Investigation of Friction Modelling and Elastic Tooling influences on the Springback Behaviour in Sheet Metal Forming Analysis

Wei Chen
Investigation of friction modelling and elastic tooling influences on the spring back behavior in sheet metal forming analysis

Summary

Sheet metal forming is one of the most common forming processes used in the industry, especially in the automotive industry. It becomes a common sense, that by increasing the accurate simulation of sheet metal forming, the industry can save dramatic cost in trial-and-error process when designing the sheet metal forming tools. In past decades, considerable studies have been done in the field of numerical analysis of metal forming processes, particularly in springback prediction. Significant progresses have been made, but the accuracy of simulation results still needs to improve. One reason is that, in the typical sheet metal forming analysis the tools are considered as absolute un-deformable rigid bodies. The deformation of the tools, which happens in the real production, is not taken into account. Another reason may be that the classic simulation considers the friction coefficient between the tools and blank as constant. However, the actual friction condition depends on a number of parameters.

The objective of this thesis work is trying to investigate how it will affect the springback prediction results when either the tool deformation or more complex friction conditions are considered. The purpose is nothing related with the precise simulation about the true problem or how accuracy the simulation results show compared to the experiment results. The work is only to give an emphasis hint that how the FE-model and friction model chosen affects the springback results when doing a numerical analysis.

A simple model called flex rail is used for sheet metal forming simulations with three different friction models. A comparison between the results clearly shows a difference when advanced friction models are applied. A 3D elastic solid model is created to compare the result with rigid model. The results show the difference when deformation of the tools is taken into account. Finally, an actual case with tools from the industry is investigated. The tools are from SAAB Cars Body Components. This case is to investigate the possibility and necessity of applying the advanced friction model and elastic tools when a complex real industry problem is faced. Further study is needed to do with comparison experimental data to verify the accuracy when these models are used.
Preface

This report is the result of my master thesis project in programme of Simulation of Manufacturing Processes at the Department of Engineering Science of University West. The work was initiated by and carried out at SAAB Automation and Production Technology Center (PTC) of University West during the winter 2010 to spring 2011.

I would like to express my appreciation to my supervisor Mats Larsson for all the help and guidance during the work of this master thesis. I would also like to thank Farhan Khan and other people at PTC for the help and understanding. Appreciation also is expressed here to all the staff at Saab Automobile AB who gave the help and support. Finally, I would like to appreciate my family and friends for all their support.

Trollhättan Sweden, May 2011

Wei Chen
Table of Contents

Summary......................................................................................... i
Preface......................................................................................... ii
Symbols and glossary ..................................................................... v
1 Introduction.................................................................................. 1
  1.1 Background............................................................................. 1
  1.2 Overview of previous works................................................... 2
    1.2.1 Introduction to the springback prediction.......................... 2
    1.2.2 Introduction to the friction modelling............................... 3
    1.2.3 Introduction to elastic tool modelling............................... 4
  1.3 Objective.................................................................................. 5
  1.4 Disposition............................................................................... 6
  1.5 Delimitation............................................................................. 6
2 Methodology.................................................................................. 7
  2.1 Method overview..................................................................... 7
  2.2 Geometry............................................................................... 8
  2.3 Software for the project......................................................... 9
  2.4 Evaluation of springback....................................................... 10
3 Theory.......................................................................................... 10
  3.1 Yield criteria.......................................................................... 10
  3.2 Time integration scheme....................................................... 12
  3.3 Friction models and application............................................. 13
    3.3.1 Coulomb Friction Model.................................................. 13
    3.3.2 V02 friction model......................................................... 14
    3.3.3 Halmstad friction model................................................ 15
4 Parameters study of the two advanced friction models.................. 17
  4.1 Friction modal v02................................................................. 17
  4.2 Halmstad friction model....................................................... 22
5 Simulation using different friction models.................................... 22
  5.1 Model description................................................................. 22
  5.2 General simulation setup........................................................ 23
  5.3 Comparison of the results....................................................... 23
    5.3.1 Punch force...................................................................... 24
    5.3.2 Blank thickening............................................................. 25
    5.3.3 Effective strain of the blank up surface............................. 26
    5.3.4 Springback analysis....................................................... 27
6 Simulations using rigid versus elastic tools................................... 29
  6.1 FE models............................................................................. 29
  6.2 General setup......................................................................... 31
    6.2.1 Elements......................................................................... 31
    6.2.2 Process parameters......................................................... 31
  6.3 Comparison of the results....................................................... 32
    6.3.1 Forming loads............................................................... 32
    6.3.2 Drawing-in................................................................. 33
    6.3.3 Blank thickening........................................................... 33
    6.3.4 Deformation of the tools................................................. 34
    6.3.5 Springback analysis....................................................... 35
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

7 Simulation with the SAAB tools.................................................................36
7.1 FE-model..................................................................................................36
7.2 Simulation with different friction models.................................................38
   7.2.1 General setup....................................................................................38
   7.2.2 Comparison of the springback results.................................................38
7.3 Simulation using rigid versus elastic models.............................................39
   7.3.1 General setup....................................................................................39
   7.3.2 Comparison of the results.................................................................40
8 Methods to decrease simulation time for elastic tools.................................41
  8.1 Deformable Rigid Bodies.........................................................................41
  8.2 Coupling of different meshes...................................................................42
9 Discussion and conclusion............................................................................43
  9.1 Different friction model simulation.........................................................43
  9.2 Elastic tools versus rigid tools simulation..............................................44
References........................................................................................................45
### Symbols and glossary

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEM</td>
<td>Finite element method</td>
</tr>
<tr>
<td>HSS</td>
<td>High strength steel</td>
</tr>
<tr>
<td>DP</td>
<td>Double precision</td>
</tr>
<tr>
<td>SMP</td>
<td>Symmetric Multi-Processing</td>
</tr>
<tr>
<td>MPP</td>
<td>Massively Parallel Processing</td>
</tr>
<tr>
<td>DOF</td>
<td>Degree of freedom</td>
</tr>
</tbody>
</table>
1 Introduction

This chapter gives the background and base knowledge and outcome of the literature study for the work.

1.1 Background

Along with the intensifying of environmental problems and energy crisis, reducing gas consumption to save energy has become the most important issues for auto-making industry. The proportion of high strength steel (HSS) used in auto-making is gradually increasing to serve the purpose of decreasing the weight of automobiles which is the most effective way to reduce gas consumption to address the energy and environmental problems. So the trend of increasing use of high and ultrahigh strength steel for bodies in white is obvious.

However, one of the main obstacles is their unknown behaviour in metal forming processes. HSS has a very high material yield stress, which requires using a very high force as well as high strength tools to form parts, and these parts will have large elastic recovery after forming. Traditionally, tools are made as stiff as possible, which generate higher costs for individual tools. It is clear though, that a tremendous amount of money could be saved by designing the tools such, that their elastic deformation during the forming process is considered. Subsequently the tool geometry may then be designed to compensate the elastic deformations. This would lead to lighter and hence less expensive tools.

In past years, numerical simulation by finite element method (FEM) has become a powerful tool to help avoiding the undesirable defects in materials technological processing, as the accuracy of predicting defects encountered in sheet metal forming has been improved dramatically. Many studies have been devoted to improve the numerical simulation of sheet metal forming processes and springback phenomenon encountered after forming. It comprises several directions of research: developing of new types of finite elements, considering materials anisotropy, developing new time-integration schemes for forming and springback stages, and introducing new constitutive models that provide a more realistic description of the material behaviour [1]. In this work, new friction models have been applied to investigate the influences of friction modelling on springback behaviour in sheet metal forming analysis.
1.2 Overview of previous works

1.2.1 Introduction to the springback prediction

In sheet metal forming, the shape of the blank obtained at the end of the forming step closely conforms to the tools’ geometry. However, as soon as the loads are removed, elastically-driven change in the blank shape takes place. This process is termed springback [2]. As is described, the phenomenon of springback introduces a deviation between the desired shape and the obtained one, which results in assembly problems. Therefore, in the product design process, it is necessary to perform an extensive and iterative experimental trial-and-error process to determine the tools’ geometry and other variables, which will enable a product of the required shape. As a result, the lead time and costs of production are increased considerably. Increasing the accuracy of prediction of springback is a common goal of the whole manufacturing industry. In past decades, a lot of research proposals have been issued to study the factors which influence the springback behaviours in sheet metal forming. From the results of previous studies, there is a general sense that to be able to efficiently and precisely use finite element software to predict springback in sheet metal forming, springback needs to be considered as a complex physical phenomenon, which is very sensitive to numerous factors. Since springback is a complicated phenomenon, it is always difficult to conclude a standard cause of the discrepancy between the magnitudes of springback obtained in simulation and reality, especially when the product geometry is complicated [3]. Springback phenomenon in metals depends on various parameters: variation of elastic properties of a material; elastic-plastic anisotropy; material hardening. If finite element modelling is employed for analysis of springback the accuracy of the obtained solution is significantly affected also by the factors that control the quality of simulation of the forming operation. The most important of these include the method of unloading, time integration scheme, choice of element, blank and tool discretization and contact algorithm (see figure 1.1) [2].
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

1.2.2 Introduction to the friction modelling

With the increased use of FE simulation in tooling departments, the forming analyses of sheet metal components are used more frequently in the design feasibility studies of production tooling. Using computer tools, it is easy to investigate the process and material parameters for the designer. However, the reliability of predicted formability and the accuracy of the estimated deformed geometry depend much on the selected computational modelling approach [4]. In recent years despite a lot of well developed material behaviour models have been launched, metal forming simulations often do not yield the correct results. This is generally because of the choice of friction model. The level of the friction is one of the largest uncertainties in the simulation. It affects the results of simulation substantially and attention has to be given to the determination of values as close to reality as possible. It is recommended to consider the sensitivity and the robustness of the results in relation to the friction level as to choose the friction model. The Coulomb friction model is a simple model frequently used in simulations. In this model, the ratio between friction force and normal force, defined as the coefficient of friction $\mu$, is considered to be constant [5]. However, particularly in lubricated systems, friction depends on a large number of parameters, e.g., the micro-geometry, the macro-geometry, the lubricant and the operational parameters: velocity, temperature and normal load [6]. If one of these parameters changes, the coefficient of friction will also change. This is a known behaviour and generally known as 'Stribeck' behavior [2]. From this it is obvious that a model which describes $\mu$ as a function of local contact conditions is needed. Two new friction models have been set and implemented in the specified LS-DYNA version, and serve the purpose...
of investigation of friction modelling influences on the springback behaviour in sheet metal forming analysis in this work.

1.2.3 **Introduction to elastic tool modelling**

As the trend of increasing use of high and ultrahigh strength steel becomes noticeable, thereby, how to reduce the stiffness of tool to save the tool production cost is the common goal of the manufacturing industry. However, reducing the stiffness of the forming tools adds also new challenges to the modelled forming system, because in that way, elastic deformations of the press equipment, i.e. the frame, table and the base foundation need to be taken into account. One possible approach is of course to model the whole press equipment by finite elements to research elastic response in detail [7]. In the same time, it must be also taken into account that the modelling effort may be very huge. Moreover usual engineering assumptions, which might already influence the results very much, need to be implemented. In other words, it might be much easier to do a limited number of real world stiffness measurements of the machine that can in turn be used as boundary conditions [7] (see Figure 1.2). The final step is to reduce the model size further to reduce the degrees of freedom (DOF) of the modelled tools, i.e. die, and punch and binder (see Figure 1.3). In overview of previous works, there are some possible approaches available in LS-DYNA to take elastic response of tools in finite element sheet metal forming simulations into account. They are namely so called deformable rigid bodies, elastic super elements and full 3D discretization of respective geometries. Details how this could be done have been discussed in [7]. Therefore, it should be emphasized that a modular setup for such forming simulations which take the elastic deformation of tools into account can be possibly achieved. Standard elastic stiffness parameters for each press combined with standard discretization techniques for the tools help to simplify the rather complex setup [7].
1.3 Objective

The purpose of this thesis is to investigate whether the springback behaviour is affected by the friction modelling in the simulation of sheet metal forming analysis. And how the springback behaviour appears when the deformation of tool surfaces is taken into account.
Some investigation and testing of available methods to decrease the needed computer capacity will be tried based on the information given in some referenced paper. Finally, an investigation will be made on an actual case with tools from SAAB Cars Body Components, which including deformation of tools and advanced friction model could identify the problem in a much more complex geometry in the actual production case. Although this work is not supposed to get a clearly generalized answer with only one material studied, some gross conclusions can be collected that whether the advanced friction modelling should be applied when springback analysis with simple geometry. Especially the conclusion can be gathered that, when high press force anticipated, geometry like “Header Upper”, if the elastic tools always are needed to be able to predict the springback behaviour.

### 1.4 Disposition

The methodology used in this work will be described after this introductory chapter. Then theory of the friction models, elastic tooling model and some numerical parameters used in simulations will be given. After this, a simple model called flex rail is used for sheet metal forming simulations with different friction models and with either elastic tooling. The results from these simulations are compared. Some of the methods to decrease the simulation time in the case of elastic tools are also mentioned. The last chapter describes the case with the tools received from SAAB. The springback results with different models are compared to investigate how the results will be affected when it is applied to the complex real industry geometries. The workflow includes description from meshing the models for FE simulations to post-processing the simulation results.

### 1.5 Delimitation

Due to limitations in condition and time, it has been not possible to verify the numerical analysis results by comparing to corresponding experiment data. So the accuracy of the springback prediction needs to be tested further. There are several methods found in the literature and subsequently described in this thesis for how to decrease the simulation time when elastic tools are included. However, due to the time limitation, we have not been able to get these methods to work as desired. So, in this work, only the total 3D discretized mesh method is used.
2 Methodology

In this chapter, methodology is described as follows.

2.1 Method overview

Since the virtual prediction of springback behavior in sheet metal forming processes includes many different parameters, this work is focused on friction model and elastic tool influences on the springback prediction results. The outline of the project is visualised through a flowchart, in Fig. 2.1 below. Each step in the process is described in detail in the following chapters.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

Use LS-DYNA to analyse the forming problem and output the dynain file

Use LS-DYNA to analyse the springback problem and compare the results

Fig. 2.1 Flowchart of the project outline

2.2 Geometry

Within this project, two different geometries have been studied.

- A flexible semi-industrial part called “Flex rail”, which was developed for researching simulation of forming processes. The geometry is shown in Fig. 2.2.
- The second geometry is a Saab Automobile tool for a part called “Header Upper” shown in Fig 2.3. This part is upgraded and compared to the existing one in the new Saab 9-5.

Fig. 2.2 Geometry of Flex rail
The “Flex rail” geometry was chosen because of its highly flexible characteristic and simple geometry which, however, could generate a complex springback behaviour representing b-pillars and side members of automobiles [1]. With this geometry not only three different positions with different strain states of the blank material have been studied but also details of how the different variations of friction models and elastic tool modelling should be decided.

"V02" and "Halmstad" are friction models implemented in LS-DYNA as part of the ongoing research program "Simupart", which is delivered by Saab Automobile during the autumn 2010. "V02" is modelled by Daniel Wiklund and Swerea IVF, "Halmstad" is modelled by Hans Löfgren Högskolan Halmstad, and both of them are still under development.

Both of the two geometries, the simple one “Flex rail” or the complex one “Header Upper” have been used to analyse the influence of both elastic tool modelling and advanced friction modelling on the springback behaviours.

### 2.3 Software for the project

A number of softwares have been utilized in this project. Main of them will be described below.

- LS-DYNA special version with new friction model implemented is used to calculate the forming and springback process.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

- LS-PREPOST is applied to analyse the results from the calculation cluster and get the animated phase picture.
- Hypermesh is used to mesh the parts and create the 3-D solid model when the elastic tool model is applied.

When the standard friction model is applied, MPP LS-DYNA version with double precision (DP) is used as the default version. While the special friction model V02 and Halmstad model was implemented in SMP LS-DYNA release 4.2 with single precision (SP) and SMP LS-DYNA release 4.2.H double precision respectively.

2.4 Evaluation of springback

For the geometry “Flex rail”, in order to achieve a similar process between FE simulations and physical experiments, the punch and blank-holder forces were measured during the experiments. Furthermore, strains were evaluated with an optical system, Argus (2007). For better presentation of the strain states, three different positions were chosen. The springback of physical experiment was evaluated. The detailed description can be found in [2]. With this information from the experiments the FE model parameters are finally set. The results are compared between both two friction models and elastic and rigid tool models, and then conclusions can be proposed for how the friction model and elastic tool influence the springback results for simple geometry like “Flex rail”.

For “Header upper” geometry, since this kind of part geometry includes a lot of complex features and manufactured in big size, so gravity influence should be taken into account when doing springback analysis. The analysis process follows steps like: gravity → forming → coarsening → springback. Different friction models, elastic and rigid tools model are also applied, difference of the results can be collected through the comparison among the different models to get a conclusion with complex geometry like “Header upper”.

3 Theory

3.1 Yield criteria

It is well known that, due to the manufacturing process, cold-rolled metal sheets get different plastic properties in the rolling, the transverse, and the thickness directions, respectively. And this initial anisotropy has a significant influence on the strain and stress distribution during forming of these sheet metals. Besides, the sheet metal is also subjected
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

to the deformation-induced anisotropy. Due to the microstructure evolution, this induced anisotropy destroys the symmetry of the initial anisotropy. However, since the magnitude of the strains during a normal sheet forming operation is moderate, the influence of the deformation-induced anisotropy is normally neglected compared to the initial one in sheet forming simulations[8]. In last decades numerous yield criteria have been proposed, which can fulfil the industrial demands regarding accuracy, easy parameter identification, and computational efficiency. Material model 133: Barlat_YLD2000 which based on the yield condition Yld2000 has been proved to fulfil these demands. The Barlat_YLD2000 yield criteria will be described below.

Here, $X_1$ represents the rolling direction, $X_2$ represents the transverse direction, and $X_3$ represents the thickness direction. Barlat’s Yld2000-2d anisotropic yield criterion $\Phi$ (Barlat et al. [2003]) can be written as

$$\Phi = \Phi' + \Phi'' = 2\sigma$$  (3.1.1)

$$\Phi' = |X'_1 - X'_2|^a$$  (3.1.2)

$$\Phi'' = [2X''_1 + X'_1] + [2X''_2 + X'_2]^a$$  (3.1.3)

where $\sigma$ represents a reference yield stress, $X'_i$ and $X''_i$ $i = 1,2$ are the principal values of two fictitious relations, whose components are related to the components of the real deviatoric stress tensor $S$ by the following matrix relations:

$$
\begin{bmatrix}
X'_1 \\
X'_2 \\
X'_3
\end{bmatrix} =
\begin{bmatrix}
C'_{11} & C'_{12} & 0 \\
C'_{21} & C'_2 & 0 \\
0 & 0 & C'_6
\end{bmatrix}
\begin{bmatrix}
s_1 \\
s_2 \\
s_3
\end{bmatrix}
$$  (3.1.4)

$$
\begin{bmatrix}
X''_1 \\
X''_2 \\
X''_3
\end{bmatrix} =
\begin{bmatrix}
C''_{11} & C''_{12} & 0 \\
C''_{21} & C''_2 & 0 \\
0 & 0 & C''_6
\end{bmatrix}
\begin{bmatrix}
s_1 \\
s_2 \\
s_3
\end{bmatrix}
$$  (3.1.5)

where $C'_{ij}$ and $C''_{ij}$ are material constants. We can calculate the material constants $C'_{11}$, $C'_{22}$, $C'_{26}$, $C''_{11}$, $C''_{12}$, $C''_{21}$, $C''_{22}$ and $C''_{66}$ from the yield stresses $\sigma_0$, $\sigma_{45}$, $\sigma_{90}$, and $\sigma_b$ and the anisotropy parameters $R_0$, $R_{45}$ and $R_{90}$ which represent the values of the yield stress
and $R$ when the tensile axis is at $0^\circ$, $45^\circ$, and $90^\circ$ from the rolling $(X_1)$ direction and $R_y$ represents the in-plane strain ratio $\varepsilon_2/\varepsilon_1$ in the equal biaxial tension test, respectively.

### 3.2 Time integration scheme

In LS-DYNA, the two main solution procedures for the simulation of sheet forming processes are the dynamic explicit and the static implicit algorithms. The dynamic explicit method is frequently used in simulations of sheet forming processes, since it reduces the computation time drastically compared to implicit methods, using a diagonal mass matrix to solve the equations of motion [9–11]. A disadvantage of this method is the conditional stability, necessitating extremely small time steps or artificial adaptations to the model. The static implicit method is unconditionally stable, but a linear set of equations must be solved repeatedly. The common basis of both methods is the discretized equation of motion [12]:

$$M \ddot{d} + F_{\text{int}} = F_{\text{ext}}$$  \hspace{1cm} (3.2.1)

where $\ddot{d}$, $F_{\text{int}}$ and $F_{\text{ext}}$ are the nodal accelerations, the internal forces and the external forces respectively. Since the internal force vector is a function of nodal displacements and accelerations, equation 3.2.1 are a non-linear equation in displacements, velocities and accelerations. Thus, a time integration algorithm must be used to solve this set of differential equations.

The Newmark integration method is an efficient and commonly used single-step integration algorithm. It establishes the relation between the state variables of the current state and the state to be calculated. The basic steps of the Newmark method are [12]:

$$d_{n+1} = d_n + \Delta t \dot{d}_n + \Delta t^2 (\frac{1}{2} - \beta) \ddot{d}_n + \beta \ddot{d}_{n+1}$$  \hspace{1cm} (3.2.2)

$$\ddot{d}_{n+1} = \dot{d}_n + \Delta t ((1 - \gamma) \ddot{d}_n + \gamma \ddot{d}_{n+1})$$  \hspace{1cm} (3.2.3)

where $n$ is the time increment and the parameters $\beta$ and $\gamma$ are free to choose. If $\beta \neq 0$, the displacement at time $t_{n+1}$ is a function of its own time derivatives and the integration scheme is called implicit. An iterative procedure is needed to solve the system of equations 3.2.1. The second term becomes equal to $F_{\text{int}} = Ku_{n+1}$, where $K$ is the stiffness matrix. In
order to iteratively solve the equations of equilibrium the computation of the stiffness matrix is required. If $\beta = 0$, the displacement at time $t_{n+1}$ is a function of variables from a previous step and therefore the integration scheme is called explicit. In this case the system of equations 3.2.1 is linear and its solution is trivial. It is important to mention that the explicit integration scheme is conditionally stable, which means that the time increment must be less than a critical value. For linear solid elements this value can be approximated by the smallest time needed for an elastic wave to cross one element [12].

The major advantage of an explicit method is its easy and straightforward calculation. There are no unbalance forces between the external and internal forces, which determine the values of nodal accelerations at the start of every time increment. So, the explicit method does not suffer from the convergence problems without the balance forces. The major disadvantage of this method is its conditional stability and prohibitively small maximum allowable time step. Usually time mass scaling is employed to serve the purpose of decreasing the total computation time. In this way the critical time step is enlarged by artificially increasing the mass of the material.

The implicit time integration method is unconditionally stable, and a time increment can be selected based on the required accuracy and the convergence behaviour. In this way, the iterative solution procedure is used to find the state variables that satisfy the equilibrium of every increment time step, which gives more reliable results. The main disadvantage of this algorithm is that the computation time for large models is significant, and sometimes reaching convergence within a time increment takes a lot of effort.

### 3.3 Friction models and application

#### 3.3.1 Coulomb Friction Model

The easiest and probably the most well known friction model is the Coulomb friction model. Though it greatly simplifies the frictional phenomena, it is widely used to describe the friction in mechanical contacts.

The friction between two contact partners is based on the Coulomb formulation in LS-DYNA. The behaviour of friction condition is based on the parameters on *CONTACT_ …card 2: static and dynamic friction coefficients, exponential decay coefficient, and coefficient for viscous friction. The interface force is updated by the underlying frictional algorithm to a trial value first, and then it computes the tangential part, the coefficient of friction and the yield force, and finally determines the frictional force.
This is equivalent to an elastic-plastic spring model [13]. The friction coefficient is calculated using the following formula:

\[ \mu = \frac{f_s}{f_n} \]  \hspace{1cm} (3.3.1.1)

where \( f_s \) denotes the shear force and \( f_n \) denotes the normal force.

The Coulomb friction law is only valid as long as the true contact area increases proportional with the normal force. Many investigations have shown, that such a relation can only be observed for relatively small normal forces. Beyond this, the increase of the true contact area is smaller than the increase of the applied force. This is due to the fact, that the true contact area cannot become greater than the geometric contact area.

### 3.3.2 V02 friction model

In LS-DYNA the standard algorithm can be modified for some specification via the keyword USER_INTERFACE_FRICTION, which invokes the subroutine defined by the user.

The friction model of V02 is based on the well known Wiklund model, which considers properties of surface topography, lubricant, sheet material, and process parameters such as sliding speed and pressure. The boundary friction level in this model was held constant and the behaviour of the boundary lubrication is partly based on empirical data from test with DP600 material. The main theory is presented as followed. More information about the model presents in [14-16].

In this model equilibrium of surface loads is evaluated, i.e. if the blank holder (apparent load \( P_A \)) and the shared load on peaks \( P_p \) and pressurised lubricant film \( P_H \) are balancing each other

\[ P_A = P_p + P_H (1 - \alpha) \]  \hspace{1cm} (3.3.2.1)

The normal force \( F_N \) is partly carried by the surface peaks \( F_P \), and partly by the lubricant \( F_L \)

\[ F_N = F_P + F_L \]  \hspace{1cm} (3.3.2.2)

Let \( \alpha \) be the fraction of contact, \( A \) the apparent area and \( P_H \). The generated hydrodynamic pressure, then:

\[ F_L = P_H (1 - \alpha) A \]  \hspace{1cm} (3.4.2.3)
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

If assume the velocity of lubricant with no-slip at each wall, then tangential force $F_T$ can be expressed as friction force on peaks and shear stress of lubricant

$$F_T = \mu_0 F_p + \tau(1-\alpha)A = \mu_0 F_p + \eta\nu(1-\alpha)A / h_f$$

(3.3.2.4)

Where $\mu_0$ is the friction in boundary lubrication and $h_f$ is the functional gap.

Furthermore, let $P_A$ be the apparent pressure, and then friction can be expressed as

$$\mu = \frac{F_T}{F_N} = \mu_0[1-\frac{P_f(1-\alpha)}{P_A}] + \eta\nu(1-\alpha)\frac{P_A}{P_A h_f}$$

(3.3.2.5)

If the pad bearing theory [17] is used to express the hydrodynamic pressure generated in the lubricant

$$P_f = \eta\nu B6W^*$$

(3.3.2.6)

Where $(6W^*)$ is the load coefficient [17], and B is the total width of bearings. Finally, the friction can be expressed as

$$\mu = \mu_0[1-\frac{\eta\nu(1-\alpha)B6W^*}{P_A h_f^2}] + \frac{\eta\nu(1-\alpha)}{P_A h_f}$$

(3.3.2.7)

In the LS-DYNA manual, there is also general introduce. When this model is applied, the IFID under *USER_INTERFACE_FRICTION must be same as CID under *CONTACT... in that way, it can be linked in corrected way. Under *CONTACT... there is a parameter for static friction ($f_s$). The number given can only be used by the model when the relative glide velocity equals 0.

3.3.3 Halmstad friction model

An implementation of a user defined frictions model for DP600 also known as the "Halmstad model" has been made in the latest version of LS-DYNA (R4.2.H). Halmstad model is a local friction model. The friction shear stress $\tau$ (parallel to the sliding direction) is given by the local contact pressure $p$. For the plate DP 600, the relation is as following.
For \( h \geq 1.5 \)

\[
p = p_c(h) + p_v(h) \quad \Rightarrow \quad h
\]

\[
\tau = \mu_c \cdot p_c(h) + \mu_v(h) \cdot p_v(h) \quad \Rightarrow \quad \tau
\]

(3.3.3.1)

and when \( h < 1.5 \)

\[
\tau = \mu_c \cdot p
\]

(3.3.3.2)

where

\[
\mu_c = 0.11 \quad \text{(Friction factor in the boundary layer area)}
\]

(3.3.3.3)

\[
\mu_v = 2.54 \cdot 10^{-8} \cdot e^{2.13h} + 5.74 \cdot 10^{-3} \cdot e^{0.486h}
\]

(3.3.3.4)

\[
p_c(h) = H \cdot 0.245 \cdot e^{-(\frac{h-0.245}{1.18})^2}
\]

(3.3.3.5)

\[
p_c(h) = \frac{\eta V}{\sigma} \cdot 774 \cdot e^{-(\frac{h+2.09}{2.91})^2}
\]

(3.3.3.6)

Where \( h \) denotes dimensionless oil film thickness; \( \eta \) denotes dynamic whisper [Pas] \( V \) denotes sliding speed [m/s] \( \sigma \) denotes surface standard deviation. \( H \) expresses hardness [Pa]. When this model invoked by \*USER_INTERFACE_FRICTION, the model has four (4) definable parameters given in the keyword. The USRFRC in \*CONTROL_CONTACT must set to four (4). Under keyword \*USER_INTERFACE_FRICTION, the input parameters, number of history variables NOC, number of variables to be initiated NOCI, number of parameters per entry segment NHSV all must be set to four(4). More information is presented in [18,19]
4 Parameters study of the two advanced friction models

As mentioned above, the two advanced friction models consider all the tribological conditions of the stamping process, and take into account lubricant properties, roughness of tool and sheet, and local contact conditions such as speed and pressure. When these models are applied, in addition to the static friction coefficient, several other parameters are needed as input. In this chapter, we study how these parameters affect the friction results of these models, in other words, how these factors which are taken into account, from the standpoint of the new friction modal, affect the friction of stamping process forward affect the springback. Each parameter is studied through varying the value 20% to check how it affects the friction. The simple flex rail geometry is used.

4.1 Friction modal v02

In this friction model, there are seven definable parameters which are involved with yield strength of the sheet material, dynamic viscosity of the lubricant oil, and roughness of the sheet and tools surface, tool anisotropic and direction and so on. Several of these parameters are studied how they affect the contact friction respectively. In all of the cases, one of the contact interfaces between die and blank is taken as the study item.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis.

Fig. 4.1.1 Friction energy as function of time of different lubricant oil dynamic viscosity.
The yield strength of the plate surface is used to calculate the effective contact area in this model which depends on the material used and the coating layer on the sheet surface. More details may be found in [14]. The lubricants oil dynamic viscosity of 2.9E-8 Pas with varying 20% up and down to 3.48E-8 and 2.32E-8 are used to study. In the figure 4.1.1, total friction energy as function of time of different lubricant oil dynamic viscosity is compared. The results show that, there is no much different in total friction energy and the history of friction energy increasing with different lubricant oil dynamic viscosity. Figure 4.1.2 and figure 4.1.3 show the von Mises stresses and shell thickness of each case. From the results no big differences can be found among the different cases. Similar analysis also have done to other parameters, there is a gross conclusion that can be gained, the factors considered in the V02 friction model are not sensitive to the value applied, in other words, there is not a separate parameter this model importantly relies on. How the influence of this model on springback will be talked about later in this work, however, the effect by each factor should be discussed related to the theory background in the future.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis.

Fig. 4.1.2 Von Mises stress results of different lubricant oil dynamic viscosity.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

Fig. 4.1.3 Shell thickness results of different tool anisotropic and direction
4.2 Halmstad friction model

There are four definable parameters in this friction modal which are involved with hardness of the sheet material, dynamic viscosity of the lubricant oil, surface standard deviation and friction factor in the boundary layer area. Here the same parameter study as the previous model has been done. The results show the same conclusion, not single parameter has an outstanding effect on this model. How the combination of all the involved factors effect on springback phenomena in each model will be studied in the next chapter.

5 Simulation using different friction models

In this chapter, previously mentioned three friction models, Coulomb, V02 and Halmstad models, are applied respectively. The influence of each model on the springback and forming process is studied. The flex rail geometry is used. All the process conditions are identical except the applied friction model among different cases.

5.1 Model description

The FE-model consists of five parts, punch, blankholder, dies (left and right) and blank. All the tools are approximated to be rigid, and therefore only the surfaces in contact with the blank are needed in the FE-model, see figure 8.1. In this model there are four interfaces in contact and the friction forces are applied between the interacting parts. The static friction
coefficient is set to 0.129, which is applied in the Coulomb friction model and other two advanced friction models only when the relative gliding velocity equals 0. A nonlinear analysis must be performed, since the blank is subjected to large strains. In sheet metal forming, the impact time is short and the time step will therefore be very small to ensure accurate solution. The most suitable approach for the forming process analysis is the explicit time integration, where many uncoupled equations are solved simultaneously. As to the springback analysis, usually implicit time integration is chosen.

### 5.2 General simulation setup

The objective of the case is to give an accurate enough result that can tell the influences of different friction model when applying in sheet metal simulation. In order to avoid noise effects as much as possible, the process parameters are input as simple as possible. To set up the simulation, the guidelines in Input Parameters for Metal Forming Simulation using LS-DYNA [20] and Input Parameters for Springback Simulation using LS-DYNA [21] were used.

As mentioned, all tool parts are modelled as rigid shell elements. The die is constrained in both displacement and rotation in all directions in the global coordinates, while the punch and blankholder are allowed to be translated in the z direction, see figure 5.1.1. The blank is modelled as material type 133. This model is ruled by yield function Barlar_yield2000. The elements used in blank sheet are fully integrated shell elements.

The contact between interfacing surfaces are modelled by using a penalty method. This penalisation reduces any residual between the two surfaces that are to be in contact, by multiplying it by some penalty factor, which thereby works like a spring in-between the two surfaces. A “CONTACT_FORMING_ONE WAY_SURFACE_TO_SURFACE” type of contact is used. The application of the two advanced models is presented in chapter 3.

The simulation of the actual forming process is performed as a so-called multi-forming simulation, which starts by the blankholder closing and fixating the blank to draw beads of the die. After the blankholder motion, the punch motion starts. Notice that, there is no gravity loading, because of the blank sheet is perfect flat.

### 5.3 Comparison of the results

The results of forming and springback simulations of the three friction models are compared in this chapter.
5.3.1 Punch force

From the result, the punch force in z direction along the punch stroke files are obtained and investigated. In Figure 5.3.1 the forces history along the stroke are compared. As shown, the forces are almost identical in the three simulations during the forming process. This fact shows that when the binder load and punch velocity and travel distance is specified, the forming loads in these three friction models are the same.
5.3.2 Blank thickening

The results of simulation show that, the two advanced friction models have different thickness distribution compared to coulomb model, and the coulomb model seems to yield more local thickening than the other two. Between the two advanced model V02 and Halmstad model, there is not much difference noticed in the results except the smallest thickness value of the blank. The results are shown in fig.5.3.2. (Fringe value scaled)

![Fig. 5.3.2 Blank thickness of simulation in Coulomb, V02 and Halmstad friction model (from left to right respectively)](image)
5.3.3 Effective strain of the blank up surface

From the results of simulation, U-surface effective strain is studied. In the figure 5.3.3 (Fringe value scaled), it can be noticed that, in the coulomb model, the blank yield the largest effective strain and in the halmstad model, the blank yield the smallest effective strain.

Fig. 5.3.3 Effective strain of blank up surface in Coulomb, V02 and Halmstad friction model (from left to right respectively)
5.3.4 Springback analysis

The springback was evaluated over the whole surface, but in order to present the comparison among the results clearly, three sections were selected to present the springback results for the three models, which is shown in figure 5.3.4. Section A is located at the area with step, section B is selected in the radius area where the step ends, while the section C is chosen at the area far away from the step and radius.

Implicit springback analysis required rigid body motions eliminated by defining constraints since dynamic inertia effects are not included in a static analysis. Without constraints, a tiny applied load would cause the entire work piece to move rigidly an infinite distance without creating any stresses [21]. Proper constraints must be defined to eliminate six rigid body motions in the model – three translations and three rotations, and the model can deform freely without developing any reaction forces at the constraint points. The usual method is to constrain selected translational degrees of freedom at three nodes, see the details in [21].

![Evaluated sections](image)

Fig. 5.3.4 Evaluated sections

The springback results were then used to study the effect of the variation in the friction model in the FE simulation of the forming processes. Figure 5.3.5 to 5.3.7 show compared results from simulations of processes in three sections respectively. It is obvious from these figures that different prediction results of springback are obtained when different friction models are applied.

From these figures, it can be noticed that, in this case coulomb friction model over predict the springback compared the other two models. While v02 and halmstad model get nearly the same result.

From figure section A and section B, through comparison the results of springback between the left and right sides of the final shape, a conclusion can be got that the right side appears less springback than the left side, which means that, the feature of the step prevents the springback in this area. The result of section C also verifies this, the
amplitudes of the springback in the right side increase because this area is far away from the step seen in figure 5.3.4.

Fig. 5.3.5 Springback results in section A

Fig. 5.3.6 Springback results in section B
6 Simulations using rigid versus elastic tools

6.1 FE models

A simple geometry called “flex rail” which is modelled as both rigid and elastic model is performed to investigate the differences in springback results affected by the deformation of the tool surfaces.

The die and the blankholder are modelled using solid elements while for simplification the punch is modelled using rigid shell elements, because die and blankholder usually suffer much more deformation than the punch. All the solid elements are 4 node-tetra elements and shell elements are 4 node-quadrilaterals. The FE model for this test is shown in Figure 6.1.1. The aim of this case is to find out how much the deformations of the elastic tool will affect the blank springback results.

Number of elements

Die: 167687 solid elements
Blankholder: 78154 solid elements
Punch: 36920 shell elements

Fig. 5.3.7 Springback results in section C
For the rigid model, to be able to compare the simulation results correctly, all the condition and process parameters should be the same as for the elastic model. So, the same contact surface, the same mesh and the same nodes of the blank and tool surfaces has to be used for the different simulations. In Hypermesh, the nodes and meshes can be collected from the surface of the solid model, and shell elements can be created. In this way, the two models have the same surface geometry, the same elements and nodes in the contact area. The rigid shell elements model is shown in Figure. 6.1.2. In both of the two models, punch and blank are the same.

Number of elements
Die: 31891 shell elements
Blankholder: 19286 shell elements
Punch: 36920 shell elements
6.2 **General setup**

6.2.1 **Elements**

Considering the deformation of the tools, elements thickness should take into account. 3-D solid mesh is applied in this case. The solid element types considered for this project were hexahedron and tetrahedron elements. Hexahedron elements are elements with six sides and eight nodes. Tetrahedron elements are elements with four sides and four nodes. Figures of these two types of element are shown below.

![Hexahedron element](image1)

![Tetrahedron element](image2)

In order to avoid the noisy effect by the element choice, both hexahedron and tetrahedron meshes are tried on this “flex rail” model. We found that the forming results with these two types of elements were similar, but the calculation time decreases dramatically with tetra compared to hexa-elements. Hexahedron meshing is also time consuming and much more effort taken to set up in comparison with tetrahedron meshing, especially considering the complex geometry of the car component “head upper” meshing later. Given that the results were similar and the processing time were noticeable shorter when using tetrahedron mesh, we decided to use tetrahedron mesh for 3D tools in this case. The mesh generated for the die tool can be seen in fig 6.1.1. As mentioned up, for the rigid punch and blank elements, there is no other better choice than quadrilateral.

6.2.2 **Process parameters**

This case aims at investigation of the influence of tool surface deformation on the springback behaviour. So the deformation of the tool is brought into analysis, elastic material model type 01 is applied to the tool. As simple as possible, the standard friction model is used. When defining the boundary constraint and loading force, much more effort is needed than we thought. When rigid tools are used as common way, the movement
boundary condition is just applied to the entire part and the force is only loaded at the geometry centre of the rigid part, and then LS-DYNA automatically distribute the boundary constrain. While the elastic 3D elements model is used, considering the un-uniform deformation distribution through the thickness direction, the movement boundary constrain and loading force have to be applied on the surface of the tools node by node. For the die, the bottom surface nodes have to be constrained the freedom degree in the z-direction and the three rotation degree to model the foundation condition. For the blankholder, the boundary movement and blankholder force are applied on the up surface nodes. Other process parameters are input in the same way as the previous case.

### 6.3 Comparison of the results

#### 6.3.1 Forming loads

In order to make sure that the including of elastic tools does not affect the forming loads, the \texttt{rforc} files are obtained and investigated. Even if the binder load and the punch velocity and travel distance are the same in the simulations, there could be a risk that the forces involved in the forming process would differ. In Figure 6.3.1 the punch forces in z direction in \texttt{rforc} file are compared. As shown, the forces are almost identical in the two simulations during the forming process. Moreover the blankholder forces and two die forces are also compared and the results are almost identical. This fact shows that the forming loads for simulations using rigid and elastic tools are the same. In other words, the possibility that the changed loading forces in the process may affect the results can be eliminated.

![Fig. 6.3.1 Punch force as function of process time](image)

Fig. 6.3.1 Punch force as function of process time
6.3.2 Drawing-in

Figure 6.3.2 shows the differences in draw-in between elastic and rigid tool models. From the results, the largest difference is found at the long side, where the difference is 2.827mm.

![Comparison of the blank draw-in, with the rigid tool forming as red and the elastic tool forming as blue](image)

Fig. 6.3.2 Comparison of the blank draw-in, with the rigid tool forming as red and the elastic tool forming as blue

6.3.3 Blank thickening

The fringe value of the figure is scaled to make it clear to compare the blank thickness distribution. From figure 6.3.3, it shows different blank thickness distribution between two models. It seems like that the simulation with the rigid tool yield more local thickening than the model with elastic tools.

![Figure 6.3.3: Blank thickness. Simulation with elastic tools to the left and rigid tools to the right](image)
6.3.4 Deformation of the tools

Usually, when performing the sheet metal forming analysis, tools are always considered as rigid bodies, which means, the tools are un-deformable, see figure 6.3.4 a. However, for the real product, tools are deformable see figure 6.3.4 a. The tiny local deformation of the tool surface may affect the stress distribution of the blank result. Given the same loading condition, the deformation of the tools should be the factor which affects the blank draw-in and thickening, which have been shown.

Figure 6.3.4 a: Plastic strain of the rigid dies

Figure 6.3.4 b: Plastic strain of the elastic blankholder (left) and elastic dies (right)
6.3.5 Springback analysis

The three sections (figure 5.3.4) are used to present the springback results. From picture 6.3.5.a to picture 6.3.5.c, the springback results of rigid and elastic model in these three sections are compared. In the first two figures, we can notice that, the rigid model over predicts the springback compared to the elastic model. While on the right side of section C, the springback amplitude of the blank with elastic model seems to be much bigger than the one with rigid model. Further study should be done to make sure the stress distribution around this area. However, from the results shown in these figures, it appears clear that, when the deformation of tools is taken into account, there is noticeable change in the results of springback prediction.

Fig. 6.3.5.a. Springback results in section A

Fig. 6.3.5.b. Springback results in section B
7 Simulation with the SAAB tools

In the following chapter the deformation of tools received from SAAB Car Body Components, which is called “head upper” will be investigated. This part is upgraded and compared to the existing one in the new Saab 9-5. With these tools, the previously studied friction models are applied. Simulation results are compared to check how the friction models influence springback results when a complex geometry is considered. To investigate the deformation of real productive tool, two simulations are made, one where the tools are modelled as rigid, and one where 3D-elastic tools model are used.

7.1 FE-model

CAD model of the tools are exported from the software Autoform, the specified simulation tool for car components manufacturing in SAAB Automobile, which considers all the tools as rigid bodies. Because of some characteristic of this software, only die can be exported, which is in IGES format. The other parts like punch, blankholder, which are used in this FE-model, are obtained through offsetting parts of the die using Hypermesh. For simplification and also base of the previous study, tetrahedron elements are used to mesh the solid tools except the punch, which is modeled as rigid shell body. The 3D models of die and blankholder are shown below.
As with the flex rail model explained earlier in this thesis, shell elements are created from the solid model surface.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

7.2 Simulation with different friction models

7.2.1 General setup
As to focus on the friction model analysis, for simplification, only the rigid shell FE-model is used. The starting point of this case is the gravity loading process. It is important when the flat, stress-free blank under its own weight after being placed on the binder surface over the die cavity, especial for the large shape and flexible blank. In LS-DYNA, it is convenient to operate a gravity analysis just including several keyword commands. More information is presented in [22]. The general setup refers to previous description in chapter 5. However, in this simulation, the die and blankholder are moving along z-direction, while the punch is fixed.

7.2.2 Comparison of the springback results
The forming result (Coulomb) without trimming is shown in figure 7.2.1. In order to show the comparison results the three models clearly, a section also is chosen, see figure 7.2.1. In the figure, we can notice that, friction model V02 and Halmstad almost have identical results, while compared to Coulomb model, the prediction results differ.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

7.3 Simulation using rigid versus elastic models

7.3.1 General setup
The general setup refers to description in chapter 6.2. As mentioned up, tetrahedron elements are used to mesh the 3D tool based on previous study and a lot of efforts are saved. Since the die and blankholder are the moving parts, more attention should be given when applying constraints and boundary conditions. Considering the deformation of the elements through the thickness direction, the loading have to be located on the surface nodes. It’s not like in reality, when closing is processing, the die is supported by some
structure applying the support forces. While in simulation, the boundary condition of drawing parts is always prescribed as motion. So, it takes a lot of efforts to avoid penetration and the dynamic effects when closing the binder and applying the loading force.

7.3.2 Comparison of the results

Contacting forces are compared to make sure the forming loads are the same.

Fig. 7.3.1 Identical forming loading between elastic and rigid mode
The same section is chosen as before to show the springback results. The result is presented in figure 7.3.2. From the result, we can notice that, in some region, it seems like the rigid tool has more punch stroke. This might cause from the deformation of the die surface. So it is obvious that, when the elastic tool is used, the results change significantly.

![Figure 7.3.2 Comparison of the springback results in the specified section](image)

**Fig. 7.3.2 Comparison of the springback results in the specified section**

### 8 Methods to decrease simulation time for elastic tools

Compared to classic rigid tool set up, the reasons why elastic tools are not included in the simulation are the increased calculation time, higher demand for memory and much more effort in meshing. In order to avoid these mentioned disadvantages, several methods have been developed.

#### 8.1 Deformable Rigid Bodies

Modal analysis is well known in linear dynamic analysis for efficient computations of the response of elastic structures. The method is to find a particularly useful problem dependent basis called eigenmodes. Compared to standard direct methods, where the response node is the analysis item, modal methods represent the response of the structure modes.

Deformable rigid bodies, or flexi rigid bodies, is a method which is based on calculation of the eigenmodes behaviour of the structure. If a complete set of modes is used, the method can be considered as a transformation into a set of generalized coordinates, see figure 8.1. Typically, only a small number of modes are used, in other words, only interesting ones are
included. So, this method becomes an approximate and simplified solution which means a substantial computational saving.

The method is performed in two phases. In the first phase, eigenmodes are obtained by making an eigenmodes simulation and output to a binary database. In the second phase, this database is used as input to supply modes for the transient dynamic analysis. This method has been tested in several studies and has been proved to give a similar solution as the one obtained from a full direct solution. The results also show that the accuracy of this method is related to the number of modes included. With more modes included, the deformation of the body is better approximated than with lesser modes. Usually, as advised in [24], six rigid body modes from the structural modes are separated by choosing the separation frequency between the sixth and the seventh eigenfrequency. The implementation of the method into LS-DYNA has been described in [25].

### 8.2 Coupling of different meshes

When investigating elastic response of forming tools, it appears that only small deformation happened on the tool surface location, which means that a fine resolution of the tool deformation is probably not necessary. Only the fine surface details demand a rather fine mesh. This method is to apply a fine surface shell mesh on the contact surfaces and use a coarse solid element to mesh the structure, see figure 8.1. The fine surface shell elements will work as contact segments on the coarse 3D meshes. The total of elements of the tool model and efforts of meshing can be decreased tremendously since the coarse 3D mesh is used. In LS-DYNA, the contact interface option of tied-offset-contact now is available to fix nodes from the fine surface mesh to corresponding segments from the coarse 3D mesh. However further investigation should be issued on the accuracy of this method since with this method the local deformation is neglected.
Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis

9 Discussion and conclusion

9.1 Different friction model simulation

As is well known, in most cases of sheet metal forming analysis, the friction between two contact partners is based on a Coulomb formulation, which is equivalent to an elastic-plastic spring model. However, lots of studies have proved that, the tribological conditions of the stamping process are more complex. Main reason for simplification of the complex condition is to save the cost. Another reason could be the limit of calculation in the specific time. Obviously, the simplification will decrease the accuracy of simulation. In this way, it might increase the cost of time and money in a trail-and-error process. So how to balance this contradiction is an interesting topic. One objective of this thesis work is to investigate how much the different friction model chosen in simulation affects the springback prediction.

Two advanced user defined friction models which consider the tribological conditions more complex are applied to compare with Coulomb model. First the simple geometry “flex rail” is used to test. The results of the two advanced friction models appear significantly different to the classic friction model. Considering the geometry effect and the industry applicability, a more complex geometry tool from SAAB car component is used for a test. There is still a considerable discrepancy between springback results. We can notice that, the choice of friction model ought to deserve more attention, whether simple geometry or complex industry production tools.

In the simulation of these cases, in order to keep the comparability, all the process parameters are considered and forced to be identical with each other. Most of the process parameters are chosen as empiricism. From the angle of accurate, some of them might need more verification study, however, if just on the friction numerical analysis, the results are still valuable and credible. These two advanced friction models are based on a
functional surface parameter for sheet metal forming simulation, further study and experiment need to be performed to verify the accuracy of the predictability.

9.2 Elastic tools versus rigid tools simulation

In this thesis, the possibilities and effect on the springback of including elastic tools in sheet metal forming simulations have been investigated. With simulations of a simple model it is obvious that there is a remarkable discrepancy between the springback results. For comparability and avoiding the interference by other factors, the shell elements of the rigid tools are just collected from the surface of the 3D solid tools. In this way, the contact conditions can keep absolute same, including the number of elements, nodes, and the detail of the surface geometry. Another effort has been done is to make sure the loading conditions are the same. Because in simulation, there is a risk that, when different FE-model included, even when the input loading parameters like forming speed and punch stroke and contact definition are the same, the loading forces like punch forces and contact forces still may differ from each other. When the process parameters and loading forces are identical, the discriminate of stress distribution between the two blanks after the forming process is only caused by the deformation in the tools.

The calculation time increases enormously when a 3D solid model is used. Even the simple model “flex rail” is tested. When the 2D rigid tool without mass scaling is used, the calculation time is 7 hours 20 minutes with 8 MPP processors. When 3D solid is applied, the calculation time is more than two days. Considering the quantity of simulation need to run and the calculations ability of the cluster, mass scaling is decided to use. With specified mass scaling, the calculation time of rigid tools decreases to almost half an hour while solid model decreases to about 8 hours. How the mass scaling affects the prediction accuracy need further investigation, while in this case, the influence of the elastic tool is obvious from the results.

There are two methods mentioned in this thesis work, which can help to decrease the calculation time compared to the total 3D discretized mesh. However since less information from the reference material, and the limited knowledge of the author, none of them can get through. The efforts what have been done are not presented here. More study need to be done in future work, since if the calculation time can be decreased without reducing the accuracy, then elastic tool can be widely used in the real production simulation.
References


Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis


Investigation of friction modeling and elastic tooling influences on the springback behavior in sheet metal forming analysis


