

**ANALYSIS TOOLS FOR THE DESIGN OF
ALUMINIUM EXTRUSION DIES**

A.J. Koopman

This research was carried out within the framework of the Simalex project, which was initiated by Boalgroup in De Lier.

De promotiecommissie is als volgt samengesteld:

Voorzitter en secretaris:

Prof.dr.ir. F. Eising	Universiteit Twente
-----------------------	---------------------

Promotoren:

Prof. dr. ir. J. Huétink	Universiteit Twente
--------------------------	---------------------

Prof. dr. ir. F.J.A.M. van Houten	Universiteit Twente
-----------------------------------	---------------------

Assisent Promotor:

Dr.ir. H.J.M Geijselaers	Universiteit Twente
--------------------------	---------------------

Leden:

Prof. dr.-Ing. A. E. Tekkaya	Universiteit Dortmund
------------------------------	-----------------------

Prof. dr. ir. F. Soetens	Technische Universiteit Eindhoven
--------------------------	-----------------------------------

Prof. dr. ir. D.J. Schipper	Universiteit Twente
-----------------------------	---------------------

Prof. dr.ir. R. Akkerman	Universiteit Twente
--------------------------	---------------------

dr. R.M.J van Damme	Universiteit Twente
---------------------	---------------------

Analysis tools for the design of aluminium extrusion dies

Koopman, Albertus Johannes

PhD thesis, University of Twente, Enschede, The Netherlands

March 2008

ISBN 978-90-9024381-8

Subject headings: Flow front tracking, Simulation based Die design

Copyright ©2009 by A.J. Koopman, Enschede, The Netherlands

Printed by Laseline, Hengelo, The Netherlands

ANALYSIS TOOLS FOR THE DESIGN OF ALUMINIUM EXTRUSION DIES

PROEFSCHRIFT

ter verkrijging van
de graad van doctor aan de Universiteit Twente,
op gezag van de rector magnificus,
prof. dr. H. Brinksma,
volgens besluit van het College voor Promoties
in het openbaar te verdedigen
op donderdag 11 juni 2009 om 15.00 uur

door

Albertus Johannes Koopman

geboren op 22 november 1976

te Brederwiede

Dit proefschrift is goedgekeurd door de promotoren:

Prof. dr. ir. J. Huétink

Prof. dr. ir. F.J.A.M. van Houten

en de assistent promotor

dr. ir. H.J.M. Geijseleers

Summary

The aluminium extrusion process is a forming process where a billet of hot aluminium is pressed through a die to produce long straight aluminium profiles. A large variety of products with different and complex cross-sections can be made. The insight in the mechanics of the aluminium extrusion process is still limited. Design of extrusion dies is primarily based on trial and error. The wasted scrap and time in these trial and error iterations, can be reduced by gaining more insight in the extrusion process. Numerical analysis is a valuable tool in obtaining that insight.

In this thesis reports new developments for the analysis of the aluminium extrusion process are treated. The subject matter is presented in four chapters. Attention is focussed on three topics:

- A comparison between experiments and simulations of container flow
- Modeling the start-up of the extrusion process in an Eulerian formulation
- Deriving a new finite element for ALE simulations

Extrusion experiments have been performed at Boalgroup to visualize the flow inside the container during extrusion. These experiments are compared with simulations. The results of the simulations are steady state results that are post-processed to be comparable to the experimental results. If the simulations are not in agreement with the experiments, the material properties used in the simulations are adapted so the results agree. With this method it is possible to determine material properties under extrusion conditions.

Correcting the dies after trial pressings is performed by die-correctors. The correctors use the first part of the profile (nose piece) to assess the work that has to be performed on the die. To be able to model this nose piece, is very valuable during designing of the die. In chapter 4 and 5 new strategies to simulate the shape of the nose piece are treated.

In the last chapter the possibilities of the proposed strategy are demonstrated

on a porthole die for a tube. The simulated nose piece is in very good agreement with the experimental results.

Samenvatting

Het aluminium extrusie proces is een vormingsproces waarbij heet aluminium door een matrijs wordt geperst om lange rechte aluminium profielen te maken. Een grote variteit aan producten met complexe doorsneden kunnen worden gemaakt. het inzicht in de mechanica van het aluminium extrusie proces is vandaag de dag beperkt. Ook het matrijs ontwerpen is voornamelijk gebaseerd op het trial and error proces. Bij deze trial and error iteraties veel aluminium afval gemaakt en er gaat kostbare tijd verloren. meer inzicht in het extrusie proces helpt om dit terug te dringen. Numerieke analyse is een waardevol gereedschap bij het verkrijgen van meer inzicht.

In dit proefschrift worden ontwikkelingen voor de analyse van aluminium extrusie behandeld. De inhoud wordt gepresenteerd in vier hoofdstukken waarbij de nadruk ligt op onderstaande drie onderwerpen:

- Een vergelijking tussen experimenten en simulaties van container flow
- Modelleren van de start-up van het extrusie proces in een Euleriaanse beschrijving
- Afleiding en implementatie van een nieuw element voor ALE simulaties

Bij Boalgroup zijn experimenten uitgevoerd om de containerstroming tijdens extrusie te visualiseren. Deze experimenten zijn vergeleken met simulaties. De resultaten van de steady-state simulaties zijn bewerkt om ze te kunnen vergelijken met de experimenten. De materiaaleigenschappen worden aangepast zodat de resultaten overeenstemmen met de experimenten. Op deze manier is het mogelijk om materiaal eigenschappen van aluminium te bepalen onder extrusie condities.

Het aanpassen van de matrijs na het proefpersen wordt uitgevoerd door correctors. Het eerste stuk profiel dat uit de matrijs komt heet het kopstuk. De correctors gebruiken het kopstuk om te beoordelen welke modificaties aan

de matrijs noodzakelijk zijn. Om vorm van het kopstuk te kunnen voorspellen is van grote waarde tijdens het matrijs ontwerp. In hoofdstukken 4 en 5 wordt een nieuwe methode behandeld om de vorm van het kopstuk te bepalen.

In het laatste hoofdstuk worden, aan de hand van een vergelijking tussen experiment en simulatie van de extrusie van een buis, de mogelijkheden getoond van de voorgestelde methode. De vorm van het kopstuk voorspeld met de methode komt goed overeen met de vorm van het kopstuk uit het experiment.

Contents

Summary	v
Samenvatting	vii
Contents	ix
1 Introduction	1
1.1 Aluminium extrusion process	2
1.1.1 Flow related defects in aluminium extrusion	4
1.2 Example: feeder hole area	5
1.3 Numerical simulations of aluminium extrusion	6
1.4 Outline of the thesis	7
2 Finite element analysis of aluminium extrusion	11
2.1 Introduction	11
2.2 Different FEM formulations	13
2.2.1 Lagrangian formulation	13
2.2.2 Eulerian formulation	14
2.2.3 ALE formulation	14
2.3 Arbitrary Lagrangian Eulerian formulation	15
2.3.1 Strong form	16
2.3.2 Weak form	17
2.3.3 Coupled and semi-coupled ALE	17
2.3.4 Material increment	20
2.3.5 Convective increment	23
2.3.6 Convection scheme	24
2.4 Modeling aluminium extrusion	28
2.4.1 Pre-processing	28
2.4.2 Equivalent bearing corner	32

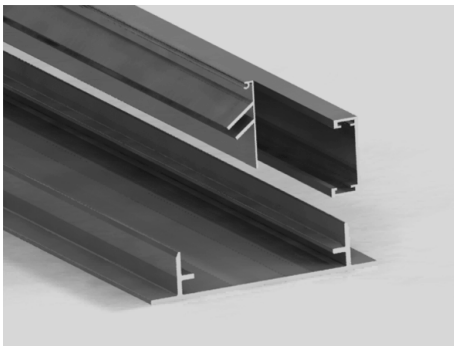
3	Front line tracking in a steady state flow	41
3.1	Introduction	41
3.2	Experiments	41
3.3	Theoretical outline for front line tracking demonstrated on Newtonian viscous flow	44
3.3.1	Viscous flow in a long parallel channel	46
3.3.2	Solutions for a discretized velocity field	48
3.3.3	Comparison of analytical and discretized results	49
3.4	Post-processing FEM results	49
3.4.1	Post-processing	50
3.4.2	A SUPG stabilization for determining front lines	50
3.4.3	Determination of the upwind parameter α	51
3.4.4	Assembly of front lines for comparison	54
3.5	Simulations of container flow in aluminium extrusion	56
3.5.1	material properties	56
3.5.2	FEM modeling	57
3.6	Comparison	58
3.7	Concluding remarks	59
4	Flow front tracking	61
4.1	Flow front tracking algorithms	62
4.2	Original coordinate tracking	63
4.3	Convection algorithm	64
4.3.1	Weighted local and global smoothing for nodal values	64
4.3.2	2D Rigid body flows	67
4.3.3	Single vortex flow field	68
4.4	Numerical model	71
4.4.1	Interface description for extrusion	71
4.4.2	Permeable boundary conditions	72
4.4.3	Modeling the pseudo material	74
4.4.4	Flow front regularization	76
4.5	2D filling of Die cavity	77
4.5.1	Results of 2D die filling	79
4.6	3D Filling of die cavity	83
4.6.1	FEM model	84
4.6.2	Results	87
4.6.3	3D Volume conservation	90
4.6.4	Visual comparison	91
4.7	Conclusions	92

5	A mixed finite element with original coordinate tracking	95
5.1	Theoretical outline	96
5.1.1	Strong formulation	96
5.1.2	Weak formulation	97
5.2	FEM discretisation	97
5.3	Convection	100
5.3.1	Courant number for triangular elements	101
5.3.2	Rigid rotation & Single vortex flow field	104
5.4	Implementation in DiekA	107
5.4.1	Incremental-iterative update	107
5.4.2	Updated Lagrangian	107
5.4.3	Implementation for ALE	108
5.5	Updated Lagrangian application: Tube expansion	110
5.5.1	Analytical	111
5.5.2	Numerical	112
5.6	Eulerian example: 2D Filling	114
5.6.1	Flow front and volume conservation of 2D filling	115
5.6.2	Extrusion force for 2D filling	116
6	Two applications	119
6.1	Container flow	119
6.2	3D die filling: 80 mm Tube	123
6.2.1	Nose piece deformation prediction	124
6.2.2	Results of the simulation	126
6.2.3	Conclusions	130
7	Conclusions & Recommendations	131
A	Weak formulation	133
A.1	Isotropic Elasto plastic material	133
A.2	Strong formulation	133
A.3	Weak formulation	134
B	Stresses in a long cylinder under internal pressure	137
	List of Symbols	141
	Bibliography	145
	Dankwoord	151

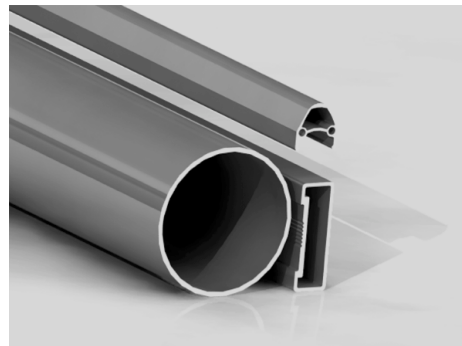
Chapter 1

Introduction

The aluminium extrusion process is a forming process where a billet of hot aluminium is pressed through a die to produce long straight aluminium profiles. A large variety of products with different and complex cross-sections can be made. Some examples are shown in figure 1.1. Extrusion products can be found in many different applications for example transport, construction, electrotechnical appliances and packaging. Aluminium extrusion is exceptionally versatile and aluminium is easy to machine.



(a) Open (flat) profiles



(b) Closed (hollow) profiles

Figure 1.1: Examples of closed and hollow profiles.

Proper die design is an important factor in order to produce high quality products, that meet the tight geometrical tolerances. Until now, designing a die has been mainly based on empirical knowledge and the experience of the die designer. The empirical knowledge is not well documented and therefore only

accessible by the die designer, die corrector and press operator.

In the recent years a trend toward the application of objective design rules can be observed. Side by side with this development is the increase of the use of automated design applications, since objective design rules are required by the design applications. Numerical methods are helpful tools to obtain quantitative information about the process.

The finite element method (FEM) is a valuable tool to gain insight in the process that cannot be obtained easily otherwise. In this thesis developments of the FEM methods for aluminium extrusion are treated. The objective is to gain more insight in the process. With this insight improved or new objective design rules can be obtained.

1.1 Aluminium extrusion process

In direct extrusion a preheated aluminium billet is placed by press operators into a heated container. Here, the ram pushes the aluminium through the die. The die is also preheated before loading the first billet.

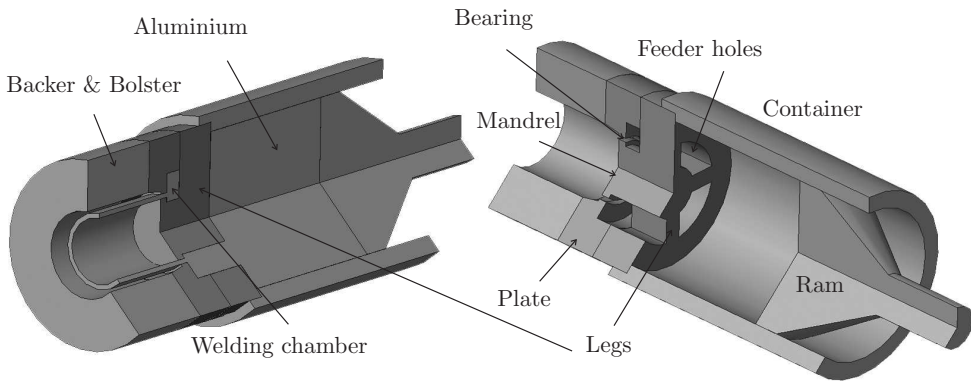


Figure 1.2: Extrusion process.

In the die the shape of the profile is determined in the bearing. The billet slides relative to the walls of the container. During the extrusion process the billet gets shorter and the friction surface between the billet surface and container liner is decreasing. Therefore the necessary ram force decreases during the extrusion of one billet.

Not all of the aluminium billet is extruded. A percentage of the compressed billet, called the discard or butt is left at the end of the extrusion cycle. When the billet is almost completely extruded, the ram and the container are retracted and the butt is sheared off. Then a new billet is inserted and the cycle is repeated. Since the next billet is placed after the previous, this process is called billet-to-billet extrusion

The profiles made using this process can be split into two groups, open and closed profiles, as in figure 1.1. Open profiles can be made with the use of flat dies and closed profiles are made with porthole dies.

A porthole die (figure 1.2) consist roughly of two parts, a mandrel to define the inner geometry of the profile and a die plate which defines the outer geometry. The core is attached to the mandrel by legs. During extrusion, the metal is split by the legs and flows through the feeder holes.

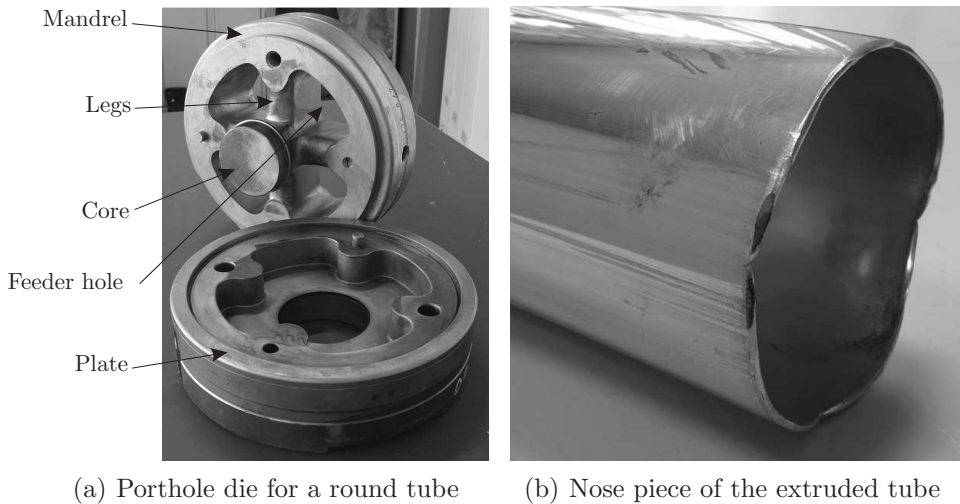


Figure 1.3: Die and profile of extrusion of a tube.

Directly behind the legs the aluminium welds together in the welding chamber. The final shape of the aluminium is determined in the bearing area. In figure 1.3 a typical porthole die and profile are shown.

One important effect in aluminium extrusion is the formation of dead metal zones (DMZ). A dead metal zone can be defined as a zone where the velocity

of the aluminium is small or equal to zero. In this thesis a DMZ is defined as a zone where the velocity is less than 10% of the ram velocity [50].

Formation of dead metal zones mainly occurs in the container, but also appears in welding chambers [45] and in sink-ins [3]. This is shown in figure 1.4. Sink-ins are pre-chambers before the bearing area.

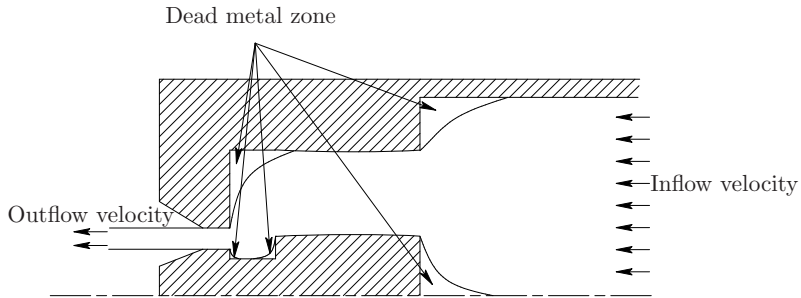


Figure 1.4: Formation of dead metal zones in a porthole die.

1.1.1 Flow related defects in aluminium extrusion

For complex profiles the challenge is to get an uniform outflow velocity over the entire cross-section of the profile. Varying profile thickness over the cross-section or flow restriction by the die design can lead to nonuniform velocities. This is even more challenging for multi hole dies. Not only must the velocity be equal over the cross-section, also the flow between the different profiles must be balanced. In figure 1.5 some typical failures due to unbalanced exit velocity are shown.

Whenever profiles exit the dies in shapes like in figure 1.5, the die is modified by die correctors. Then new trial pressings are performed and often more corrections are necessary. This is repeated until the die produces correct profiles. A reduction of trial pressings will reduce the amount of scrap. It takes energy to recycle the scrap and therefore reducing the number of trial pressings leads to energy savings. Another advantage is the reduction of production time by reducing the amount of trial pressings.

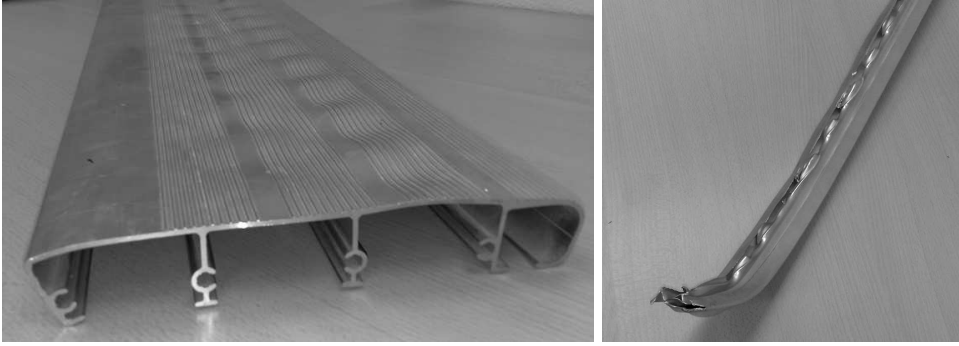


Figure 1.5: Flow related defects in aluminum extrusion.

Research over the last years within Boalgroup has shown that a consistent application of objective design rules, leads to better performing dies [68]. It also shows that in some cases the performance of the dies deteriorate with the use of these design rules. This calls for new or improved design rules and is the main motivation behind this research.

1.2 Example: feeder hole area

The die designer has several methods to control the exit velocity. In both porthole and flat dies the shape and size of the bearing has the strongest influence on the velocity [43]. In flat dies the flow can also be controlled by one or more pre-chambers (or sink-in). In porthole dies the flow can be controlled by leg shape and feeder hole design.

One other aspect of feeder hole design is the size of the feeder holes relative to each other. In figure 1.6 a quarter of a porthole die is shown. In this quarter one profile (dotted lines) and two feeder holes can be recognized.

The inner feeder hole has cross-sectional area A_0 and the outer feeder hole has area A_1 . It is assumed that both inner and outer feeder hole each feed half of the profile. If both feeder holes would have a equal area, the inner feeder hole will have a higher flow compared to the outer hole. The phenomenon that the flow slows down in the proximity of the container wall is called the container effect.

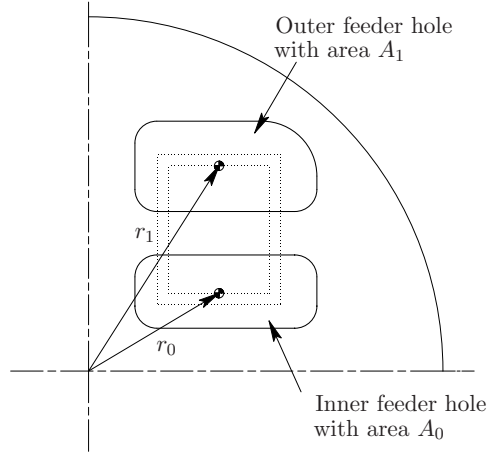


Figure 1.6: Back view of quarter of a four hole porthole die.

This is the effect of the friction between the aluminium and the container wall on the container flow. This effect is known both from literature and experience.

1.3 Numerical simulations of aluminium extrusion

The Finite Element Method (FEM) is nowadays widely used for the analysis of aluminium extrusion as well as for other forming processes. Both two and three dimensional aspects of extrusion can be investigated with this valuable tool. The used codes can be commercial packages (Forge, HyperXtrude, Qform, Deform) and also non commercial codes are used (DiekA, PressForm).

In table 1.1 a summary of some work on aluminium extrusion over the last decade is given, however this table is far from complete. Since papers over more than a decade have been listed, improvements over time can be seen. The table is based on the work of Mooi and Lof [43, 45] and work presented during Extrusion Technology 2004 and Extrusion Bologna 2007 [1, 2].

It is clear that the main focus is on aluminium flow simulations with rigid dies. During the period from 2004 till today the developers of commercial packages all have included some sort of steady state solver to simulate the aluminium flow. Even when a steady state solution may not be the best solution, the calculation times are appealing.

From the comparison with the overview in Mooi [45] can be recognized that over the last ten to fifteen years the use of 3D simulations has become the standard. Computer power and improved numerical techniques make it possible to capture the start-up of the process up to steady state within reasonable simulation times [6, 13].

Furthermore can be concluded that most commercial packages have the ability to deal with thermo-mechanically coupled simulations. Remarkable is the lack of information in literature about thermal properties, modeled tools and other boundary conditions. Hardly any information is available about how to model heat transfer to the dies, ram, container and rest of the press.

In the modeling of friction large differences in the used models and values can be found. During the first extrusion benchmark in Zürich 2005 [54], this was recognized as one of the main problems to match experiments with simulations. It must be remarked that in this benchmark extremely long parallel bearings were used, obviously making friction one of the important phenomena.

1.4 Outline of the thesis

This thesis consists of five main chapters. Chapters three, four and five are rewritten papers which have been submitted for publication elsewhere. In chapters five and six simulations described in previous chapters are extended with more sophisticated elements. For the sake of readability some overlap in subject matter is present. Below an outline of the five main chapters is given.

Chapter 2: In this thesis all the simulations are performed with the Finite Element Method. In chapter 2 an introduction in FEM methods is given. Also the used boundary conditions for Aluminium extrusion are treated. Specific attention is paid to the bearing area and streamlining the pre-processing of a complex 3D simulation.

Chapter 3: In chapter 3 a comparison is made between simulations and experiments of container flow. The steady state results of FEM simulations are used as the first step in a two step procedure to create simulation results that are comparable with the experimental results. In the second step the steady state results are post-processed to front lines. This method can also be used for inverse material modeling

Chapter 4: The start-up of the extrusion process can feed important information to the designer. The tracking of a free surface with original coordinate functions to follow the flow front in an Eulerian description is introduced. The results of the simulations are compared with experiments.

Chapter 5: Meshing complex 3D geometry with hexahedral elements from chapter four proves to be difficult. To be able to mesh with hexahedral elements, many details are lost in the modeling. In this chapter both 2D triangular and 3D tetrahedral MINI elements are developed to be able to model more detail. The tracking algorithm is implemented for the MINI elements and the simulations are repeated.

Chapter 6: To assess the reliability of the simulations, it is important to have good experimental validation of the simulations. In chapters 3 and 4 experiments are shown and in this chapter numerical results for the MINI elements are compared to these experiments. The application of ALE options in a mesh generated with the use of automatic meshers is not straight forward. Extra attention is placed on automatic ALE option generation. The trend in the numerical results is in agreement with the experiments. Small differences can be attributed to the neglecting thermal aspects and other material data.

Reference	Package	Dimension	Formulation	Thermal	Process	Friction	Element
Mooi [45],1996	DiekA	2D,3D	ALE	Yes	Aluminium Flow, Thermal Flow, Die deformation	Stick, $\mu = 0.1$	Q4,H8
Lof [43],2000	DiekA	2D,3D	ALE	No	Flow,	Stick , $\mu = 0.5$	Q4, TET10
Halvorsen [25], 2004	MSC Super-Form	3D	Lagrangian	No	Profile buckling	Stick or full slip	H8
Flitta, Velay [19, 70], 2004	Forge2 (2D)	2D	Lagrangian	Yes	Temperature evolution in Container	$0.1 < \mu < 0.9$	T3
Moroz [46], 2004	Qform	2D	Lagrangian	Yes	Flow, Temperature in Container & Extrusion Force	?	?
Li, Leśniak, Donati [14,38,40],2004	DEFORM3D, DEFORM2D	2D,3D	Lagrangian	Yes	Pocket Designs of Die; Transverse and seam welds	$\mu = 0.6$	Q4, TET-MINI
Reddy [55], 2004	HyperXtrude	3D	Eulerian, ALE	Yes	Bearing length optimization	?	H8
Müller [49],2004	PressForm	3D	ALE	?	Curvature prediction	?	?
Biba [6], 2007	Qform	3D	Eulerian, Lagrangian	Yes	Curvature prediction	combination model	TET
Donati, Li,Liu [13, 39, 41], 2007	DEFORM3D	3D	Lagrangian, ALE	Yes	Filling (Lagrangian), Profile velocity (ALE)	$m = 0.7$ [41]	H8, TET-MINI
Koopman [34, 35], 2007	DiekA	2D,3D	Eulerian ALE	No	Filling (Eulerian), Container flow	Stick or full slip	Q4, H8, TET-MINI
Kloppenborg [33], 2007	HyperXtrude	3D	Eulerian, ALE	Yes	Bearing length optimization	?	TET

Table 1.1: Summary of some recent aluminum extrusion simulations.

Chapter 2

Finite element analysis of aluminium extrusion

2.1 Introduction

The Finite Element Method (FEM) is widely accepted as an effective tool to research many of the extrusion flow and extrusion die properties. Over the years the main interest is in homogenizing the outflow velocity. Other than that topic also weld seam quality prediction, prediction of temperature distribution and prediction of die wear and lifetime is performed.

The finite element method can also be used to optimize the extrusion profile design. Aluminium is widely used in construction. A better quantification of the structural properties through FEM simulations can lead to significant material reduction. An example of FEM analysis on profiles can be found in [47]. In that research profiles for window framing are optimized for a minimum of thermal conductivity to obtain higher insulation values.

This thesis is focused on FEM analysis of the aluminium flow in extrusion dies. In this chapter the basic outline of these simulations is given. For FEM analysis it is important to distinguish three different phases of the billet-to-billet extrusion process: Start-up, steady state and butt-end. These phases are shown in figure [2.1](#).

start-up

The first phase is the start-up of the process, see figure 2.1(a). In this phase the die cavity is filled with aluminium. When the second or next billet is loaded, the die is already filled with aluminium and the process is restarted. In this thesis, start-up is defined as the die filling and first meters of the profile.

This first length of profile is commonly used by the die correctors to evaluate the adjustments to be made on the die to get a homogeneous outflow velocity. This first length is called the nose piece. In chapter 4 an algorithm is treated that is able to capture this part of the process in a FEM analysis.

steady state

After start-up, the process can be regarded as a steady-state process. There is a steady inflow of aluminium into the deformation zone and a steady outflow of aluminium in the form of the profile. The uniform and constant inflow only holds when the ram is far away from the deformation zone. This should be taken into account when modeling a steady state flow. Since the inflow of

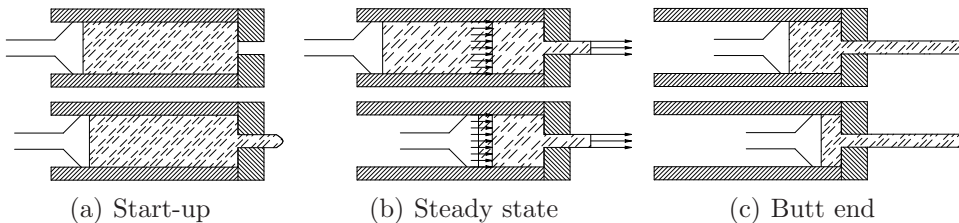


Figure 2.1: Three process parts in billet to billet extrusion. The begin at the top row (l) and end (l) of each part is at the bottom row.

aluminium is constant, only a part of the billet needs to be modeled. At the inflow side of the modeled part a constant inflow velocity, equal to the ram speed, is applied. This is shown in figure 2.1(b). The length of the billet to be modeled L_{bil} is chosen between one and one and a half of the billet diameter D_{bil} . The constant inflow velocity should develop into a plug flow that usually appears inside the container away from the die. In our experience the chosen billet length is sufficient to allow for this, before entering the deformation zone.

At the outflow side many conditions can be applied. Default is chosen for a free outflow condition. This condition allows for the material to flow out

with non homogeneous velocity. The velocity profile tells something about the effectiveness of the die-design. Another condition can be constraints that invoke the velocity to be constant over the cross-section. This resembles a profile that is clamped in a puller and therefore forced to be straight. Also thickening or thinning because of velocity effects can be described with this condition.

butt-end

In the final phase the ram gets close to the die. Here it starts to interfere with the deformation zone. This marks the end of the second phase because the steady state conditions are no longer valid. Usually the next phase is not simulated, since it is only a small part of the process. The last centimeters of the billet are not extruded, since it contains the build up contamination and oxidation of the complete billet [36]. This last part, the butt-end, is sheared off at the die face. The next billet is placed and the loop is started over.

In this thesis only the start-up and steady state simulations are shown. Steady state simulations are convenient since they are much less time consuming than the transient (start-up and butt-end) simulations. However start-up gives the designer very valuable information about the effectiveness of the die. The actual nose piece can be derived and used for evaluation of the die. In the next sections the framework for both start-up and steady state simulations are described.

2.2 Different FEM formulations

Aluminium extrusion can be modeled with different types of FEM. In this section some types of FEM formulations are discussed and assessed on their suitability for modeling aluminium extrusion. The different phases in billet-to-billet extrusion raise their own demands. In the start-up phase the formulation should be able to describe large deformations and free surfaces. In the steady state phase, the ability to model a free surface is not required.

2.2.1 Lagrangian formulation

In forming processes a Lagrangian formulation is often used. The mesh is fixed to the material and deforms accordingly. In this formulation the frame of reference is equal to the initial geometry (Total Lagrangian) or equal to the deformed geometry and moving with the material (Updated Lagrangian).

Since the mesh is fixed to the material, free surfaces are accurately followed. A drawback of the updated Lagrangian formulation is that when the material undergoes large deformations, the mesh can become severely distorted. Distorted elements will lead to less accurate results or when elements flip inside out lead to a premature end of the simulation.

To accommodate these problems to be able to model aluminium extrusion with an updated Lagrangian formulation, remeshing techniques have been developed. Remeshing is widely used for simulation of forming processes. The most important issues in remeshing are the creation of a new mesh and the remapping of the state variables from the old mesh onto the new mesh.

The main drawback of using an Updated Lagrangian formulation with remeshing for the simulation of aluminium extrusion is the time involved in remeshing and remapping.

2.2.2 Eulerian formulation

In an Eulerian formulation material flows through the mesh. The mesh is kept at the initial location. This will avoid all of the problems involving the mesh distortions. However since free surfaces are almost never equal to the element edges, tracking the free surface is not as straight forward as in Lagrangian formulations. In chapter 4 some procedures for tracking free surfaces are treated. Where in Lagrangian formulations the history dependent state variables are readily known, in Eulerian formulations they have to be convected through the mesh every step.

2.2.3 ALE formulation

The two formulations described above can be combined to the Arbitrary Lagrangian Eulerian (ALE) formulation. In an ALE formulation the update of the frame of reference is neither equal to the material displacement (Lagrangian) nor zero (Eulerian), but can be chosen arbitrarily, independent of the material displacement [16, 29, 30]. Like in Eulerian formulations history dependent state variables have to be convected.

With ALE it is possible to simulate aluminium extrusion in an Eulerian formulation with the possibility to model some free surfaces. This is only possible if the free surface coincides with the free surface of the mesh. In figure 2.2 this is shown for the bearing area and the profile from [43]. The mesh displacement

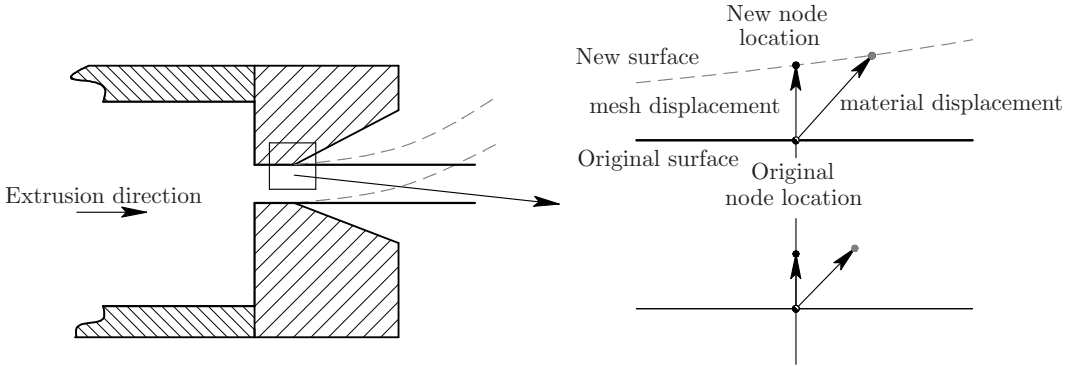


Figure 2.2: Control of the mesh on free surfaces inside the bearing channel and profile.

on the free surfaces in the bearing is defined to be perpendicular to the original surface. This means that material is allowed to flow through the mesh in extrusion direction, but normal to the surface the mesh follows the new surface.

The nodes on the inside of the bearing area, between the two surfaces can be fully Eulerian in the bearing area since displacements are small. In the profile the displacements perpendicular to the extrusion direction can be much greater. The mesh control for the inside nodes, ensures that these nodes are always equally spaced between two opposing surface nodes. This is done by interpolation of the mesh displacement of the two surface nodes.

2.3 Arbitrary Lagrangian Eulerian formulation

In an ALE formulation both material and mesh displacements have to be calculated. To follow path dependent properties, the total load is applied in a number of time steps.

In the derivation of the ALE formulation in this thesis the work of Wiselink [74] is closely followed. It is assumed that the state at time $t = t^n$ is known on the domain Ω_g^n . At the beginning of the step the material domain is equal to the grid domain $\Omega_g^n = \Omega_m^n$. The grid and material displacements during a step from $t = t^n$ to $t = t^{n+1}$ for point $\mathbf{x}$ are given by equations (2.1)

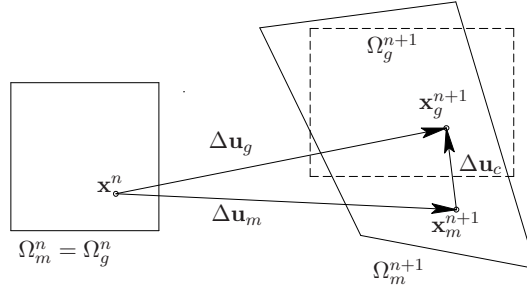


Figure 2.3: Material and grid domain displacements.

and (2.2).

$$\mathbf{x}_m^{n+1} = \mathbf{x}^n + \mathbf{v}_m \Delta t = \mathbf{x}^n + \Delta \mathbf{u}_m \quad (2.1)$$

$$\mathbf{x}_g^{n+1} = \mathbf{x}^n + \mathbf{v}_g \Delta t = \mathbf{x}^n + \Delta \mathbf{u}_g \quad (2.2)$$

With $\mathbf{v}_g$ the grid velocity and $\mathbf{v}_m$ the material velocity as shown in figure 2.3.

Using a Lagrangian formulation the new material domain is equal to the new grid domain $\Omega_m^{n+1} = \Omega_g^{n+1}$ and $\Delta \mathbf{u}_m = \Delta \mathbf{u}_g$. Using an Eulerian formulation the new grid domain is equal to the old grid domain $\Omega_g^{n+1} = \Omega_g^n$, $\Delta \mathbf{u}_g = 0$ and the incremental convective displacement $\Delta \mathbf{u}_c = -\Delta \mathbf{u}_m$.

In an ALE formulation the state variables have to be calculated in the new grid points at $t = t^{n+1}$. The state variables are calculated using the grid time derivatives $\dot{\xi}^*$. The grid time derivative consists of a material time derivative $\dot{\xi}$ and a convective part.

$$\xi_g^{n+1} = \xi_g^n + \int_t^{t+\Delta t} \dot{\xi}^* \, dt \quad (2.3)$$

$$= \xi_g^n + \int_t^{t+\Delta t} \dot{\xi} \, dt - \int_t^{t+\Delta t} (\mathbf{v}_m - \mathbf{v}_g) \cdot \nabla \xi \, dt \quad (2.4)$$

$$= \xi_g^n + \Delta \xi_g \quad (2.5)$$

2.3.1 Strong form

When FEM is applied to forming simulations the objective is to find the solution for the stresses $\boldsymbol{\sigma}$ and displacements $\mathbf{u}_m$ that fulfills the equilibrium

equations and boundary conditions at t^{n+1} . The surface is split into a part Γ_u where the displacements are prescribed: $\mathbf{u}_0^{n+1}$, and a part Γ_t where the forces are prescribed: $\mathbf{t}^{n+1}$.

$$\begin{aligned} \boldsymbol{\sigma}^{n+1} \cdot \nabla &= 0 && \text{in } \Omega_g^{n+1} \\ \mathbf{u} &= \mathbf{u}_0^{n+1} && \text{on } \Gamma_u \\ \boldsymbol{\sigma}^{n+1} \cdot \mathbf{n} &= \mathbf{t}^{n+1} && \text{on } \Gamma_t \end{aligned} \quad (2.6)$$

with $\mathbf{n}$ the outward normal on the boundary. Furthermore inflow conditions are set for state variables and displacements at the boundary Γ_f where material flows into the domain.

$$\begin{aligned} \mathbf{u} &= \mathbf{u}_f && \text{on } \Gamma_f \\ \xi &= \xi_f && \text{on } \Gamma_f \end{aligned} \quad (2.7)$$

2.3.2 Weak form

In FEM the equilibrium equations are only weakly enforced. The constitutive relations are the material rate of the stress-strain relations. Therefore the rate of the weak form is used. In appendix A the rate of the weak form for ALE is derived. Here only the result is given:

$$\begin{aligned} \int_{\Omega} \mathbf{w} \nabla : [-\mathbf{L}_g \cdot \boldsymbol{\sigma} + \dot{\boldsymbol{\sigma}} - (\mathbf{v}_m - \mathbf{v}_g) \cdot \nabla \boldsymbol{\sigma} + \boldsymbol{\sigma} \text{tr}(\mathbf{L}_g)] \, d\Omega = \\ \int_{\Gamma} \mathbf{w} \cdot \dot{\mathbf{t}} \, d\Gamma - \int_{\Gamma} \mathbf{w} \cdot [(\mathbf{v}_m - \mathbf{v}_g) \cdot \nabla \boldsymbol{\sigma}] \mathbf{n} \, d\Gamma \end{aligned} \quad (2.8)$$

With $\mathbf{w}$ the weighing function, $\mathbf{L}_g$ the gradient of the grid velocity and the integration is over the spatial coordinates. For an Updated Lagrangian formulation, since $\mathbf{v}_m = \mathbf{v}_g$, this degenerates to:

$$\int_{\Omega} \mathbf{w} \nabla : [-\mathbf{L} \cdot \boldsymbol{\sigma} + \dot{\boldsymbol{\sigma}} + \boldsymbol{\sigma} \text{tr}(\mathbf{L})] \, d\Omega = \int_{\Gamma} \mathbf{w} \cdot \dot{\mathbf{t}} \, d\Gamma \quad (2.9)$$

With $\mathbf{L}$ the gradient of the material velocity.

2.3.3 Coupled and semi-coupled ALE

For a coupled ALE formulation equation (2.8) is discretized. In a coupled formulation both material and convective terms are solved simultaneously and therefore a coupled formulation yields the best possible solution of the ALE problem. In figure 2.4(a) the solution strategy for a coupled ALE formulation is shown. The complete derivation can be found in Appendix A.

The velocity is assumed constant during a step:

$$\mathbf{v}_m = \frac{\Delta \mathbf{u}_m}{\Delta t}; \quad \mathbf{v}_g = \frac{\Delta \mathbf{u}_g}{\Delta t} \quad (2.10)$$

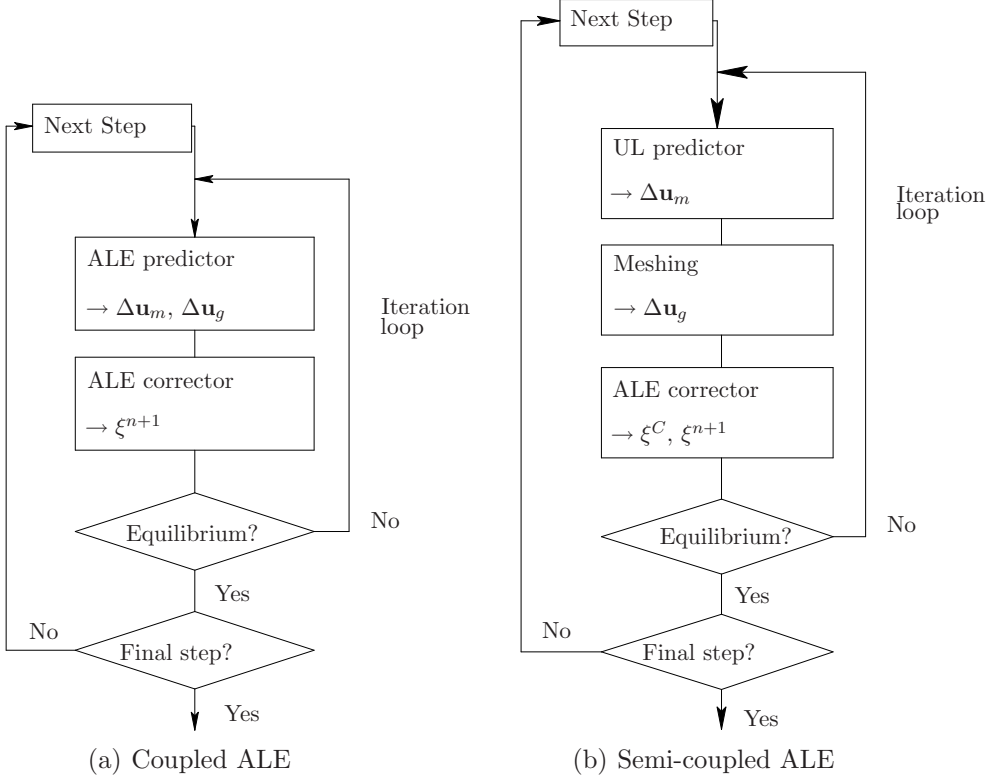


Figure 2.4: Material and grid domain displacements.

The discretized form, equation (2.11), now contains two unknown displacement increments, the material displacement increment $\Delta \mathbf{u}_m$ and the grid displacement increment $\Delta \mathbf{u}_g$.

$$\begin{bmatrix} K_m & K_g \\ A & B \end{bmatrix} \begin{Bmatrix} \Delta \mathbf{u}_m \\ \Delta \mathbf{u}_g \end{Bmatrix} = \begin{Bmatrix} \Delta F_{ext} \\ C \end{Bmatrix} \quad (2.11)$$

The matrices A, B and the vector C define the relation between $\Delta \mathbf{u}_m$ and $\Delta \mathbf{u}_g$. These are necessary to solve the equations.

To solve this set of nonlinear relations an iterative scheme, figure 2.4(a) is used. The scheme consists of a predictor and a corrector part. The first approximation for the incremental displacements $\Delta \mathbf{u}_m^1$ and $\Delta \mathbf{u}_g^1$ is found by linearizing equation (2.11) with the known state variables at the beginning of the step. Assembling and solving these equations is called the predictor. The superscript i denotes the iteration number.

$$\begin{bmatrix} K_m^0 & K_g^0 \\ A & B \end{bmatrix} \begin{Bmatrix} \Delta \mathbf{u}_m^i \\ \Delta \mathbf{u}_g^i \end{Bmatrix} = \begin{Bmatrix} \Delta F_{ext} \\ C \end{Bmatrix} \quad \text{for } i = 1 \quad (2.12)$$

In the corrector the new state variables are calculated based on the results of the predictor. The calculation is the integration of the state variables in time as in equation (2.5). This integral consists of two parts, a material part and a convective part. In sections 2.3.4 and 2.3.5 these parts will be treated.

Based on the new stresses from the corrector step the nodal internal reaction forces $\{F_{int}\}$ can be calculated. The difference between these forces and the prescribed forces $\{F_{ext}\}$ is the unbalance after the iteration step.

$$\{R^i\} = \{F_{ext}\} - \{F_{int}\} \quad (2.13)$$

When the unbalance ratio, ϵ^i is smaller than an allowed tolerance ϵ_t the system is in equilibrium and the iteration process is stopped.

$$\epsilon^i = \frac{\|\{R^i\}\|}{\|\{F_{int}\}\|} \leq \epsilon_t \quad (2.14)$$

If the unbalance criterion is not fulfilled a next iteration is performed. The matrices in the predictor will be calculated based on the state variables from the last iteration. The unbalance force from the previous iteration is taken to the next iteration:

$$\begin{bmatrix} K_m^{i-1} & K_g^{i-1} \\ A & B \end{bmatrix} \begin{Bmatrix} \Delta \Delta \mathbf{u}_m^i \\ \Delta \Delta \mathbf{u}_g^i \end{Bmatrix} = - \begin{Bmatrix} \Delta R^{i-1} \\ 0 \end{Bmatrix} \quad \text{for } i > 1 \quad (2.15)$$

Semi-coupled ALE

An advantage of a semi-coupled ALE formulation compared to the coupled ALE formulation is the flexibility in defining the new grid, as in semi-coupled ALE the new grid is defined after the Lagrangian step [74]. In this thesis a semi-coupled formulation is used.

In figure 2.4(b) the scheme for a semi-coupled ALE formulation is shown. With respect to the coupled ALE scheme, the ALE predictor is replaced by a UL predictor by setting $\Delta \mathbf{u}_g = \Delta \mathbf{u}_m$ and discretizing equation (2.9). After the UL predictor a meshing step is performed to calculate the grid displacement increment $\Delta \mathbf{u}_g$.

In the corrector step first the convective increment of the state variables ξ^C is calculated based on the values at the beginning of the step.

$$\xi^C(\mathbf{x}_g^{n+1}) = \xi^0(\mathbf{x}_m^n - \Delta \mathbf{u}_c) \quad (2.16)$$

The problem in calculation of $\xi^C(\mathbf{x}_g^{n+1})$ is that ξ^0 is only known in the integration points and the field is discontinuous. In section 2.3.5 this problem is treated in more detail.

Next the material increment is calculated by integration of the material rate in time:

$$\xi^{n+1} = \xi^C + \int_t^{t+\Delta t} \dot{\xi} dt \quad (2.17)$$

With these new state variables the unbalance is calculated. By comparison with the unbalance criterion is checked whether the new state variables are in equilibrium.

The advantage of the semi coupled ALE method is the flexibility in meshing compared to the coupled ALE. Also the equilibrium is fulfilled at the end of a step. The semi-coupled formulation is less efficient than the decoupled, since the convection and meshing are performed each iteration step.

In the next paragraphs is shown how the material and convective increment resp. are calculated.

2.3.4 Material increment

The constitutive equations describe the relation between the stresses and the strains. In this section the constitutive elasto-viscoplastic model is presented, which has been implemented into the FEM code.

The strain is decomposed in an elastic and a viscoplastic part. The equation is given in incremental form, obtained by integrating the constitutive relations in rate form. The integration is over a time increment $[t_n, t_{n+1}]$ with

an implicit Euler backward scheme.

$$\Delta \boldsymbol{\varepsilon} = \Delta \boldsymbol{\varepsilon}^e + \Delta \boldsymbol{\varepsilon}^{vp} \quad (2.18)$$

Hooke's law relates the Cauchy stress tensor $\boldsymbol{\sigma}$ to the elastic strain tensor. The Cauchy stress tensor relates to the material frame of reference. In appendix A the stress tensor for a co-rotational frame of reference is given. In equation (2.19) $\mathbf{E}$ is a fourth order tensor based on the Young's modulus E and the poisson ration ν .

$$\Delta \boldsymbol{\sigma} = \mathbf{E} : (\Delta \boldsymbol{\varepsilon} - \Delta \boldsymbol{\varepsilon}^{vp}) \quad (2.19)$$

The viscoplastic strain increment is assumed to be in the direction of the deviatoric stress. The plastic multiplier $\Delta \lambda$ is used to scale the viscoplastic strain rate.

$$\Delta \boldsymbol{\varepsilon}^{vp} = \Delta \lambda \mathbf{s}^{n+1} \quad (2.20)$$

with $\mathbf{s}$ the deviatoric stress tensor defined as $\boldsymbol{\sigma} = \mathbf{s} - p\mathbf{I}$ and the hydrostatic pressure as $p = -\text{tr}(\boldsymbol{\sigma})$. The limit function describes the rate dependent behavior of the material in the plastic domain.

$$g_{n+1}(\boldsymbol{\sigma}_{n+1}, \kappa_{n+1}, \dot{\kappa}_{n+1}) = \bar{\sigma}_{n+1} - \sigma_y(\kappa_{n+1}, \dot{\kappa}_{n+1}, T) \quad (2.21)$$

The Von Mises criterion is used to define the effective stress $\bar{\sigma}$.

$$\bar{\sigma} = \sqrt{\frac{3}{2} \mathbf{s} : \mathbf{s}} \quad (2.22)$$

The flow stress σ_y is a function of the state variables, κ and $\dot{\kappa}$, representing the equivalent plastic strain and equivalent plastic strain rate respectively.

$$\kappa_{n+1} = \kappa_n + \sqrt{\frac{2}{3} \Delta \boldsymbol{\varepsilon}^{vp} : \Delta \boldsymbol{\varepsilon}^{vp}} \quad (2.23)$$

$$\dot{\kappa}_{n+1} = \frac{\sqrt{\frac{2}{3} \Delta \boldsymbol{\varepsilon}^{vp} : \Delta \boldsymbol{\varepsilon}^{vp}}}{\Delta t} \quad (2.24)$$

From equation (2.20) and (2.24) the relation between the plastic multiplier and the incremental equivalent plastic strain can be derived.

$$\begin{aligned} \Delta \kappa_{n+1} &= \sqrt{\frac{2}{3} \Delta \boldsymbol{\varepsilon}^{vp} : \Delta \boldsymbol{\varepsilon}^{vp}} = \sqrt{\frac{2}{3} \Delta \lambda \mathbf{s}^{n+1} : \Delta \lambda \mathbf{s}^{n+1}} \\ \Delta \lambda \sqrt{\frac{2}{3} \mathbf{s}^{n+1} : \mathbf{s}^{n+1}} &= \frac{2}{3} \Delta \lambda \sqrt{\frac{3}{2} \mathbf{s}^{n+1} : \mathbf{s}^{n+1}} = \frac{2}{3} \Delta \lambda \bar{\sigma}_{n+1} \end{aligned} \quad (2.25)$$

The material can be in two states. In the elastic state the plastic multiplier is zero and the limit function is smaller than or equal to zero. In the viscoplastic state the plastic multiplier is greater than zero and the limit function is equal to zero. This is captured by the Kuhn-Tucker loading-unloading conditions.

$$\Delta\lambda \geq 0, \quad \Delta\lambda g_{n+1} = 0, \quad g_{n+1} \leq 0 \quad (2.26)$$

In the correction step the new total strain increments are calculated. Based on these strains a new elastic trial stress is calculated. If the trial stress fulfills the Kuhn-Tucker condition this is assumed the actual stress and the material is elastic.

If the Kuhn-Tucker relations are not satisfied by the elastic trial stress, the material is in the plastic state. The stresses are then calculated by enforcing that the limit function $g_{n+1} = 0$. Looking at the deviatoric stresses and strains only, equation (2.19) can be written as:

$$\mathbf{s}^{n+1} = \mathbf{s}^n + 2G(\Delta\mathbf{e} - \Delta\lambda\mathbf{s}^{n+1}) \quad (2.27)$$

and for the limit function in equation (2.21):

$$g_{n+1} = \sqrt{\frac{3}{2}\mathbf{s}^{n+1} : \mathbf{s}^{n+1}} - \sigma_y(\kappa_{n+1}, \dot{\kappa}_{n+1}, T) = 0 \quad (2.28)$$

With G the shear modulus and $\Delta\mathbf{e}$ the deviatoric part of the total strain increment.

Sometimes an explicit equation for the plastic multiplier $\Delta\lambda$ can be found, but in most cases an iterative approach is necessary to determine $\Delta\lambda$. For this purpose a Newton-Raphson method can be used, but calculation of the derivative of g can be expensive to compute. In this work a secant method is used, which requires only the evaluation of g [43].

To obtain objective integration the derivation of Lof [44] is closely followed. Lof chooses to use a corotational formulation. The initial deviatoric stress $\mathbf{s}^{n+1}$ is replaced by $\bar{\mathbf{s}}^{n+1}$:

$$\bar{\mathbf{s}}^{n+1} = \mathbf{R}\mathbf{s}^{n+1}\mathbf{R}^T \quad (2.29)$$

Where $\mathbf{R}$ follows from the polar decomposition of the incremental deformation gradient $\mathbf{F}$.

Material model

The constitutive behavior of aluminium is represented by the limit function equation (2.21) and a relation for the flow stress. In this work the simulations are performed using the Sellars-Tegart law [43].

$$\sigma_y = s_m \operatorname{arcsinh} \left(\left(\frac{\dot{\kappa}}{A} \exp \left(\frac{Q}{RT} \right) \right)^{\frac{1}{m}} \right) \quad (2.30)$$

The relation is given between the flow stress σ_y , $\dot{\kappa}$ and T . s_m and m are strain sensitivity parameters. These are assumed to be constant. Q is the apparent activation energy of the deformation process during plastic flow. R is the universal gas constant, T the absolute temperature and A is a factor, depending explicitly on the Magnesium and Silicium content in the aluminium matrix. Because of the high temperatures during extrusion, the strain hardening can be neglected [31].

In table 2.1 the material properties for AA6063 alloy are summarized. In this work simulations are performed using these properties, unless mentioned otherwise.

Table 2.1: Material parameters for AA6063 alloy [43].

	Symbol	value
Elastic properties ($T=823\text{K}$)	E [MPa]	40000
	ν [-]	0.4995
Plastic properties	s_m [MPa]	25
	m [-]	5.4
	A [1/s]	$6 \cdot 10^9$
	Q [J/mol]	$1.4 \cdot 10^5$
	R [J/molK]	8.314

The reason for $\nu = 0.4995$ is to reduce the amount of elastic compression at the start of the simulations. With an value that makes the aluminium nearly incompressible, the simulation reaches the steady state much faster.

2.3.5 Convective increment

For semi-coupled ALE the convective increment has to be calculated every iteration. This convective increment can be written as a convection or interpolation problem or a mix of those two. Since the velocity is assumed constant

during a step the convective increment can be written as:

$$\Delta\xi^C = \int_t^{t+\Delta t} -\mathbf{v}_c \cdot \nabla\xi dt \approx -\Delta\mathbf{u}_c \cdot \nabla\xi \quad (2.31)$$

The main difficulty is to calculate the gradient $\nabla\xi$. Stresses and strains are only evaluated in the integration points and are discontinuous over the element boundaries. This means that the gradient $\nabla\xi$ cannot be determined locally, since this would ignore the jump over the element boundaries. Therefore information of neighboring elements is necessary to construct a global gradient.

The history dependent state variables, like equivalent plastic strain and stress, are known in the integration points of the elements. These state variables have to be remapped from the old location of the integration points to the new locations. Nodal values, like displacements, are not necessary for the FEM analysis, but can be useful for the analysis of the results.

Courant number

For the calculation of the convective increment the Courant number is used. In this thesis the Courant number is defined as the difference in local coordinates between the old and the new integration point locations, divided by the characteristic element length l . This depends on the element type of choice.

$$C_r = \frac{1}{nr_{ip}} \sum_{ip=1}^{nr_{ip}} \frac{|\Delta r_{ip}|}{l}, \quad C_z = \dots \quad (2.32)$$

The three courant numbers C_r , C_z and C_h are calculated for the different directions in every integration point. The element average is given in equation (2.33). The total courant number is written as:

$$C = \sqrt{C_r^2 + C_z^2 + C_h^2} \quad (2.33)$$

2.3.6 Convection scheme

Many methods for the calculation of $\Delta\xi^C$ have been proposed. Two main methods can be distinguished. The *finite element method* used by [29, 74] and the *finite volume method* used by [28].

Usage of the finite element method for the convective increment in an ALE

method has the main advantage that the Lagrangian step is also performed with the finite element method, which facilitates the implementation of the method. One of the Taylor-Galerkin methods [15] is the WLGS scheme proposed by [29]. This scheme requires a continuous distribution of the state variable across the element boundaries. Creating this continuous field tends to smooth the gradients between the elements.

The discontinuous Galerkin method, which is a generalization of the finite volume method, [21,23,37,53] allows for discontinuities across element boundaries. In [23] a second order Discontinuous Galerkin scheme is presented. The 2D convection tests yield far better accuracy than standard discontinuous Galerkin methods and the WLGS scheme. The scheme is also attractive in an ALE method since it uses the existing mesh of the Lagrangian step. Development and implementation of 3D versions of this scheme would be an interesting topic.

In this research the focus is mainly on the convection of nodal values. Since nodal values are already continuous, it can be expected the WLGS scheme yields more accurate results than for integration point values. In chapter 4 the accuracy of the WLGS scheme for nodal values is shown.

Weighted local and global smoothing

The new integration point values are calculated by the use of the weighted local and global smoothing (WLGS) scheme described in [29,71,74]. In this thesis a summary of the WLGS scheme presented in Wisselink [74] is given.

The WLGS is an interpolation approach, which uses an intermediate step to create a smoothed continuous field from the integration point values. There are two steps to be distinguished in the WLGS scheme:

1. At the end of each step a continuous field is constructed from the integration point data values. The same field used at the beginning of the next step
 - a First the integration point values are extrapolated to the nodes, with a *local smoothing* factor
 - b From the extrapolated values the nodal averages are calculated
 - c These averaged values are interpolated back to the integration points

2. The integration point values are calculated in the new integration points locations using the continuous field
 - a First the Courant numbers are calculated
 - b Based on the Courant numbers the *global smoothing* factor is calculated
 - c The integration point values in the new integration point locations are calculated using the global smoothing factor and the continuous fields

The extrapolation from the integration points to a node is performed using the shape functions and the local smoothing factor β

$$\xi_{node} = \sum_{k=1}^{nr_{node}} N^k(r_{ip}\beta_r, z_{ip}\beta_z, h_{ip}\beta_h)\xi_{ip}^k \quad (2.34)$$

In figure 2.5 the extrapolation with the local smoothing factor β is explained. When β is chosen equal to zero the element average is used as integration point and nodal values. When β is chosen equal to $\sqrt{3}$ the integration point values are extrapolated without smoothing. After the extrapolation a nodal value is

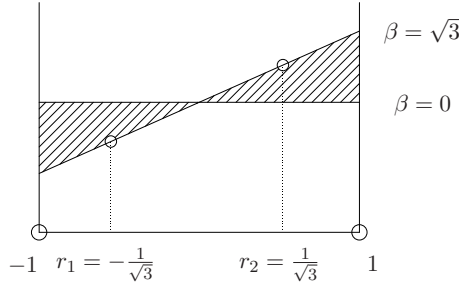


Figure 2.5: Local smoothing factor.

derived. This is done by averaging the values of the surrounding elements of a node.

$$\xi_{node}^* = \frac{1}{ne} \sum_{k=1}^{ne} \xi_{node}^k \quad (2.35)$$

In figure 2.6 the smoothing effect of β can be found. It is clear when choosing $\beta = 0$, the continuous field is much smoother than when no local smoothing is applied. Wisselink shows that local smoothing is necessary for a stable convec-

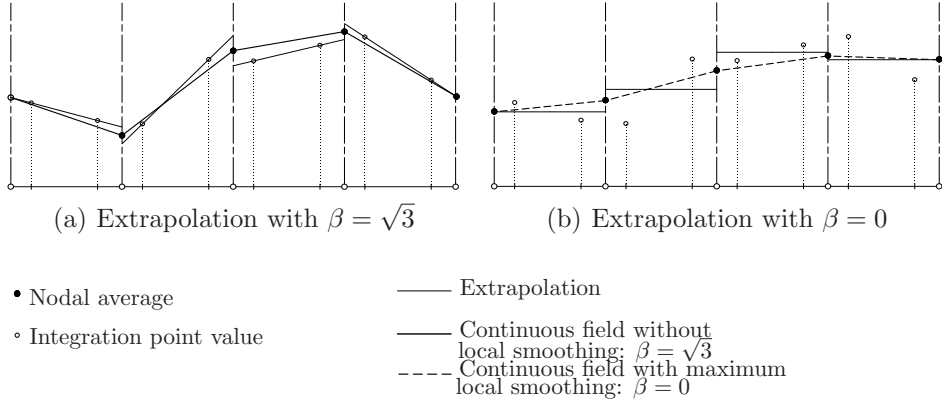


Figure 2.6: Local smoothing and nodal averaging.

tion. Choosing β equal for all directions leads to much cross-wind diffusion. To prevent that, a flow direction dependent β is derived. This leads to an orthotropic local smoothing. An isotropic smoothing parameter $\bar{\beta}$ is applied, to allow a little cross-wind diffusion.

$$\beta_r = (1 - C_r)(1 - \left| \frac{C_r}{C} \right|) \cdot \bar{\beta} \cdot \sqrt{3}; \quad \beta_z = \dots \quad (2.36)$$

The second part of the WLGS scheme is to calculate the state variable values in the new locations of the integration points $(r, z, h)^{new}$.

$$(r, z, h)^{new} = (r, z, h)^{old} - (\Delta_r, \Delta_z, \Delta_h) \quad (2.37)$$

The state variables can be calculated by either an interpolation approach or a convection approach. The first approach is very diffusive and the second approach shows spurious oscillations [29]. The state variables are calculated as a weighted combination of the two methods.

$$\xi_{ip}^{new} = \sum_{node=1}^{nr_{node}} N^k((r, z, h)^{new}) \xi_{node}^{*k} + \quad (2.38)$$

$$(1 - \alpha) \left(\xi_{ip}^{old} - \sum_{node=1}^{nr_{node}} N^k((r, z, h)^{old}) \xi_{node}^{*k} \right) \quad (2.39)$$

The factor α is the global smoothing factor. In [74] an appropriate choice for α for H8 elements is given:

$$\alpha = \min(1.64C^{1.2}, 1) \quad (2.40)$$

Convective increment for nodal values

The total displacements are nodal values. For some elements pressures are nodal values as well instead of integration point values. The convection of nodal values is treated slightly different compared to integration point values. This is discussed in chapter 4

2.4 Modeling aluminium extrusion

In this section specific aspects of FEM analysis of aluminium extrusion are treated. First is described how a 3D FEM model for extrusion is created. In this pre-processing some simplifications are made to the die design to be more time efficient in both pre-processing and solving the equations.

2.4.1 Pre-processing

A FEM model consists of a mesh with nodes and elements representing the geometry and of boundary conditions to model the interaction with the environment. Pre-processing a complex 3D geometry for a FEM simulation can be a time consuming process. The time can be reduced if the necessary steps for preparing a simulation are automated. In figure 2.7 those steps are shown for a porthole die used for extrusion of a tube.

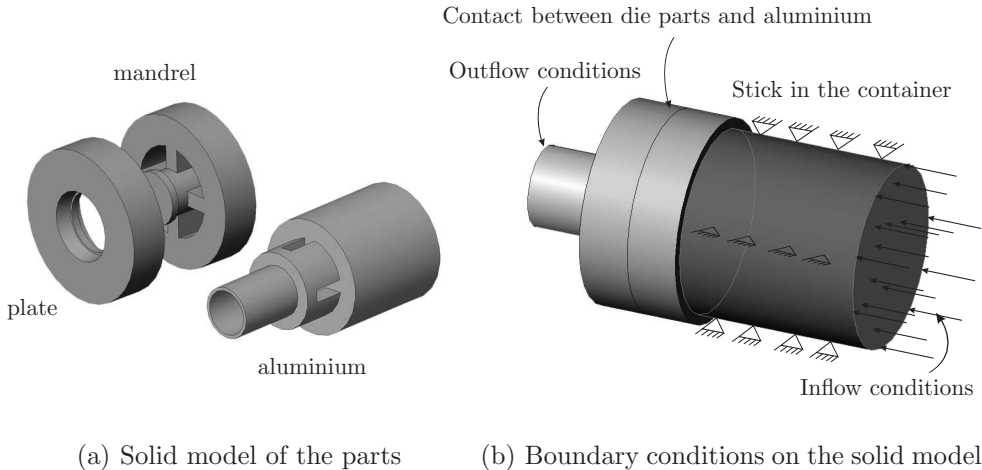


Figure 2.7: Solid model and boundary conditions.

The starting point for the pre-processing is the 3D geometry of the dies and the

aluminium. In this research SolidWorks is used for modeling the 3D geometry. The aluminium inside the dies is taken as a negative of the die cavity. Parts of the profile and billet are added to that negative.

Before the mesh is created the boundary conditions are applied to the solid model as in figure 2.7(b). All the boundary conditions are applied to the surface areas or surface lines. The most important boundary conditions are:

Inflow of aluminium At the inflow surface of the billet the material velocity is prescribed uniform and equal to the ram speed. The outer edge of the inflow surface touches the container wall. Here a choice has to be made whether to have a sticking condition here or a slip condition with material velocity. In this thesis, the outer edge has the material inflow velocity and a free slip condition with the container. This condition ensures that the whole inflow surface has a uniform velocity and the inflow is determined exactly. This is convenient for setting the ram speed and testing the volume conservation properties.

Outflow of aluminium At the outflow surface of the billet the material velocity can be constrained to be uniform or otherwise constrained. In the simulations in this thesis, no special conditions are applied at the outflow, unless mentioned otherwise.

Contact between aluminium and the container Contact between aluminium and container is always modeled as stick. In other words, the material displacement is equal to zero in all directions. That this assumption is allowed has been shown in [43].

Contact between aluminium and die The contact between aluminium and die parts is also modeled as stick, except for the bearing area. Here special conditions are applied. These conditions are treated in the next paragraphs.

Definition of bearing area To transfer the special conditions in the bearing area from the solid model to the FEM model, the bearing area has to be defined. The boundary conditions on the bearing area are special and regular pre-processors don't have the ability to apply these conditions. Therefore these conditions are applied in a next phase. In this phase dummy forces are placed on the nodes on the bearing surface. In the next phase these dummy forces are used as markers and not transferred as force.

Definition of bearing corner The boundary on the bearing corner also needs special attention. The definition of these conditions is treated in the next paragraph. On the bearing corner also a marker is placed.

In the next phase of pre-processing the mesh is generated and the boundary conditions are translated to the nodes and/or elements. Since the meshing can be done with standard 4 or 10 node tetrahedral elements and only simple boundary conditions are applied, the FEM plug-ins available in most standard 3D CAD programs are sufficient. In this research the SolidWorks plug-in CosmosWorks is used. Exporting a FEM model is much more straight forward than a solid model, since the geometry has been discretized into elements.

The boundary conditions prescribed on the solid model are now transferred to the element model. On the faces where components are in contact, the opposing faces get equal node distributions. If the aluminum flow is simulated with rigid dies, the die deformation can be calculated afterwards by transferring the loads from the aluminium flow to the die parts. When it concerns contact between die parts, the generated contact elements can be used.

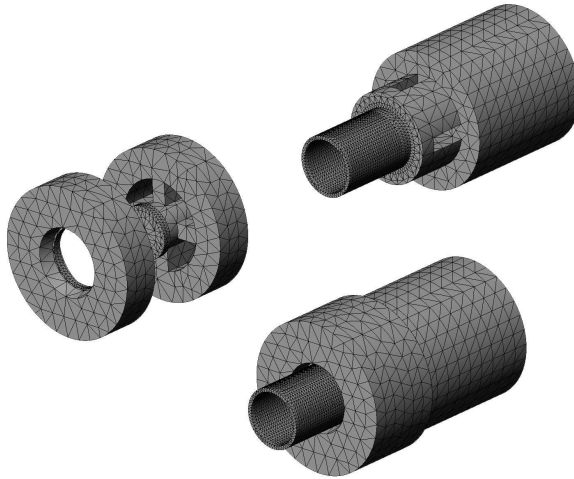


Figure 2.8: FEM model of parts and total model.

Nodes on a marked surface get the same markers as the surface. The final step is to translate the mesh and boundary conditions to input for our in-house FEM code DieKA [29]. This translation mainly consists of rewriting the data in the right form.

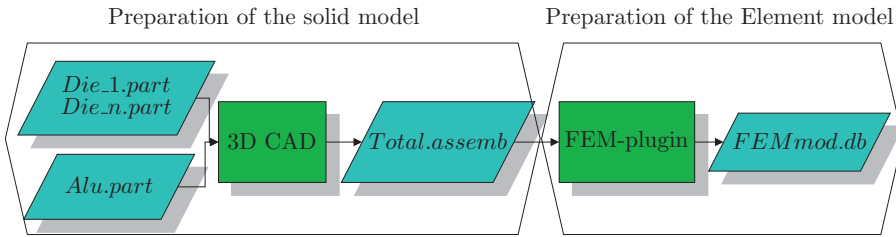


Figure 2.9: Dataflow in solid and element modeling.

In this step the marked nodes on the bearing surface get their final boundary conditions as well. The data flow in the solid modeling is illustrated in figure 2.9. The format of the FEM model *FEMmod.db* depends on the export possibilities of the FEM plugin.

The input file for the final pre-process step is the *FEMmod.db* database file. This input is translated to DiekA input with a macro and some additional information is given to the macro. The additional information is:

Temperature In this thesis all the simulations are performed isothermally, so only the initial billet temperature is input

Element type The output from the chosen FEM plug-in, CosmosWorks, are 10 node tetrahedral elements. In the translation to DiekA the connectivity of 10 node elements can be used as it is or be used to create linear tetrahedral elements.

Ram speed Here the ram speed can be set. The ram speed that is prescribed to the solid model and transferred the FEM model is used or a new value can be set.

Symmetry The XZ-plane and YZ-plane can be used as symmetry plane. The nodes in these planes are suppressed in the direction normal to the plane.

Material properties The material properties for the equation (2.30) are input.

The dataflow and output from the final steps in pre-processing are represented in figure 2.10. The five output files from the translator are:

Dieka.inv This is the main file where the other files from the pre- processing are referred. Furthermore material properties and analysis options are

defined in this file. Also the boundary condition in the bearing area is prescribed here.

Alu_model.node Aluminium nodal data

Alu_model.elem Aluminium element data

Alu_model.supp Aluminium suppressed degrees of freedom. All the suppressed degrees of freedom are joined together in this file.

Alu_model.disp Ram displacement

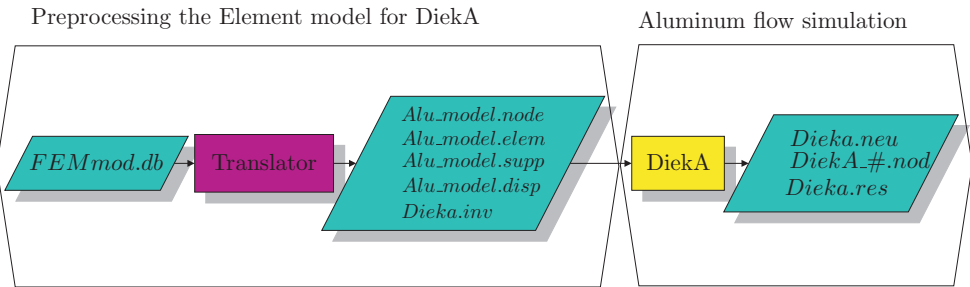


Figure 2.10: Dataflow during pre-processing and simulation.

Die deformation

The forces of the aluminium acting on the die, cause it to deform. Since the dies are also meshed and exported with the *FEMmod.db*, in the translator these parts are also written in DiekA input. The missing loads on the die can be extracted from the aluminium simulation and written into the input file for the die deformation simulation.

2.4.2 Equivalent bearing corner

A specific topic of interest in FEM analysis for aluminium extrusion is the bearing area. In practice a small fillet can be found on the bearing corner. In a FEM simulation this fillet can be modeled with a number of small elements, see (figure 2.11(a)). However, this will make the simulation very time consuming. Therefore an equivalent bearing model is presented that will overcome this problem. It models the bearing area with fewer elements, as in figure 2.11(b).

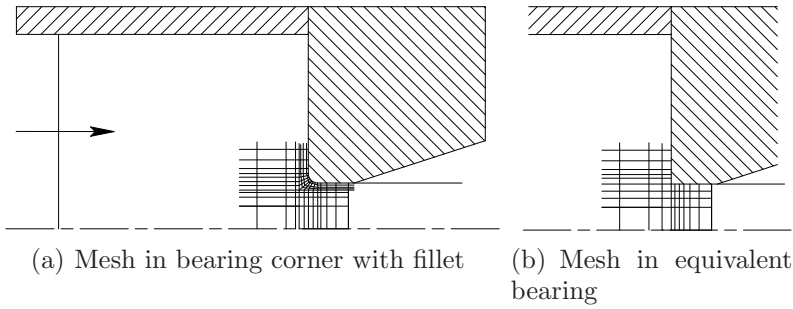


Figure 2.11: Bearing corner and a equivalent bearing corner.

In the past modeling of the bearing corner has been researched by [43, 69]. Both authors give good alternatives to model the bearing corner with a few amount of elements. However the triple node used by Lof [43] gives time consuming challenges in pre-processing the simulation model.

The weighted normal chosen by van Rens [69] is easier to pre-process, but the method requires information from the solution. Hence it is an implicit method where constraints between degrees of freedom are dependent on nearby velocities. To implement this in FEM simulation in most cases the FEM program of choice would have to be modified. Besides that, it can be proven that a weighted calculated normal is not necessarily volume conservative. At the end of this section that is shown.

The main concern in this chapter is to get a fast and robust bearing model that is easily implemented in the main available FEM packages.

Two alternatives are investigated and compared with a reference calculation and the triple node method. The first alternative is a simplification of the weighted average of Van Rens. In the Van Rens method the normal is calculated by the velocities. The simplified normal is a normal that remains the same throughout the simulation. The second alternative is to substitute a chamfer in the location of the fillet, with special conditions at both sides of the fillet.

Reference The reference calculation is built from selectively reduced integrated 4 node bilinear elements. Along the container wall a non slip sticking condition is modeled. On the die face and in the bearing a coulomb type friction with a factor of 0.4 is used. All nodes are Eulerian

except for the nodes on the contact surface, they are ALE with only node movement perpendicular to the surface.

Triple The triple node calculation is built from 9 node quad elements, as in figure 2.12. Along the container wall and die face a stick condition is applied. In the bearing area frictionless slip is modeled. The node in the corner is a triple node. The three nodes on top of each other are constrained in such a manner that material can flow around the corner. All nodes in the simulation are Eulerian.

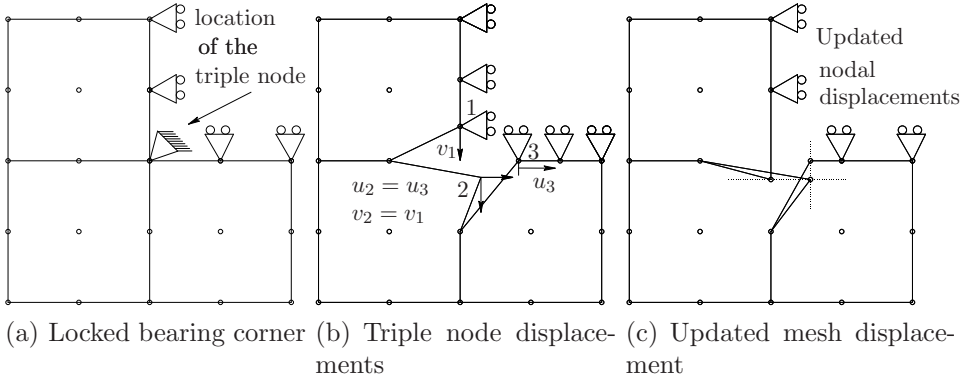


Figure 2.12: Triple node in bearing corner.

Chamfer In the chamfered simulation the nodes of the 9 node quad element are all Eulerian. The fillet in the bearing corner is represented by a chamfer. Along the chamfered edge two elements are placed. The nodes on each side of the chamfer are constrained to have only material displacement tangential to the adjacent surface. The material displacement of the other three nodes (two midside nodes are not shown in the figure) on the side of the chamfer is constraint to be in the direction of the chamfer. The other boundary conditions are equal to those in the Triple node variant.

Normal The elementype is equal to the triple node variant. The bearing corner is modeled by a single node that is constrained to have material displacement with a certain normal (φ in figure 2.13) to the bearing surface. This test is performed for $\varphi = 5, 30, 45, 60, 85$ degrees.

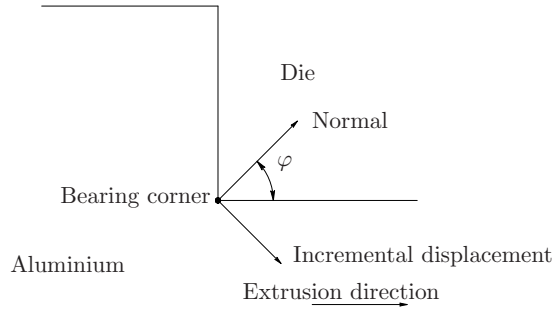


Figure 2.13: Normal on bearing corner.

The quality of the equivalent bearing corner is judged by two parameters. The first parameter is a streamline error. The second measure is the extrusion force, also compared to the reference simulation. The meshes for the four options are shown in figure 2.14.

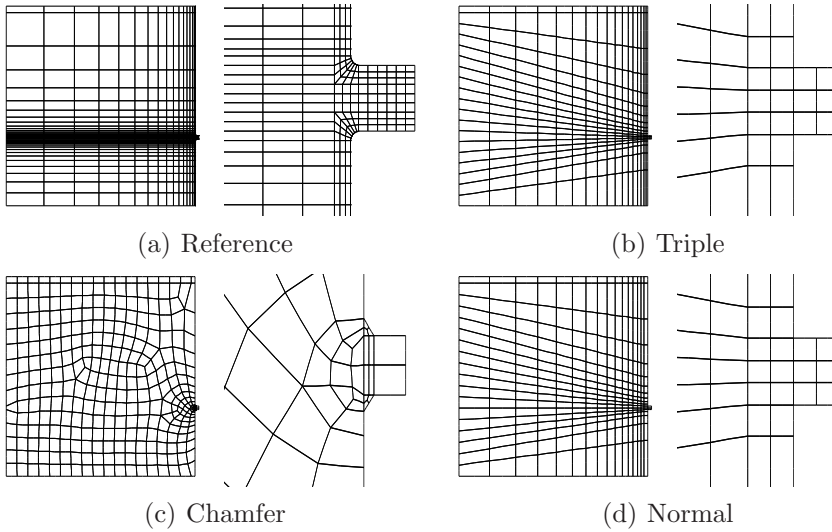


Figure 2.14: Meshes for the four bearing models.

Streamlines

Starting from 5 points in each simulation in the aluminium domain, streamlines are created for every simulation. From the steady state solution stream-

lines can be calculated. In figure 2.15 the departure locations of 5 streamlines are plotted. The superscript i denotes the i^{th} bearing model and the l denotes the l^{th} streamline. The dotted streamlines in this figure are sketched.

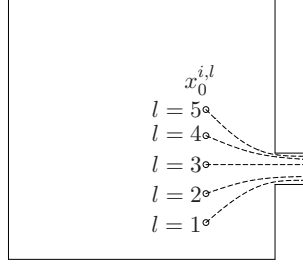


Figure 2.15: Departure locations of the 5 streamlines.

In figure 2.17 the calculated streamlines are plotted for all the bearing corner variants. The error between the streamlines in the equivalent simulations with respect to the streamlines from the reference simulation is the error measure. It is visually almost impossible to determine the 'best' option. Therefore the measure for the error is defined as an euclidean norm of the difference vector Δx , averaged over n steps.

$$E^{i,l} = \frac{1}{n} \sum_{step=1}^n |\Delta x_{step}^{i,l}| = \frac{1}{n} \sum_{step=1}^n |x_{step}^{i,l} - x_{step}^{ref,l}| \quad (2.41)$$

From the simulations it can be found that particles travel the same path as in the reference simulation, but their velocity is higher or lower. With the error measure chosen not only deviations perpendicular to the streamline is taken into account, also the velocity difference contributes to the error (figure 2.16).

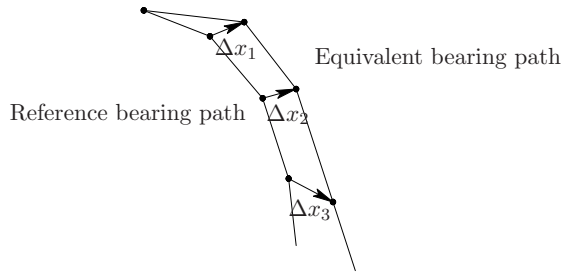


Figure 2.16: particle path difference.

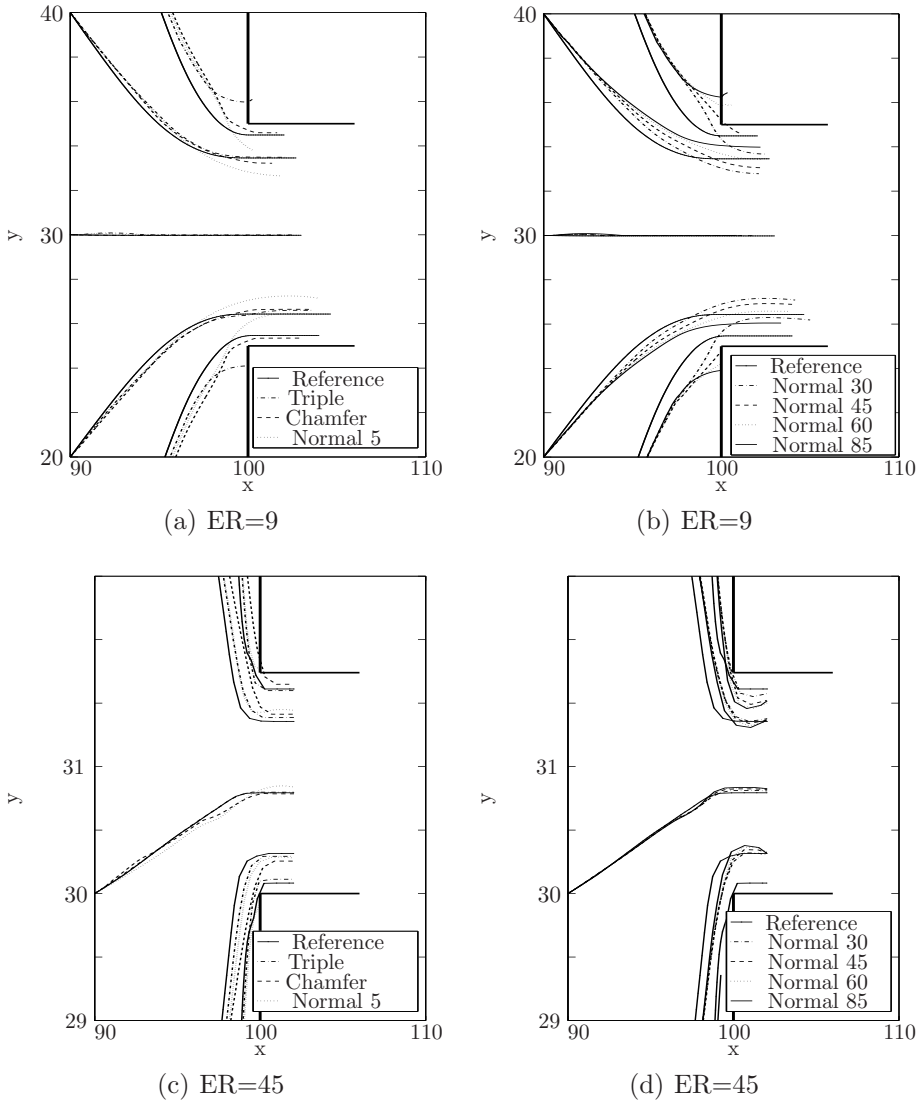


Figure 2.17: Streamlines.

In figure 2.18 the error is plotted. At one axis the 5 streamlines are expanded at the other axis the 8 bearing node variants.

Observing the results for extrusion ratio 9, the triple node out performs all other variants. This effect is less severe for an extrusion ratio of 45. From

figure 2.17 can be seen that the error of streamline 3 is mainly a velocity error.

Extrusion force

In figure 2.19 the ram force is plotted. It can be seen that the triple node always results in a slight under estimation of the extrusion force. This can be explained by the fact that between the elements, that contain one of the triple nodes, friction less slip occurs. According to an upper bound criterion there should be shearing forces at those interfaces.

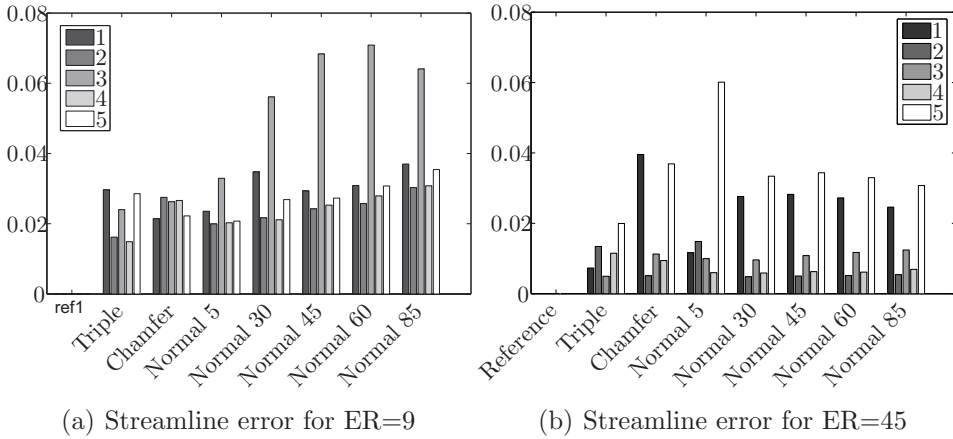


Figure 2.18: Averaged streamline errors for the 8 simulations and two extrusion ratios (a & b).

The chamfered variant gives good results, although only the best solution is shown. The extrusion force depends on the chamfer size and angle, making the success of this variant dependent on the engineer's choices. This can be a desirable feature for advanced engineers. Observing the results for the simulations with the normals, it can be seen that for 45 degrees the extrusion force is in quite good agreement with the reference force, for both extrusion ratios. Because we want to pre-process simple, straight forward and with a minimum of user input, this option is chosen.

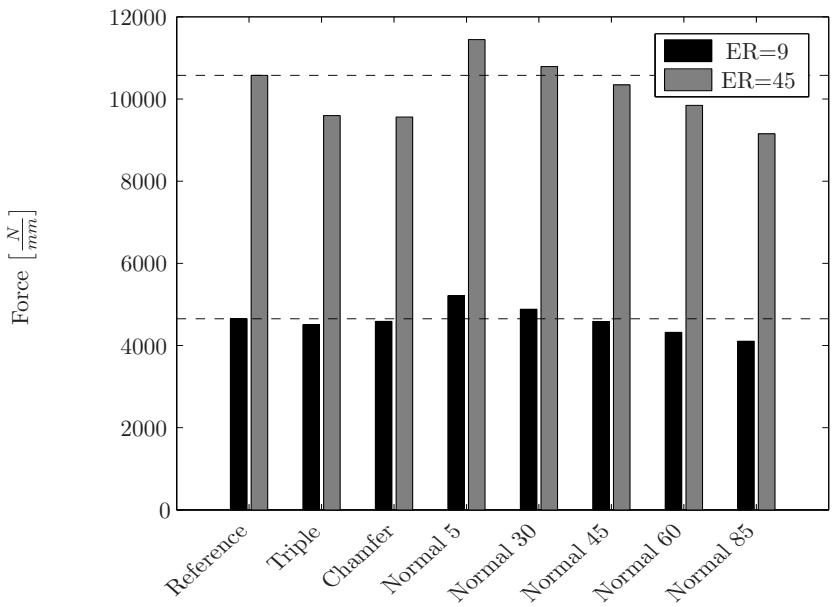


Figure 2.19: Extrusion force for different equivalent bearing options.

Volume conserving properties of the prescribed or averaged normal

When performing ALE simulation volume can be lost or gained in the bearing corner.

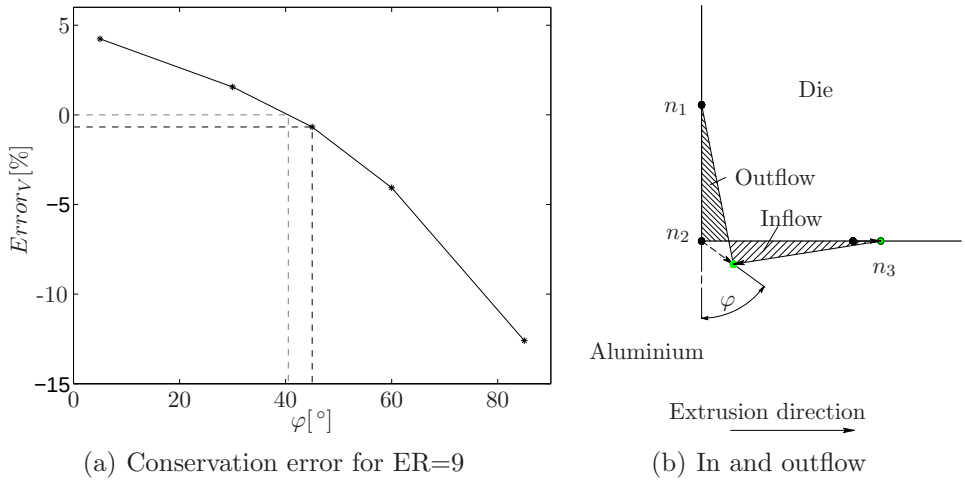


Figure 2.20: Numerical error in volume conservation.

With meshing the nodes to their new positions, the Eulerian part of the step, after the updated solution material can "flow" in or out, as shown in figure 2.20(b)). Depending on the normal, mesh size and incremental displacement of node 3, the bearing corner will be more or less volume conserving. When the normal is chosen horizontal ($\varphi = 0$), there can only be an inflow. When simulations are performed with $\varphi = 90$ there will certainly be volume loss.

In figure 2.20(a) the volume conserving properties are shown for the normal simulation with extrusion ratio 9. The elements on both sides of the corner are approximately the same size and the velocity of node 3 is relatively small compared to the element size. Therefore for approximately 45 degrees the volume is conserved. From figure 2.20(b) it can be seen that when the size of the element in the bearing (distance $n_2 - n_3$) is bigger than the element on the opposite side (distance $n_1 - n_2$), more volume will flow in. This is something worthwhile to remind while pre-processing the mesh with a normal condition applied in the bearing corner.

This infow or outflow mechanism is the reason why the weighted normal option of van Rens can lead to volume loss or gain when used in an Eulerian simulation. Since normal variants are convenient for pre-processing purposes, these options are favored, despite their slightly lesser performance in streamline behavior. The extrusion force is well predicted for a 45 degree angle in both extrusion ratio simulations. There is no reason to believe that in between these extrusion ratios the 45 degree normal will not yield accurate results. When the size of the elements on both sides of the bearing corner will be approximately equal, the volume conservation is reasonably accurate for a 45 degree normal. This is shown in figure 2.20(a). In [5] an improved normal bearing corner is treated that derives the normal to maintain volume conservation.

Chapter 3

Front line tracking in a steady state flow

3.1 Introduction

In this chapter aluminium flow inside the container is studied. Extrusion experiments are performed at Boalgroup to visualize the flow inside the container during extrusion.

These experiments are compared with simulations. In section 3.3 a procedure is treated to obtain comparable numerical results. The procedure is first demonstrated on a Newtonian viscous flow.

The numerical results are fitted to the experiments by adapting the material parameters. To yield accurate flow results in a simulation, accurate material properties are necessary. In this chapter it is shown that the procedure can be used for an inverse determination of material parameters. Determining the material properties is usually done by compression or torsion tests [43]. However these tests do not give results under extrusion conditions of high pressure and shear strain. Comparison of aluminium flow between experiments and simulations [65], [66] inside a container, can lead to better material properties, among other simulation parameters.

3.2 Experiments

Extrusion experiments are performed with aluminium billets prepared as in figure 3.1. First a cylindrical billet of 210 mm long is cut into slices of 15

mm thick (step 1 and 2). Next a copper foil grid is placed between the slices. The slices, with the copper foil in between, are spot welded together and the preparation of the billet is complete (steps 3 and 4). After preheating the billet, container and dies, the billet it is placed in the container and extruded (step 5). Extrusion is stopped and the billet and extrudate are removed from the container and cut in half in the extrusion direction step 6). After cutting the copper foils between the slices show the front lines.

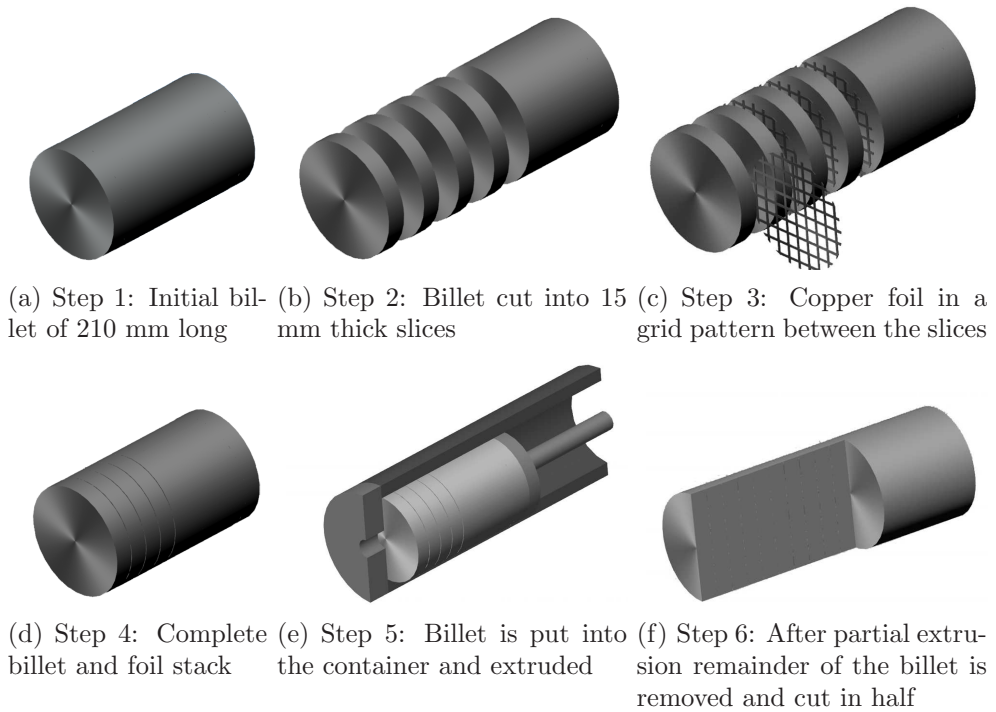


Figure 3.1: Billet preparation before (steps 1,2, 3 and 4), during (step 5) and after (step 6) extrusion.

The experiments have been performed at Boalgroup. A total of 3 billets of 210 mm were extruded and each was stopped at different remaining billet length (see table 3.1). The diameter of the billet and extrudate are 92 mm and 20 mm respectively. This is an extrusion ratio of approximately 21 which is slightly lower than what is believed to be optimal in every day practice. The ram speed in the experiments was held constant at approximately 1 mm/s and the initial temperature of the billet was approximately 723K. The exit temperature was

not measured. Based on experience the estimated temperature rise due to the work done is about 100K. After the stop, the billets were pressed out of the container. The deformation of the specimen, visible on the photos, is due to this extraction. Note that this deformation will work throughout the entire billet and will also deform the position of the copper foil.

Experiment	Extrusion time	Remaining billet length
	[s]	[mm]
Experiment 1	24	186
Experiment 2	50	160
Experiment 3	90	120

Table 3.1: Three experiments with prepared billets.

It is not known whether the discretisation of the billet with copper foil inserted will cause a shear between the slices and therefore a different velocity field. To minimize this effect, not a continuous foil is used but a grid of copper. In between this grid the aluminium can weld together due to pressure and friction.

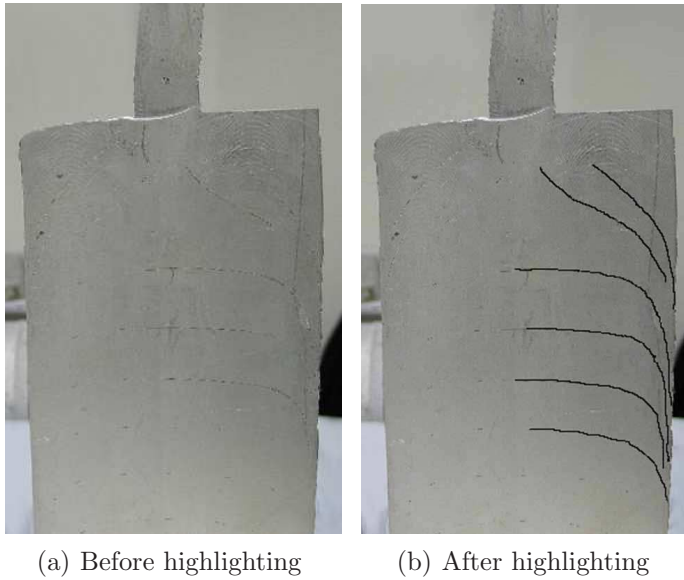


Figure 3.2: Experiment 1 after cutting.

Because of the use of a copper grid the resulting lines in the billet are discontinuous, which is visible in figure 3.2a. In figure 3.2b the lines are highlighted to improve the visibility. In figure 3.3 the results of the three experiments are shown.

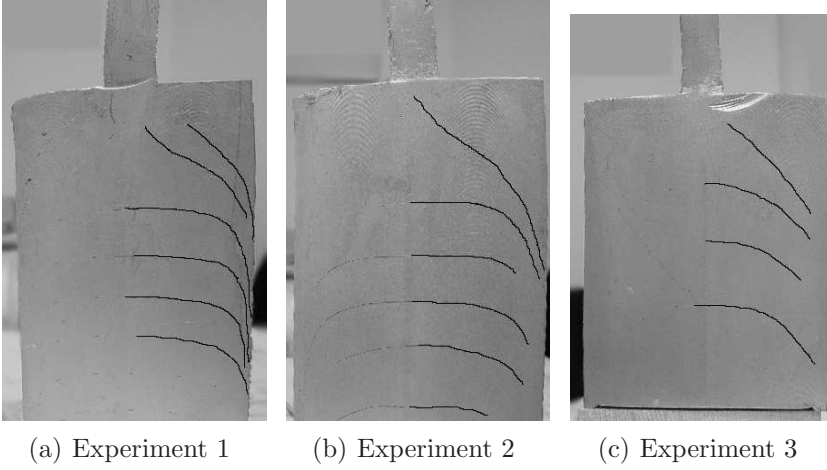


Figure 3.3: Results of the three experiments.

3.3 Theoretical outline for front line tracking demonstrated on Newtonian viscous flow

In figure 3.4(a) a line in the aluminium, a front line, at the beginning of extrusion is shown. During extrusion the front line deforms inside the container and the new shape of the front line at $t = 10s$ is as in figure 3.4(b). In this section the determination of such front lines in simulations is treated. The method is demonstrated on a Newtonian viscous flow.

The front lines from figure 3.4(c) can be created based on numerical simulations. One method is to use the results of a transient simulation. After performing the transient simulation the total displacements at different times hold all the information to create the front lines. A major drawback of this approach is the time it takes to perform a transient simulation. This extrusion process is mainly in steady state. In the next section another, faster, and just as accurate method is treated to create the front lines based on the steady state velocity field. From the steady state velocity field a transition time field f is

calculated. The simulation to obtain a steady state velocity field takes much less time compared to a transient simulation.

Combining these front lines at different times during extrusion in one figure shows the transition of a front line through the billet (figure 3.4(c)). Another way of looking at figure 3.4(c) would be that one front line is an isoline in a field f that represents the transition time from a point on the $t = t_0$ front line to the $t = t_n$ front line.

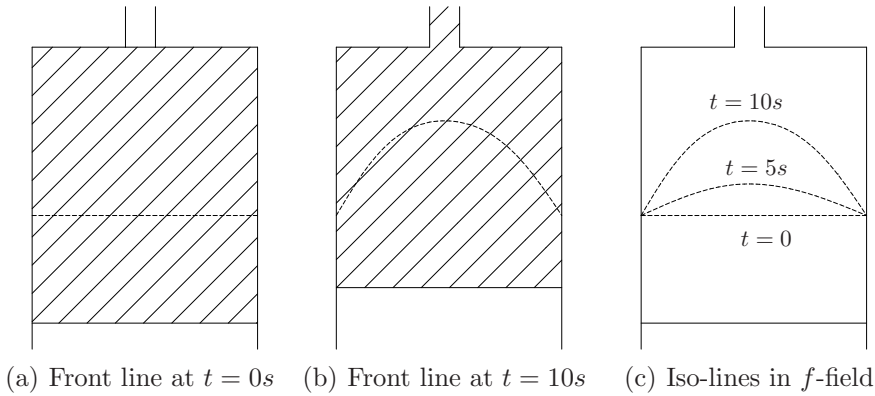


Figure 3.4: Front lines and f -field in extrusion.

In this section the theoretical background is given of the calculation of front lines with a steady state velocity field. It is assumed that a steady state velocity field $\mathbf{v}(\mathbf{x})$ or $\mathbf{v}$ is known as input.

Consider the following convection equation over a domain Ω with boundary Γ .

$$\frac{Df}{Dt} = \frac{\partial f}{\partial t} + \mathbf{v} \cdot \nabla f = S \quad (3.1)$$

f is the convected quantity and $\mathbf{v}$ the velocity. S is a source term. Since the velocity field is in steady state the convected quantity is not time dependent.

$$\mathbf{v} \cdot \nabla f = S \quad (3.2)$$

Now defining the source term as 1, the solution of the equation, $f = f(\mathbf{x})$, is a time, which we call the "transition-time".

$$\mathbf{v} \cdot \nabla f = 1 \quad (3.3)$$

An iso-line of the f -field represents the location of a front line at time $f = t$ that had the initial location on the boundary Γ_f where $f = 0$. The differential equation (3.3) is a first order equation therefore one boundary condition is necessary. The necessary boundary condition for this equation is:

$$f = 0 \quad \text{on} \quad \Gamma_f \quad (3.4)$$

Γ_f can be freely defined inside Ω . Γ_f is the location of the initial front that is convected through the domain, as can be seen in figure 3.5.

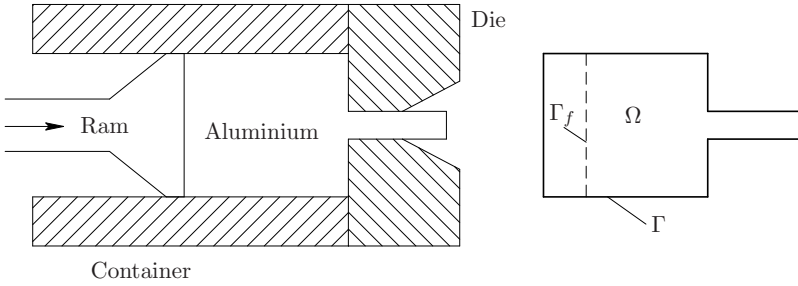


Figure 3.5: Aluminium domain and boundaries.

To demonstrate this method in the next section it is applied to a Newtonian viscous flow.

3.3.1 Viscous flow in a long parallel channel

In figure 3.6 a viscous flow in a channel is shown. The velocity field of this flow is known from the Stokes equations. In a fully developed poiseuille flow the velocity is parabolic in y direction and independent of the x coordinate.

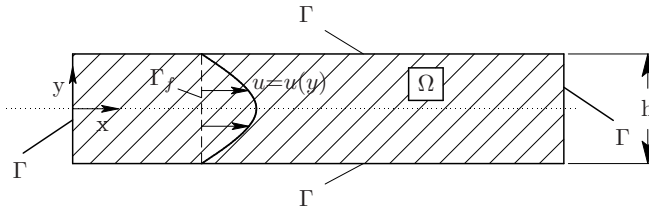


Figure 3.6: Viscous flow in a long parallel channel.

With $u = u(x, y)$ the velocity in the x direction and $v = v(x, y)$ the velocity in the y direction.

$$u(x, y) = u(y) = v_0(-y^2 + \frac{h^2}{4}) \quad (3.5)$$

$$v(x, y) = 0 \quad (3.6)$$

Equation (3.5) v_0 determines the magnitude of the velocity. The velocity field can be used to solve equation (3.3).

$$\mathbf{v} \cdot \nabla f = u \frac{\partial f}{\partial x} + v \frac{\partial f}{\partial y} = 1 \quad (3.7)$$

Since $v = 0$ this can be rewritten into:

$$\frac{\partial f}{\partial x} = \frac{1}{u} \quad (3.8)$$

The velocity u in the x direction only depends on the y location, therefore integrating with respect to x leads to:

$$f(x, y) = \frac{x}{v_0(-y^2 + \frac{h^2}{4})} + C \quad (3.9)$$

The initial condition for this problem is at the vertical line Γ_f at x_f where $f(x_f) = 0$. With this condition the constant C can be established.

$$C = -\frac{x_f}{v_0(-y^2 + \frac{h^2}{4})} \quad (3.10)$$

The total equation for f is:

$$f(x, y) = \frac{x - x_f}{v_0(-y^2 + \frac{h^2}{4})} \quad (3.11)$$

From this equation the $f(x, y) = t_i$ iso-line can be derived.

$$y = \pm \sqrt{-\frac{x - x_f}{v_0 t_i} + \frac{h^2}{4}} \quad (3.12)$$

This analytical solution can now be used to validate numerical results that are created by solving the transport equation numerically. In the next paragraph the numerical solution is calculated and the results for viscous flow in a long parallel channel are compared with the analytical values.

3.3.2 Solutions for a discretized velocity field

The domain Ω is discretized in quadrilateral 4 node elements (Q4) as in figure 3.7. At the nodes the analytical velocity is prescribed.

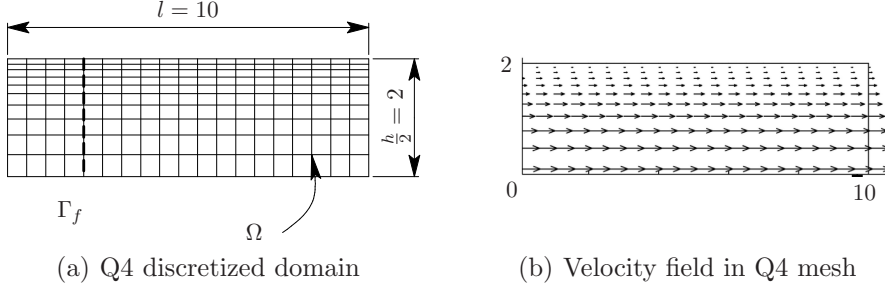


Figure 3.7: Mesh and velocity field for comparison with analytical results.

In this example the geometry and boundary condition are chosen as: $h = 4, x_f = 2, v_0 = \frac{3}{8}$. The equation in (3.3) is rewritten in a weak form. The left and right hand side of equation (3.3) are multiplied with a weighing function w and integrated over the domain.

$$\int_{\Omega} (w)(\mathbf{v} \cdot \nabla f) d\Omega = \int_{\Omega} (w) \quad d\Omega \quad (3.13)$$

It is known from literature that when for the weighing function w the same interpolation functions are chosen as for f the solution of this equation is in most cases not stable. In section 3.4.2 stabilization of these equations is treated. Here the derivation is shown for a Galerkin approach: $w = \sum N_i w_i$ and $f = \sum N_j f_j$

$$\sum_{ij} [w_i \int_{\Omega_E} (N_i)(\mathbf{v} \cdot \nabla N_j) d\Omega_E] f_j = \sum_{ij} w_i \int_{\Omega_E} (N_i) \quad d\Omega_E \quad \forall \quad w_i \quad (3.14)$$

This can be simplified into:

$$\sum_j [\int_{\Omega_E} (N_i)(\mathbf{v} \cdot \nabla N_j) d\Omega_E] f_j = \sum_j \int_{\Omega_E} (N_i) \quad d\Omega_E \quad (3.15)$$

From solving the total set of equations the nodal values for f_j are found. The solution of these equations is stable for the shown mesh and velocity field from figure 3.7, since the velocity has only one component and is aligned with the mesh.

3.3.3 Comparison of analytical and discretized results

In figure 3.8 the iso-lines are plotted at $f = [1, 2, 3]$. The dotted results are the analytical iso-lines from equation (3.12). The numerical results are in perfect agreement with the analytical values.

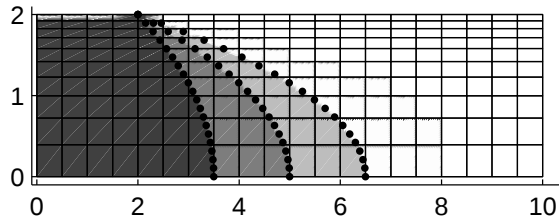


Figure 3.8: Comparison of iso-lines from numerical results with analytical results.

3.4 Post-processing FEM results

The method described in the previous paragraph only requires a steady state velocity field to determine the front lines. After the steady state velocities are determined by a FEM simulation, the front lines are calculated in a post-processing step. In figure 3.9 an overview is shown of the method. The FEM mesh and velocities are then input in a post-processing in which an f -field (based on nodal values) is calculated. This f -field represents the transition time of material from where $f = 0$ to the location where $f = t_i$. Front lines at multiple times can be plotted as iso-lines in one f -field.

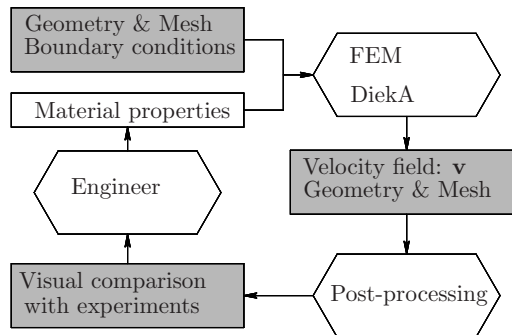


Figure 3.9: Feedback to material parameters.

The main advantage is the decrease of the total simulation time to get front lines. The post-processing step to create the front lines is fast and cheap as well as the flow simulations for a steady state flow.

The experiments from paragraph can be compared to numerical results with this method. Also can this method be used as an inverse approach to determine material properties of aluminium under extrusion conditions.

First the material properties have to be guessed from literature or from other alloys. With these properties the simulations are run and the post-processing is performed as in figure 3.9. After visual comparison the material properties are modified. In the flow chart an engineer chooses the parameters to change. With the new material properties the simulations, postprocessing and comparison is performed again. The loop is repeated until the simulations are best fitted to the experiments.

3.4.1 Post-processing

In this section we focus on the determination of the f -field based on a velocity field from a FEM simulation. The input for this post-processing step are the steady state velocities from a FEM simulation. Based on these velocities the convection equation (3.3) is solved for the f . For the solution of the convection equation is chosen for a FEM formulation with a Stream Upwind stabilization. In the derivation we closely follow Brooks [9]

3.4.2 A SUPG stabilization for determining front lines

Consider the transport equation for (3.3) over a domain Ω with boundary Γ .

$$(\mathbf{v} \cdot \nabla f) = 1 \quad (3.16)$$

The weak stabilized form of equation (3.16) can be written as follows:

$$\int_{\Omega} (w + \alpha l \frac{\mathbf{v}}{\|\mathbf{v}\|} \cdot \nabla w) (\mathbf{v} \cdot \nabla f) d\Omega = \int_{\Omega} (w + \alpha l \frac{\mathbf{v}}{\|\mathbf{v}\|} \cdot \nabla w) d\Omega \quad (3.17)$$

With α the SUPG stabilizing parameter which is chosen constant for every element and l the typical element length in velocity direction. We define l as the length of the projection of an element on the velocity vector as in figure 3.10.

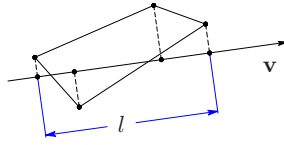


Figure 3.10: Projection of element on $\mathbf{v}$.

Next the weighing function w is interpolated in the same order as the state variable f : $w = \sum N_i w_i$ and $f = \sum N_j f_j$

$$\begin{aligned} \sum_{ij} w_i \left[\int_{\Omega} (N_i + \alpha l \frac{\mathbf{v}}{\|\mathbf{v}\|} \cdot \nabla N_i) (\mathbf{v} \cdot \nabla N_j) d\Omega \right] \{f_j\} = \\ \sum_{ij} w_i \left\{ \int_{\Omega} (N_i + \alpha l \frac{\mathbf{v}}{\|\mathbf{v}\|} \cdot \nabla N_i) d\Omega \right\} \quad \forall \quad w_i \end{aligned} \quad (3.18)$$

which implies:

$$\begin{aligned} \sum_j \left[\int_{\Omega} (N_i + \alpha l \frac{\mathbf{v}}{\|\mathbf{v}\|} \cdot \nabla N_i) (\mathbf{v} \cdot \nabla N_j) d\Omega \right] \{f_j\} = \\ \sum_j \left\{ \int_{\Omega} (N_i + \alpha l \frac{\mathbf{v}}{\|\mathbf{v}\|} \cdot \nabla N_i) d\Omega \right\} \end{aligned} \quad (3.19)$$

3.4.3 Determination of the upwind parameter α

In literature however several remarks can be found that the stabilization is not very sensitive to variations in the upwind parameter. In most cases it is impossible to derive an exact value for the optimal upwind parameter. Here the optimal upwind parameter is determined by 2D tests with the viscous flow problem from the previous paragraph.

The post-processor provided with a mesh and nodal velocities $\mathbf{v}$ that are equal to the analytical velocities $v_{ana}(x)$ or $\mathbf{v}_{FEM}$. From the analytical velocity field the exact solution of $f_a(x)$ can be derived. This is compared with the post-processed $\mathbf{f}_{FEM}$ with both the analytical velocities $\mathbf{v}_{ana}$ and FEM velocities $\mathbf{v}_{FEM}$ as input, see figure 3.11.

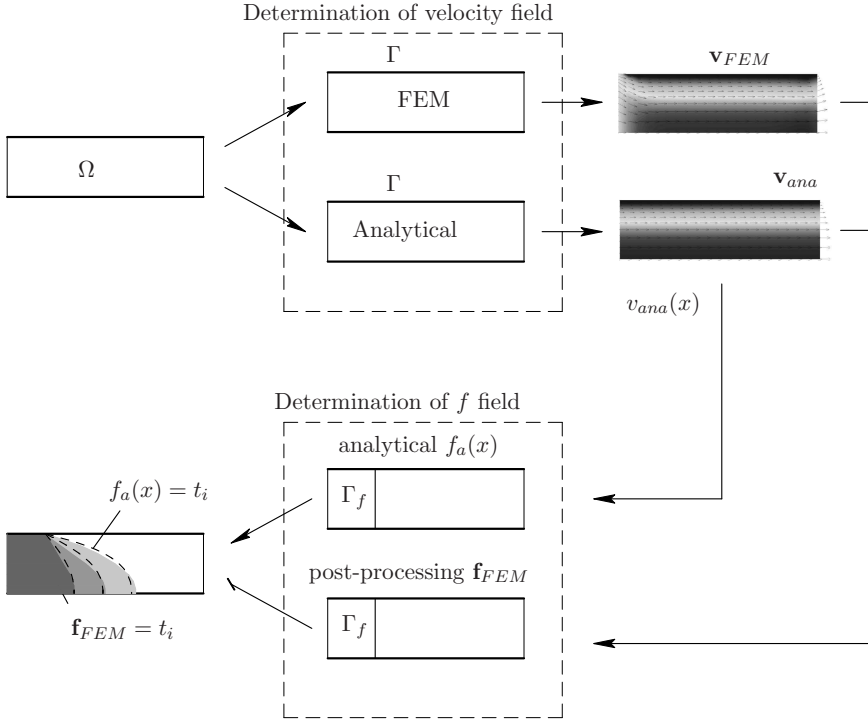


Figure 3.11: Accuracy test of SUPG algorithm.

In figure 3.12 results of the post-processor are shown. The analytical velocity field from equation (3.6) is used as input in the post-processor to create the results in the left column. As mentioned before this solution is stable even without upwind stabilization. From equation (3.19) we can derive that for this specific case there is no need for stabilization and it even has no contribution. This effect is apparent in the left column of figure 3.12. Here the post-processor gives exact results for any upwind parameter.

The calculation of the f -field in right column is based on velocities from FEM calculations $\mathbf{v}_{FEM}$. The inflow condition in the FEM calculation is an uniform inflow velocity on the left side which will develop to a poiseuille flow. This velocity field will give instabilities for $\alpha = 0$ as can be seen in the right column of figure 3.12. Choosing an upwind parameter in the range between 0.1 and 1 will give good results. Higher values will diffuse the field excessively.

In figure 3.12 the results for structured meshes is shown. In figure 3.13 the f -field for an analytical velocity field as in equation (3.6) is applied to an unstructured triangle (T3) mesh. For this reason even for the analytical velocities, there is a need for stabilization. Again we find that for the same range of α the results are stabilized but not diffused.

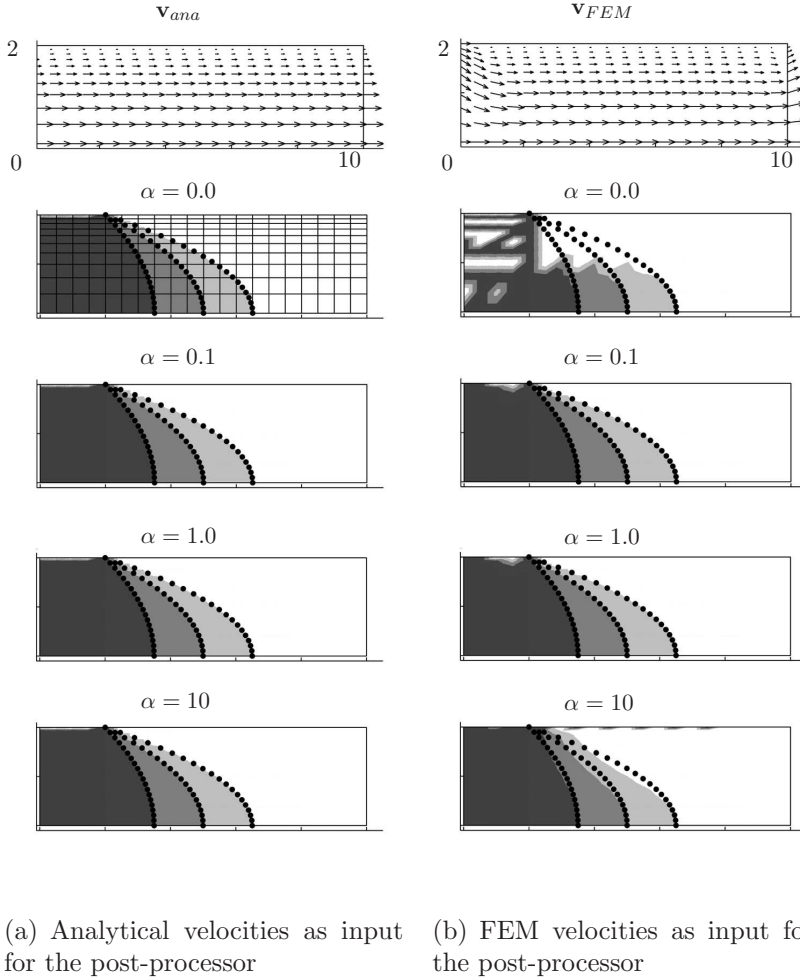


Figure 3.12: Viscous flow in channel with Q4 elements.

In figure 3.13 for $\alpha = 1.0$ the f -field upstream from the initial location of the front line is clearly not well predicted. It is not visible, but this is the case for

all simulations. Upstream from the front the solution is unstable, but this is not a problem for creating front lines.

A difference between the Q4 and T3 elements can be found at the symmetry line for $\alpha = 10$. When post-processing the Q4 elements with velocities $\mathbf{v}_{FEM}$, there is no diffusive effect apparent at the symmetry line. This is in contrast with the T3 mesh. In the T3 mesh the element(s) downstream from a node on the symmetry line can have a different contribution to the nodal value for f than an upstream element. In the Q4 case these contributions are equal and therefore the stabilizing part vanishes.

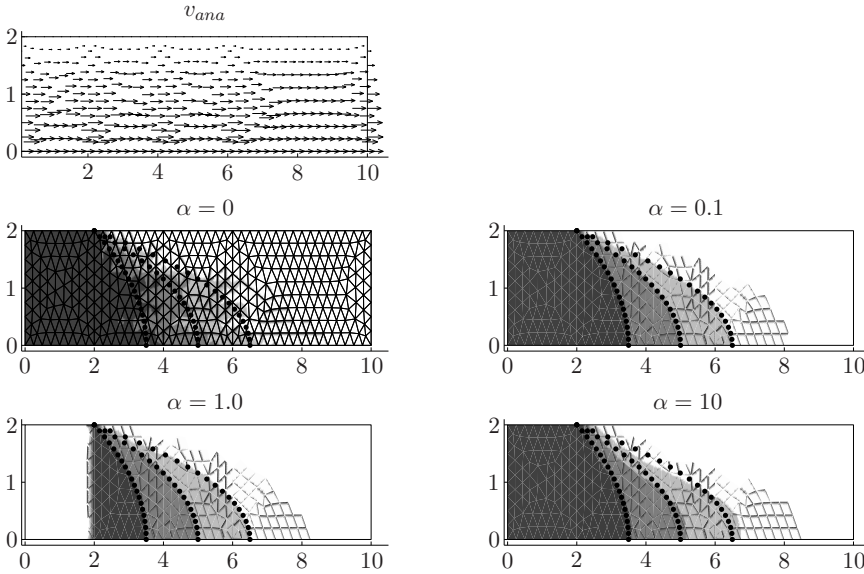


Figure 3.13: Viscous flow in channel with T3 elements.

In conclusion it can be said that if the direction of the upwinding is chosen in the velocity direction, the post-processor is very accurate for both structured and unstructured meshes.

3.4.4 Assembly of front lines for comparison

When the velocity field is post-processed, with Γ_f^1 as initial location of the front, the front lines at different extrusion times can be plotted. The results from the experiments however show multiple front lines, started from different

locations, at one extrusion time.

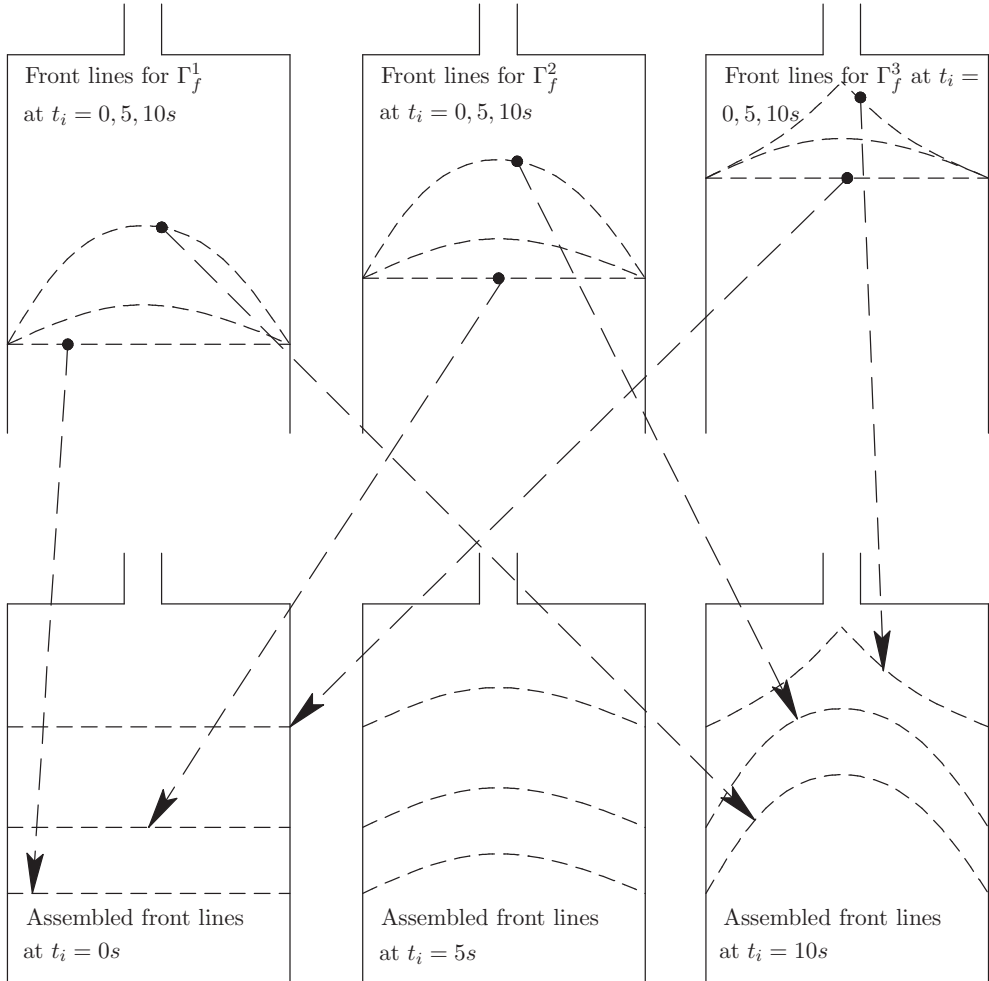


Figure 3.14: Assembly of front lines from different f -fields for comparison.

To get front lines starting from different locations, based on one velocity field, the post-processing has to be repeated for the different initial locations Γ_f^i . In the top row of figure 3.14 three post-processing results are shown starting from three different initial fronts. In the bottom row the results are assembled for three different extrusion times. The results in the bottom row are now directly comparable to the experimental results.

3.5 Simulations of container flow in aluminium extrusion

In the post-processor we use the steady state velocities and assemble the matrix and right-hand-side as described before and solve the system. We perform a number of analyses with Γ_f^i at different equidistant locations. For every analysis the matrix and right hand side are identical, but since Γ_f^i is different the system has to be solved for every analysis. From each analysis contour plots of f are made. Next the results are assembled as described in paragraph 3.4.4. These plots show the locations of different foils after extrusion time $t_i = t$. The assembled plots are made at $t=24, 50$ and 90 s (See table 3.1). An assembled iso-line plot is shown in figure 3.16b.

3.5.1 material properties

The material behavior is described rigid visco-plastic by the Sellars-Tegart law:

$$\sigma_y = s_m \operatorname{arcsinh} \left(\left(\frac{\dot{\kappa}}{A} \exp \left(\frac{Q}{RT} \right) \right)^{\frac{1}{m}} \right) \quad (3.20)$$

The material properties from AA6061, used for the experiments, under extrusion conditions are scarcely available from literature [10]. The properties for AA6063 can be found in Lof [43] and are shown in table 3.2. Here is chosen to determine the properties for AA6061 by modifying the properties of AA6063 and determining which combination shows the best comparison with the experiments. From all parameters, m has the strongest effect on the shape of the front lines. m is fitted by comparing the results from the simulations on Experiment 1 (figure 3.3(a)). The value for m from [43] is 5.4, the best fit for AA6061 was found at $m = 10$.

	Symbol	Value
Plastic properties (T=773K)	s_m [MPa]	25
	m [-]	5.4 (10)
	A [1/s]	$6 \cdot 10^9$
	Q [J/mol]	$1.4 \cdot 10^5$
	R [J/molK]	8.314

Table 3.2: Material parameters for AA6063 alloy [43].

The flow from the experiments show a plug flow in the container. The results with $m = 5.4$ show a more parabolic front line. The results with $m = 10$ are

in better agreement with the plug flow. This can be explained by equation (3.20). Making m higher reduces the increase of flow stress at higher strain rates, making it easier for the aluminium to shear near the container wall.

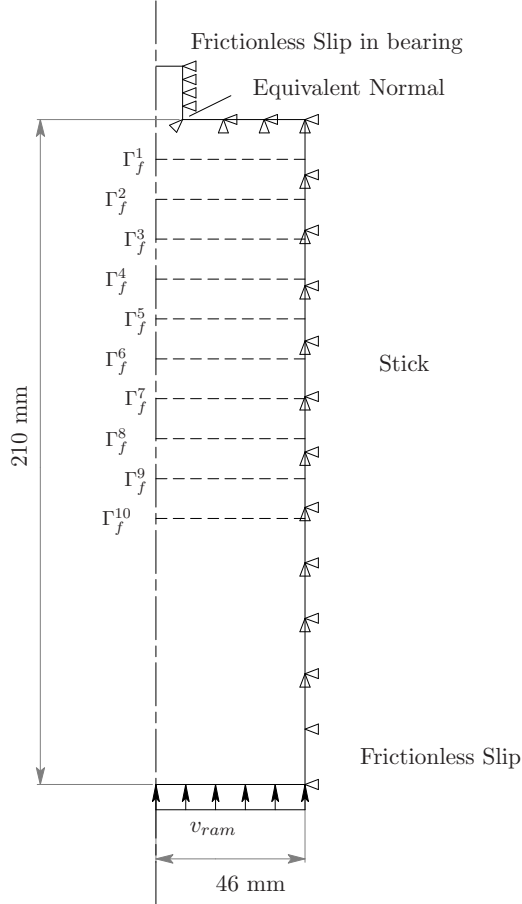


Figure 3.15: Mesh and boundary conditions.

3.5.2 FEM modeling

The simulations are performed with the use of an ALE / Eulerian formulation in the in-house code DiekA. The aluminium domain is chosen axial symmetric as in figure 3.5. The dimensions are chosen similar to the dimensions of the experiments, described in section 3.2. The aluminium domain is meshed with

4-node elements with bilinear interpolation function (Q4). In figure 3.15 the boundary conditions on Γ are shown. It is assumed that the aluminium sticks to the container wall and die, except for the bearing area where a frictionless slip situation is modeled.

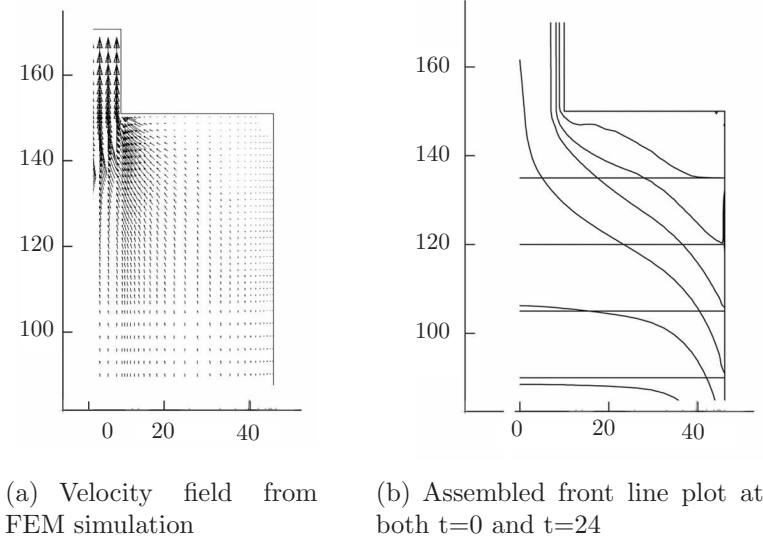


Figure 3.16: Numerical results from simulation 1.

The time-iso-lines are plotted for initially straight fronts at y -positions from 0 up to 135 ($\Gamma_f^1 \dots \Gamma_f^{10}$) with slice thickness increments of 15 mm. To be able to set the necessary condition in the post-processing part at locations Γ_f^i , during meshing is made sure that nodes are placed at Γ_f^i .

3.6 Comparison

In this section we compare the results of the simulation with the experiments. In the simulations the deformation of the first ten front-lines are calculated. The deformations of the lines 3,4 and 5 are used to fit the material properties for experiment 1, so they correspond well with the experimental results. Since the simulation is done with an Eulerian description the movement of the ram is not included. The difference between the experimental results and numerical lines can be attributed to several effects. Firstly the velocity field can be inaccurate. The difference can be caused by the fact that the determination of

the velocity field is performed iso-thermally and in the experiment the billet will heat up locally due to deformation near the container wall, near and in the die. This will affect the yield stress and therefore the velocity field. Secondly the exact ram speed is not known from the experiment, so therefore deviations can occur. Also the Streamline Upwind diffusion in the post-processing step can give deviations with the reality. Specially in regions where the velocity gradient is high a bigger error can occur. Furthermore, in the experiments the extraction of the billet has caused deformation. This deformation is apparent from the non rectangular shape of the billet in figure 3.17.

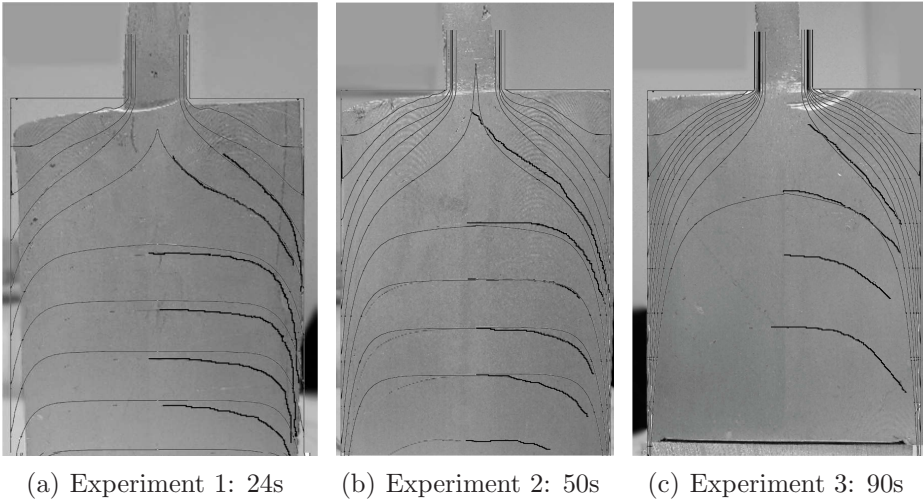


Figure 3.17: Comparison between experiments and simulations.

3.7 Concluding remarks

Based on the results we can conclude that the accuracy of the method is mainly determined by the accuracy of the velocity field. Extrusion is most of the times a non iso-thermal process and the extension to simulations with heat generation will improve the accuracy of the velocity field.

Even for this iso-thermal simulation the results are in very good agreement with the experiments. With the proposed procedure new material parameters for AA6061 under extrusion conditions are determined.

Chapter 4

Flow front tracking

During filling of the die at the startup of the extrusion process, the location and velocity of the flow front will have a great influence on the deformation of the die. This especially applies to porthole dies. This deformation during initial filling will affect the performance of the die in the rest of the process.

When die correctors work on the die after trial pressings, their main information where to improve the die comes from the shape of the nose piece. When assessing a die numerically by FEM analysis the prediction of the shape of the nose piece can provide the same information. The prediction of the nose piece shape requires the simulation of the full transient start-up of the process. This includes the filling of the die.

In general an analysis of the filling stage requires the solution of the flow equations and tracking the free surface or interface. The many interface tracking methods can be categorized in two groups. Lagrangian methods are based on a moving mesh and Eulerian strategies are based on a fixed mesh. In Floryan & Rasmussen [20] an overview of various existing methods and their limitations is given.

In moving mesh strategies the material domain is meshed and the mesh deforms according to the material. The free surface is represented by the free boundary of the mesh. In processes with high deformations, like e.g. extrusion, this strategy is limited by mesh distortion and contact problems. The unavoidable remeshing and searching for contact areas makes this strategy inflexible and time consuming.

In fixed mesh strategies the complete domain is meshed and material with the interface flows through the mesh. In extrusion the domain exists of a part of the billet, the die cavity and a part of the profile. In regions where high gradients are expected the mesh can be refined. The difficulty of fixed mesh methods is the tracking of the interface. In the next section interface tracking methods are described.

A fixed mesh method is preferred because time consuming remeshing can be avoided. Besides that, when during the simulation the die is filled a fixed mesh strategy will converge to the steady state.

4.1 Flow front tracking algorithms

The three most popular strategies for tracking are the volume of fluid method (VOF) [27], the level set method [51,59] and the pseudo-concentration method [63,64]. In all approaches the function ϕ is transported using the convection equation

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \nabla \phi = 0 \quad (4.1)$$

With $\mathbf{u}$ the velocity field. It is well known in literature that the Galerkin formulation of equation (4.1) gives non-physical oscillations. Adding numerical stabilization can prevent this effect.

In VOF the function $\phi(x, y, z)$ indicates the fraction of a fluid in a cell. If $\phi = 1$ the cell is filled with fluid A, if $\phi = 0$ the cell is filled with fluid B. The interface is at $0 < \phi < 1$. There are several ways to transport this function. The main advantage in the VOF method is that the volume conservation is ensured, however reconstruction of the interface is time consuming and complex for 3D cases [12].

In 1986 Thompson introduced the pseudo-concentration method [63], [64] to track interfaces in two phase flows. The pseudo concentration method uses a function ϕ that is 1 for fluid A and 0 for fluid B. At $\phi = 0.5$ the interface can be found. The material properties are calculated as:

$$\mu = \phi \mu_A + (1 - \phi) \mu_B \quad (4.2)$$

The main disadvantage of both pseudo-concentration function and VOF methods is the steep gradient in the ϕ -field that will be diffused in the numerical procedures. To minimize these effect mesh refinements have been proposed in

the proximity of the interface [24, 62, 73].

The discontinuity of the pseudo-concentration function around the flow front introduces inaccuracies or oscillations during the convection through the FEM mesh. Dvorkin [18] already notices that if the initial distribution of the concentration function is smooth enough, no smoothing algorithm is necessary later in the simulation.

In the level set method the function ϕ is an initial smooth function. In [48] the best initial function for ϕ was found to be a signed distance from the front. The zero level set of the function $\phi = 0$ defines the interface. The drawback of the level set method is, as in the pseudo-concentration function, that it does not ensure volume conservation rigorously. Besides that describing complex initial interfaces with one distance function is difficult. In level set methods several regularization algorithms of the distance function ϕ have been proposed [59, 60].

In the next section a method is described that resembles the level set method. However the distance function ϕ is built from two or three (2D or 3D) initial linear functions: The original coordinates.

4.2 Original coordinate tracking

In this section a new way to describe an interface is defined. The interface is no longer described by one field, but can be described by several fields. The chosen initial fields should be smooth and easy to convect. As concentration functions we chose the initially linear functions of the original coordinates. The original coordinate $\mathbf{X}_0$ in node n is calculated by subtracting the material displacement $\mathbf{u}_m^n$ from the current coordinate $\mathbf{X}^n$.

$$\mathbf{X}_0^n = \mathbf{X}^n - \mathbf{u}_m^n \quad (4.3)$$

This can be done in two or three directions, according to the dimension of the simulation. For example the location of an initial circular interface in a 2D simulation with center $(0,0)$ and radius R can be described as the zero level of ϕ in equation (4.4).

$$\phi = R^2 - [(X_0 - X_{cent})^2 + (Y_0 - Y_{cent})^2] \quad (4.4)$$

$[X_{cent}, Y_{cent}]$ is the location of the initial centre of the circular interface. In this scheme not the function ϕ is convected but the separate total displace-

ments fields are. How the total displacement $\mathbf{U}$ is convected in our Eulerian formulation is described in the next section.

4.3 Convection algorithm

In ALE and Eulerian methods, a convective increment has to be calculated in each step or iteration. This convective increment can be calculated by interpolation or a convection method.

Normally the history dependent state variables such as equivalent plastic strain, stress and damage parameters are known in the integration points.

For the re map of state variables, known in integration points, is chosen for a convection approach. The convective velocity $\mathbf{v}_c$ is regarded constant during a step.

$$\Delta u = \int_t^{t+\Delta t} -\mathbf{v}_c \cdot \nabla u \, dt \approx -\Delta \mathbf{u}_c \cdot \nabla u \quad (4.5)$$

The convective increment is calculated with the "Weighted local and global smoothing" (WLGS) scheme of [29]. In [74] is mentioned that this scheme is not the most accurate scheme for integration point data. This is shown from results with molenkamp tests [8, 56, 58, 67]. The WLGS scheme [29] has been tested for accuracy on integration point data in [74]. In this thesis the convection of nodal data, that is initially a smooth field, by the WLGS scheme is treated.

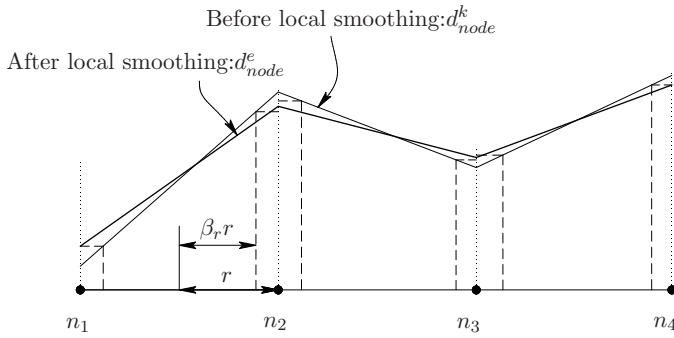
The total material displacements in a Eulerian mesh are not related to the mesh position. For a standard simulation it is not necessary to calculate the total material displacements. However for the calculation of the original coordinates the material displacement field is a necessary variable. The total material displacements are nodal values. The procedure to convect the total material displacements is a little different from the WLGS scheme and is described in the next paragraph.

4.3.1 Weighted local and global smoothing for nodal values

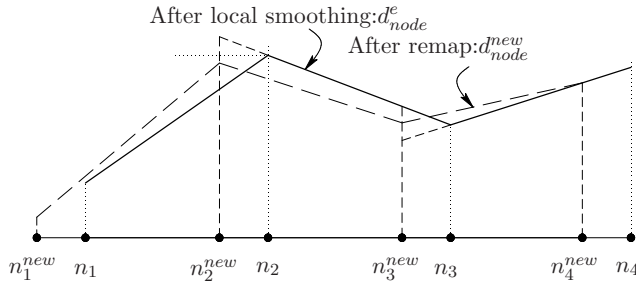
In the WLGS scheme of Huétink and Wisselink [29, 74] the state variables known in the integration points have to be extrapolated to the nodes. This extrapolation will give to a discontinuous field. New, continuous nodal values

are calculated by nodal averaging. Using this continuous field the new integration point values are subsequently calculated based on a convection scheme. This scheme will add the global smoothing. This scheme has been treated in chapter 2.

For the total material displacements another scheme is used. In figure 4.1 the WLGS for nodal values is shown. First the nodal values are "extrapolated" to the nodes. After this extrapolation the nodal value are averaged to create a new continuous field. These two steps are the equivalent of the local smoothing step for integration point data. Both steps are shown in figure 4.1(a).



(a) Local smoothing and nodal averaging



(b) Remap of nodal values

Figure 4.1: Weighted Local and Global smoothing for nodal values.

Next the values in the new nodal locations are calculated by interpolation. This is shown in figure 4.1(b). Finally the new nodal values are averaged to get the new smooth nodal field. These four steps are written in equations

(4.6) and (4.8).

$$d_{node}^e = \sum_{k=1}^n N^k(\beta_r r, \beta_z z, \beta_h h) d_{node}^k; \quad (4.6)$$

With r, z, h the location of the nodes in isoparametric (local) coordinates of an element. To prevent cross- wind diffusion the parameters $\beta_r, \beta_z, \beta_h$ can be chosen as [74]

$$\beta_r = (1 - C_r), \dots \quad (4.7)$$

The new discontinuous field is then averaged.

$$d_{node}^* = \frac{1}{ne} \sum_{k=1}^{ne} d_{node}^e{}^k \quad (4.8)$$

The smoothed nodal values are remapped to the new mesh location as in 4.1(b). In (4.9) this interpolation scheme is used to calculate the new nodal values.

$$d_{node}^{new} = \sum_{k=1}^n N^k((r, z, h)_{old} - (C_r, C_z, C_h)) d_{node}^*{}^k \quad (4.9)$$

In these equations n is the number of nodes to one element and ne is the number of elements attached to one node. After the convection step the nodal values are discontinuous. The final step is nodal averaging to create a continuous field.

When the incremental displacements are equal to zero ($C_r=0, \dots$), then $d_{node}^{new} = d_{node}^e$.

Courant number

The courant number is defined as the difference in local coordinates between the old and new integration points, divided by the elements size (2x2x2 in local coordinates). The three Courant numbers are calculated for the different directions at every integration point. The integration point values are then averaged to one value for the whole element.

$$C_r = \frac{1}{n_{ip}} \sum_{i=1}^{n_{ip}} \frac{|\Delta r_{ip}|}{l} \quad (4.10)$$

and the total courant number

$$C = \sqrt{C_r^2 + C_z^2 + C_h^2} \quad (4.11)$$

4.3.2 2D Rigid body flows

In this section the convection scheme for nodal values is tested on rigid body movements. In figure 4.2 two rigid body movements are shown, a rigid translation in the top row and a rigid rotation on the bottom row. The rigid body translation is of a circular interface with $R = 0.30$ and initial center at $[X_{cent}, Y_{cent}] = (-0.6, -0.6)$ in a domain of $\Omega = [(-1, -1); (1, 1)]$. Since the original coordinates are used to describe the interface, the circle can be described as the zero level of ϕ

$$\phi(x, y) = R^2 - [(X_0(x, y) - X_{cent})^2 + (Y_0(x, y) - Y_{cent})^2] \quad (4.12)$$

The two linear fields X_0 and Y_0 are calculated based on the total material displacements. The total material displacements are convected through a mesh with equally sized (bi-)linear elements.

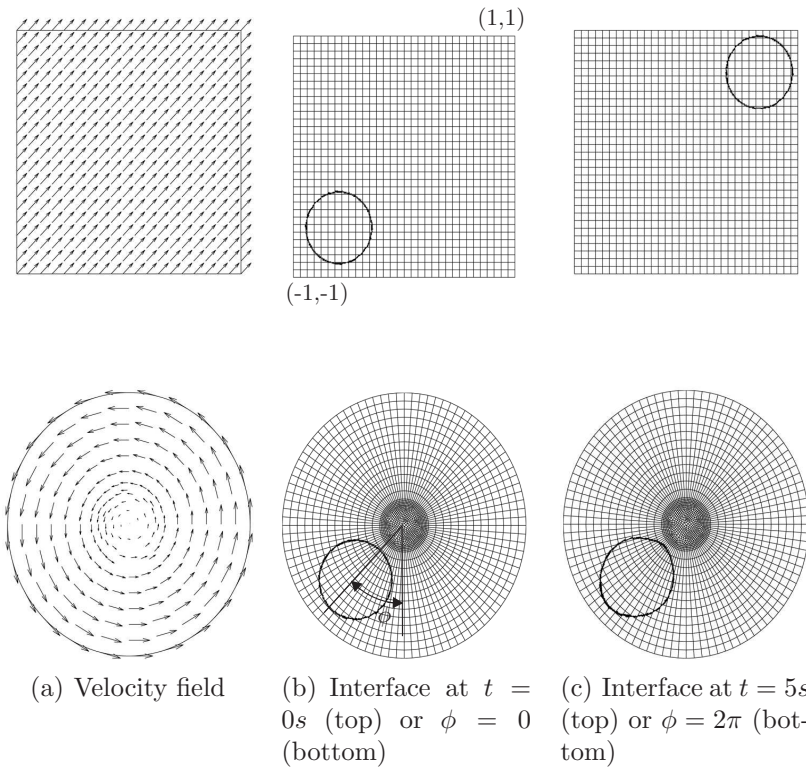


Figure 4.2: Velocity field and interface convection for rigid translation (top) and rigid rotation (bottom).

Using the convection scheme previously described, the results are without diffusion. In this case the circle is convected from $t=0$ to $t=5$ in 200 time increments ($C_r = 0.1$). Even if the circle is convected outside the domain and then back into the domain, the circle ends up without any distortion (not shown in the figure 4.2). No other interface tracking method does posses this behavior. Due to the constant displacements fields in both directions and the linear interpolation, the inflow is predicted correct and the two original coordinate fields that depend on the total displacements remain linear.

For rigid rotation a circular mesh is created as in figure 4.2 with radius. The rigid body rotation is of a circular interface with $R = 0.15$ and initial center at $[X_{cent}, Y_{cent}] = (-0.2, -0.2)$ in a circular domain with radius $R_{mesh} = 0.5$. In the results of the rigid rotation at bottom row of figure 4.2 a little numerical diffusion can be observed. This is because the displacement field is linear and for exact convection of this field higher order terms should be accounted for.

4.3.3 Single vortex flow field

The vortex flow problem is a test commonly used in literature [12, 24, 62]. A circular interface with center $[X_{cent}, Y_{cent}] = (\frac{\pi}{2}, \frac{\pi}{4})$ and radius $R = 0.18\pi$ is placed in a rectangular domain of $\Omega = [(0, \pi); (0, \pi)]$. A single vortex with a velocity field is defined as:

$$\begin{aligned} u &= -\sin(x)\cos(y) \\ v &= \sin(y)\cos(x) \end{aligned} \tag{4.13}$$

The circular object in this field stretches and spirals around the center of the domain. After a certain time the velocities are reversed and the spiral interface should end up at the original location.

In figure 4.3(a) and 4.3(b) the velocity field and the original coordinate field X_0 at $t = 0$ are shown respectively. In figure 4.3(c) the distorted X_0 at $t=5.6s$ is shown. After $t=5.6$ the velocities are reversed and at $t = 11.2s$ the X_0 field should be in the original position. In figure 4.3(d) the X_0 field is plotted at $t = 11.2s$.

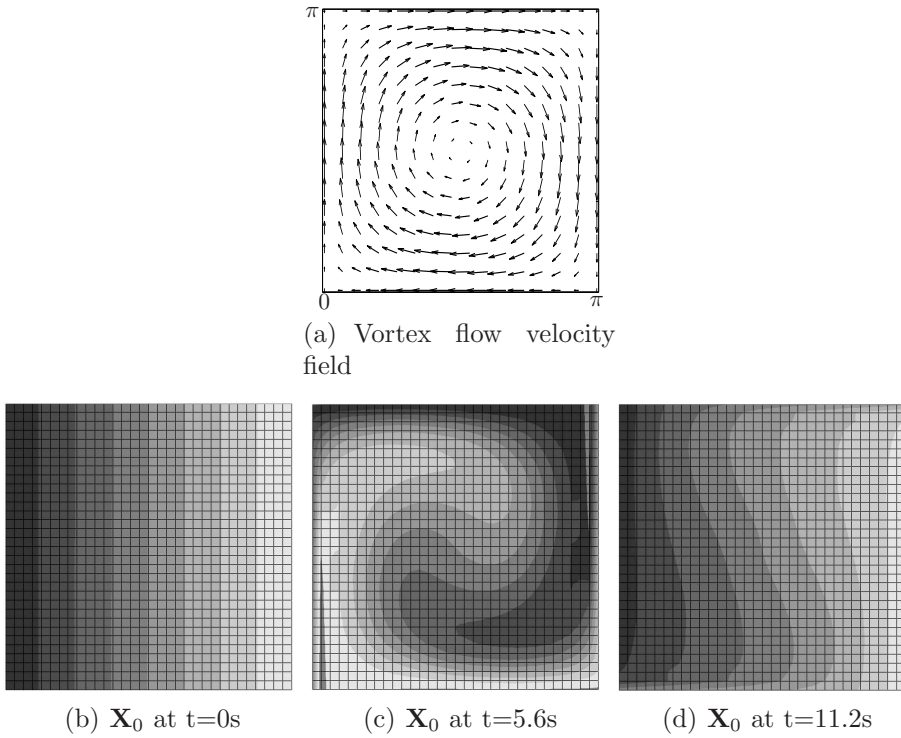


Figure 4.3: Vortex flow velocity field (a) and Original Coordinate Fields X_0 at $t=0.0s$, $5.6s$ and $11.2s$ (b,c,d) for a 32×32 grid.

In table 4.1 the test cases that were performed are shown. For all simulations the maximum courant number is kept constant at 0.06. The simulation is performed up to $t = 5.6s$ then the velocities are reversed and the simulation is continued till $t = 11.2s$.

Grid	# elements	C_r	Time steps	CPU time
16 x 16	256	.064	900	12
32 x 32	1024	.064	1800	101
64 x 64	4096	.064	3600	892

Table 4.1: Test cases for the single vortex flow fields.

In figure 4.4 the analytical solution and the results of the three test cases are shown. It is evident that with a finer mesh the prediction of the mesh improves. Even without adaptive remeshing, the predictions for D32 and D64

are quite good. In figure 4.3 it is clear that the original coordinate field at $t = 5.6s$ is no longer linear and diffusive effects will appear in the original coordinate field and therefore in the location of the interface.

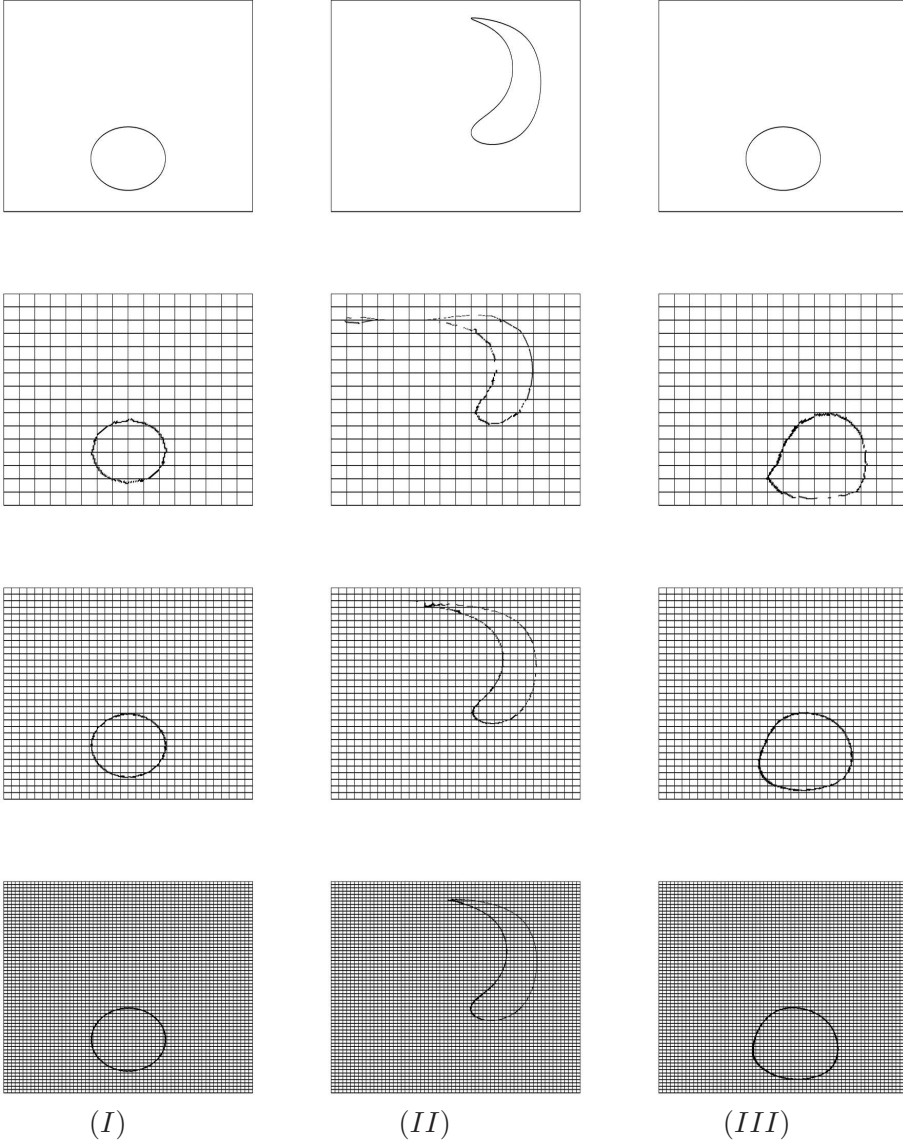


Figure 4.4: Interface tracking in a vortex flow, at $t=0$ (I), $t=5.6s$ (II), $t=11.2s$ (III).

From these results can be concluded that the convection scheme in combination with the original coordinate tracking shown good agreement with the analytical location of the interface. The results in 16x16 grid are somewhat oscillatory, but taking the coarseness of the grid into account, the results are reasonable. Mesh refinement shows that the numerical results converge to the exact solution.

4.4 Numerical model

In the previous section the accuracy of the original coordinate interface tracking scheme was demonstrated. In this section the numerical framework, specific for interface tracking is shown. These specific features concern the update of material properties during the simulation, boundary conditions and the description of the interface during extrusion.

4.4.1 Interface description for extrusion

At the start of an extrusion process the die is still filled with air and the front of a billet is straight and perpendicular to the extrusion direction. This makes the definition of the interface in original coordinates very simple. A function, dependent on the original coordinate X_0 in extrusion direction and a initial location of the front X_{ini} is defined.

$$\phi = X_{ini} - X_0 \quad (4.14)$$

For this specific application the function itself could be seen as a typical level set function. However not the function is convected but the total displacements are convected. The total displacements are initially constant equal to zero, whereas the initial level set function is linear. From the previous convection tests can be concluded that the convection of a constant field shows less diffusion than the convection of a linear field.

Through convection high gradients can appear the function ϕ . These high gradients will deteriorate the accuracy of the convection of the function. In many level set approaches the function is regularized to stay a distance function from the interface.

This is not the case for the original coordinate function. This function is regularized in such a manner that it will remain the original coordinate function. In paragraph the regularization technique used for the original coordinate function is treated.

The material properties in the 'air' domain should have zero or negligible stiffness in both deviatoric and hydrostatic sense. In equation (4.2) the material properties are calculated as weighted average of the two material parameters. In this work is chosen for a different approach. The material parameter ξ^i in an integration point is not a weighted average of the two phase parameters, but has either aluminium or pseudo material properties.

$$\xi^i = \begin{cases} \xi_{pseudo} & \text{if } \phi^i < 0 \\ \xi_{alu} & \text{if } \phi^i > 0 \end{cases} \quad (4.15)$$

Since the function ϕ is based on total material displacements known in nodes, the integration point value is interpolated from the nodal data.

$$\phi^i = \sum_{n=1}^k N_i^n \phi^n \quad (4.16)$$

The material parameters ξ_{alu} are defined through the Sellars-Tegart law (equation (2.30)). This gives a strain rate dependent material behavior. The bulk modulus C_b multiplied with the volumetric strain, determine the hydrostatic part. The optimal material parameters ξ_{pseudo} for the pseudo material, are discussed in the next section.

4.4.2 Permeable boundary conditions

When the die is filled and aluminium is extruded through the die, it is commonly accepted to model the aluminium sticking to the tools. In the bearing area this condition is often changed to slip (with or without friction).

If these conditions are also applied during filling, the pseudo material will stick to the wall and will not leave the die completely. Whenever one of the integration points in an element is in the aluminium domain the stiffness of the complete element is in the order of the stiffness of the aluminium. In a simulation this effect can be seen in a layer of elements at the boundary which are filled for approximately 50%. This effect is shown in figure 4.5(a). This effect is also described by Dvorkin [17, 18].

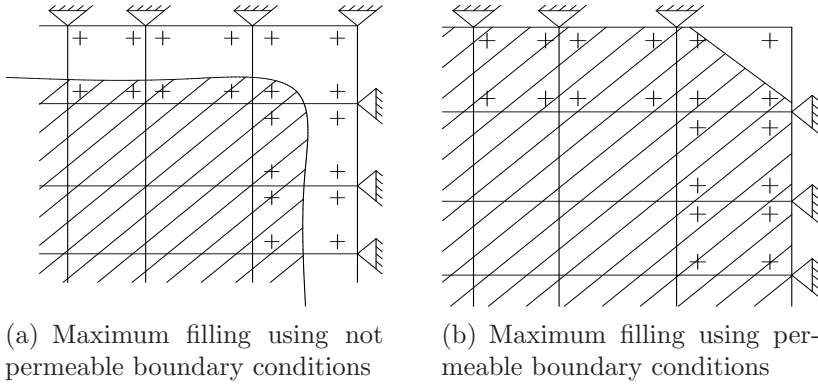


Figure 4.5: Difference between not permeable versus permeable boundary conditions during filling.

To avoid the appearance of a boundary layer, permeable boundary conditions are used as in [17, 18]. The boundary conditions are original coordinate function ϕ dependent. Using these conditions allows the 'air' to flow out freely, while aluminium slips or sticks the wall. These boundary conditions are shown in figure 4.6

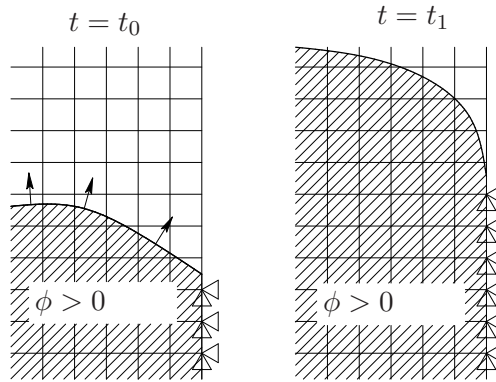


Figure 4.6: Permeable boundary conditions.

The boundary conditions on the boundary Γ are written as:

$$\begin{cases} \mathbf{u}_m & \text{is free} & \text{if } \phi^i < 0 \\ \mathbf{u}_m & = 0 & \text{if } \phi^i > 0 \end{cases} \quad (4.17)$$

Only using permeable boundary conditions would be a problem when air gets

trapped inside the aluminium. This can happen if porthole dies are used and aluminium has to weld together after it has passed the portholes. This effect can also appear during simulations. Therefore also the pseudo material should be adjusted so that it is compressible.

Another advantage of the permeable boundary conditions is the ability to use bigger time increments during the first part of filling. The material velocity is highest in the region upwind from the bearing area. Also the gradients are high in that region. Therefore the mesh is refined in this region. It is clear that the highest courant numbers are found there. The maximum step size for the model is determined by those elements. Using permeable boundary conditions the high exit velocity at the bearing area vanishes. The pseudo material is then not forced to leave the domain through the bearing area, but can flow out everywhere downstream from the front. This effect will lead to smaller courant numbers and the ability to use bigger increments.

4.4.3 Modeling the pseudo material

The pseudo material can be modeled in many different ways. In literature only the viscosity is mentioned as specific pseudo parameter. Here we explored 26 different models.

In simulation 0 (see table 4.2) the pseudo material is modeled as a viscous material. The viscosity is chosen low with respect to the aluminium to ensure that the pseudo material acts with low shearing forces on the aluminium. To make the pseudo material compressible the bulk modulus is also small compared to the aluminium bulk modulus.

In the other 25 simulations the material model for the pseudo material is an 'adapted' aluminium model (equation (2.30)) and bulk modulus. This adapting of the parameters can be done on different levels in DiekA. In figure 4.7 a simplified scheme of the implementation of a FEM program is given. It is an incremental, iterative procedure as in figure 2.4(b). Within a time increment DiekA iterates toward a solution of the non-linear equilibrium equations.

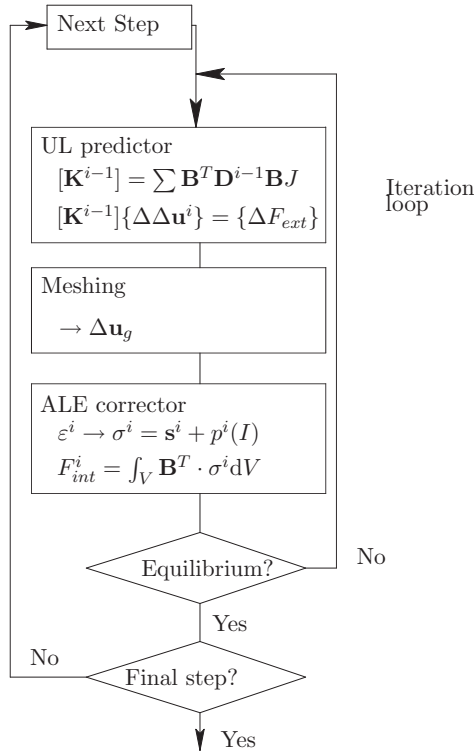


Figure 4.7: Semi-coupled ALE implementation.

The iteration scheme is a Newton-Raphson iteration in which a solution is predicted in the 'predictor' step. In the next part of the scheme, the 'corrector', the stresses are updated according to the solution from the predictor. If the unbalance after correction is within tolerance, DiekA has found a solution.

The five different factors used to adapt the pseudo material are:

- ρ_{yld}

The yield stress as defined in equation (2.30) is multiplied with the factor ρ_{yld} . This acts on the material stiffness $\mathbf{D}$ used in the predictor step as well as the stress calculated in the radial return routine in the corrector step.

$$\sigma_y^*(\dot{\kappa}) = \begin{cases} \rho_{yld} \cdot \sigma_y(\dot{\kappa}) & \text{if } \phi^i < 0 \\ \sigma_y(\dot{\kappa}) & \text{if } \phi^i > 0 \end{cases} \quad (4.18)$$

- ρ_{blk}

The bulk modulus is multiplied with the factor ρ_{blk} . This also impacts the material stiffness $\mathbf{D}$ in the predictor step and it reduces the hydrostatic pressure part of the stress p in the corrector step.

$$C_b^* = \begin{cases} \rho_{blk} \cdot C_b & \text{if } \phi^i < 0 \\ C_b & \text{if } \phi^i > 0 \end{cases} \quad (4.19)$$

- ρ_{sig}

The deviatoric stress $\mathbf{s}$ is multiplied with the factor ρ_{sig} in the corrector step during the update only.

$$\mathbf{s}^* = \begin{cases} \rho_{sig} \cdot \mathbf{s} & \text{if } \phi^i < 0 \\ \mathbf{s} & \text{if } \phi^i > 0 \end{cases} \quad (4.20)$$

- ρ_{hyd}

The hydrostatic pressure p is multiplied with the factor ρ_{hyd} in the corrector step only.

$$p^* = \begin{cases} \rho_{hyd} \cdot p & \text{if } \phi^i < 0 \\ p & \text{if } \phi^i > 0 \end{cases} \quad (4.21)$$

- ρ_{det}

The determinant of the Jacobian J is multiplied with the factor ρ_{det} in both predictor when calculating the contribution of an integration point to the stiffness matrix $[K]$ and in the corrector step in the calculation of the internal force vector.

$$J^* = \begin{cases} \rho_{det} \cdot J & \text{if } \phi^i < 0 \\ J & \text{if } \phi^i > 0 \end{cases} \quad (4.22)$$

The factors that act on both corrector and predictor ρ_{yld} , ρ_{blk} and ρ_{det} are preferred because of conservation of consistency between predictor and corrector. These options should not deteriorate the rate of convergence of the iteration scheme.

4.4.4 Flow front regularization

To smooth oscillations in the original coordinates a regularization of the total material displacements is applied. The used regularization is a special case of the local smoothing used in the convection of the total displacements as in

equation (4.6). The regularization is performed when convergence rate of the global iteration slows down.

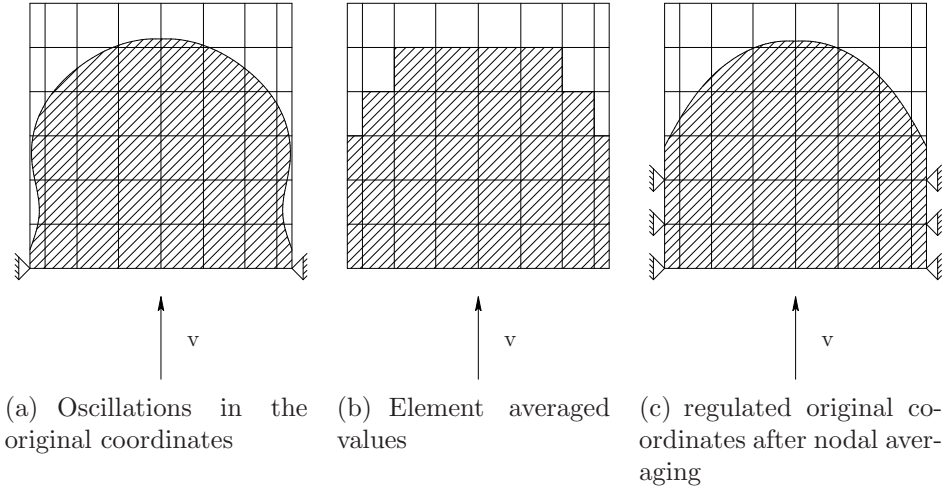


Figure 4.8: Regularization of the flow front.

For all elements the element average value for original coordinates is calculated and written to the nodes as in figure 4.8(b). The created discontinuous field is made continuous by nodal averaging. This creates the field as in figure 4.8(c). Note that extra boundary conditions appear by this regularization step.

With the factor β from equation (4.7) the smoothing can be less radical as using the element average as nodal values. In our implementation, when a smoothing step is applied the element average is put in the nodes. ($\beta = 0$). In the next section in figure 4.20 is shown that this improves the volume conservation behavior drastically. Also an improvement in convergence speed is observed.

4.5 2D filling of Die cavity

In this section a comparison is made of different option to model the pseudo material. Different values and combinations of the factors from paragraph 4.4.3 are used.

The geometry for the test is a 2D plane strain model and is shown in figure

4.9. The aluminium has to flow around an obstacle and merge back together in the welding chamber. This geometry is a representation of aluminium flow in a porthole die in 2D.

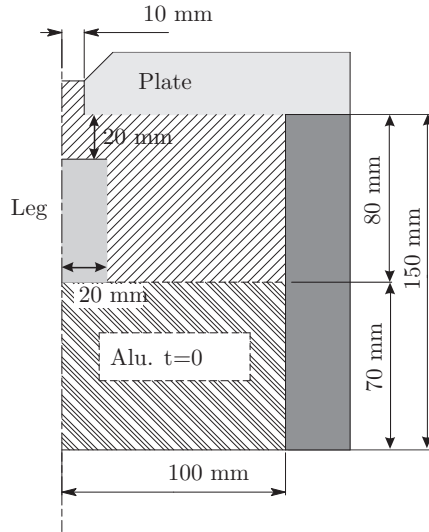


Figure 4.9: 2D geometry representing a porthole die.

The single crosshatched section is the domain inside the die filled with air at the start of the simulation. The double crosshatched section is the domain inside the container already filled with aluminium. At the container and die walls sticking conditions are assumed, except for the bearing where frictionless slip is assumed. Also the permeable boundary conditions are applied.

The complete domain is meshed with Q4, bilinear elements. The elements are selectively reduced integrated to avoid locking problems. No stabilization is implemented to stay clear of spurious modes. Some spurious effects can be found in elements at the boundary where the flow front has just entered.

The different combinations are summarized in table 4.2. Simulation 0 is with the pseudo material modeled as a viscous material with low viscosity. Since the viscous material model is linear the global iteration is expected to converge faster than with the use of a non-linear material model.

Simulation 19 is the simulation in which all the factors are equal to one, mak-

ing the pseudo material properties equal to the aluminium properties. This is for simulation times the reference simulation. Without the permeable boundary conditions this would be a regular extrusion simulation.

Two different mechanisms occur. The first mechanism shows the oscillations of figure 4.8(a). The other effect is also observed at the edge of the domain. When in an element at the edge of the domain both the interface has entered and one node stays permeable while the other is changed to fixed, the permeable node can have a (large) spurious displacement. When this happens in some subsequent increments the element fills up too fast.

4.5.1 Results of 2D die filling

The best option will be chosen based on the results of volume conservation, flow front shape, ability to fill corners and the simulation times.

Table 4.2: Simulations.

Simulation	ρ_{yld}	ρ_{blk}	ρ_{sig}	ρ_{hyd}	ρ_{det}	run
0	Viscous, $\mu = 10^{-6} m^2 s^{-1}$, $C_b = 1 MPa$					OK
1	0.1	1.0	1.0	1.0	1.0	Incomplete
2	0.01	1.0	1.0	1.0	1.0	Fails
3	0.001	1.0	1.0	1.0	1.0	Fails
4	1.0	0.1	1.0	1.0	1.0	OK
5	1.0	0.01	1.0	1.0	1.0	OK
6	1.0	0.001	1.0	1.0	1.0	OK
7	0.01	0.01	1.0	1.0	1.0	Incomplete
8	0.01	0.001	1.0	1.0	1.0	OK
9	0.01	0.00001	1.0	1.0	1.0	Fails
10	1.0	1.0	1.0	1.0	0.1	OK
11	1.0	1.0	1.0	1.0	0.01	Incomplete
12	1.0	1.0	1.0	1.0	0.001	Incomplete
13	1.0	1.0	0.667	1.0	1.0	OK
14	1.0	1.0	0.333	1.0	1.0	OK
15	1.0	1.0	0.000	1.0	1.0	Fails
16	1.0	1.0	-0.333	1.0	1.0	Fails
17	1.0	1.0	-0.667	1.0	1.0	Fails
18	1.0	1.0	-1.000	1.0	1.0	Fails
19	1.0	1.0	1.0	1.0	1.0	OK
20	1.0	1.0	1.0	0.667	1.0	OK
21	1.0	1.0	1.0	0.333	1.0	OK
22	1.0	1.0	1.0	0.000	1.0	Fails
23	1.0	1.0	1.0	-0.333	1.0	Fails
24	1.0	1.0	1.0	-0.667	1.0	Fails
25	1.0	1.0	1.0	-1.000	1.0	Fails

Not all options were able to complete the filling simulation. Especially the simulations where the consistency is destroyed, the convergence is slow or the solution diverges. The simulations that did not converge were ignored. In figure 4.10 the simulation times of completed simulations are shown. The simulation times of the incomplete simulations are set to 0.

The simulation with the pseudo material modeled as a viscous material converges properly. Simulation 1 does run, up to the point where the die is almost filled except for the bearing and "welding chamber" and will be ignored. Simulations 2, 3, and 9 do not converge before reaching the maximum iterations at the first step. Simulation 11 appears to give nice results, however this simulation only fills the die half before diverging and will therefore be ignored. Similar problems happen for simulations 15 to 18 and 22 to 25.

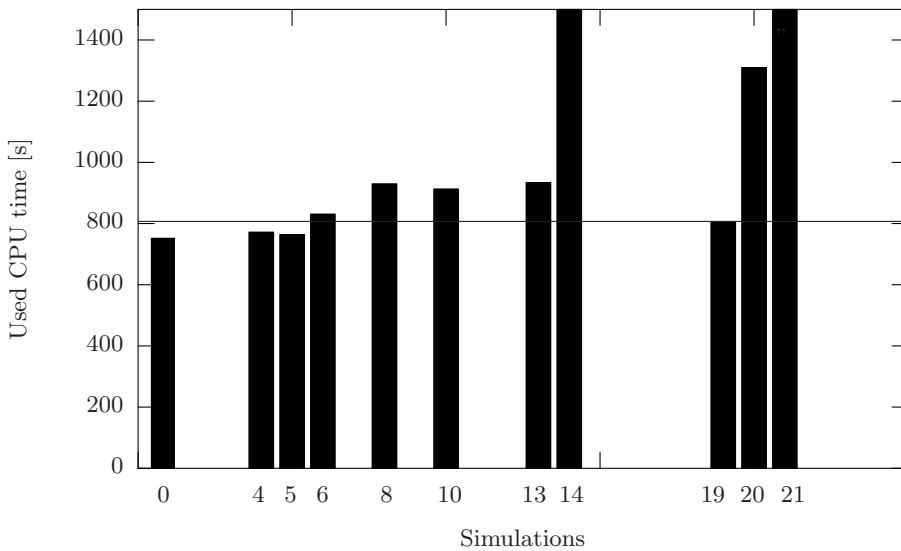


Figure 4.10: Used CPU time per simulation.

Simulation 19 is the reference simulation in which all the factors are 1.0, meaning that both pseudo material and aluminium are modeled as aluminium. In simulation 20 & 21 the hydrostatic stress is affected only mildly, so the consistency between predictor and corrector is not disturbed too badly, but convergence is slow and time consuming.

However the flow front is also not different from the flow front in simulation 19 as can be seen in figure 4.11. Only simulation 0 and simulation 8 give, a good representation of the flow front.

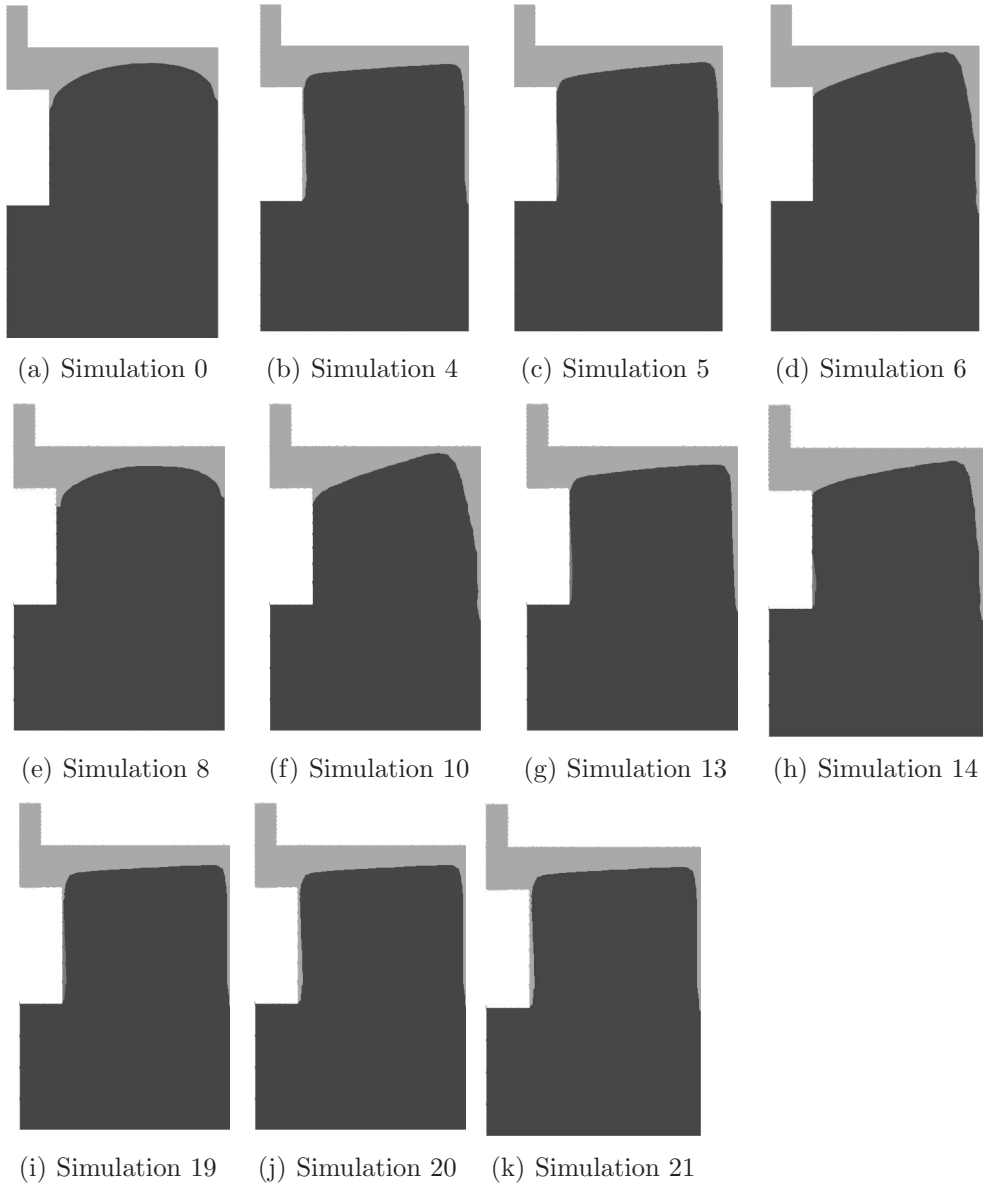


Figure 4.11: Flow front shapes.

The volume conservation error is calculated with respect to the total inserted volume.

$$Error^t = \frac{\text{Numerical volume} - \text{Analytical volume}}{\text{Analytical volume} - \text{Initial volume}} = \frac{V_{sim}^t - V_{an}^t}{V_{an}^t - V_{an}^{t=0}} \cdot 100 \quad (4.23)$$

The initial volume $V_{an}^{t=0}$ is easily calculated by the size of the domain that is filled. The analytical inserted volume can be calculated by multiplying the prescribed inflow velocity with the inflow area of the billet.

The simulated volume is calculated based on the nodal values of ϕ . In a simple post-processing step, the contribution of every element to the aluminium domain is calculated. The contribution of an element is numerically integrated, using the gauss integration points. If an integration point is in the pseudo domain the contribution is set to zero.

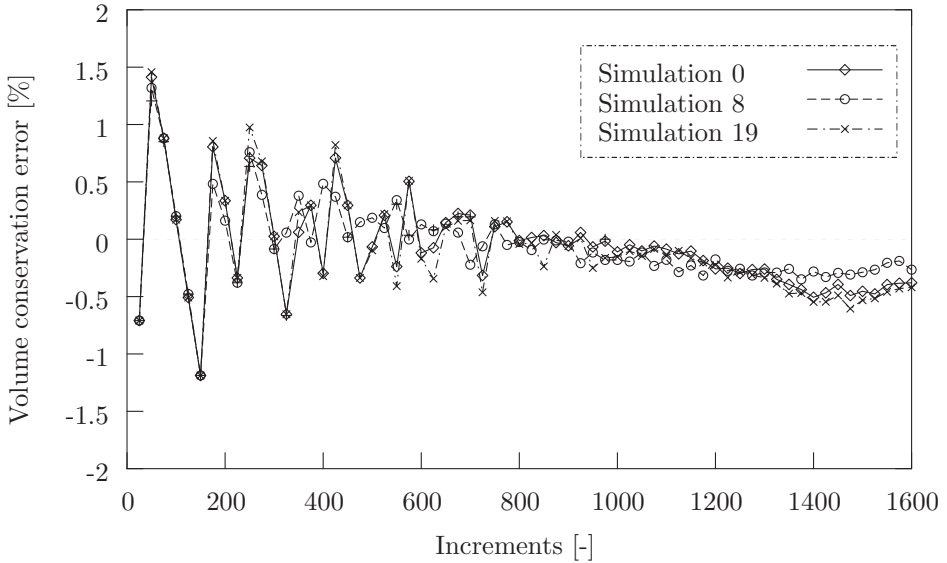


Figure 4.12: Volume conservation of simulations 0,8 and 19.

In the figure 4.12 the volume conservation is plotted against the steps. Only the results from simulations 0, 8 and 19 are plotted. It can be concluded that these simulations are accurate in volume conservation. The oscillations in volume conservation in the beginning of the simulations can be explained by the relatively coarse mesh that is used. Since the contribution of one element

is only integrated with 4 integration points, the volume V_{sim} can have rather big steps in time. Since the inflow V_{an} is calculated exactly and is small at first, rather large oscillations should appear.

4.6 3D Filling of die cavity

Aluminium extrusion of complex profiles is hardly ever a problem that can be reduced to a 2D simulation.

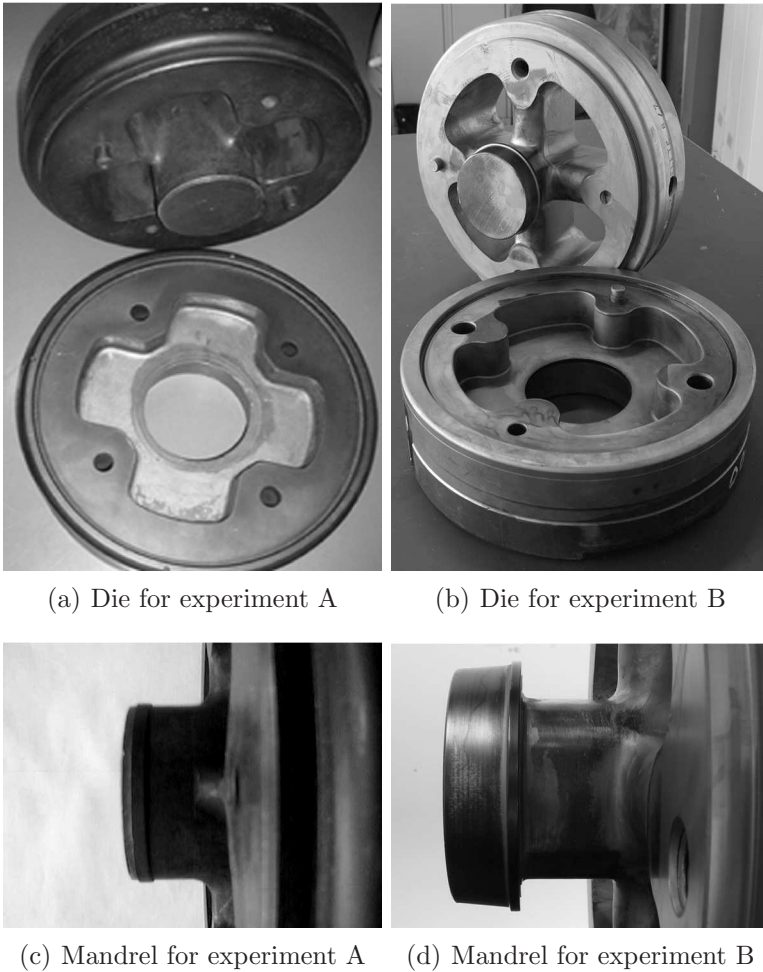


Figure 4.13: The dies for the tube extrusion experiment A & B.

In this section two different die designs, for the same profile, are compared. Both simulations and experiments have been performed and the nose piece of experiments and simulations are compared. The nose piece is the first part of the profile that exits the die.

The two different die designs are shown in figure 4.13. The extruded profile is a tube with diameter of 80 mm and a wall thickness of 1.72 mm. The dies both have four legs to support the mandrel. The main difference between the two dies is the big undercut in the mandrel in experiment B compared to a small undercut in the Mandrel in experiment A (see figure 4.13 c&d).

Another difference is the 5 mm bearing length in experiment A compared to 3 mm in experiment B.

For both experiments a ram speed of 5 mm/s is used. The alloy used is AA6063. The material properties for this alloy can be found in table 2.1.

4.6.1 FEM model

In this paragraph the FEM model and the interface tracking are described. The used method is the original coordinate tracking as prescribed in the previous sections.

The used elements are 3D 8 node hexahedral elements, see figure 4.14. The integration for this element type is selectively reduced to avoid volume locking. The performance of this element type for (nearly) incompressible material models is better than linear tetrahedral elements. A drawback of this element type is difficulties in mesh generation. In this case the domain is divided into many brick or wedge shaped sub domains. These sub domains can easily meshed by a mapping or sweeping method. The geometry of the dies is simplified to be able to create the sweepable sub domains. The main simplifications concern the rectangular legs, also all the fillets and small details are removed.

The most obvious difference between die A and die B is the large undercut just before bearing area in the mandrel in die B. Designers from Boalgroup attributed the improved performance of die B to this property, therefore this is the only geometrical difference between simulation A and B.

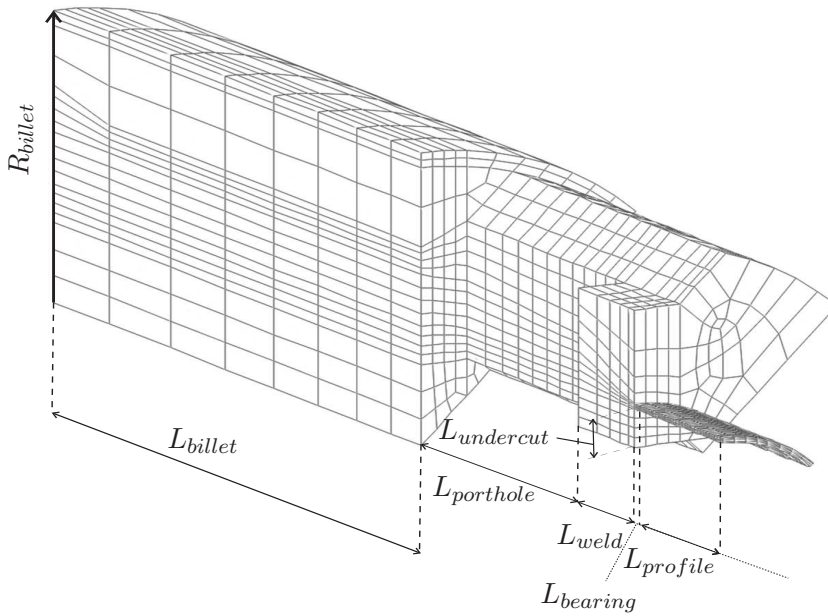


Figure 4.14: The used dimensions and mesh.

In the container and portholes the deformation occurs at the wall. In the interior the material moves as a plug. This is taken into account when creating the mesh. The main dimensions of the models can be found in table 4.3.

Table 4.3: Geometrical parameters simulation A & B.

	R_{billet} [mm]	L_{billet} [mm]	$L_{porthole}$ [mm]	L_{weld} [mm]	$L_{bearing}$ [mm]	$L_{profile}$ [mm]	$L_{undercut}$ [mm]
A	95	200	85.5	22	5	45	1
B	95	200	85.5	30	3	45	9

In figure 4.15 a & b the simplified geometry and a zoom of the FEM mesh (c & d) are shown. Since the domain is Eulerian the billet length does not change during extrusion. The billet length is chosen 200 mm, approximately equal to the diameter of the billet.

The boundaries in the billet, legs and welding chamber are permeable during the filling process as previously described. When the filling of the bearing area starts the nodes in the profile are no longer fully Eulerian, but grid displace-

ments are allowed. The grid displacement of the surface nodes is perpendicular to the surface and equal to the material displacement.

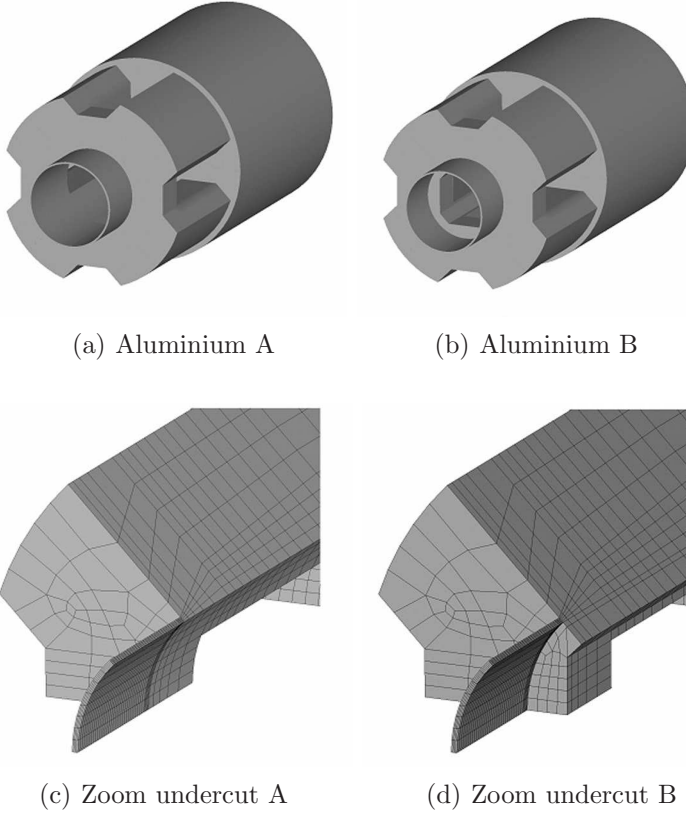


Figure 4.15: FEM model and the symmetry expanded Solid model.

The grid displacements of the nodes between the two outer surfaces are linearly interpolated between the outer nodes. This makes it possible to simulate distortions of the nose piece when it leaves the die. These surface options are shown in figure [4.16](#)

In simulation A high gradients develop in the original coordinate function and these give rise to deviations in the location of the front. To smooth these gradients the discussed regularization of the original coordinate function is applied. This is described in paragraph 4.4.4. In this case when a regularization step is applied the nodal values are averaged. For simulation A the regularization is performed at $t=3, 5, 7$ and 9 seconds.

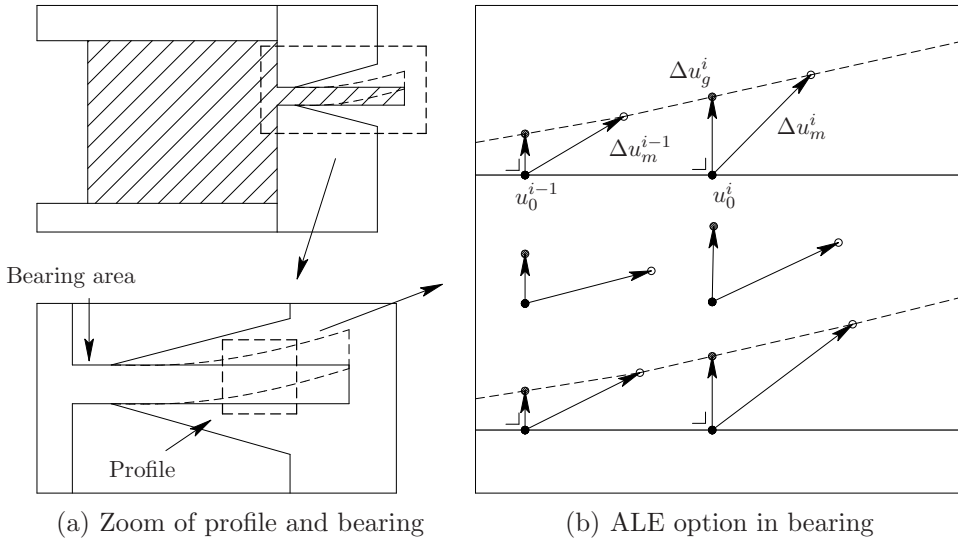


Figure 4.16: Special ALE surface options on the profile to model deflections.

For simulation B no spurious effects appeared, therefore no smoothing was necessary.

4.6.2 Results

In figures 4.17 & 4.18 the results of simulation A are shown. In figure 4.17a-c the evolution of the flow front is shown. It can be seen that the front progresses through the porthole. The contact line at the porthole walls also progresses with time.

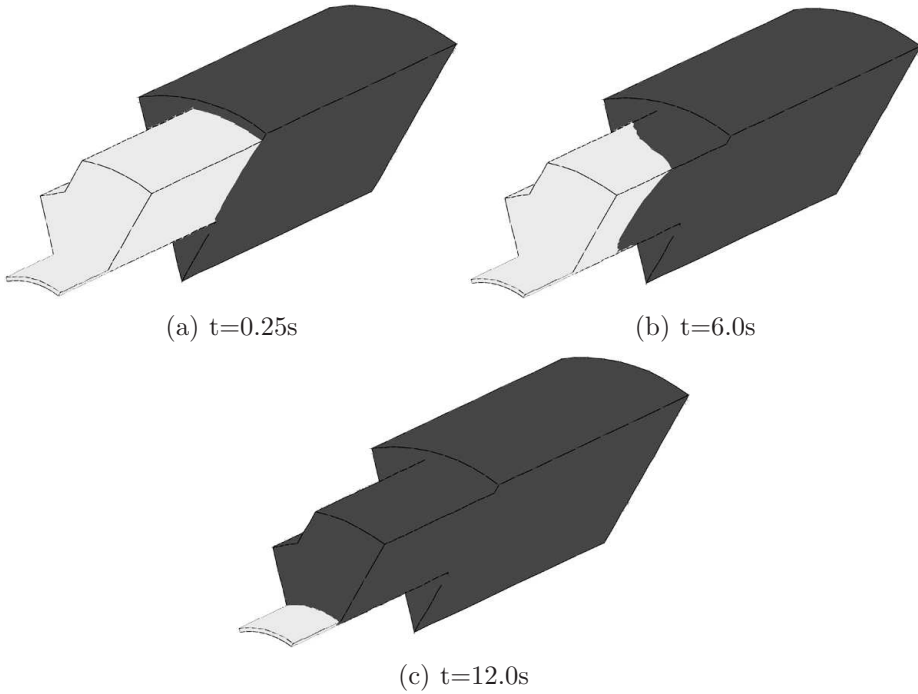


Figure 4.17: Aluminium interface evolution.

In figure 4.18 the evolution of the equivalent stress and pressure are plotted. It can be seen that the pressure in the billet increases during filling. The equivalent stress in the billet remains almost constant. The pressure and equivalent stress in the porthole are close to zero for $t=0.25$. The values increase steadily during filling.

In figure 4.19 the extrusion force for both simulations is shown. Since the porthole is equal for both simulations, the expected extrusion force should be the same up to the point of filling the welding chamber. Also the sudden increase in force at $t \approx 12s$ shows a clear start when filling is ended and extrusion starts. Simulation B starts extrusion approximately one second after simulation A, since the welding chamber is a little bigger.

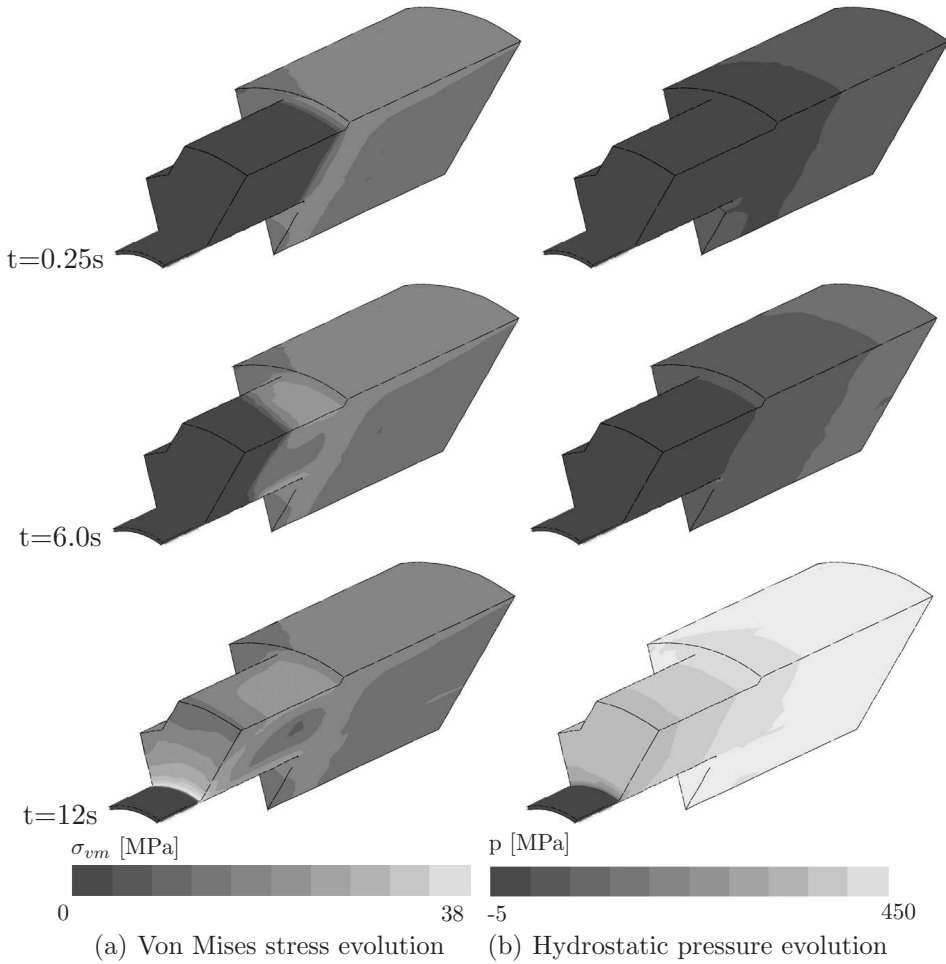


Figure 4.18: State variables evolution from simulation A at $t=0.25s$, $t=6s$ and $t=12s$.

The drops in the force in simulation A is because of the regularization. When the extrusion is stopped and the front is regularized the force in that 'artificial' step is equal to zero. Since the forces are only plotted at some increments it appears as a drop in the force.

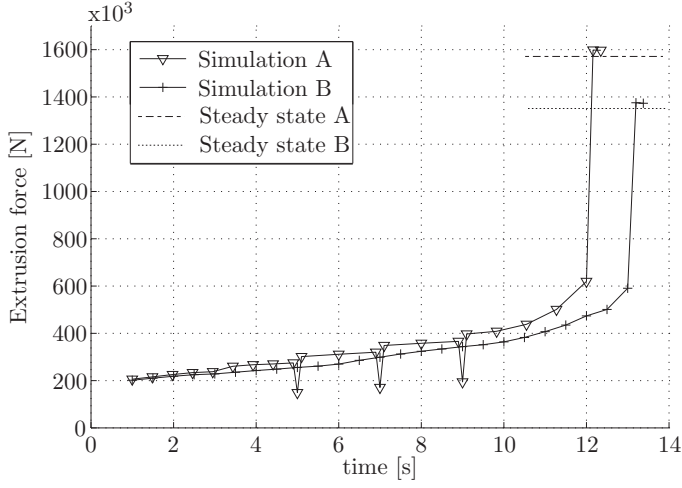


Figure 4.19: Extrusion force from both simulations.

Without the regularization the force is a continuous function. However after the regularization the force is higher than just before the regularization. The regularization makes the force curve discontinuous. This can be attributed to the way the front is repositioned after regularization. In figure 4.8 the effect of regularization is shown.

From figure 4.8(c) can be observed that due to the regularization nodes, that were permeable, are fixed after regularization. This is the cause for the increase in extrusion force, after regularization.

4.6.3 3D Volume conservation

Volume conservation is a measure for the accuracy of the flow front. For both simulations the volume of aluminium is checked and compared with the analytical volume. In figure 4.20 the error in the volume conservation is plotted in percentage against the time, see equation (4.23). Remarkable is the improvement of volume conservation due to the regularization steps.

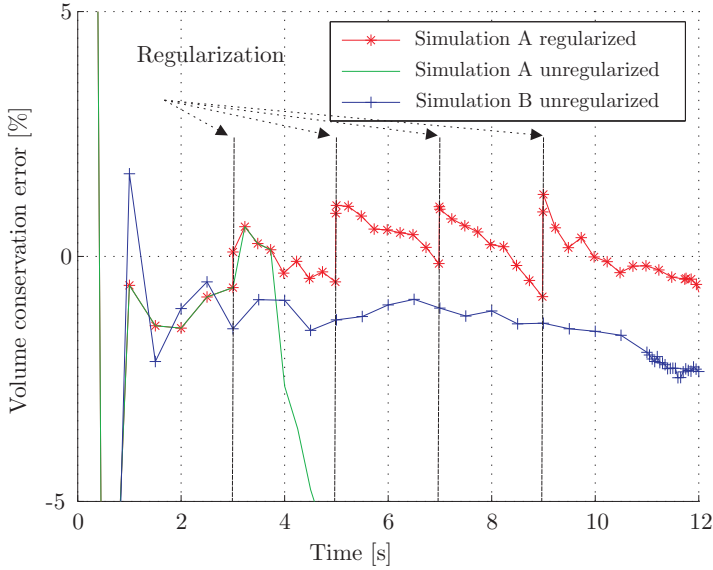


Figure 4.20: Volume conservation of both simulations.

4.6.4 Visual comparison

The most obvious result from the experiments is the improvement of uniformity of the outflow velocity of the tube using Die B, see figure 4.21. This improvement is attributed to the undercut in the mandrel and a deeper welding chamber.

From the comparison of the experiments with the simulations the conclusion can be drawn that the method will predict the front shape quite reasonably. In simulation A the profile shows a good agreement with the experiment. At the location of a leg the profile lags and in the middle of a porthole the profile bends in. It is noticed that due to the regularization of the front during filling the lagging of the profile behind a leg is smoothed. In simulation B some abnormalities are noticed at the symmetry plane. Looking at the fastest part of the profile, the peak is a numerical artifact and appears just before extrusion is started. Numerical diffusion does smooth this over a broader area, however at the front around the symmetry plane the profile still shows a peak spot.

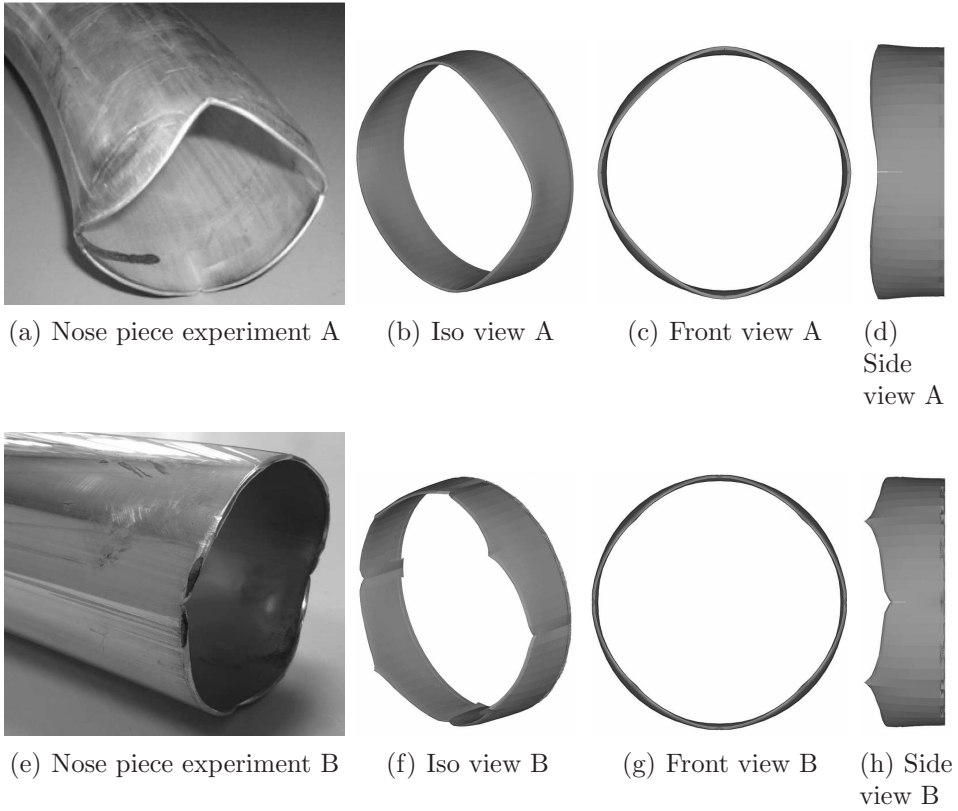


Figure 4.21: Simulation results of Nose piece A & B.

The prediction of the transversal bending of the front of the profile predicts the right trend. In experiment A the bending is more than in B. In simulation A the bending is approximately twice as much as in simulation B. The difference between experiments and simulations can be explained by the simplification of the geometry in both cases. In the simulations the different legs shapes and small fillets are ignored.

4.7 Conclusions

In this chapter is shown that the Original coordinate function is suitable to make efficient simulations of the filling of aluminium extrusion dies. For 3D the volume conservation is accurate and the predicted nose pieces show similarities with the experimental ones. The major drawback of the proposed method is

the use of the hexahedral elements. To be able to mesh the domain with topological correct hexahedral elements, the geometry had to be simplified to an extent that no correct prediction of the nose piece could be expected. In the next chapter the interface tracking algorithm is implemented in a tetrahedral element. Meshing with tetrahedral elements allows the user to add much more detail.

Chapter 5

A mixed finite element with original coordinate tracking

Creating finite element meshes of complex geometries is often done by commercial software tools. Many of these packages prefer triangular or tetrahedral elements over quadrilateral or hexahedral elements. The automatic mesh generation is easier for these elements. For computation time sake the elements are to be of low order. This also makes contact issues less problematic.

Although low order elements have been available since decades, it is well known that they do not yield accurate results. Especially when an extra physical constraint of (nearly) incompressibility is added, volume locking occurs. This failure may be assessed numerically by patch tests [32, 76] or mathematically verified by the Babuška-Brezzi condition [7].

A possibility to improve standard linear elements is by adding extra degrees of freedom such as pressure or strains. The degrees of freedom consist of a mix of displacement and other degrees of freedom: a mixed element. This technique was first introduced by Herrmann [26] and Chorin [11] to solve nearly incompressible linear elastic problems and incompressible viscous flow problems respectively. Later Schneider [57] introduced equal order interpolations for displacement field and pressure field, regarding both fields as separate fields. When both pressure and displacement fields are chosen linear and continuous, it is well known that these elements fail the patch tests [75].

One way to stabilize the pressure field is to augment the linear displacement interpolation functions by a bubble function. This was first introduced by Arnold et. al. [4] to solve the Stokes problem. This type of element is called

MINI element. In this thesis MINI elements are referred to as T3P1⁺P1 (triangular) and T4P1⁺P1 (tetrahedral). Coupez et. al. [22, 53] uses an element from the MINI family to solve 3D mold filling problems for viscous incompressible fluids. The bubble displacements are four piecewise linear functions to maintain exact integration property of the element. Coupez solves the transport equation decoupled from the flow problem. Taylor [61] introduced the use of bubbles in a mixed-enhanced tetrahedral element for small and finite elastic strains.

Meshing and contact problems are easier with linear triangles or tetrahedral elements. From the literature in the above paragraphs the behavior of the MINI elements show good results. However not much literature about the use of MINI elements in ALE formulations is available. In the next sections, our implementation of two MINI elements is described. In this chapter many different Element types are mentioned by their abbreviation. In table 5.1 an overview these elements and abbreviations is given.

	Abbr.	Elem. shape	# nodes	DOFS	Interpolation
2D	T3	Triangular	3	u,v	linear
	Q4	Quadrilateral	4	u,v	bi-linear
	T3P1 ⁺ P1	Triangular	4 (3+1)	u,v,p + u,v	linear + bubble
3D	TET4	Tetrahedral	4	u,v,w	linear
	H8	Hexahedral	8	u,v,w,	bi-linear
	T4P1 ⁺ P1	Tetrahedral	5 (4+1)	u,v,w,p + u,v,w	linear + bubble

Table 5.1: Element types and abbreviations.

5.1 Theoretical outline

In chapter 2 the basic outline of a semi-coupled ALE simulation is shown. In the next sections the theoretical outline and implementation of a MINI element in a semi-coupled ALE formulation are treated.

5.1.1 Strong formulation

The strong form of the equilibrium equations with the boundary conditions is given in equation (5.1).

$$\begin{aligned}
 \boldsymbol{\sigma} \cdot \boldsymbol{\nabla} &= \mathbf{0} & \text{in } \Omega_g \\
 \mathbf{u} &= \mathbf{u}_0 & \text{on } \Gamma_u \\
 \boldsymbol{\sigma} \cdot \mathbf{n} &= \mathbf{t} & \text{on } \Gamma_t
 \end{aligned} \tag{5.1}$$

5.1.2 Weak formulation

Since a semi-coupled ALE description is used, the Updated Lagrangian formulation of the weak form is used in the predictor step. In Appendix A the weak form for an Updated Lagrangian formulation is derived. Here the result is written as:

$$\int_{\Omega} \mathbf{w} \nabla : (- \boldsymbol{\sigma} \cdot \mathbf{L}^T + \dot{\boldsymbol{\sigma}} + \boldsymbol{\sigma} \text{tr} \mathbf{L}) d\Omega = \int_{\Gamma_t} \mathbf{w} \cdot \dot{\mathbf{t}} d\Gamma_t + \int_{\Gamma_t} \mathbf{w} \cdot \mathbf{t} \dot{J}_{\Gamma} d\Gamma_t \quad (5.2)$$

5.2 FEM discretisation

The finite element method is applied to discretize the weak formulations. The domain is split into a finite number of sub domains. Integrals of the weak form are subdivided into a finite number of sub integrals. For each element the incremental displacement is described by the interpolation functions N and the nodal incremental displacements $\Delta \mathbf{u}$. The pressure is interpolated using the interpolation functions M . Within one increment the velocity and (rate of the) pressure are assumed constant making $\mathbf{v} = \frac{\Delta \mathbf{u}}{\Delta t}$ and $\dot{\mathbf{p}} = \frac{\Delta \mathbf{p}}{\Delta t}$.

$$\Delta \mathbf{u} = \sum \Delta \mathbf{u}_i N_i \quad ; \quad \mathbf{x} = \sum \mathbf{x}_i N_i \quad (5.3)$$

$$\Delta p = \sum \Delta p_i M_i \quad (5.4)$$

N_i are linear for the corner nodes and cubic or quartic for the bubble node. Since p_i is chosen linear the shape functions M_i are linear shape functions and chosen identical to the linear part of N_i .

The derivation for the MINI element start with the same weak form as in paragraph 2.3.2. For the MINI elements the stresses are split into a deviatoric and a hydrostatic part. For the weighing functions the Galerkin method is applied here, which means that the same interpolation functions used for the displacements are also used for the weighing functions.

$$\mathbf{w} = \sum \mathbf{w}_i N_i \quad ; \quad p^* = \sum p_i^* M_i \quad (5.5)$$

$$\mathbf{L} = \mathbf{v} \nabla = \sum \mathbf{v}_i N_i \nabla \quad (5.6)$$

$$\mathbf{D} = \frac{1}{2} (\mathbf{L} + \mathbf{L}^T) = \sum \frac{1}{2} (\mathbf{v}_i N_i \nabla + \nabla N_i \mathbf{v}_i) = \sum \mathbf{B}_i \mathbf{v}_i \quad (5.7)$$

And the same for the weighing functions:

$$\mathbf{L}^* = \mathbf{w} \overleftarrow{\nabla} = \sum \mathbf{w}_i N_i \overleftarrow{\nabla} \quad (5.8)$$

$$\mathbf{D}^* = \frac{1}{2}(\mathbf{L}^* + \mathbf{L}^{*T}) = \sum \frac{1}{2}(\mathbf{w}_i N_i \overleftarrow{\nabla} + \overrightarrow{\nabla} N_i \mathbf{w}_i) = \sum \mathbf{B}_i \mathbf{w}_i \quad (5.9)$$

The Jaumann stress rate can be written as:

$$\overset{\nabla}{\sigma} = \overset{\nabla}{s} - \dot{p} \mathbf{1} = \mathbf{Y} : \mathbf{D} - \boldsymbol{\sigma} \cdot \mathbf{W} + \mathbf{W} \cdot \boldsymbol{\sigma} - \boldsymbol{\sigma} \text{tr} \mathbf{D} - \dot{p} \mathbf{1} \quad (5.10)$$

with $Y = 2G\mathbf{I} - \frac{3G\mathbf{S}\mathbf{S}}{(1+h)\sigma_y^2} - \frac{2}{3}G\mathbf{1}\mathbf{1}$. Equation (5.10) can be substituted into equation (5.2):

$$\begin{aligned} \int_{\Omega} \mathbf{w} \overleftarrow{\nabla} : (-\boldsymbol{\sigma} \cdot \mathbf{L}^T + \mathbf{Y} : \mathbf{D} - \boldsymbol{\sigma} \cdot \mathbf{W} + \mathbf{W} \cdot \boldsymbol{\sigma}) \, d\Omega - \\ \int_{\Omega} \mathbf{w} \overleftarrow{\nabla} : \dot{p} \mathbf{1} \, d\Omega = \\ \int_{\Gamma_t} \mathbf{w} \cdot \dot{\mathbf{t}} \, d\Gamma_t + \int_{\Gamma_t} \mathbf{w} \cdot \mathbf{t} \dot{J}_{\Gamma} \, d\Gamma_t \end{aligned} \quad (5.11)$$

This can be rewritten into:

$$\begin{aligned} \int_{\Omega} \mathbf{w} \overleftarrow{\nabla} : (\mathbf{L} \cdot \boldsymbol{\sigma} + (\mathbf{Y} : \mathbf{D} - \boldsymbol{\sigma} \cdot \mathbf{D} - \mathbf{D} \cdot \boldsymbol{\sigma})) \, d\Omega - \\ \int_{\Omega} \mathbf{w} \overleftarrow{\nabla} : \dot{p} \mathbf{1} \, d\Omega = \\ \int_{\Gamma_t} \mathbf{w} \cdot \dot{\mathbf{t}} \, d\Gamma_t + \int_{\Gamma_t} \mathbf{w} \cdot \mathbf{t} \dot{J}_{\Gamma} \, d\Gamma_t \end{aligned} \quad (5.12)$$

By using equations (5.3)-(5.9) and keeping the velocity and rate of pressure constant during the increment we get:

$$\begin{aligned} \sum_{ij} \mathbf{w}_j \left[\int_{\Omega} N_j \overleftarrow{\nabla} : \boldsymbol{\sigma} : N_i \overleftarrow{\nabla} \, d\Omega + \right. \\ \left. \int_{\Omega} \mathbf{B}_j^T : (\mathbf{Y} - \boldsymbol{\sigma} \cdot \mathbf{I} - \mathbf{I} \cdot \boldsymbol{\sigma}) : \mathbf{B}_i \, d\Omega \right] \Delta \mathbf{u}_i - \mathbf{w}_j \int_{\Omega} \mathbf{B}_j^T : \mathbf{1} M_i \, d\Omega \Delta p_i = \\ \sum_{ij} \mathbf{w}_j \left[\int_{\Gamma_t} N_j \cdot \Delta \mathbf{t} \, d\Gamma_t + \int_{\Gamma_t} N_j \cdot \mathbf{t} \dot{J}_{\Gamma} \, d\Gamma_t \right] \quad \forall \mathbf{w}_j \end{aligned} \quad (5.13)$$

Which implies

$$\begin{aligned} & \sum_i \left[\int_{\Omega} N_j \overleftarrow{\nabla} : \boldsymbol{\sigma} : N_i \overleftarrow{\nabla} \, d\Omega + \right. \\ & \left. \int_{\Omega} \mathbf{B}_j^T : (\mathbf{Y} - \boldsymbol{\sigma} \cdot \mathbf{I} - \mathbf{I} \cdot \boldsymbol{\sigma}) : \mathbf{B}_i \, d\Omega \right] \Delta \mathbf{u}_i - \int_{\Omega} \mathbf{B}_j^T : \mathbf{1} M_i \, d\Omega \Delta p_i = \quad (5.14) \\ & \sum_i \left[\int_{\Gamma_t} N_j \cdot \Delta \mathbf{t} \, d\Gamma_t + \int_{\Gamma_t} N_j \cdot \mathbf{t} \dot{J}_{\Gamma} \, d\Gamma_t \right] \end{aligned}$$

The constraint for the constitutive equation of the pressure gives the second equation:

$$\int_{\Omega} p^* \left(\text{tr}(\mathbf{D}) + \frac{1}{C_b} \dot{p} \right) d\Omega = 0 \quad (5.15)$$

Using the interpolations functions gives and integration in time gives:

$$\sum_{ij} p_j^* \left[\int_{\Omega} M_j^T \mathbf{1} : \mathbf{B}_i \, d\Omega \mathbf{u}_i + \frac{1}{C_b} \int_{\Omega} M_j^T M_i \, d\Omega p_i \right] = 0 \quad (5.16)$$

In this integration over time the volume strain is assumed to be elastic.

$$p = -C_b \frac{\Delta \mathbf{v}}{\mathbf{v}} = -C_b J \approx \sum_i \mathbf{1} : \mathbf{B}_i \mathbf{u}_i \quad (5.17)$$

With $J = \det(\mathbf{F})$ and $\mathbf{F}$ the deformation gradient tensor. The determinant of the deformation gradient tensor is linearized as $J = \mathbf{1} : \mathbf{B}_i \mathbf{u}_i$. That this linearisation is allowed, is shown later in this chapter when volume conservation is compared between simulations with linearized volume strain and simulations with nonlinear volume strain implemented.

In incremental formulation with $\mathbf{u}^0$ and $\mathbf{p}^0$ the displacements and pressures at the beginning of the increment.

$$p_j^* \left[\int_{\Omega} M_j^T \mathbf{1} : \mathbf{B}_i \, d\Omega \Delta \mathbf{u}_i + \frac{1}{C_b} \int_{\Omega} M_j^T M_i \, d\Omega \Delta p_i \right] = \quad (5.18)$$

$$-p_j^* \left[\int_{\Omega} M_j^T \mathbf{1} : \mathbf{B}_i \, d\Omega \mathbf{u}_i^0 - \frac{1}{C_b} \int_{\Omega} M_j^T M_i \, d\Omega p_i^0 \right] \quad \forall p_j^* \quad (5.19)$$

Which implies

$$\int_{\Omega} M_j^T \mathbf{1} : \mathbf{B}_i \, d\Omega \Delta \mathbf{u}_i + \frac{1}{C_b} \int_{\Omega} M_j^T M_i \, d\Omega \Delta p_i = \quad (5.20)$$

$$- \int_{\Omega} M_j^T \mathbf{1} : \mathbf{B}_i \, d\Omega \mathbf{u}_i^0 - \frac{1}{C_b} \int_{\Omega} M_j^T M_i \, d\Omega p_i^0 \quad (5.21)$$

The displacements consist of linear and bubble displacements:

$$\Delta \mathbf{u} = \left\{ \begin{array}{c} \Delta \mathbf{u}_l \\ \Delta \mathbf{u}_b \end{array} \right\} \quad (5.22)$$

In the further description no distinction is made between the bubble and linear displacements. Equation (5.14) can be written in the following matrix notation:

$$\begin{bmatrix} K_{uu} & K_{up} \\ K_{pu} & K_{pp} \end{bmatrix} \left\{ \begin{array}{c} \Delta \mathbf{u} \\ \Delta p \end{array} \right\} = \left\{ \begin{array}{c} R_u \\ R_p \end{array} \right\} \quad (5.23)$$

with:

$$K_{uu} = \left[\int_{\Omega} N \overleftarrow{\nabla} : \boldsymbol{\sigma} : N \overleftarrow{\nabla} \, d\Omega + \int_{\Omega} \mathbf{B}^T : (\mathbf{Y} - \boldsymbol{\sigma} \cdot \mathbf{I} - \mathbf{I} \cdot \boldsymbol{\sigma}) : \mathbf{B} \, d\Omega \right] \quad (5.24)$$

$$K_{up} = K_{pu}^T = - \int_V \mathbf{B}^T : \mathbf{1} M \, dV \quad (5.25)$$

$$K_{pp} = \frac{1}{C_b} \int_V M^T M \, dV \quad (5.26)$$

and

$$R_u = F_u - \int_V \mathbf{B}^T \boldsymbol{\sigma}^0 \, dV \quad (5.27)$$

$$R_p = - \int_{\Omega} M^T \mathbf{1} : \mathbf{B} \, d\Omega \mathbf{u}^0 - \frac{1}{C_b} \int_{\Omega} M^T M \, d\Omega p^0 \quad (5.28)$$

5.3 Convection

The goal is to add the possibility to do semi-coupled ALE simulations with this element. In the previous section the formulas for the updated Lagrangian part are derived. In this section the calculation of the convective increment is treated. The scheme is mostly similar to the WLGS scheme described in

section 4.3.1. In the next paragraph differences for on the one hand T3P1⁺P1 & T4P1⁺P1 and the other hand Q4 and H8 elements are treated.

The WLGS scheme consists of 2 main steps. First a continuous smoothed field is created based on integration point or nodal values from the Updated step. In the second step the values in nodes in the new locations are calculated using the continuous smoothed field. In the calculation of the continuous field and remap of the state variables for the MINI elements, the bubble displacement is ignored. In other words the elements are treated as regular linear T3 or TET4 elements. The total bubble displacement itself is set to zero after each step when equilibrium is satisfied. This choice will be explained in paragraph 5.4.3.

5.3.1 Courant number for triangular elements

The Courant number is calculated in local coordinates. The calculation is given in equation (5.29).

$$C_r = \frac{1}{n_{ip}} \sum_{i=1}^{n_{ip}} \frac{|\Delta r_{ip}|}{l} \quad (5.29)$$

With Δr_{ip} the shift of a point in the element in iso parametric (local) coordinates and l the characteristic element length in local coordinates. The total courant number is calculated as:

$$C = \sqrt{C_r^2 + C_z^2 + C_h^2} \quad (5.30)$$

The courant number is calculated as element average with l the characteristic element length. For quadrilateral and hexagonal elements l is in all local coordinate directions the same. The characteristic element length for triangular and tetrahedral elements is not so well defined.

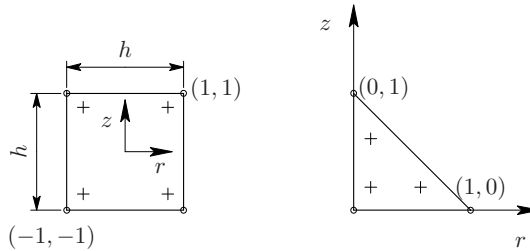


Figure 5.1: Characteristic element length for Q4 and T3 elements in local coordinates.

This is demonstrated in figure 5.1 for Q4 and T3 elements. For Q4 elements it is clear that $l = 2$. For the triangle element the element edge at the r and z axis is 1 long and the length is 0 at $(r, z) = (1, 0) \& (0, 1)$. It seems logical to choose the typical element length equal to the square root of the area of the element for 2D elements or cube root for 3D elements.

$$l_{T3} = \sqrt{A} = \frac{1}{2}\sqrt{2} \approx 0.71 \quad (5.31)$$

$$l_{TET4} = \sqrt[3]{V} = \sqrt[3]{\frac{1}{6}} \approx 0.55 \quad (5.32)$$

Convection test for characteristic element length

Since the Courant number is used to determine local smoothing, the choice for l will have an effect on the diffusion. To determine the optimal characteristic element length the convection test in figure 5.2 is performed.

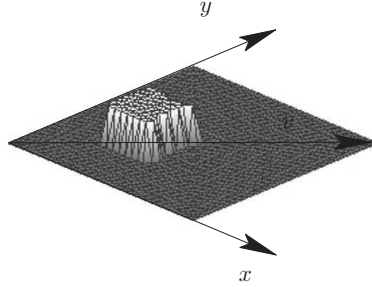


Figure 5.2: Mesh and initial values for the nodal pressures.

In a square 2D mesh $\Omega = [(0,0); (1,1)]$ the initial values for a nodal state variable are set to 0 except for a square block $\Omega_p = [(0.25, 0.25); (0.5, 0.5)]$ that get an initial value of 1. In 300 steps the state variable is convected through the mesh. The displacements are prescribed constant in the $x=y$ direction.

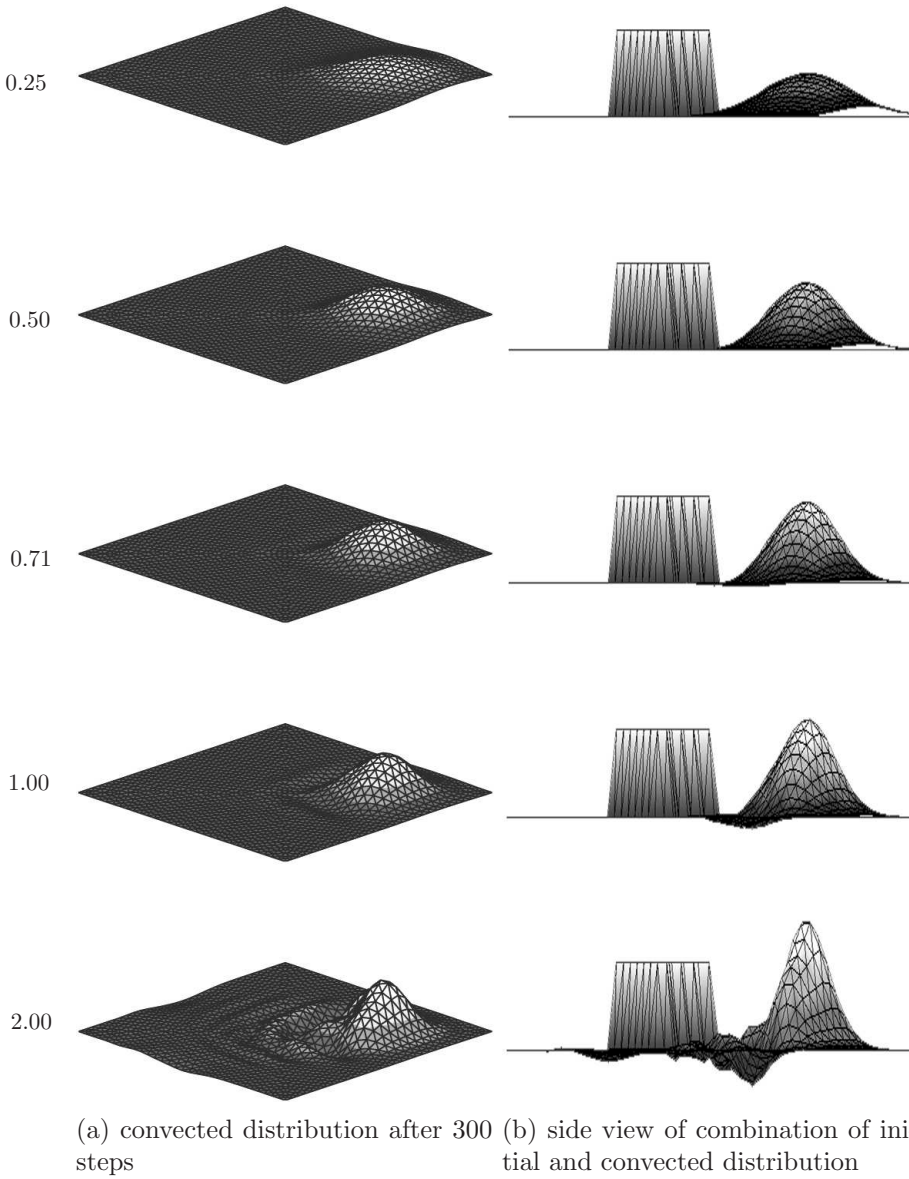


Figure 5.3: Initial and convected distribution after 300 increments. From top to bottom $l = [0.25, 0.50, 0.71, 1.00, 2.00]$.

The simulation is repeated five times with different characteristic element lengths ($l = [0.25, 0.50, 0.71, 1.00, 2.00]$). In figure 5.3 the results from the

simulations are shown.

In the left column the values of l are indicated. In the second and third column the initial and convected pressure field respectively are plotted. In the right column a side view of both initial and convected values are plotted. From the results can be observed that the WLGS scheme will remain stable for characteristic element lengths up to $l \approx 0.71$. A higher characteristic element length will lead to spurious oscillations in the pressure field. Small values of h will diffuse the results excessively.

The same simulations is performed with a 5 times higher courant number. These simulations are not shown, but the results are highly similar and the optimal value for l is the same.

5.3.2 Rigid rotation & Single vortex flow field

In chapter 4 the WLGS convection scheme is tested with the rigid rotation simulation in paragraph 4.3.2 and the vortex flow problem in paragraph 4.3.3. In this section these simulations are repeated for an unstructured 2D T3P1+P1 mesh. The circular interface is rotated 360° in 400 steps. The maximum courant number is found at the edge of the domain $C \approx 0.35$. Since the element size is approximately constant the courant number decrease linearly towards 0 in the center.

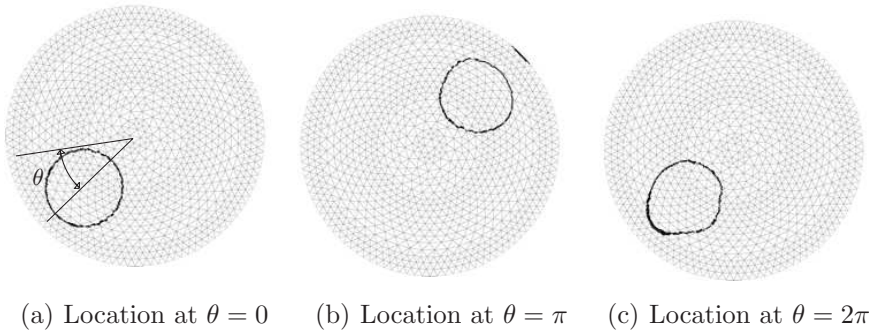


Figure 5.4: Rigid rotation in an unstructured T3P1+P1 mesh.

The results for the T3P1+P1 mesh are shown in figure 5.4. The same simulation is performed to test the convection scheme with the 3D T4P1+P1 element. In figure 5.5 the mesh is shown.

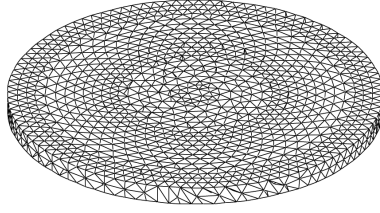


Figure 5.5: Mesh for convection test in an unstructured T4P1⁺P1 mesh.

The results are shown in figure 5.6

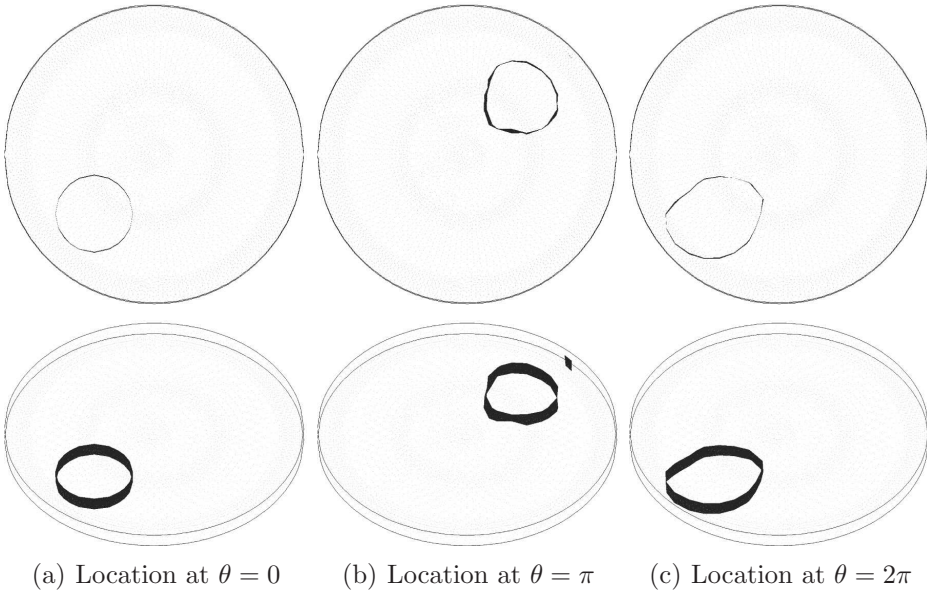


Figure 5.6: Rigid rotation in an unstructured T4P1⁺P1 mesh.

The overall result for this course mesh is very good. From these simulations can be observed that in this linear prescribed displacement field some diffusion is introduced through the WLGS scheme.

Vortex flow

The vortex flow problem described in chapter 4 is repeated for unstructured triangular meshes. The velocity field is prescribed in rectangular domain of

$\Omega = [(0, \pi); (0, \pi)]$ by:

$$\begin{aligned} u &= -\sin(x)\cos(y) \\ v &= \sin(y)\cos(x) \end{aligned} \quad (5.33)$$

In figure 5.7 the results are shown. For the fine mesh a smaller step size is used, so that the courant number for both simulations is comparable.

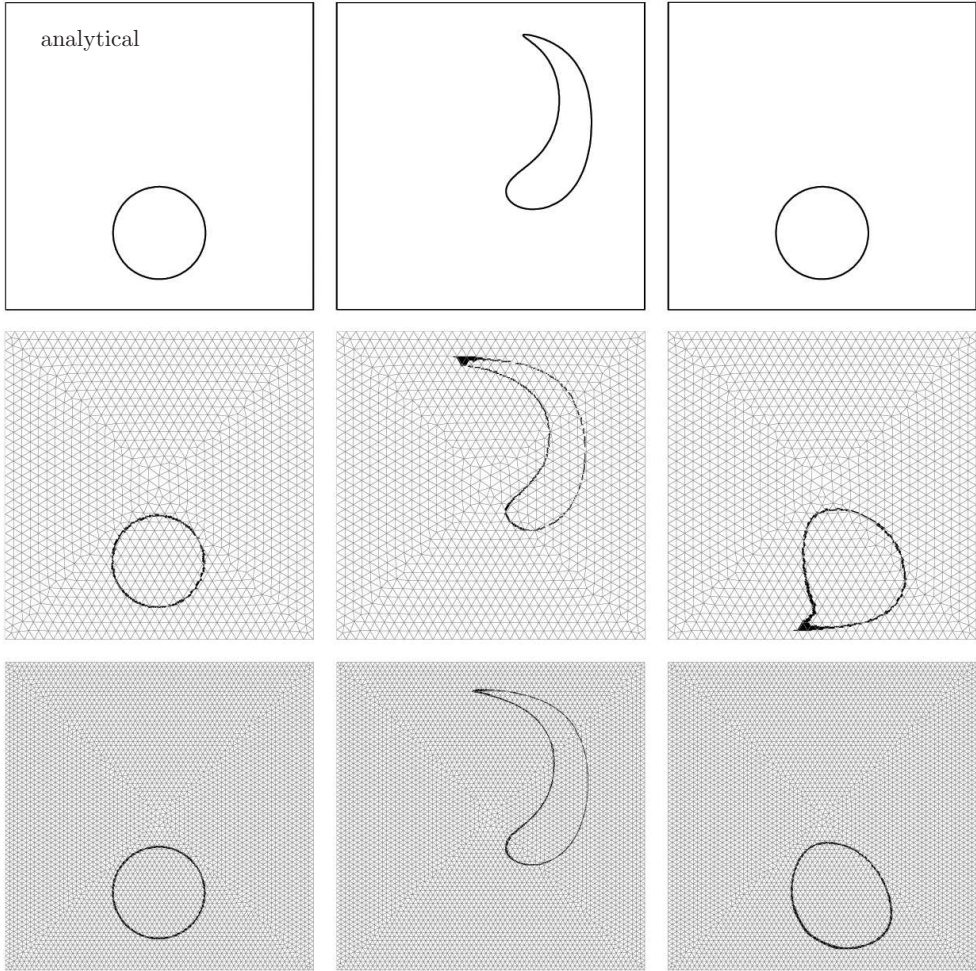


Figure 5.7: Vortex flow in an unstructured T3P1⁺P1 mesh.

This simulation is performed for the two mesh densities in figure 5.7, since in the previous chapter is shown that with courser mesh density the results are

not in good agreement with the analytical solution.

From the results of this simulation can be concluded that even for highly inhomogeneous displacements fields the convection of the field is reasonable. The deviation in the final shape is caused by diffusive effects. This effect reduces when less local smoothing is applied, however then oscillations in the field at the edges of the domain appear.

5.4 Implementation in DiekA

5.4.1 Incremental-iterative update

In the previous section the main equations for the FEM discretisation are derived. In this section some extra implementation aspects are treated. These aspects are specific for these type of elements.

Integration The T3P1⁺P1 element is integrated using 3 integration points and the T4P1⁺P1 is integrated using 4 integration points.

5.4.2 Updated Lagrangian

The total stress is decomposed into a deviator and a hydrostatic part. The deviatoric stresses are stored as integration point values while the hydrostatic pressure are stored as nodal values. The total stress in an integration point in the element can be written as:

$$\boldsymbol{\sigma}_{ip} = \mathbf{s} + \mathbf{I} \sum_i M_i p_i \quad (5.34)$$

In the corrector step only the deviatoric stress is updated and the total stress is calculated using the new deviatoric stresses and the pressures from the solution from the predictor step, as in equation (5.34).

The calculation of the Jacobian matrix is performed without the bubble node displacements. Omitting the bubble node displacements makes the Jacobian equal for all integration points.

The bubble degrees of freedom can be condensed on element level. This would reduce calculation time in the predictor step. However, the determination of the bubble displacements for calculating the stresses in the corrector step would take more time. Here is chosen not to condense the bubble degrees of freedom.

5.4.3 Implementation for ALE

In the ALE step all the integration point variables and nodal variables have to be convected to the new grid location. For the total pressures and linear displacements this is performed as described in the previous section. This also means that the total bubble displacements are not known after convection. This has an impact on the calculation of the residual on the pressures.

Calculation of residual on pressure

From equation (5.26) can be found that the residual on the pressure R_p consists of the total displacements and total pressures at the beginning of the step. The calculation of the correct total bubble displacements after convection is not a straight forward procedure.

$$R_p = R_p^0 + \Delta R_p + \Delta R_p^c \quad (5.35)$$

$$R_p = \frac{1}{C_b}(p_0 + \Delta p + \Delta p^c) - \mathbf{1} : \mathbf{B}(\mathbf{u}_0 + \Delta \mathbf{u} + \Delta \mathbf{u}^c) \quad (5.36)$$

For an Updated Lagrangian simulation $\Delta \mathbf{u}$ and Δp are known and of course $\Delta \mathbf{u}^c = 0$ and $\Delta p^c = 0$. For an ALE simulation incremental values of the convective part are added. The convective increment of the linear displacements and pressures is calculated. To get accurate results, the bubble displacements have to be calculated correctly. Calculation of the convective part of the bubble proves to be difficult and results are poor.

Several methods to calculate the residual on the pressure have been investigated in this research . The extrusion force for the simulation as in paragraph 5.6 is used to compare the results. In all options the calculation of the convective bubble increment is treated differently from the calculation of the convective linear displacement increment. Therefore equation (5.36) is rewritten as:

$$R_p = \frac{1}{C_b}(p_0 + \Delta p + \Delta p^c) - \mathbf{1} : \mathbf{B} \left(\mathbf{u}_0 + \Delta \mathbf{u} + \left\{ \begin{array}{c} \Delta \mathbf{u}_l^c \\ \Delta \mathbf{u}_b^c \end{array} \right\} \right) \quad (5.37)$$

To calculate the residual on the pressure different options are described below.

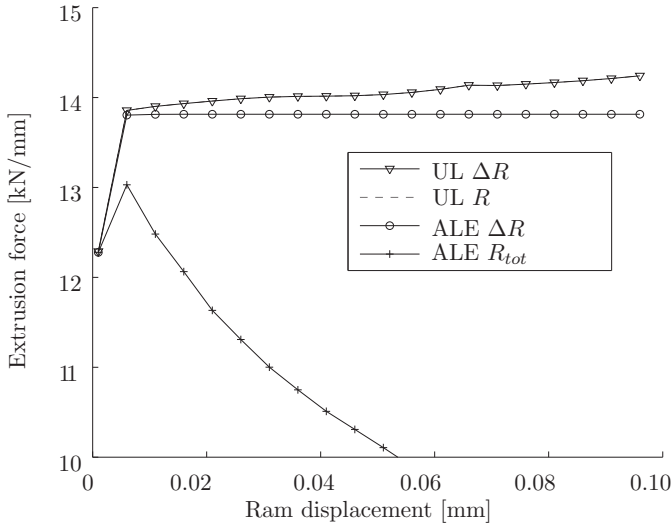


Figure 5.8: Extrusion force for residual on pressure.

The first option is to calculate a residual a zero convective contribution for the bubble displacements in the remap of the variables:

$$R_p = \frac{1}{C_b}(p_0 + \Delta p + \Delta p^c) - \mathbf{1} : \mathbf{B} \left(\mathbf{u}_0 + \Delta \mathbf{u} + \left\{ \begin{array}{c} \Delta \mathbf{u}_l^c \\ 0 \end{array} \right\} \right) \quad (5.38)$$

The total bubble displacement before and after remap are the same. Since the equilibrium is checked after convection in our semi-coupled approach, the equilibrium is still fulfilled. This option is depicted in figure 5.8 as R . For this option the updated simulation is unchanged. The ALE simulation shows a strong decay of the force over time. This decay is also found in the pressure in the container.

The second option is to convect the linear displacements and pressures. Then based on these values the bubble displacements that minimizes the "convected" residual is calculated.

$$R_p^{ALE} = \frac{1}{C_b}(p_0 + \Delta p + \Delta p^c) - \mathbf{1} : \mathbf{B} \left(\mathbf{u}_0 + \Delta \mathbf{u} + \left\{ \begin{array}{c} \Delta \mathbf{u}_l^c \\ \Delta \mathbf{u}_b^{min} \end{array} \right\} \right) \quad (5.39)$$

The optimal convective bubble displacement is calculated by least squares fitting. This option shows an improvement in force decay with respect to the

previous option, however the results are still not in good agreement with the expected values. The results for this option are not shown in the figure.

The third option is to ignore the residual on the pressures from the previous step by setting $R_p^0 = 0$. The residual on the pressures is now only calculated based on the incremental displacements and pressures. The total bubble displacement is set to zero at the beginning of each step.

$$R_p^{ALE} = \frac{1}{C_b}(\Delta p + \Delta p^c) - \mathbf{1} : \mathbf{B} \left(\Delta \mathbf{u} + \left\{ \begin{array}{c} \Delta \mathbf{u}_l^c \\ 0 \end{array} \right\} \right) \quad (5.40)$$

Within an iteration the updated incremental bubble displacement is also used as convected incremental bubble displacement. This option is depicted in the figure as ΔR . This option shows the proper force displacement curves for both Updated and ALE simulations.

From the previous results can be concluded that in both UL and ALE the residual on the pressure from the previous step can be omitted. This option is implemented in DiekA.

With this implementation the calculation of the total nodal bubble displacements $\mathbf{u}_b$ can be ignored completely. At the end of every increment $\mathbf{u}_b$ is set to zero.

$$R_u = F - \int_V \mathbf{B}^T \boldsymbol{\sigma}_0 dV \quad (5.41)$$

$$R_p = \frac{1}{C_b}(\Delta p + \Delta p^c) - \mathbf{1} : \mathbf{B}(\Delta \mathbf{u} + \Delta \mathbf{u}^c) \quad (5.42)$$

5.5 Updated Lagrangian application: Tube expansion

A long tube is loaded with an internal pressure p . The axial displacement is suppressed, making it a plane strain situation. The material model for the tube is chosen elastic rigid plastic.

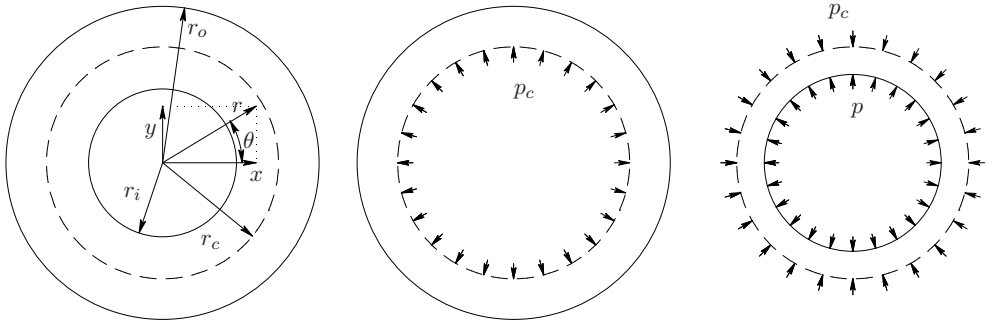


Figure 5.9: Tube with inner pressure.

When the tube is loaded with the critical pressure, at the inner wall the tube start to deform plastically. Upon increasing the load the plastic region start to grow. The location as in figure 5.9 of the transition from plastic to elastic can be determined analytically. Also the stress profile has an analytical solution. This is derived in appendix B.

The analytical results can be compared with the numerical results for both T3P1+P1 and T4P1+P1.

5.5.1 Analytical

The tube has inner radius r_i and outer radius r_o , and the transition radius is r_c . The inner pressure is p and the pressure at the transition location is p_c .

The material is chosen nearly incompressible with $\nu = 0.4995$ and the perfect plastic yield stress $\sigma_0 = 100MPa$). In appendix B the analytical stresses in the tube as function of the internal pressure p are derived. In this derivation the Tresca flow rule is used.

For the Elastic part of the tube the stresses are taken from equations (B.22),(B.23) and (B.24):

$$\sigma_r = -\sigma_0 \frac{r_c^2}{r_o^2 - r_c^2} \left(\frac{r_o^2}{r^2} - 1 \right) \quad (5.43)$$

$$\sigma_\theta = \sigma_0 \frac{r_c^2}{r_o^2 - r_i^2} \left(\frac{r_o^2}{r^2} + 1 \right) \quad (5.44)$$

$$\sigma_z = \nu(\sigma_r + \sigma_\theta) \quad (5.45)$$

The stresses in the plastic part are taken from equation (B.16),(B.17) and (B.18):

$$\sigma_r = \sigma_0 \ln\left(\frac{r}{r_i}\right) - p \quad (5.46)$$

$$\sigma_\theta = \sigma_0 \ln\left(\frac{r}{r_i}\right) - p + \sigma_0 \quad (5.47)$$

$$\sigma_z = \nu A = \nu 2\sigma_r + \sigma_0 \quad (5.48)$$

The interface between the elastic and plastic part can be found at the location where elastic and plastic stresses are equal. With this condition the location of r_c can be found.

In the simulations, treated in the next paragraph, the Von Mises flow criterion is used. The stresses at which the material will yield are lower for the Tresca flow criterion. This is compensated for in the analytical calculation to be able to compare the results. In the simulations the Von Mises flow stress is set to 100MPa . In the analytical calculation σ_0 should be chosen as:

$$\sigma_0 = \frac{2}{\sqrt{3}}\sigma_{vm} \approx 1.15 \cdot 100[\text{MPa}] \quad (5.49)$$

This is derived in Appendix B.

5.5.2 Numerical

The numerical models are shown in figure 5.10. For this simulation a quarter of the tube is modeled with symmetry conditions on x and y axis. The 2D simulation is in plane strain condition and for the 3D simulation the displacements in axial direction are suppressed at the two edges. in between the edges the z-displacements are free.

The tube is incrementally loaded until the elastic-plastic interface is approximately in the middle of the tube wall. The stresses plotted in 5.10, are the stresses at the end of the 10th increment.

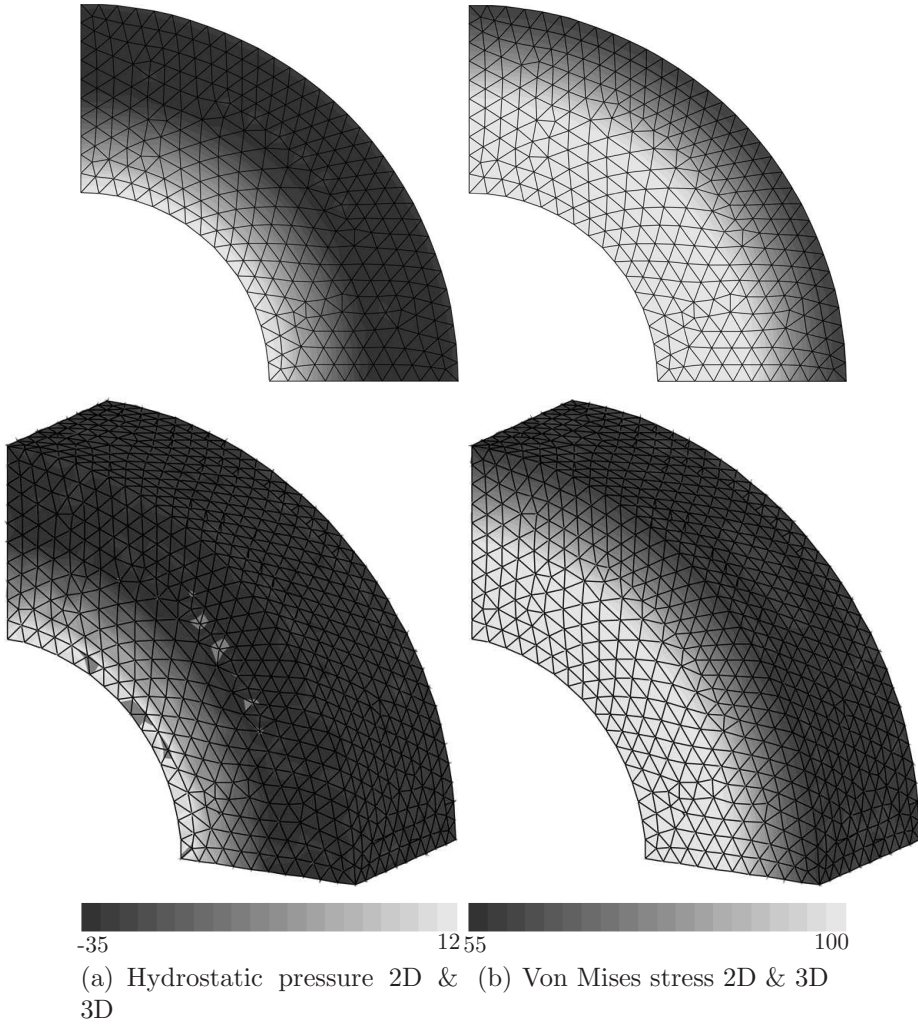


Figure 5.10: Pressure and Von Mises result plots of both T3P1+P1 (top) & T4P1+P1 (bottom).

The stresses in all nodes are rotated to the r and θ direction and plotted in figure 5.11. The numerical results for the Von Mises stress are in good agreement with the analytical prediction.

The pressure calculated with the T4P1+P1 elements shows some scatter, especially at the loaded interface. This effect is also noticed in [22].

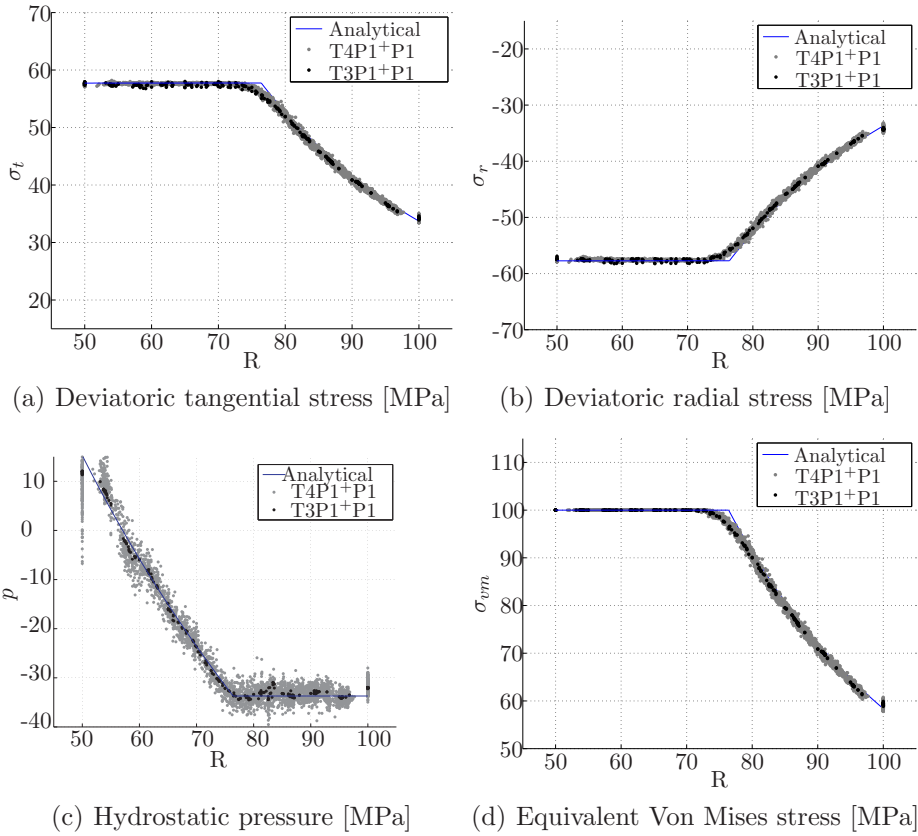


Figure 5.11: Numerical and analytical stresses.

5.6 Eulerian example: 2D Filling

In chapter 4 the original coordinate tracking algorithm was treated for Q4 and H8 elements. In this section the original coordinate tracking algorithm is used for T3P1+P1 elements.

In this trial the boundaries are permeable as in the previous simulations. The geometry of paragraph 4.5 is meshed with a coarse and a finer mesh of T3P1+P1 elements. The material properties are chosen as in table 2.1.

The material properties of the "air" domain are updated according to the scheme described in paragraph 4.4.3. From the five properties only the yield

stress is reduced by a factor 100 ($\rho_{yld} = 0.01$).

5.6.1 Flow front and volume conservation of 2D filling

In figure 5.12 the results of the two trials are shown. The mesh in the top row consist of 596 elements and the mesh in the bottom row of 2212 elements.

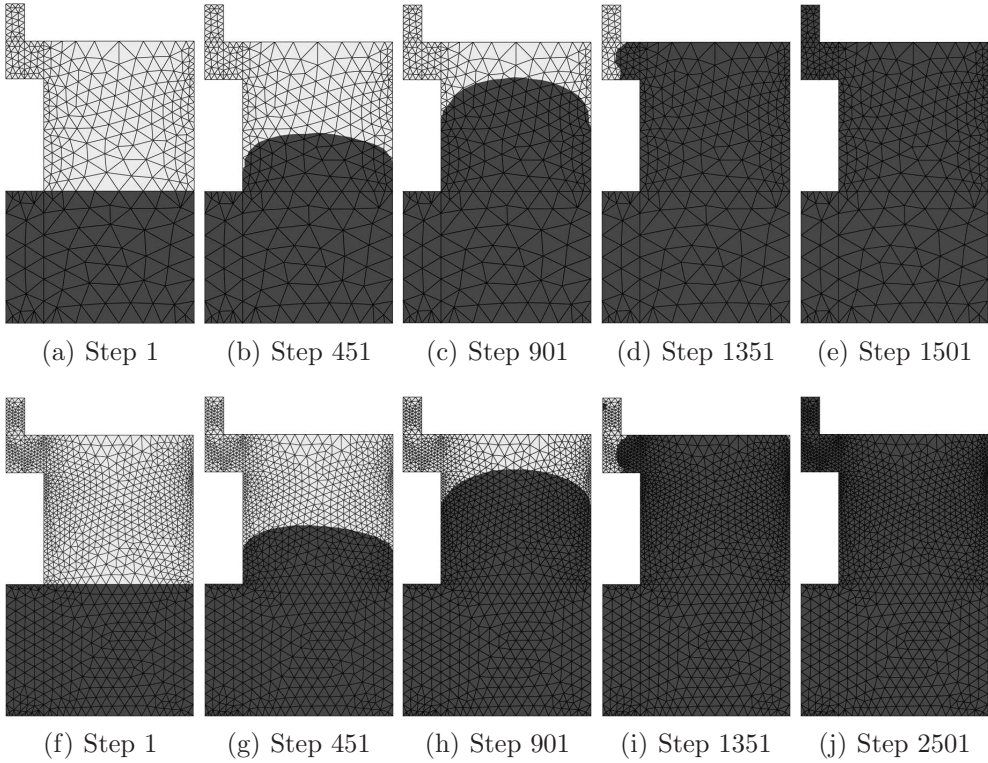


Figure 5.12: 2D filling of a die for two meshes densities. The coarse mesh (mesh 1) on the top row and the fine mesh (mesh 2) is found on the bottom row.

The flow front shape is in good agreement with the expected flow front shape from figure 4.11. Because of the use of permeable boundary conditions the complete domain fills with aluminium.

The spurious contact phenomenon observed in chapter 4 is not observed with these simulations. This is an improvement with the simulations from chapter 4.

The volume conservation error is calculated as in equation (4.23) and plotted in graph 5.13.

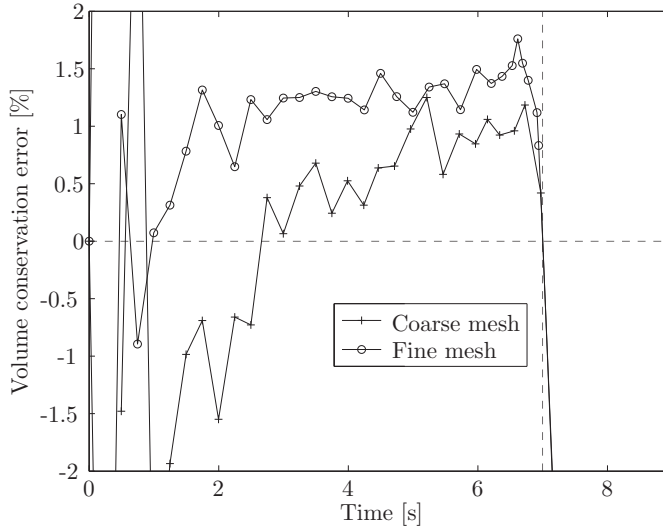


Figure 5.13: Volume conservation of 2D filling of a die.

For both simulations can be observed that the volume conservation is very good. The errors are in the same order as for the Q4 elements. The conservation for the simulation for both mesh densities appears to be comparable. From these figures can be concluded that the linearisation of the volume strain, in equation (5.17), does not introduce high volume loss or gain.

5.6.2 Extrusion force for 2D filling

In figure 5.14 the extrusion force, the force on the inflow boundary, is plotted. In both simulations the extrusion force after filling converges immediately to the steady state force. The fine mesh shows a slightly lower value of the extrusion force. A coarse mesh at the sticking boundaries, for example in the bearing area, creates a numerical restriction of the velocity profile and restricts the flow. This effect also appears in the feeder holes. The effect causes the extrusion force to be slightly higher.

In the next chapter the results of two 3D simulations with T4P1+P1 elements are treated.

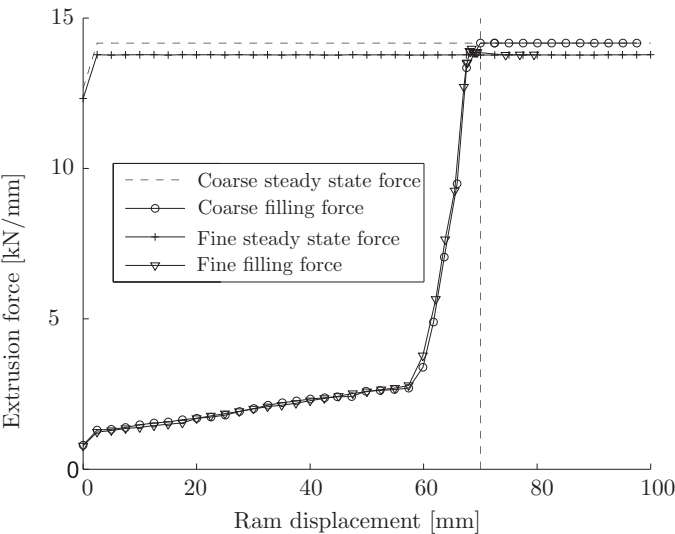


Figure 5.14: Extrusion force of 2D filling of a die.

Chapter 6

Two applications

In this chapter two applications with the MINI elements combined with the original coordinate tracking algorithm are considered. The elements and tracking algorithm are implemented in the implicit FEM code DiekA. From both examples the experimental results are known and treated in previous chapters.

From the first simulation it is shown that the $T4P1^+P1$ elements give good results for a large number of increments. The second simulation is an improvement of the 3D filling simulation from chapter 4. In this chapter the simulation is performed using $T4P1^+P1$ elements. Since automatic meshing with $T4P1^+P1$ elements is available, it is easier to add detail to the simulation. The other reason for using $T4P1^+P1$ elements is the spurious effects found with filling with H8 elements.

The last part of the filling simulation is the forming of the nose piece. To be able to model this nose piece, the use of special ALE options is treated.

6.1 Container flow

In chapter 3 front lines in container flow (figure 6.1) during rod extrusion were created based on steady state numerical results and compared with experiments.

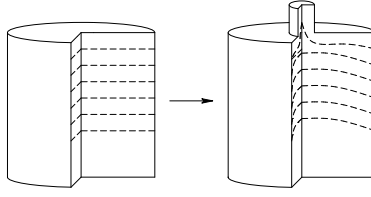


Figure 6.1: Front lines in container during extrusion.

In this section two transient 3D simulations with $T4P1+P1$ elements are performed to create the similar front line results. The dimensions in both simulations are equal to the dimensions from chapter 3 summarized in table 6.1. One simulation is performed with a coarse mesh and one simulation with a finer mesh. See figure 6.2.

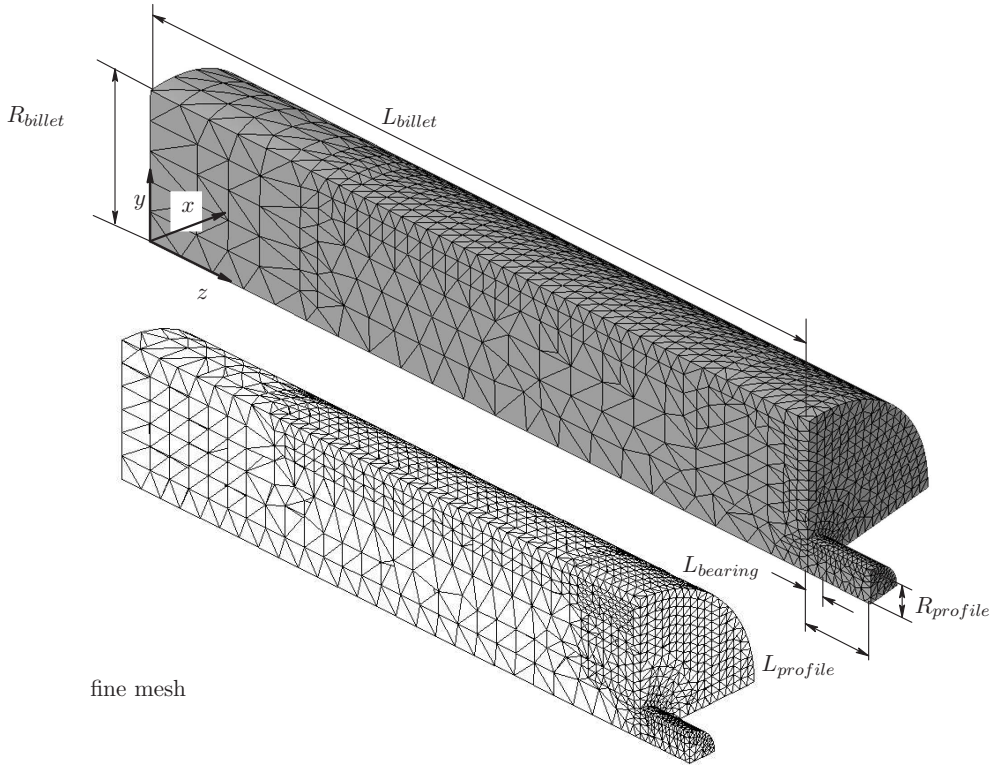


Figure 6.2: geometry and mesh for transient front line simulation.

The total length of the billet is 210 mm. Over the complete length of the billet a stick condition is assumed. Also at the die face and in the bearing a stick condition is applied.

R_{billet}	$R_{profile}$	L_{billet}	$L_{profile}$	$L_{bearing}$
46 mm	10 mm	210 mm	20 mm	5 mm

Table 6.1: Dimensions for simulation of front lines in container flow during rod extrusion.

The material properties from table 3.2 are used, with an exception to $m = 10$, as found to be the best solution from previous simulations. In table 6.2 more information about both models can be found.

results

Both simulations have been run up to $t = 90s$. The largest possible step size for the coarse mesh is approximately 2.5 time bigger than for the fine mesh. Since the experimental results are known at $t = 24s$, $t = 50s$ and $t = 90s$ these results are shown in figure 6.3

Mesh	Elements	DOFS	Increment at $t = 90s$	CPU time at $t = 90$
Coarse	9910	36262	3600	0.36e5
Fine	24126	92878	8950	2.94e6

Table 6.2: Simulation information of both meshes.

From this figure it can be seen that the results of the simulation with the fine mesh is more like a plug flow than the coarse mesh. This can be explained by the large (linear) elements at the container wall. These elements are not able to describe the shear zone accurately and the result is that the flow is restricted here numerically.

This behavior is improved by reducing the elements in the fine mesh, however when the results are compared with the experiments (figure 6.4) the front lines still show a deviation larger than the simulations from chapter 3.

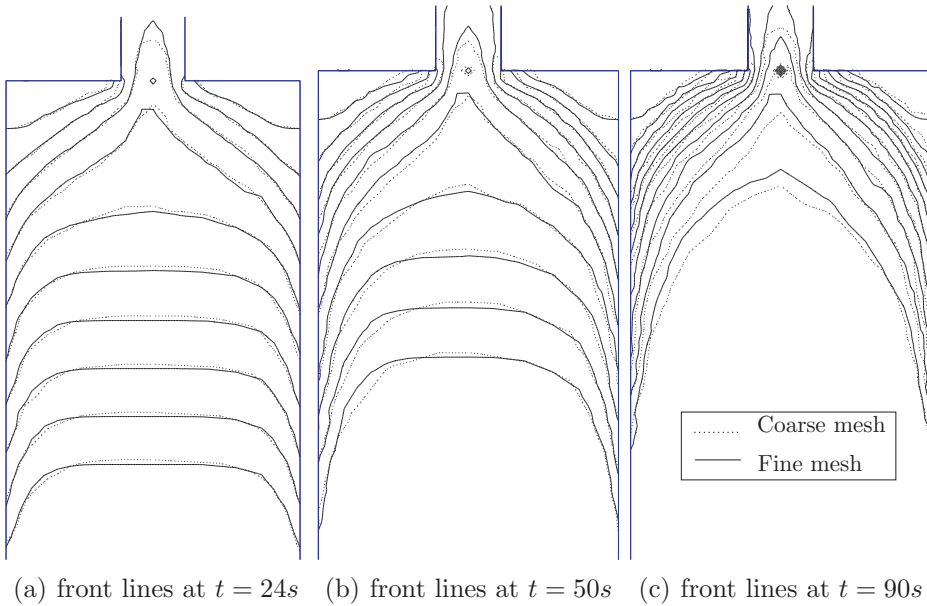


Figure 6.3: Front lines calculated with both coarse mesh and fine mesh.

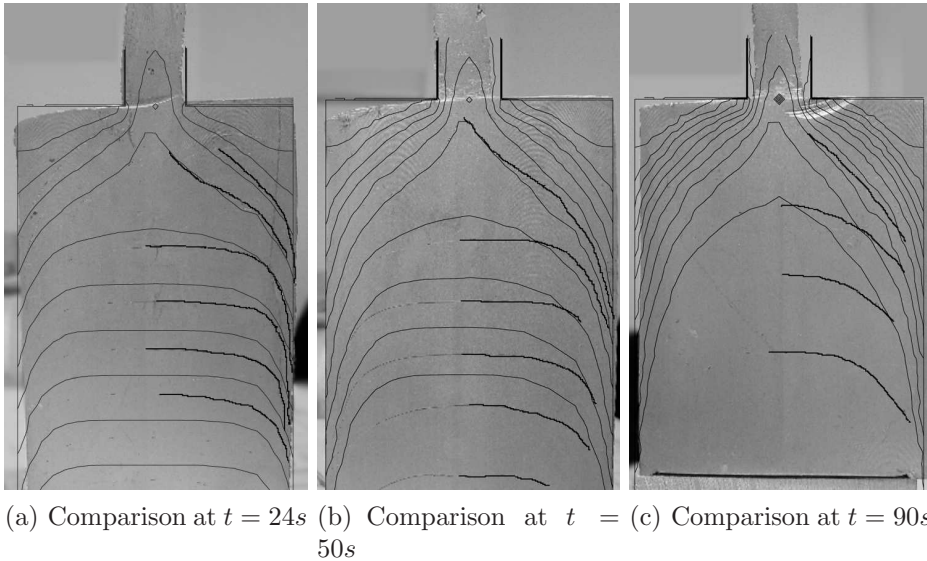


Figure 6.4: Results of both coarse and fine mesh.

The fine mesh in the 3D simulation is still a lot coarser at the edges than the mesh used in chapter 3. The simulations show that the T4P1+P1 elements give comparable results with the experiments and that with mesh refinement the prediction improves.

The comparison for $t = 90s$ shows a significant difference between experiment and simulation. This deviation is explained by the fact that in the simulation the inflow surface (the ram) does not move. In the proximity of the ram, the aluminium slips along the container wall and does not stick as modeled. This leads to a much straighter front than what is simulated.

6.2 3D die filling: 80 mm Tube

Dies and experimental results for tube extrusion are shown in chapter 4 section [4.6](#). The simulations shown in that section are performed with H8 elements.

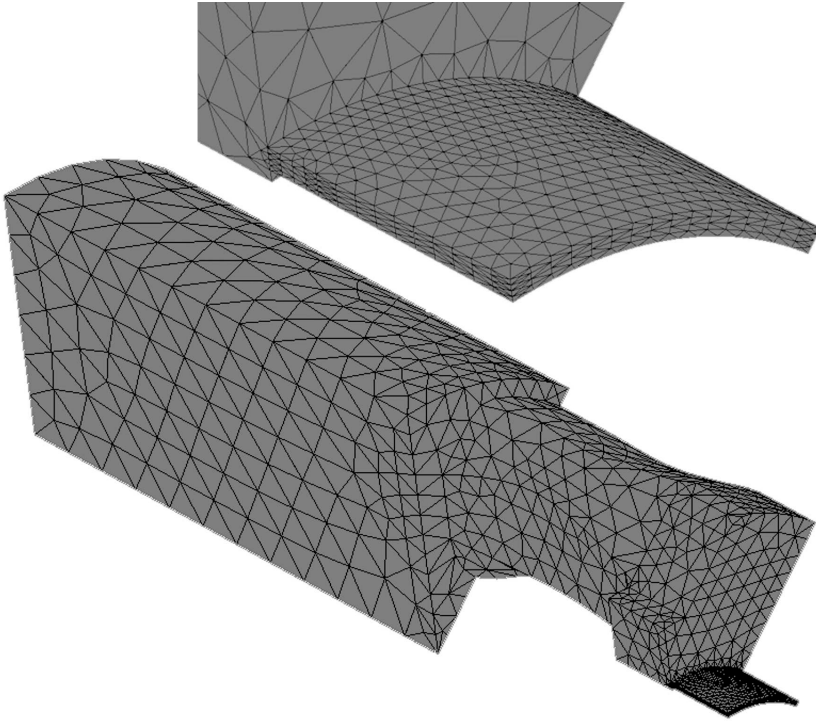


Figure 6.5: geometry and mesh for tube extrusion simulation.

In this section a similar model is used, now meshed with $T4P1^+P1$ elements. Using these elements enables us to model more detail as fillets and curved surfaces. In figure 6.5 the used mesh is shown. This is a mesh that corresponds with the dimensions from Experiment A in table 4.3. In this simulation again symmetry is applied and only one eighth of the model is meshed. The ram speed is set to 10 mm/s.

In the profile a minimum of 4 elements over the thickness are used, as done in the H8 model from chapter 4. During filling all the nodes are Eulerian and the boundary conditions are permeable. When a surface node is in the aluminium domain, a stick is applied to that node, in the air domain the material is free to flow out. When the flow front reaches the bearing area, the nose piece is formed. The prediction of the shape of the nose piece requires special attention

6.2.1 Nose piece deformation prediction

The objective is to predict the shape of the nose piece. Therefore the nodes on the boundary of the profile should be able to move with the aluminium. The necessary ALE options are treated here, since they are not as straight forward as for a structured mesh.

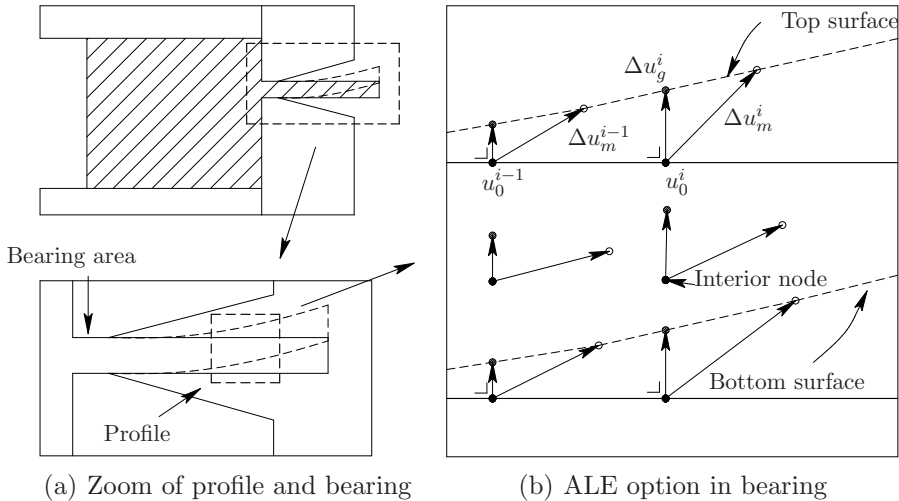


Figure 6.6: Special ALE surface options on the profile to model deflections.

The displacement of the node on the surface of the profile, perpendicular to

the surface, should be equal to the material displacement. In the extrusion direction the displacement of the node should be equal to zero. This mesh movement is a typical Arbitrary Lagrangian-Eulerian description. The nodes in the interior of the profile should move with the surface nodes, so that they will remain in between them. In figure 6.6 these options are explained.

For the calculation of the displacement of a node, perpendicular to the surface, the data from an element upstream is used. For every surface node in the profile the adjacent elements are searched. From each adjacent element the surface triangle is determined. For some adjacent elements no surface triangles are present since they are in the interior of the profile. From all the surface triangles adjacent to the surface node, the center of gravity is determined.

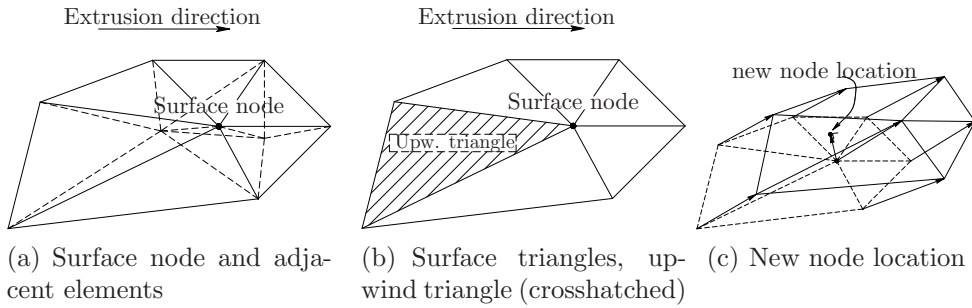


Figure 6.7: Search for upwind surface triangle.

The surface triangle that is upstream from the surface node is used to calculate the node displacement perpendicular to the surface. This process is shown in figure 6.7.

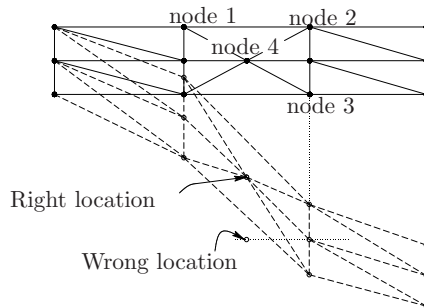


Figure 6.8: Determination of new interior node location.

A node in the interior is interpolated between two surface nodes (master nodes). Here some caution is necessary that the right master nodes are chosen. In figure 6.8 the procedure is shown.

For the interpolation of the interior nodes that are at a location where nodes on both surfaces are found, for example the interior node between node 2 and 3, the interpolation is done between the two surface nodes. When an interior node, as node 4 in the figure, does not have corresponding nodes, interpolation should be between two node on the same surface. For example node 1 and 2. If would have been chosen to determine the location between node 2 and 3, node four would have ended up outside the profile. In extrusion not a great deal of thickening or thinning is expected. Therefore interpolation based on one surface is allowed.

6.2.2 Results of the simulation

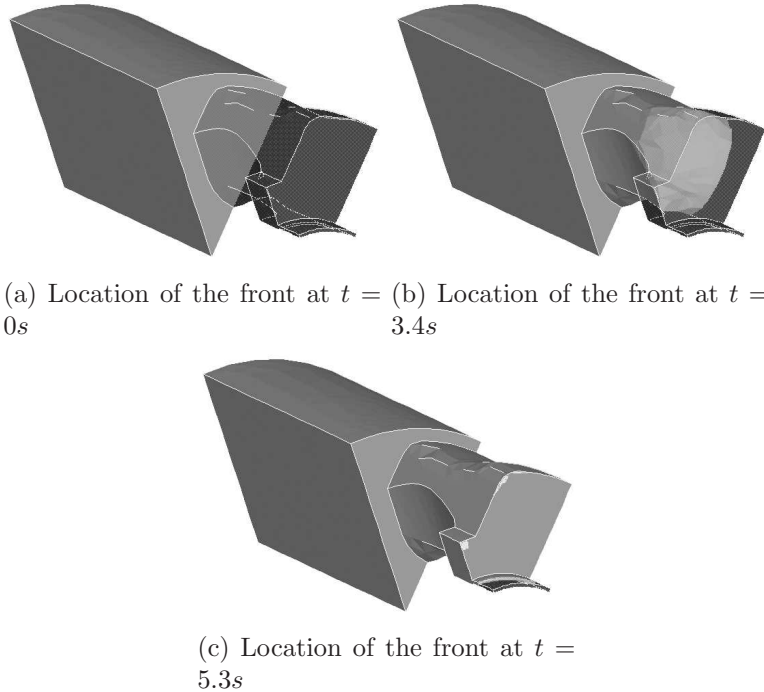


Figure 6.9: Location of the front during filling.

In figure 6.9 the evolution of the front during filling is shown. After 5.3 seconds the die is (almost) completely filled and extrusion starts.

During filing no spurious boundary effects are noticed and the flow front is free from oscillations as shown in figure 4.8. Therefore no regularization of the front was necessary.

Extrusion force and volume conservation

The extrusion force during the filling is plotted in figure 6.10. The force increases until it reaches the steady state force when extrusion begins.

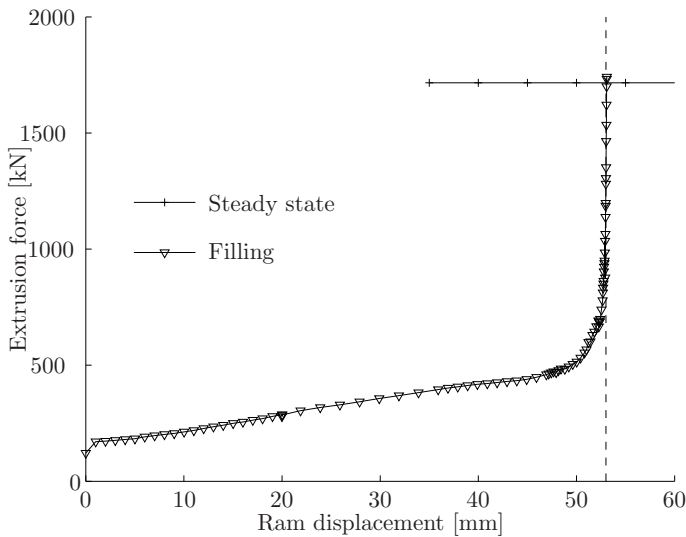


Figure 6.10: Extrusion force of 3D filling compared with steady state.

The overall trend is similar to the force displacement curves of the simulations in chapter 4. The force is approximately 10% higher due to a doubled ram speed. This can be explained by the Sellers-Tegart law. In figure 6.11 the extrusion force of the T4P1+P1 simulation of geometry A is compared to the H8 simulations from chapter 4.

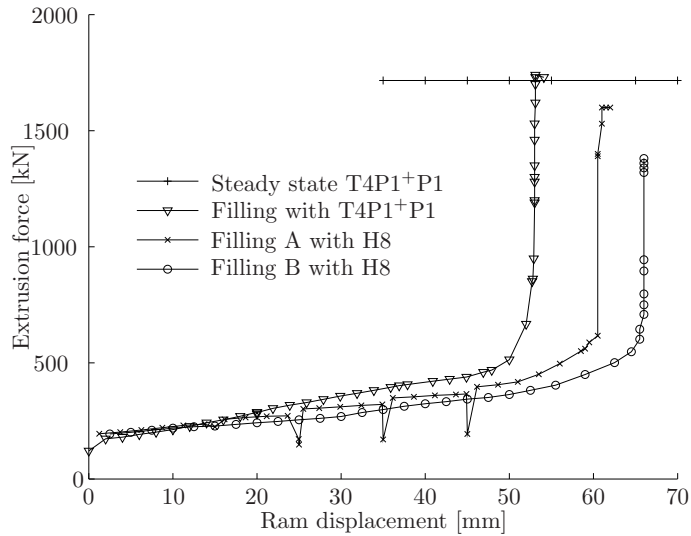


Figure 6.11: Extrusion force of 3D filling compared with the H8 simulations.

From the figure becomes clear that the volume in the portholes in the geometry for the T4P1+P1 simulation is slightly smaller then the geometry from chapter 4. Therefore the extrusion starts sooner.

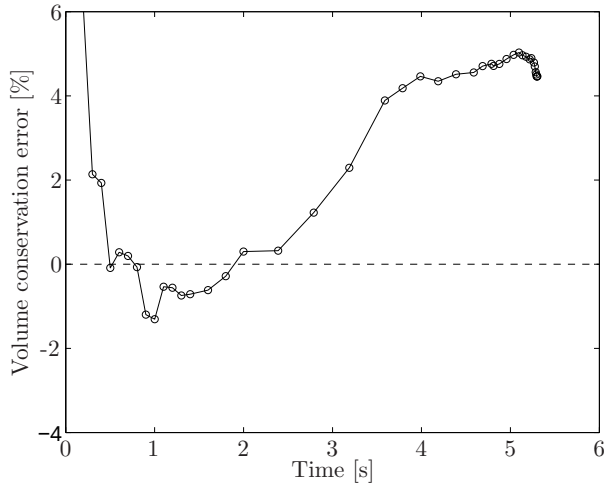


Figure 6.12: Volume conservation of 3D filling of a die.

The volume conservation is calculated as in equation (4.23). In figure 6.12 the error in volume conservation is plotted. Here the same trend as for T3P1+P1 elements can be observed. For T3P1+P1 and T4P1+P1 elements the tracking algorithm slightly over predicts the volume. In other words the location of the flow front is probably slightly more advanced than it should be.

However a volume conservation error of less than 5% in this coarse mesh can still be considered as very good.

Visual comparison

In figure 6.13 the simulation results of the nose piece are shown. It can be seen that with a full stick bearing the deformation of the front is not in agreement with the experimental results.

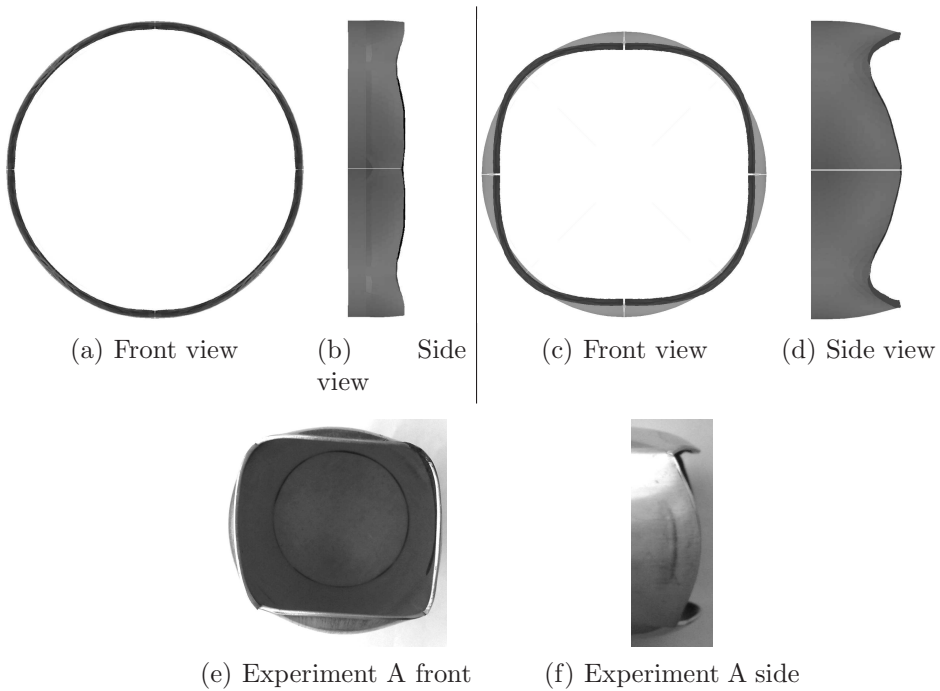


Figure 6.13: Nose piece after filling of a porthole with a full stick bearing (a & b) and a zero bearing (c & d).

From this figure it is clear that the nose piece deformation is under predicted severely. The prescribed stick condition in the bearing with only 4 node in

the profile thickness direction restricts the flow. The full stick condition is one extreme of the bearing conditions. Applying a full slip condition would show the results of the other extreme condition.

In order to see the maximum of deflection possible with this mesh, the bearing is chosen full slip except for the nodes on the bearing corner, those are prescribed fixed. This type of bearing we call a 'zero' bearing. In figure [6.13\(c\)](#) and [6.13\(f\)](#) the results for a zero bearing are shown. The results from this simulation are in very good agreement with the nose piece of experiment A from chapter 4.

6.2.3 Conclusions

The main improvement in the simulations with $T4P1^+P1$ elements are found in the stable flow front propagation. The spurious effects found with H8 elements do not appear. Therefore the regularization steps can be omitted. Also the shape prediction of the front shows an improvement compared to the simulations in chapter 4. This can be related to the addition of more detail to the geometry.

Chapter 7

Conclusions & Recommendations

In this research numerical simulations of aluminium extrusion has been studied. Especially simulation of the aluminium flow during the filling of the die and creation of the nose piece has been treated.

In general can be said that most effects in extrusion can be qualitatively described by the simulations. Hence it can be said that these models provide insight in the extrusion process. However quantitatively the results deviate from the experimental results.

The general conclusions for steady state simulations are:

In chapter 3 was shown that the velocity field from steady state simulations can be accurately post-processed to the transient front lines. This method has been used as an inverse method to determine material properties under extrusion conditions.

The general conclusions for interface tracking simulations are:

The initial coordinate function is an appropriate function to track interfaces in two phase flows. In chapter 4 an ALE/Eulerian formulation with initial coordinate interface tracking has been successfully used for the simulation of the transient start-up of aluminium extrusion process. The initial coordinate tracking method shows good volume conservation. The trend in the prediction of the nose piece shape is in agreement with the experiments. The quantitative distortions are under predicted.

An element from the MINI-family was successfully developed for a semi-coupled ALE formulation. The initial interface tracking algorithm works well with the MINI element.

The spurious effects in the flow front prediction found with H8 elements is not apparent in simulations with T4P1⁺P1 elements. Therefore also the regularisation can be omitted in filling simulations with T4P1⁺P1 elements.

Meshing with triangular or tetrahedral elements from the MINI-family increases the possibility to include details as radii. From the tube extrusion simulation in chapter 6 can be concluded that increasing detail increases the quantitative agreement of the simulations with experiments.

Some general recommendations are:

In simulations in this thesis the dies are assumed rigid. It is well known that die deflection can play an important role in aluminium flow through the die, especially in the bearing area. Further research on die deflection can give useful insight for a better equivalent bearing model and better understanding of stresses in the die during start-up and steady state.

The flow stress and other material properties of aluminium at extrusion temperatures are sensitive for small differences. Thermo-mechanically coupled simulations will improve the quantitative agreement between simulations and experiments.

Material properties under extrusion temperatures are essential for accurate simulations. The development of a validated material properties database and scatter analysis of different alloys would be a helpful tool for further developments in extrusion die design based on simulations.

The used regularisation of the original coordinate function is a diffusive step. Other and better regularisation techniques can be found in the level set approaches. These better regularization techniques would improve the accuracy and speed of the simulations.

The current state of developments in aluminium extrusion requires more experimental validation of numerical results to increase quantitatively accuracy. In this thesis two methods have been presented to compare experiments with numerical results. These methods help to improve material models and to gain insight in aluminium flow during start-up.

Appendix A

Weak formulation

A.1 Isotropic Elasto plastic material

$$\boldsymbol{\sigma} = \mathbf{E} : \boldsymbol{\varepsilon} - 2G\boldsymbol{\varepsilon}^{pl} \quad (\text{A.1})$$

To obtain objective integration the derivation of Lof [44] is closely followed. Lof chooses to use a corotational formulation. The initial deviatoric stress $\boldsymbol{\sigma}$ is replaced by $\bar{\boldsymbol{\sigma}}$:

$$\bar{\boldsymbol{\sigma}} = \mathbf{R}\boldsymbol{\sigma}\mathbf{R}^T \quad (\text{A.2})$$

Where $\mathbf{R}$ follows from the polar decomposition of the incremental deformation gradient $\mathbf{F}$. From the differentiation of this equation with respect to time follows the Jaumann stress rate

$$\overset{\nabla}{\boldsymbol{\sigma}} = 2G(\mathbf{I} - \frac{1}{3}\mathbf{1}\mathbf{1}) : \mathbf{D} - \frac{3G}{(1+h)\sigma_y^2} \text{ss} : \mathbf{D} + C_b \mathbf{1}\mathbf{1} : \mathbf{D} - \boldsymbol{\sigma} \text{tr} \mathbf{D} \quad (\text{A.3})$$

$$\overset{\nabla}{\boldsymbol{\sigma}} = \mathbf{Y} : \mathbf{D} + C_b \mathbf{1}\mathbf{1} : \mathbf{D} - \boldsymbol{\sigma} \text{tr} \mathbf{D} \quad (\text{A.4})$$

A.2 Strong formulation

$$\begin{aligned} \boldsymbol{\sigma}^{n+1} \cdot \nabla &= 0 && \text{in } \Omega_g^{n+1} \\ \mathbf{u} &= \mathbf{u}_0^{n+1} && \text{on } \Gamma_u \\ \boldsymbol{\sigma}^{n+1} \cdot \mathbf{n} &= \mathbf{t}^{n+1} && \text{on } \Gamma_t \end{aligned} \quad (\text{A.5})$$

A.3 Weak formulation

In the finite element formulation the equilibrium equations are only weakly enforced. This weak form is equal to the virtual power equation if the weighing functions $\mathbf{w}$ is equal to the virtual velocity $\delta \mathbf{v}$. When bodyforces are neglected, the so-called weak form can be written as:

$$\delta W_{in} = \delta W_{ext} \quad (\text{A.6})$$

$$\int_{\Omega^{n+1}} (\mathbf{w} \nabla) : \boldsymbol{\sigma}^{n+1} d\Omega = \int_{\Gamma^{n+1}} \mathbf{w} \cdot \mathbf{t}^{n+1} d\Gamma_t \quad (\text{A.7})$$

Since the material model is given as the material rate of the stress-strain relation, the rate or the weak form is used. The increase in virtual work from Ω_n at t^n to Ω_{n+1} at t^{n+1} can be approximated by:

$$\delta W_{in}^{n+1} \approx \delta W_{in}^n + \delta \dot{W}_{in} \Delta t \quad (\text{A.8})$$

$$\delta W_{ext}^{n+1} \approx \delta W_{ext}^n + \delta \dot{W}_{ext} \Delta t \quad (\text{A.9})$$

from which the updated lagrangian formulation can be derived:

$$\delta \dot{W}_{in} = \delta \dot{W}_{ext} \quad (\text{A.10})$$

Using [42] and [72] the derivation for the rate of weak form is given. The first step is to translate the weak form A.7 into referential coordinates.

$$\int_{\Omega_{\chi}} \frac{\partial \mathbf{w}}{\partial \boldsymbol{\chi}} \frac{\partial \boldsymbol{\chi}}{\partial \mathbf{x}} : \boldsymbol{\sigma} J^{\Gamma} d\Omega_{\chi} = \int_{\Gamma_{\chi}} \mathbf{w} \cdot \mathbf{t} J^{\Gamma} d\Gamma_{\chi} \quad (\text{A.11})$$

with the surface Jacobian J^{Γ} and:

$$J^{\chi} = \det \left(\frac{\partial \mathbf{x}}{\partial \boldsymbol{\chi}} \right) \quad (\text{A.12})$$

Using the following definitions for the deformation gradient $\mathbf{F}$ and the velocity gradient $\mathbf{L}$:

$$\mathbf{F} = \frac{\partial \mathbf{x}}{\partial \boldsymbol{\chi}} \quad (\text{A.13})$$

$$\mathbf{L} = \dot{\mathbf{v}} \nabla \quad (\text{A.14})$$

The velocity gradient $\mathbf{L}$ can be decomposed into a symmetric part and an skew-symmetric part:

$$\mathbf{L} = \mathbf{D} + \mathbf{W} \quad (\text{A.15})$$

With the symmetric part $\mathbf{D}$ and the skew-symmetric part $\mathbf{W}$:

$$\mathbf{D} = \frac{1}{2}(\mathbf{v}\overleftarrow{\nabla} + \overrightarrow{\nabla}\mathbf{v}) \quad (\text{A.16})$$

$$\mathbf{W} = \frac{1}{2}(\mathbf{v}\overleftarrow{\nabla} - \overrightarrow{\nabla}\mathbf{v}) \quad (\text{A.17})$$

$$\delta\dot{W}_{in} = \int_{\Omega_{\chi}} \frac{\partial \mathbf{w}}{\partial \chi} : \left(\dot{\mathbf{F}}^{-1} \cdot \boldsymbol{\sigma} J^{\chi} + \mathbf{F}^{-1} \cdot \dot{\boldsymbol{\sigma}} J^{\chi} + \mathbf{F}^{-1} \cdot \boldsymbol{\sigma} j^{\chi} \right) d\Omega_{\chi} \quad (\text{A.18})$$

The rates of the weighing functions vanish [29]. The three time derivatives are:

$$\dot{\mathbf{F}}^{-1} = -\mathbf{F}^{-1}\mathbf{L} \quad (\text{A.19})$$

$$j^{\chi} = J^{\chi} \text{tr}(\mathbf{L}) \quad (\text{A.20})$$

$$\dot{\boldsymbol{\sigma}} = \overrightarrow{\boldsymbol{\sigma}} - \boldsymbol{\sigma} \cdot \mathbf{W} + \mathbf{W} \cdot \boldsymbol{\sigma} \quad (\text{A.21})$$

Substitution and transforming back to the spatial coordinates gives:

$$\delta\dot{W}_{in} = \int_{\Omega} \mathbf{w}\overleftarrow{\nabla} : (-\mathbf{L} \cdot \boldsymbol{\sigma} + \dot{\boldsymbol{\sigma}} + \boldsymbol{\sigma} \text{tr} \mathbf{L}) d\Omega \quad (\text{A.22})$$

The virtual external work written in referential coordinates:

$$\delta W_{ext} = \int_{\Gamma_t} \mathbf{w} \cdot \mathbf{t} d\Gamma_t = \int_{\Gamma_{\chi}} \mathbf{w} \cdot \mathbf{t} J^S d\Gamma_{\chi} = \int_{\Gamma_{\chi}} \mathbf{w} \cdot \boldsymbol{\sigma} \cdot \mathbf{n} J^{\Gamma} d\Gamma_{\chi} \quad (\text{A.23})$$

The time derivative of the virtual work is:

$$\delta\dot{W}_{ext} = \int_{\Gamma_{\chi}} \mathbf{w} \cdot (\dot{\boldsymbol{\sigma}} J^{\Gamma} + \boldsymbol{\sigma} j^{\Gamma}) \cdot \mathbf{n} d\Gamma_{\chi} \quad (\text{A.24})$$

Transforming back the the spatial coordinates using $\dot{\boldsymbol{\sigma}} \cdot \mathbf{n} = \dot{\mathbf{t}}$ gives:

$$\delta W_{ext} = \int_{\Gamma_t} \mathbf{w} \cdot \dot{\mathbf{t}} d\Gamma_t + \int_{\Gamma_t} \mathbf{w} \cdot \mathbf{t} j^{\Gamma} d\Gamma_t \quad (\text{A.25})$$

The complete weak form can now be written as:

$$\int_{\Omega} \mathbf{w}\overleftarrow{\nabla} : (-\mathbf{L} \cdot \boldsymbol{\sigma} + \dot{\boldsymbol{\sigma}} + \boldsymbol{\sigma} \text{tr} \mathbf{L}) d\Omega = \int_{\Gamma_t} \mathbf{w} \cdot \dot{\mathbf{t}} d\Gamma_t + \int_{\Gamma_t} \mathbf{w} \cdot \mathbf{t} j^{\Gamma} d\Gamma_t \quad (\text{A.26})$$

Appendix B

Stresses in a long cylinder under internal pressure

In this appendix the stresses for an elastic-perfect plastic tube under inner pressure are derived. The derivation closely follows the solution from Pettersson [52].

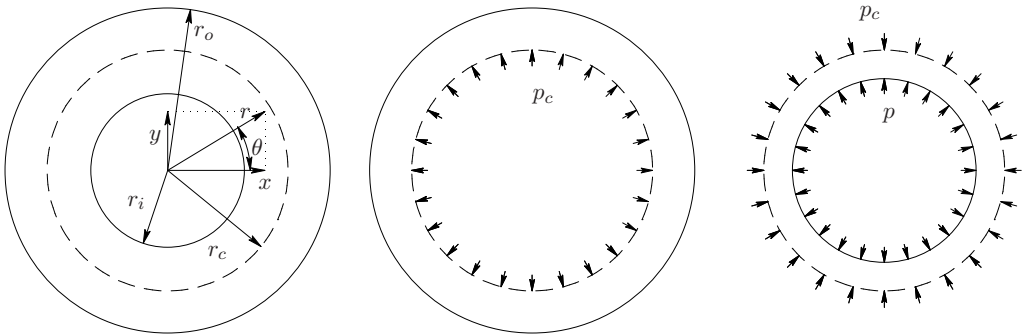


Figure B.1: Vortex flow for triangles

The stresses are derived with a plane strain condition and with the use of the tresca flow criterion.

The stress-strain relations are

$$\epsilon_r = \frac{1}{E}(\sigma_r - \nu(\sigma_z + \sigma_\theta)) + \epsilon_r^p \quad (\text{B.1})$$

$$\epsilon_\theta = \frac{1}{E}(\sigma_\theta - \nu(\sigma_z + \sigma_r)) + \epsilon_\theta^p \quad (\text{B.2})$$

$$\epsilon_z = \frac{1}{E}(\sigma_z - \nu(\sigma_r + \sigma_\theta)) + \epsilon_z^p \quad (\text{B.3})$$

From equilibrium in radial direction can be derived:

$$\frac{d\sigma_r}{dr} = \frac{\sigma_\theta - \sigma_r}{r} \quad (\text{B.4})$$

And the compatibility equation

$$\frac{d\epsilon_\theta}{dr} = \frac{\epsilon_r - \epsilon_\theta}{r} \quad (\text{B.5})$$

In a elastic region $\epsilon_z = 0$ and $\epsilon_z^p = 0$. (B.3) can be rewrtitten into:

$$\sigma_z = \nu(\sigma_r + \sigma_\theta) \quad (\text{B.6})$$

Combining equations (B.2),(B.3),(B.4) and (B.6) leads to:

$$\frac{d\sigma_\theta}{dr} = -\frac{d\sigma_r}{dr} \quad \text{or} \quad (\text{B.7})$$

$$\sigma_\theta = -\sigma_r + A \quad (\text{B.8})$$

This combined with (B.4) yields

$$\frac{d\sigma_r}{dr} + \frac{2\sigma_r}{r} = \frac{A}{r} \quad (\text{B.9})$$

Equation (B.9) together with the boundary conditions $\sigma_r(r_i) = -p, \sigma_r(r_o) = 0$ give for the elastic case:

$$\sigma_r = -p \frac{r_i^2}{r_o^2 - r_i^2} \left(\frac{r_o^2}{r^2} - 1 \right) \quad (\text{B.10})$$

$$\sigma_\theta = p \frac{r_i^2}{r_o^2 - r_i^2} \left(\frac{r_o^2}{r^2} + 1 \right) \quad (\text{B.11})$$

$$\sigma_z = 2p\nu \frac{r_i^2}{r_o^2 - r_i^2} \quad (\text{B.12})$$

For the plastic part Tresca's flow hypothesis $\sigma_\theta - \sigma_r = \sigma_0$ is used. This can be substituted in (B.4) and integrated.

$$\frac{d\sigma_r}{dr} = \frac{\sigma_0}{r} \quad (\text{B.13})$$

$$\sigma_r = \sigma_0 \ln(r) + B \quad (\text{B.14})$$

With the boundary condition $\sigma_r = -p$ at $r = r_i$

$$\sigma_r = \sigma_0 \ln\left(\frac{r}{r_i}\right) - p \quad (\text{B.15})$$

This can be substituted in Tresca's flow rule to get σ_θ . The complete set of stresses for the plastic part is written as:

$$\sigma_r = \sigma_0 \ln\left(\frac{r}{r_i}\right) - p \quad (\text{B.16})$$

$$\sigma_\theta = \sigma_0 \ln\left(\frac{r}{r_i}\right) - p + \sigma_0 \quad (\text{B.17})$$

$$\sigma_z = \nu(\sigma_r + \sigma_\theta) \quad (\text{B.18})$$

With r_c the radius of the interface between plastic and elastic, and $p_c = -\sigma_r|_{r_c}$ the pressure at the interface the equations for the elastic part can be given. Using

$$\sigma_r = -p_c \frac{r_c^2}{r_o^2 - r_c^2} \left(\frac{r_o^2}{r^2} - 1 \right) \quad (\text{B.19})$$

$$\sigma_\theta = p_c \frac{r_c^2}{r_o^2 - r_i^2} \left(\frac{r_o^2}{r^2} + 1 \right) \quad (\text{B.20})$$

$$\sigma_z = \nu(\sigma_r + \sigma_\theta) \quad (\text{B.21})$$

With these formulas it is possible to determine $p_c = \sigma_0$.

$$\sigma_r = -\sigma_0 \frac{r_c^2}{r_o^2 - r_c^2} \left(\frac{r_o^2}{r^2} - 1 \right) \quad (\text{B.22})$$

$$\sigma_\theta = \sigma_0 \frac{r_c^2}{r_o^2 - r_i^2} \left(\frac{r_o^2}{r^2} + 1 \right) \quad (\text{B.23})$$

$$\sigma_z = \nu(\sigma_r + \sigma_\theta) \quad (\text{B.24})$$

The simulations in chapter 5 use a Von Mises flow criterion. The Von Mises flow criterion for this problem is written as:

$$\sigma_{vm} = \sqrt{\sigma_r^2 + \sigma_z^2 + \sigma_\theta^2 - \sigma_r \sigma_\theta - \sigma_r \sigma_z - \sigma_z \sigma_\theta} \quad (\text{B.25})$$

$\sigma_z = \nu(\sigma_r + \sigma_\theta)$ and this can be substituted in the Von Mises flow criterion.

$$\sigma_{vm} = \sqrt{\sigma_r^2 + \nu^2(\sigma_r + \sigma_\theta)^2 + \sigma_\theta^2 - \sigma_r\sigma_\theta - \sigma_r\nu(\sigma_r + \sigma_\theta) - \nu(\sigma_r + \sigma_\theta)\sigma_\theta} \quad (\text{B.26})$$

Since the material is nearly incompressible $\nu = 0.4995 \approx 0.5$ the Von Mises flow criterion can be simplified.

$$\sigma_{vm} = \frac{1}{2}\sqrt{3}\sqrt{\sigma_r^2 + \sigma_\theta^2 - 2\sigma_r\sigma_\theta} = \quad (\text{B.27})$$

$$\frac{1}{2}\sqrt{3}\sqrt{(\sigma_r - \sigma_\theta)^2} \quad (\text{B.28})$$

And this can be related to the Tresca flow criterion as in equation (B.30).

$$\sigma_{vm} = \frac{1}{2}\sqrt{3}\sqrt{(\sigma_r - \sigma_\theta)^2} = \frac{1}{2}\sqrt{3}\sigma_0 \quad (\text{B.29})$$

In the simulations the use yield stress is 100Mpa. To be able to compare results with analytical values, σ_0 should be chosen as:

$$\sigma_0 = \frac{2}{\sqrt{3}}\sigma_{vm} \approx 1.15 \cdot 100[\text{MPa}] \quad (\text{B.30})$$

List of Symbols

Scalars

A_i	cross sectional area of surface i	[m ²]
A	material parameter in Sellars-Tegart law	[1/s]
C_b	bulk modulus	[MPa]
C, C_r	Courant number	[-]
D_{bil}	diameter of the billet	[m]
E	Young's modulus	[MPa]
f_j	nodal transition time for node j	[s]
g	limit function	[MPa]
G	shear modulus	[MPa]
J	Jacobian	[-]
l	characteristic element length	[-]
L_{bil}	length of the billet	[m]
m	shear friction factor	[-]
m	material parameter in Sellars-Tegart law	[-]
M	interpolation function	[-]
N	interpolation function	[-]
p	hydrostatic pressure	[MPa]
Q	activation energy in Sellars-Tegart law	[J/mol]
r, z, h	local cartesian coordinates	[-]
δr_{ip}	displacement of local cartesian coordinates	[-]
r_i	radius i	[-]
R	universal gas constant	[J/molK]
s_m	material parameter in Sellars-Tegart law	[MPa]
S	source term	[-]
t	time	[s]
Δt	time increment	[s]
T	temperature	[K]
u, v, w	displacements in Cartesian coordinates	[m]

w_i	weighing function for node i	[-]
W	work	[J]
x, y, z	Cartesian coordinates	[m]
X_0	original coordinate	[m]
X	current coordinate	[m]
α	weight factor for global smoothing,upwind parameter	[-]
β, β_r	local smoothing parameter	[-]
$\Delta\lambda$	plastic multiplier	[1/MPa]
ϵ_r	radial strain component	[-]
Γ	surface	[-]
Γ_f	location of initial front line	[-]
κ	equivalent plastic strain	[-]
$\dot{\kappa}_0$	parameter to include elastic region in Sellars-Tegart law	[1/s]
μ	friction coefficient	[-]
ν	Poisson ratio	[-]
Ω	domain	[-]
φ	normal angle in bearing	[°]
ϕ	concentration function	[-]
ρ_{yld}	multiplication factor for pseudo material	[-]
σ_y	yield stress	[MPa]
σ_r	radial stress component	[MPa]
$\bar{\sigma}$	effective stress	[MPa]
θ	angle	[°]
ξ	state variable	[-]

Vectors

$\mathbf{f}$	nodal transition times	[s]
F_{int}	internal force vector	[N]
F_{ext}	external force vector	[N]
$\mathbf{n}$	normal	[-]
R^i	residual force vector of the i^{th} iteration	[N]
$\mathbf{t}$	traction forces	[N/m ²]
$\mathbf{u}_g$	nodal grid displacements	[m]
$\mathbf{u}_m$	nodal material displacements	[m]
$\mathbf{u}_c$	nodal convective displacements	[m]
$\Delta\mathbf{u}_g$	incremental nodal grid displacements	[m]
$\Delta\mathbf{u}_m$	incremental nodal material displacements	[m]
$\Delta\mathbf{u}_c$	incremental nodal convective displacements	[m]
$\mathbf{v}_g$	nodal grid velocities	[m/s]

$\mathbf{v}_m$	nodal material velocities	[m/s]
$\mathbf{v}_c$	nodal convective velocities	[m/s]
$\mathbf{w}$	weighing function	[-]
$\mathbf{x}$	Spatial coordinate	[m]
$\mathbf{X}_0$	original coordinate	[m]
$\mathbf{X}$	current coordine	[m]

Tensors

$\mathbf{B}$	tensor relating $\mathbf{D}$ to $\mathbf{v}$	[1/m]
$\mathbf{D}$	rate of deformation	[1/s]
ϵ	small deformation strain	[-]
$\Delta\epsilon$	total strain increment	[-]
$\Delta\epsilon^e$	elastic strain increment	[-]
$\Delta\epsilon^{vp}$	(visco)plastic strain increment	[-]
$\mathbf{E}$	elastic tensor	[MPa]
$\Delta\mathbf{e}$	deviatoric strain increment	[-]
$\mathbf{F}$	deformation gradient	[-]
$\mathbf{I}$	4 th order unity tensor	[-]
$\mathbf{1}$	2 th order unity tensor	[-]
$\mathbf{L}_g$	grid velocity gradient	[1/s]
$\mathbf{L}$	material velocity gradient	[1/s]
σ	Cauchy stress	[MPa]
$\Delta\sigma$	stress increment	[MPa]
$\overset{\nabla}{\sigma}$	corotational stress rate	[MPa/s]
$\mathbf{s}$	deviatoric stress	[MPa]
$\mathbf{W}$	Spin tensor	[1/s]
$\mathbf{Y}$	Yield tensor	[MPa]

Matrices

K	tangential stiffness matrix	[N/m]
-----	-----------------------------	-------

Abbreviations

ALE	Arbitrary Lagrangian Eulerian
CPU	Central Processing Unit
DG	Discontinuous Galerkin
DMZ	Dead Metal Zone

DOF	Degree Of Freedom
DOFS	Degrees Of Freedom
ER	Extrusion Ratio
FEM	Finite Element Method
SUPG	Stream Upwind Petrov Galerkin
WLGS	Weighted Local and Global Smoothing

Miscellaneous

∇x	Gradient of scalar gives vector
$\nabla \mathbf{x}$	Gradient of vector gives tensor
$\nabla \cdot \mathbf{x}$	=Divergence of vector gives scalar
$div \mathbf{x}$	
$\dot{\mathbf{x}}$	time derivative of $\mathbf{x}$
$\mathbf{x}^T$	transpose of $\mathbf{x}$
$tr \mathbf{x}$	trace of $\mathbf{x}$

Bibliography

- [1] Proceedings of Eight International Aluminium Extrusion Technology Seminar. In *Eight International Aluminium Extrusion Technology Seminar*, Florida, 2004.
- [2] proceedings of Extrusion Bologna. In *Extrusion Bologna*, Bologna, 2007.
- [3] R. Akeret and W. Strehmel. Control of metal flow in extrusion dies. In *Fourth International Aluminum Extrusion Technology Seminar*, 1988.
- [4] D.N. Arnold, F. Brezzi, and M. Fortin. A stable finite element for the Stokes equations. *Calcolo*, 21(4):337–344, 1984.
- [5] W. Assaad, H.J.M. Geijselaers, and J. Huétink. Boundary conditions applied on bearing corner in direct aluminum extrusion, 2009.
- [6] N. Biba, A. Vlasov, and S. Stebounov. The simulation of the extrusion process using QForm3D. In *Latest advances in extrusion technology in Europa and second extrusion benchmark*, Bologna, 2007.
- [7] F. Brezzi. On the existence, uniqueness and approximation of saddle-point problems arising from Lagrangian multipliers. *RAIRO Anal. Numer.*, 8:129–151, 1974.
- [8] D. Brokken. *Numerical modelling of ductile fracture in blanking*. PhD thesis, Eindhoven University of Technology, 1999.
- [9] A.N. Brooks and T.J.R. Hughes. Streamline upwind/Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations. *Computer Methods in Applied Mechanics and Engineering*, 32(1-3):199, 1982.
- [10] T. Chanda, J. Zhou, and J. Duszczek. A comparative study on iso-speed extrusion and isothermal extrusion of AA6061 alloy using 3D FEM simulation. *Journal of Materials Processing Technology*, 114(2):145–153, 2001.
- [11] A.J. Chorin. A numerical method for solving incompressible viscous flow problems. *Journal of Computational Physics*, 2(1):12–26, 1967.
- [12] C. Devals, M. Heniche, F. Bertrand, R.E. Hayes, and P.A. Tanguy. A finite element strategy for the solution of interface tracking problems. *International Journal for Numerical Methods in Fluids*, 49(12):1305–1327, 2005.

- [13] L. Donati and L. Tomesani. Seam welds modeling and mechanical properties prediction in the extrusion of AA6082 alloy. In *Latest advances in extrusion technology in Europa and second extrusion benchmark*, Bologna, 2007.
- [14] L. Donati, L. Tomesani, and V. Giacomelli. Evaluation Of A New FEM Criterion For Seam Welds Quality Prediction In Aluminum Extruded Profiles. In *Proceedings of the eight international alumnium extrusion technology seminar*, volume II, pages 221–234, 2004.
- [15] J. Donea. Taylor-Galerkin method for convective transport problems. *International Journal for Numerical Methods in Engineering*, 20(1):101–119, 1984.
- [16] J. Donea, S. Giuliani, and J.P. Halleux. An arbitrary lagrangian-eulerian finite element method for transient dynamic fluid-structure interactions. *Computer Methods in Applied Mechanics and Engineering*, 33(1-3):689, 1982.
- [17] E.N. Dvorkin, M.A. Cavaliere, and M.B. Goldschmit. A three field element via Augmented Lagrangian for modelling bulk metal forming processes. *Computational Mechanics*, 17(1-2):2–9, 1995.
- [18] E.N. Dvorkin and E.G. Petocz. An effective technique for modeling 2D metal forming processes using an eulerian technique. *Engineering Computations*, 10(4):323–336, 1993.
- [19] I. Flitta and T. Sheppard. Temperature Changes and their Effect on Deformation during Extrusion using FEM. In *Proceedings of the eight international alumnium extrusion technology seminar*, volume I, pages 269–283, 2004.
- [20] J.M. Floryan and H. Rasmussen. Numerical methods for viscous flows with moving boundaries. *Appl. Mech. Rev.*, 42(12):323–341, 1989.
- [21] M. Fortin and A. Fortin. A new approach for the FEM simulation of viscoelastic flows. *Journal of Non-Newtonian Fluid Mechanics*, 32(3):295–310, 1989.
- [22] C. Gay, P. Montmitonnet, T. Coupez, and J.L. Chenot. Test of an element suitable for fully automatic remeshing in 3D elastoplastic simulation of cold forging. *J. Mater. Process. Technol.*, 45:683–688, 1994.
- [23] H.J.M Geijselaers. *Numerical simulations of stresses due to solid state transformations*. PhD thesis, University of Twente, 2003.
- [24] D. Greaves. Simulation of interface and free surface flows in a viscous fluid using adapting quadtree grids. *International Journal for Numerical Methods in Fluids*, 44(10):1093, 2004.
- [25] F. Halvorsen and T. Aukrust. Studies of the Effects of Flow Imbalance on Shape Variations in Aluminum Extrusion by use of Lagrangian FEM Software. In *Proceedings of the eight international alumnium extrusion technology seminar*, volume I, pages 243–251, 2004.
- [26] L.R. Herrmann. Elasticity equations for incompressible, or nearly incompressible materials by a variational principle theorem. *AIAA*, 3(10):1896–1900, 1965.

- [27] C.W. Hirt and B.D. Nichols. Computational method for free surface hydrodynamics. *Journal of Pressure Vessel Technology, Transactions of the ASME*, 103(2):136, 1981.
- [28] A. Huerta, F. Casadei, and J. Donea. ALE stress update in transient plasticity problems. *International Conference on Computational Plasticity*, pages 1865–1876, 1995.
- [29] J. Huétink. *On the simulation of thermomechanical forming processes*. PhD thesis, University of Twente, 1986.
- [30] T.J.R. Hughes, W.K. Liu, and T.K. Zimmermann. Lagrangian-Eulerian finite element formulation for incompressible viscous flows. *Computer Methods in Applied Mechanics and Engineering*, 29(3):329, 1981.
- [31] F.J. Humphreys and M. Hatherly. *Recrystallization And Related Annealing Phenomena*. Elsevier Science Ltd, 1996.
- [32] B.M. Irons and A. Razzaque. Experience with the patch test for convergence of finite elements. *The Mathematical Foundations of the Finite Element Method with Applications to Partial Differential Equations*, 1972.
- [33] T. Kloppenborg, M. Schikorra, M. Schomäcker, and A.E. Tekkaya. Numerical optimization of bearing length in composite extrusion processes. In *Latest advances in extrusion technology in Europa and second extrusion benchmark*, Bologna, 2007.
- [34] A.J. Koopman and H.J.M. Geijselaers. Flow front tracking in ALE/Eulerian formulation FEM simulations of aluminium extrusion. *Key Engineering Materials*, 367(1):39–46, 2008.
- [35] A.J. Koopman, H.J.M. Geijselaers, and J. Huétink. A SUPG approach for determining frontlines in aluminium extrusion simulations and a comparison with experiments. *AIP Conference Proceedings*, (1):602–607, 2007.
- [36] K. Laue and H. Stenger. *Extrusion : processes, machinery, tooling*. American Society for Metals, Metals Park, Ohio, 1981.
- [37] P. Lesaint and P.A. Raviart. On a finite element method for solving the neutron transport equation. *Mathematical Aspects of Finite Elements in Partial Differential Equations*, pages 89–123, 1974.
- [38] D. Lesniak, W. Libura, and J. Zaradzinski. Experimental and numerical investigations of aluminium extrusion through pocket dies. In *Proceedings of the eight international aluminium extrusion technology seminar*, volume 2, 2004.
- [39] G. Li, W. Wu, P. Chigurupati, J. Fluhrer, and S. Andreoli. Recent advancement of extrusion simulation in DEFORM-3D. In *Latest advances in extrusion technology in Europa and second extrusion benchmark*, Bologna, 2007.

- [40] Q. Li, C. Harris, and M.R. Jolly. FEM Investigations for Practical Extrusion Issues, Extrusion Process for Complex 3-D Geometries. In *Proceedings of the eight international aluminium extrusion technology seminar*, volume II, pages 151–169, 2004.
- [41] G. Liu, J. Zhou, K. Huang, and J. Duszczek. Analysis of Metal Flow through a Porthole Die to Produce a Rectangular Hollow Profile with Longitudinal Weld Seams. In *Latest advances in extrusion technology in Europa and second extrusion benchmark*, Bologna, 2007.
- [42] W. H. Liu, H. Chang, J.-S. Chen, and T. Belytschko. Arbitrary lagrangian-eulerian petrov-galerkin finite elements for nonlinear continua. *Computer Methods in Applied Mechanics and Engineering*, 68(3):259–310, 1988.
- [43] J. Lof. *Developments in finite element simulations of aluminium extrusion*. PhD thesis, University of Twente, 2000.
- [44] J. Lof, G. Klaseboer, J. Huétink, and P.T.G. Koenis. FEM simulations of aluminium extrusion using an elasto-viscoplastic material model, 2000.
- [45] H.G. Mooi. *Finite element simulations of aluminium extrusion*. PhD thesis, University of Twente, 1996.
- [46] B. Moroz, S. Stebunov, N. Biba, and K. Mueller. Results of an Investigation of Forward and Backward Extrusion with FEM Program QForm. In *Proceedings of the eight international aluminium extrusion technology seminar*, volume I, pages 285–293, 2004.
- [47] P.M. Muessig. Thermal-efficient products through computer simulation software. In *Proceedings of the ninth international aluminium extrusion technology seminar*, 2008.
- [48] W. Mulder. Computing interface motion in compressible gas dynamics. *Journal of Computational Physics*, 100(2):209–228, 1992.
- [49] K.B. Müller. Production of High-Curved Extruded Profiles Directly During The Extrusion Process. In *Proceedings of the eight international aluminium extrusion technology seminar*, volume II, pages 437–445, 2004.
- [50] T.J. Nieuwenhuis. *Aluminium porthole die design*. MSc thesis, University of Twente, 2008.
- [51] S. Osher and J.A. Sethian. Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. *Journal of Computational Physics*, 79(1):12, 1988.
- [52] K. Pettersson. Relation between diametral expansion and internal pressure of thick-walled tubes of elastic perfectly plastic materials. *Journal of Nuclear Materials*, 107(2-3):347–349, 1982.

- [53] E. Pichelin and T. Coupez. Finite element solution of the 3D mold filling problem for viscous incompressible fluid. *Computer Methods in Applied Mechanics and Engineering*, 163(1-4):359, 1998.
- [54] Handouts from presentations. Extrusion Zürich, 2005.
- [55] M.P. Reddy, R. Mayavaram, D. Durocher, H. Carlsson, and O. Bergqvist. Analysis and Design Optimization of Aluminum Extrusion Dies. In *Proceedings of the eight international aluminium extrusion technology seminar*, volume II, page 171, 2004.
- [56] G. Rekers. *Numerical simulations of unsteady viscoelastic flow of polymers*. PhD thesis, University of twente, 1995.
- [57] G.E. Schneider, G.D. Raithby, and M.M. Yovanovich. Finite element analysis of incompressible fluid flow incorporating equal order pressure and velocity interpolation. pages 89–102, 1978.
- [58] H.C. Stoker. *Developments of the arbitrary Lagrangian-Eulerian method in non linear solid mechanics, applications to forming processes*. PhD thesis, University of Twente, 1999.
- [59] M. Sussman. A level set approach for computing solutions to incompressible two-phase flow. *Journal of Computational Physics*, 114(1):146, 1994.
- [60] M. Sussman, E. Fatemi, P. Smereka, and S. Osher. An improved level set method for incompressible two-phase flows. *Computers and Fluids*, 27(5-6):663, 1998.
- [61] R.L. Taylor. A mixed-enhanced formulation for tetrahedral finite elements. *International Journal for Numerical Methods in Engineering*, 47(1-3):205–227, 2000.
- [62] A. Theodorakakos and G. Bergeles. Simulation of sharp gas-liquid interface using VOF method and adaptive grid local refinement around the interface. *International Journal for Numerical Methods in Fluids*, 45(4):421, 2004.
- [63] E. Thompson. Use of pseudo-concentrations to follow creeping viscous flows during transient analysis. *International Journal for Numerical Methods in Fluids*, 6(10):749–761, 1986.
- [64] E. Thompson. Transient analysis of forging operations by the pseudo-concentration method. *International Journal for Numerical Methods in Engineering*, 25(1):177–189, 1988.
- [65] H. Valberg. Metal flow in the direct axisymmetric extrusion of aluminium. *Journal of Materials Processing Technology*, 31(1-2):39, 1992.
- [66] H. Valberg and T. Malvik. Experimental investigation of the material flow inside the bearing channel in aluminium extrusion. *International Journal of Materials and Product Technology*, 9(4-6):428, 1994.

- [67] P.N. Van Der Helm, J. Huétink, and R. Akkerman. Comparison of artificial dissipation and limited flux schemes in arbitrary Lagrangian-Eulerian finite element formulations. *International Journal for Numerical Methods in Engineering*, 41(6):1057, 1998.
- [68] G. Van Ouwerkerk. *CAD implementation of design rules for aluminium extrusion dies*. PhD thesis, University of Twente, 2009.
- [69] B. J. E. Van Rens, W. A. M. Brekelmans, and F. P. T. Baaijens. Modelling friction near sharp edges using an Eulerian reference frame: application to aluminium extrusion. *International Journal for Numerical Methods in Engineering*, 54(3):453–471, 2002.
- [70] X. Velay. Prediction of material flow pattern in the hot extrusion of aluminium alloys by the finite element method. *Proceedings of the eight international aluminium extrusion technology seminar*, 2:6, 2004.
- [71] P.T. Vreede. *A finite element method for the simulations of 3-dimensional sheet metal forming*. PhD thesis, University of Twente, 1990.
- [72] J. Wang and M. S. Gadala. Formulation and survey of ALE method in nonlinear solid mechanics. *Finite Elements in Analysis and Design*, 24(4):253–269, 1997.
- [73] Z. Wang and Z.J. Wang. The level set method on adaptive Cartesian grid for interface capturing. In *AIAA Paper*, page 3990, 2004.
- [74] H.H. Wisselink. *Analysis of guillotining and slitting*. PhD thesis, University of Twente, 2000.
- [75] O. C. Zienkiewicz, S. Qu, R. L. Taylor, and S. Nakazawa. Patch test for mixed formulations. *International Journal for Numerical Methods in Engineering*, 23(10):1873, 1986.
- [76] O.C. Zienkiewicz and R.L. Taylor. *The finite element method*, volume 1. Elsevier Butterworth-Heinemann, 2005.

Dankwoord

Een kort dankwoord is wel op zijn plaats na het afmaken van een proefschrift. Het voltooien van dit proefschrift is alleen geslaagd door de enorme inzet, bereidwilligheid en hulp van veel mensen. Ik kijk met veel dankbaarheid terug op de afgelopen jaren. Ik hoop dan ook dat ik niemand te kort doe in dit dankwoord en als dat wel het geval is, dan bied ik op voorhand mijn excuses aan.

Veel dank ben ik verschuldigd aan Prof. Dr. Ir. J. Huétink, mijn promotor. Beste Han, het begon met jouw uitnodiging om dit onderzoek te gaan doen en het eindigt met dit boekje als resultaat. Het was niet altijd even makkelijk, maar ik heb me altijd gesteund en gemotiveerd gevoeld door alle gesprekken. Je liet me ook steeds weer versteld staan van je enorme kennis en inzicht, niet alleen van je eigen vakgebied maar ook ver daarbuiten. Daarnaast is jou betrokkenheid bij iedereen onder je hoede een geweldig voorbeeld.

Ik dank ook Dr. Ir. H.J.M. Geijselaers voor zijn begeleiding afgelopen jaren. Geijs, toen ik zeven jaar geleden afstudeerde, kwam promoveren kort ter sprake. Jij sprak toen de woorden: "Dat is niets voor jou!". Gelukkig was ik die woorden vergeten toen ik twee jaar later toch nog begon aan een promotieopdracht opdracht. Gedurende mijn promotie heb ik er regelmatig aan gedacht, en je had gelijk. Gelukkig is het dankzij jou ondersteuning en hulp toch gelukt.

Ik wil ook Kjell Nilsen en Peter Koenis van Boalgroup bedanken voor het financieel mogelijk maken van dit onderzoek en hun bijdrage aan het experimenteel deel van mijn proefschrift. Gijs van Ouwerkerk wil ik als projectpartner bedanken voor het meedenken aan oplossingen en het gezelschap naar de conferenties. Onze muzieksmaak loopt uiteen, maar met de humor zitten we op een lijn. Als we moeten lachen, dan gieren we het uit.

Tanja en Debbie, ontzettend bedankt voor de gesprekjes en alle (last minute) regelwerk dat jullie voor ons promovendi doen. Bovendien vrolijken jullie het geheel van stoffige wetenschappers flink op!

Ik wil ook alle collega's waarmee ik afgelopen jaren heb gewerkt bedanken voor alle steun en gezelligheid. Lunchwandelaars, het was wat minder frequent de afgelopen tijd, maar ik hoop dat ik nog een paar jaar mee mag wandelen. Ekke, als we (twee-jaren-) plannen maken dan voeren we ze ook goed uit! Ik heb veel van je geleerd en ik hoop dat je weet hoe dankbaar ik ben dat ik jou heb leren kennen. Ook dank ik de studenten die aan dit onderzoek hebben meegewerkt. Johan, Bastiaan, Kees Durk en Thijs. Het was goed om jullie als klankbord en hulp te mogen hebben.

De laatsten zullen de eersten zijn. Lieve ouders en zus, heel erg bedankt voor jullie steun, klushulp bij verhuizen, financiële steun bij krapte en onvoorwaardelijke liefde en begrip als ik weer aan het werk moest en niet veel tijd maakte om thuis mee te dragen.

Lieve Wilma, het viel niet mee hè, afgelopen jaar met mij. Je had wel de lasten, maar niet de lusten van ons kersverse huwelijk. Ik ben dankbaar dat je het toch hebt aangedurfd om met me te trouwen zo vlak voor het einde van mijn promotie. Naast al je liefde, ben ik je ook veel verschuldigd voor alle zorgzaamheid thuis. Je bent echt een hulp voor mij geweest afgelopen jaar!

Bert Koopman, mei 2009