DATABASE_MATSUM
$# dt binary lcur ioopt
0.1800E-4 0 0 1
*DATABASE_NODOUT
$#  dt  binary  lcur  ioopt  dthf  binhf
0.1800E-4  0  0  1  0.000  0

*DATABASE_SPCFORC
$#  dt  binary  lcur  ioopt
0.1800E-4  0  0  1

*DATABASE_SECFORC
0.1800E-4

*DATABASE_BINARY_D3PLOT
$#  dt  lcdt  beam  npltc  psetid
2.0000E-4  0  0  0  0
$#  ioopt
0

*DATABASE_EXTENT_BINARY
$#  neiph  neips  maxint  strflg  sigflg  epsflg  rltflg  engflg
59  0  3  1  1  1  1  1
$#  cmpflg  ieveparameter  beamip  dcomp  shge  stssz  n3thdt  ialemat
0  0  4  1  1  1  2  0
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$#  nintsld  pkp_sen  sclp  unused  msscl  therm  intout  nodout
0  0  1.000000  0  0  0  OSTRESS  STRESS

*DATABASE_HISTORY_NODE_SET
3
$ Section definitions - start

*DATABASE_HISTORY_SHELL_SET
1

*DATABASE_CROSS_SECTION_SET_ID
1  InnerCSedge
2  0  0  1

*DATABASE_HISTORY_SHELL_SET
2

*DATABASE_CROSS_SECTION_SET_ID
2  SpearEdge
5  0  0  2

*DATABASE_HISTORY_SHELL_SET
3

*DATABASE_CROSS_SECTION_SET_ID
3  TransCircToSpear
7  0  0  3

*DATABASE_HISTORY_SHELL_SET
4

*DATABASE_CROSS_SECTION_SET_ID
4  SecInnerCSedge

74
6 0 0 4
*DATABASE_HISTORY_SHELL_SET
5
*DATABASE_CROSS_SECTION_SET_ID
5 SpearSec
8 0 0 5
$ Section definitions - end
*BOUNDARY_SPC_SET_BIRTH_DEATH
$# nsid cid dofx dofy dofz dofrx dofrx dofrz
d1 0 1 1 0 1 1 1
$# birth death
0.0001.0000E+20
*CONTACT_2D_AUTOMATIC_SINGLE_SURFACE
0 0 1.5000000 1 0.2500000 0.2250000 0.0000000 0
0.0000000 1.0000E20 1.0000000 1.0000000 0 0 0 0
*PART
$# title
$# pid secid mid eosid hgid grav adpopt tmid
4 2 2 0 0 0 0 0
*PART
$# title
$# pid secid mid eosid hgid grav adpopt tmid
3 1 1 0 0 0 0 0
*PART
$# title
$# pid secid mid eosid hgid grav adpopt tmid
5 2 3 0 0 0 0 0
*SECTION_SHELL_TITLE
Section
$# secid elform shrf nip propt qr/irid icomp setyp
1 15 1.000000 2 1 0 0 0 1
$# t1 t2 t3 t4 nloc marea idof edgset
0.000000 0.000000 0.000000 0.000000 0.000 0.000 0.000 0
*SECTION_SHELL_TITLE
Section
$# secid elform shrf nip propt qr/irid icomp setyp
2 15 1.000000 2 1 0 0 0 1
$# t1 t2 t3 t4 nloc marea idof edgset
0.000000 0.000000 0.000000 0.000000 0.000 0.000 0.000 0
*HOURGLASS
1 1
*HOURGLASS
 2 2
*MAT_ELASTIC
   27.85000E-9 210000 0.30 0 0
*MAT_ELASTIC
   31.06448e-6 210000 0.30 0 0
*INITIAL VELOCITY
$# nsid nsidex boxid irigid icid
 4 0 0 0 0
$# vx vy vz vxr vyr vzr
 0.000-4430.0000 0.000 0.000 0.000 0.000
*DEFINE_CURVE SMOOTH TITLE
Smoooooooooth
$# lcid sidr dist tstart tend trise v0
 1 0 5.0000E+6 0.000 0.000 1.0000E-6 1.000000
*END
C.4.2 Excerpt from the geometry file geommdt.k

```
$# LS-DYNA Keyword file created by LS-PrePost 3.2 - 16Jan2012(01:28)
$# Created on May-01-2012 (09:35:56)
*KEYWORD
*TITLE
$# title
Geometry: Droptower modelled as steel tip with extra weight.
*SET_NODE_LIST_TITLE
Rigid
$# sid   da1   da2   da3   da4   solver
   1  0.000  0.000  0.000  0.000MECH
$#   nid1   nid2   nid3   nid4   nid5   nid6   nid7   nid8
   17093  17094  17095  17096  17097  17098  17099  17100
   17101  17102  17103  17104  17105  17106  17107  0
*SET_NODE_LIST_TITLE
InnerCSedge
$# sid   da1   da2   da3   da4   solver
   2  0.000  0.000  0.000  0.000MECH
$#   nid1   nid2   nid3   nid4   nid5   nid6   nid7   nid8
  19151  19150  19149  19148  19147  19146  19145  19144
  19143  18986  18978  18970  18962  18954  18946  18938
  18930  18922  18897  0  0  0  0  0
*SET_NODE_LIST_TITLE
SpearTIP
$# sid   da1   da2   da3   da4   solver
   3  0.000  0.000  0.000  0.000MECH
$#   nid1   nid2   nid3   nid4   nid5   nid6   nid7   nid8
  19171  0  0  0  0  0  0  0
*SET_NODE_LIST_TITLE
Spear
$# sid   da1   da2   da3   da4   solver
   4  0.000  0.000  0.000  0.000MECH
$#   nid1   nid2   nid3   nid4   nid5   nid6   nid7   nid8
  18827  18828  18829  18830  18831  18832  18833  18834
  18835  18836  18837  18838  18839  18840  18841  18842
  18843  18844  18845  18846  18847  18848  18849  18850
  18851  18852  18853  18854  18855  18856  18857  18858
  18859  18860  18861  18862  18863  18864  18865  18866
  18867  18868  18869  18870  18871  18872  18873  18874
  18875  18876  18877  18878  18879  18880  18881  18882
  18883  18884  18885  18886  18887  18888  18889  18890
  18891  18892  18893  18894  18895  18896  18897  18898
```
<table>
<thead>
<tr>
<th>*SET_NODE_LIST_TITLE</th>
<th>SpearEdge</th>
</tr>
</thead>
<tbody>
<tr>
<td>$#$ sid da1 da2 da3 da4 solver</td>
<td></td>
</tr>
<tr>
<td>5 0.000 0.000 0.000 0.000MECH</td>
<td></td>
</tr>
<tr>
<td>$#$ nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8</td>
<td></td>
</tr>
<tr>
<td>19170 19171 19169 19168 19167 19166 19165 19164</td>
<td></td>
</tr>
<tr>
<td>19163 18984 18976 18968 18960 18952 18944 18936</td>
<td></td>
</tr>
<tr>
<td>18928 18920 18895 0 0 0 0 0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>*SET_NODE_LIST_TITLE</th>
<th>secondInnerCSedge</th>
</tr>
</thead>
<tbody>
<tr>
<td>$#$ sid da1 da2 da3 da4 solver</td>
<td></td>
</tr>
<tr>
<td>6 0.000 0.000 0.000 0.000MECH</td>
<td></td>
</tr>
<tr>
<td>$#$ nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8</td>
<td></td>
</tr>
<tr>
<td>19161 19160 19159 19158 19157 19156 19155 19154</td>
<td></td>
</tr>
<tr>
<td>19153 18985 18977 18969 18961 18953 18945 18937</td>
<td></td>
</tr>
<tr>
<td>18929 18921 18896 0 0 0 0 0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>*SET_NODE_LIST_TITLE</th>
<th>TopCircleSection</th>
</tr>
</thead>
<tbody>
<tr>
<td>$#$ sid da1 da2 da3 da4 solver</td>
<td></td>
</tr>
<tr>
<td>7 0.000 0.000 0.000 0.000MECH</td>
<td></td>
</tr>
<tr>
<td>$#$ nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8</td>
<td></td>
</tr>
<tr>
<td>18923 18924 18925 18926 18927 19003 19004 19005</td>
<td></td>
</tr>
<tr>
<td>19006 19007 19008 19009 19010 19011 0 0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>*SET_NODE_LIST_TITLE</th>
<th>CSspear</th>
</tr>
</thead>
<tbody>
<tr>
<td>$#$ sid da1 da2 da3 da4 solver</td>
<td></td>
</tr>
<tr>
<td>8 0.000 0.000 0.000 0.000MECH</td>
<td></td>
</tr>
<tr>
<td>$#$ nid1 nid2 nid3 nid4 nid5 nid6 nid7 nid8</td>
<td></td>
</tr>
<tr>
<td>18844 18845 18846 18847 18848 18849 18850 18851</td>
<td></td>
</tr>
<tr>
<td>18852 18853 18854 18855 18856 18857 18858 18859</td>
<td></td>
</tr>
<tr>
<td>18860 0 0 0 0 0 0 0</td>
<td></td>
</tr>
</tbody>
</table>
### *SET_PART_LIST>Title*

**Plate**

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>pid1</th>
<th>pid2</th>
<th>pid3</th>
<th>pid4</th>
<th>pid5</th>
<th>pid6</th>
<th>pid7</th>
<th>pid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### *SET_PART_LIST>Title*

**Spear**

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
<th>solver</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>MECH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>pid1</th>
<th>pid2</th>
<th>pid3</th>
<th>pid4</th>
<th>pid5</th>
<th>pid6</th>
<th>pid7</th>
<th>pid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### *SET_SHELL_LIST>Title*

**RefEl**

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>eid1</th>
<th>eid2</th>
<th>eid3</th>
<th>eid4</th>
<th>eid5</th>
<th>eid6</th>
<th>eid7</th>
<th>eid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>8124</td>
<td>8123</td>
<td>8122</td>
<td>8121</td>
<td>8120</td>
<td>8119</td>
<td>8118</td>
<td>8117</td>
</tr>
<tr>
<td>8116</td>
<td>7994</td>
<td>7987</td>
<td>7973</td>
<td>7980</td>
<td>7966</td>
<td>7952</td>
<td>7959</td>
</tr>
<tr>
<td>7945</td>
<td>7938</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### *SET_SHELL_LIST>Title*

**SpearEdge**

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>eid1</th>
<th>eid2</th>
<th>eid3</th>
<th>eid4</th>
<th>eid5</th>
<th>eid6</th>
<th>eid7</th>
<th>eid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>8142</td>
<td>8141</td>
<td>8140</td>
<td>8139</td>
<td>8138</td>
<td>8137</td>
<td>8136</td>
<td>8135</td>
</tr>
<tr>
<td>7992</td>
<td>8134</td>
<td>7985</td>
<td>7978</td>
<td>7971</td>
<td>7957</td>
<td>7964</td>
<td>7950</td>
</tr>
<tr>
<td>7943</td>
<td>7936</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### *SET_SHELL_LIST>Title*

**SpearLowEL**

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>eid1</th>
<th>eid2</th>
<th>eid3</th>
<th>eid4</th>
<th>eid5</th>
<th>eid6</th>
<th>eid7</th>
<th>eid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>7939</td>
<td>7940</td>
<td>7941</td>
<td>7942</td>
<td>7999</td>
<td>8000</td>
<td>8001</td>
<td>8002</td>
</tr>
<tr>
<td>8003</td>
<td>8004</td>
<td>8005</td>
<td>8006</td>
<td>8007</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### *SET_SHELL_LIST>Title*

**VerEl**

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>eid1</th>
<th>eid2</th>
<th>eid3</th>
<th>eid4</th>
<th>eid5</th>
<th>eid6</th>
<th>eid7</th>
<th>eid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>8133</td>
<td>8131</td>
<td>8132</td>
<td>8130</td>
<td>8129</td>
<td>8128</td>
<td>8127</td>
<td>8126</td>
</tr>
<tr>
<td>8125</td>
<td>7993</td>
<td>7986</td>
<td>7979</td>
<td>7972</td>
<td>7965</td>
<td>7958</td>
<td>7951</td>
</tr>
<tr>
<td>7944</td>
<td>7937</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
*SET_SHELL_LIST_TITLE
ElSpear

\$# sid da1 da2 da3 da4
5  0.000  0.000  0.000  0.000

\$# eid1  eid2  eid3  eid4  eid5  eid6  eid7  eid8
7872  7873  7874  7875  7876  7877  7878  7879
7880  7881  7882  7883  7884  7885  7886  7887

*ELEMENT_SHELL

\$# eid pid n1 n2 n3 n4 n5 n6 n7 n8
6280  3  17108  17109  17210  17209  0  0  0  0
   .  .  .  .  .  .  .  .  .  .
   .  .  .  .  .  .  .  .  .  .