European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS 2000
Barcelona, 11-14 September 2000

ON FINITE ELEMENT SIMULATION OF SHEET METAL FORMING PROCESSES IN INDUSTRY

Kjell Mattiasson

Volvo Car Corporation
405 31 Göteborg, Sweden
e-mail: vcc2.kmat@memo.volvo.se, web page: http://www.volvocars.com/

and
Department of Structural Mechanics
Chalmers University of Technology
412 96 Göteborg, Sweden
e-mail: kjell@sm.chalmers.se, web page: http://www.sm.chalmers.se/

Key words: Sheet metal, forming, simulation, Finite Element

Abstract. *During the last few years an enormous progress in the industrial use of numerical tools for the simulation of sheet forming processes has taken place. This is especially true for the automotive industry. The present paper tries to give a brief introduction to the subject of practical sheet metal forming, and to describe some of those forming defects, which can occur. Thereafter, a state-of-the-art review of methods and procedures for sheet forming simulation in practical use today will be presented. This concerns especially various Finite Element formulations used for the current application, including a brief historical review on the subject. Finally, some shortcomings of today's simulation technology will be described.*

1 INTRODUCTION

An often-cited statement is that sheet metal forming during the last decade has turned from being an art to being a science. The background to this statement is that sheet metal forming, from ancient to modern times, has been the task of skilled craftsmen rather than of theorists and scientists. Very few theoretical aids have been available for facilitating the die designers in their task, but they have had to rely on their own experience and simple guidelines. The design and tryout of forming tools have, thus, been a time consuming trial and error process.

However, the demand for shorter lead times, especially in the automotive industry, has accentuated the need for a computerized simulation aid, in which the forming process can be simulated, analyzed, and optimized, before any hard tools are built. During the last few years this desire has partially become reality, and today the simulation technique has been integrated in the die design process at many automotive and tool manufacturers.

The aim of the present paper is to give a state-of-the art review of current sheet forming simulation methods. The focus will be on the industrial implementation of these methods, rather than on current academic achievements. In order to provide a historical perspective on the subject, a brief review of the developments in this area during the last three decades will also be given.

2 SOME PRACTICAL ASPECTS ON SHEET METAL FORMING

2.1 Sheet forming processes

By far the most common sheet forming process is *stamping*, which especially is used in the huge automotive industry. In the stamping process the metal sheet is formed by rigid tools, which consist of a punch (male part), a die (female part), and, finally, a blankholder. The role of the blankholder is to press the blank against the die and prevent it from wrinkling, and also through friction forces control the material flow into the die cavity during the stroke. The main advantage of the stamping process is its high productivity, which is a very important quality in the highly efficient and automated car manufacturing industry. In Fig. 1 a Finite Element (FE) model of a stamping operation is displayed.

In *hydroforming* processes a hydraulic pressure replaces one of the rigid tools in the stamping process. For instance, in the *flex-forming* process the punch is replaced by a hydraulic pressure, which presses the blank down into the die cavity. Flex-forming is typically used in the aircraft industry, and for manufacturing of prototype parts in the automotive industry. The advantage of most hydroforming processes is that they allow parts to be manufactured, which would have been impossible in ordinary stamping. The main disadvantage is their low productivity, which makes them unsuitable to use in the automotive industry.

One type of hydroforming processes has, however, gained a great deal of popularity in the automotive industry in later years. This is the *tube hydroforming* process. In this process beam type parts with closed cross sections are manufactured. A tube-shaped work piece is first bent, and then formed to its final shape by an internal hydraulic pressure against an enclosing rigid die. Although tube hydroforming is a rather slow process, it has gained a wide appreciation, since, due to its good formability, it allows a part to be manufactured in one piece, instead of being welded together of several stamped parts. A FE model of a tube hydroforming part can be seen in Fig. 2.

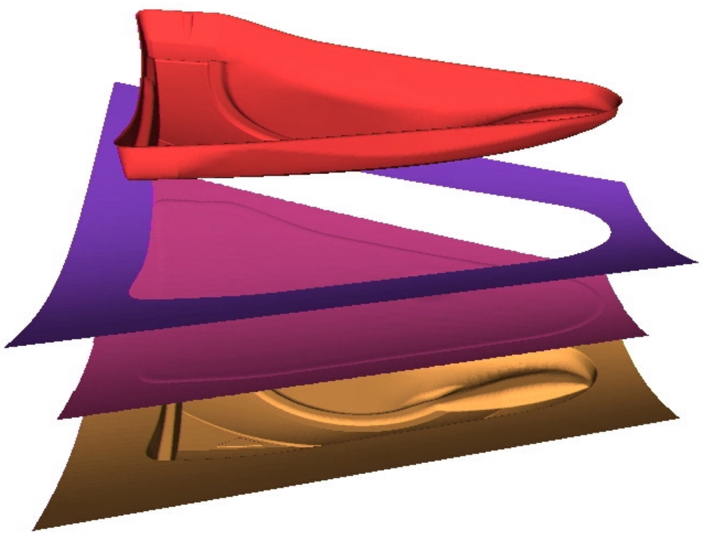


Figure 1   FE model of a stamping process. The parts are from top to bottom: punch, blankholder, blank, and die.

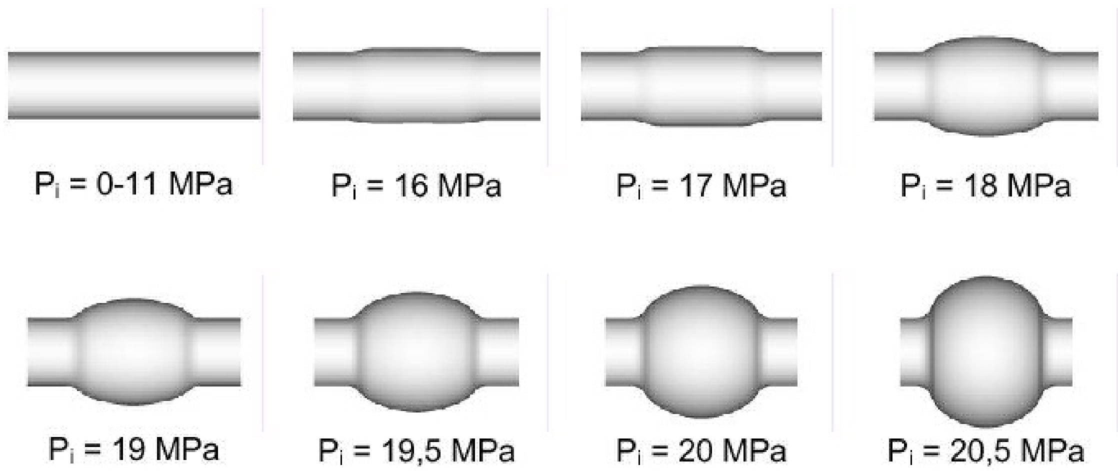


Figure 2   FE simulation of a tube hydroforming process

2.2 Forming defects

During the tryout of a forming process various types of defects are usually appearing in the formed part. Examples of such defects are:

- *Fracture* in the material, usually preceded by a marked strain localization.
- Excessive *thinning* in some areas of the blank
- *Wrinkling,* which implies the formation of bulges with relatively short wavelength due to high compressive stresses.
- *Buckling,* a term used for bulges with long wavelength, preferably appearing in unsupported areas of the blank with small compressive stresses.
- *Springback* is a term for those deformations that take place when a work piece is removed from the tools after completed forming.
- Various *surface defects*, which usually are due to insufficient stretching of the material.

It is the purpose of the die design and process layout work, and the subsequent tool tryout, to optimize the forming process in such a way that these defects can be avoided.

3 FINITE ELEMENT METHODS FOR SHEET FORMING SIMULATION

3.1 Introduction

The aim is that the simulation code should be able to simulate the complete forming process and to be able to unveil any possible defects. It should also be possible in the simulation to vary all those parameters, which in a practical case are used to optimize the process.

Sheet forming simulations tend to be very time consuming. One reason for this is that the process itself is computationally very complicated, involving effects such as nonlinear material behaviour, large deformations, and complicated contacts between tools and work piece. Another reason is that the FE models usually are very big, containing tens and hundreds of thousands elements. In the development of FE codes for sheet forming simulation, computational efficiency has therefore always been a primary concern.

3.2 Finite Element formulations

Through the years a number of different FE formulations for sheet forming simulation have been presented. These can differ from each other in several respects, such as FE types, kinematic description, constitutive description, and solution methodology. The bases for FE analysis of large deformation problems were not established until the mid 70's, and it was not until then the first procedures for FE simulation of sheet forming were presented.

The Hill'48 material model To be able to review the different FE formulations, we will first have a look at different ways of expressing the constitutive equations. The most commonly used constitutive relation in sheet metal forming contexts is the model of Hill[1] from 1948. It can describe orthonormal anisotropy of the material. The model is also known as Hill's quadratic model, since the stress terms describing the yield surface are all squared. The effective stress can in matrix form be expressed as

$$\bar{\sigma} = \left(\{\sigma\}^T [A]\{\sigma\}\right)^{1/2} \tag{1}$$

where [A] is a matrix with constants describing the anisotropy of the material.

The normality condition can for associated plasticity be written

$$\{\dot{\varepsilon}^p\} = \dot{\lambda}\left\{\frac{\partial f}{\partial \sigma}\right\} \tag{2}$$

In the special case of a quadratic yield condition this can, in view of Eq. (1), be expressed as

$$\{\dot{\varepsilon}^p\} = \frac{\dot{\lambda}}{\bar{\sigma}} [A]\{\sigma\} \tag{3}$$

Inverting this expression, and noting that $\dot{\lambda} = \dot{\bar{\varepsilon}}^p$, we get

$$\{\sigma\} = \frac{\bar{\sigma}}{\dot{\bar{\varepsilon}}^p} [A]^{-1} \{\dot{\varepsilon}^p\} \tag{4}$$

Note that this equation expresses *total* stress in terms of *rate* of plastic strain. Note also that it is only for quadratic yield conditions that the normality condition can be inverted to this form.

If the *plastic* strain rates in Eq. (4) are replaced by *total* strain rates, i.e. the elastic part of the deformation is ignored, this equation will form the basis of the *rigid-plastic* theory. A couple of the earlier FE formulations for sheet forming simulation were based on this form of the constitutive equations.

The flow formulation The *flow-formulation* for sheet metal forming is based on the above rigid-plastic material law. It uses a kind of Updated Eulerian formulation with nodal velocities as primary unknowns. The geometry is fixed in each time step, while the equilibrium is iteratively solved for. The geometry is then updated based on the calculated velocities.

It is interesting to note that there exists a complete analogy between the equations of the flow approach, and of those of small strain, linear elasticity. The only differences being that strain rate and nodal velocity in the flow formulation take the place of strain and nodal displacement in linear elasticity, and that the elasticity modulus in the elastic constitutive equations corresponds to a nonlinear 'viscosity' term in the rigid-plastic equations.

One of the main advantages of the flow approach is, thus, that the governing equations get a very simple appearance. A disadvantage of the approach is that problems occur when there are undeformed zones in the body, where $\dot{\bar{\varepsilon}} = 0$, and the 'viscosity' turns to infinity. It takes some artificial actions to cure that problem. Another obvious disadvantage is of course that no phenomena related to elasticity, such as springback, can be simulated.

See for instance Onate et.al.[2] for further references on the subject.

The rigid-plastic formulation In the *rigid-plastic approach* the same rigid-plastic constitutive relations as in the flow approach are used. However, some writers have preferred to rewrite these relations in terms of *increments* of strain. This leads naturally to a Langrangian FE formulation with nodal displacements as primary unknowns.

The disadvantages of such an approach are of course the same as for the flow approach. However, the present formulation do also lack the simplicity of the flow formulation, since the kinematic relations in a Lagrangian formulation are much more complicated than those in an Eulerian one.

Refs.[3,4,5] are examples of works in which the rigid-plastic approach has been employed.

The static-implicit method A sheet forming formulation, which is based on a Lagrangian description of motion and an elastic-plastic or elastic-viscoplastic constitutive law, is termed the *solid approach*. It is a formulation of considerable theoretical complexity, but has the advantage of being able to simulate also phenomena related to elasticity, such as springback. In contrast to the previous two approaches, this one is not restricted to quadratic yield conditions.

The resulting system of equations is normally solved by the Newton-Raphson method, or some similar technique. The method is in that case also known as the *static-implicit method.* In later years the importance of the concept of consistent linearisation has been realized. This concept is essential in order to preserve the quadratic rate of convergence of the Newton method, and has applicability both on stress integration as well as on contact/friction procedures. The use of consistent linearisation has implied a dramatic improvement of the performance of the solid approach, both with regard to efficiency as well as to robustness.

The main approximation introduced in the solid approach originates from the integration of the rate constitutive equations in order to calculate stresses.

The present approach has been used by numerous researchers, and the reader is referred to the proceedings from some of the recent conferences on the subject of metal forming simulation for further references. See for instance Sünkel et.al.[6], and Tang and Hu[7].

The static-explicit method The previous approaches have all been *implicit* in the sense that an iterative procedure has been employed in each step in order to fulfill the static equilibrium conditions. However, some authors have used a technique in which no iterations at all are performed. The updating of the geometry is just based on the result of the first iteration in each step. This implies that equilibrium is never satisfied. In order to reduce the errors involved, very small steps have to be taken. Several thousand steps are common for an ordinary simulation.

An advantage of this approach is that it is quite robust, since there are no iterative processes that have to converge. Even instability phenomena like wrinkling have been simulated by means of this procedure. The procedure is called the *static-explicit* approach in order to distinguish it from the better known dynamic-explicit approach.

This procedure has been particularly popular among Japanese researchers. A couple of recent papers on this subject are Nakamachi[8], and Kawka and Makinouchi[9].

The dynamic, explicit method Metal forming problems can generally be considered to be quasi-static problems, i.e. inertia forces do not have any major influence on the processes. All the previous approaches can be considered as 'natural' in the sense that they are quasi-static. Despite this fact an approach, in which the problem is treated as a transient, dynamic one, has become the most popular method in later years. This particular type of method has previously been used to simulate highly transient problems like explosions, projectiles penetrating targets, automobile crashes, and so on.

The reasons for using a method like this in metal forming problems are twofold: The method is extremely robust, and it is very efficient, especially for large-scale problems.

The discretized dynamic equations are integrated by the central difference explicit time integration scheme. Furthermore, lumped mass matrices are used, which implies that the mass matrix is diagonal, and no system of equations has to be solved. The critical time step is approximately equal the time for a bending or compression wave to travel through the smallest element in the mesh. A typical time step in a sheet forming analysis is therefore of the order of a microsecond. The number of time steps in a typical sheet forming simulation is normally several tens of thousands.

Other advantages of the dynamic, explicit method are, for instance, that, because of the small time steps, the kinematic and contact conditions become very simple. The memory and data storage requirements are, furthermore, relatively small. The method is well adapted for vectorization and parallelization.

In the dynamic explicit method the computing time is directly proportional to the duration of the analyzed event. In order to speed up the computations it is customary to use a fictitious time scale and/or a fictitious density. It is, however, essential to control that the inertia forces do not influence the solution.

A majority of the most popular commercial codes for sheet metal forming simulation are based on the dynamic, explicit method. See for instance Hallquist et.al.[10], Haug et.al.[11], Mercer et.al.[12], and Aberlenc et.al.[13].

On-step methods The so-called *one-step methods* are variants of the static-implicit method, where the complete solution is performed in one single step under the assumption of linear strain paths. The history dependency of material and contacts are thus neglected. The main advantage of these methods is of course the short computing time, which is a fraction of the one for any of the previous methods.

In spite of the considerable simplifications introduced in these methods, they still have proven useful in some applications. Especially in early phases of the tool design process, even the rough predictions from a code like this can be a valuable aid. However, in later evaluations of process and die designs more accurate methods have to be used. Recent presentations of one-step methods can be found in Batoz et.al.[14] and El Mouatassim et.al.[15].

The AUTOFORM approach The approach used in the commercial code AUTOFORM is another variant of the static-implicit method. Normally, quasi-static, implicit codes make use of direct, linear solvers. The disadvantage of such solvers is that the computing time increases roughly with the second to the third power of the number of equations, which makes them less suitable for large scale problems. Iterative solvers, on the other hand, for which the computing time increases almost linearly with the size of the problem, are inappropriate for sheet metal forming problems, since the resulting system of equations is highly ill-conditioned. A condition for an efficient utilization of an iterative solver is that the system of equations is well conditioned.

AUTOFORM uses basically membrane element, but bending can be considered as a secondary effect. The special feature of this code is that, in each time step, the motions of the nodes perpendicular to the tool surfaces are uncoupled from motions tangential to these surfaces. In each new step a form of the sheet is first sought, that satisfies the boundary conditions determined by the tools. Thereafter equilibrium is determined iteratively. Within this process the nodes have only two degrees of freedom each - two translation components in a tangent plane to the tool surface. The resulting system of equations is well conditioned, and an iterative solver can effectively be utilized.

The advantage of the present approach is that it is highly efficient and robust. The disadvantage is that, since it is based on membrane theory, some approximations are introduced in the solution, and phenomena related to bending, such as wrinkling, cannot be directly simulated.

For a more detailed description of this approach the reader is referred to, for instance, Kubli and Reissner[16].

4 THE PRACTICAL USE OF SHEET METAL FORMING SIMULATION IN A HISTORICAL PERSPECTIVE

The bases for FE analysis of large deformation problems were not established until the mid 70's, and it was not until then the first procedures for FE simulation of sheet forming were presented. Early attempts to simulate sheet metal forming processes by means of the Finite Element method were usually based on 2D, or axisymmetric models. The 'flow' and the 'rigid-plastic' approaches were more popular than the 'solid' one, mainly because it was possible to advance the solution in much bigger increments in these approaches.

In 1978 Wang and Budiansky[17] published the first complete 3D formulation for sheet forming problems, based on a membrane formulation and a 'static-explicit' approach. The practical application of sheet forming simulation was, however, during many years hampered by too unstable numerical procedures and excessive computing times, even for very small problems.

Ten years later, in 1988, Tang et.al.[18] published results from practical applications of a code, developed at Ford, to the simulation of stamping of real 3D automotive parts. This code was based on large strain shell theory and a 'static-implicit' approach. Models with up to 400 elements were analyzed, and the reported computing time was about 20 hours.

In 1989 results from a Volvo/Control Data project was presented (Honecker and Mattiasson[19]), in which the 'dynamic-explicit' approach was evaluated in application to sheet metal stamping. The results from this study were very promising. Problems with up to 10,000 shell elements could be solved within 1.5 hour on a super computer. Also the robustness of this approach was found to be widely superior to that of any other method.

From that time the practical utilization of sheet forming simulations within the industry has shown an explosive development. Most companies within the automotive industry are today performing sheet stamping simulations on a regular basis. Dynamic explicit codes, such as LS-DYNA, PAM-STAMP, OPTRIS, ABAQUS/Explicit, and others, are dominating the software market. Exceptions can be found in Japan, where also a couple of codes based on the 'static-explicit' approach have found some industrial usage. The highly specialized code AUTOFORM (see Sect. 3.2) is also widely used, often as a complement to other codes. Various one-step codes are frequently used as preliminary design tools.

There are several reasons for the breakthrough of simulation aids in the sheet forming industry in later years. One reason is of course the development of efficient and robust simulation methods. Another equally important factor is the rapid development of computer hardware, which makes it possible for most companies to perform simulations of complex production parts in reasonable time and to a reasonable cost. However, the forming simulation is just one activity in a chain of activities. A necessary condition for the success of forming simulations has also been the rapid development of the softwares used before and after the forming simulation in this chain of actions. For instance, a necessary condition is the availability of CAD systems in which the geometry of the products can be numerically described and easily modified. Another necessary condition is the availability of efficient tools for creating FE meshes on the CAD surfaces. Finally, the development of computer graphics and efficient post-processors make it possible to easily evaluate the huge amount of output data from the simulation codes.

5 MATERIAL MODELING

5.1 Introduction

The cold rolling of the sheet material generates crystallographic textures, which is observed as a mainly orthogonal plastic anisotropy. The anisotropy normal to the sheet surface is known to be the most significant one, and is known as *normal anisotropy.* If also the anisotropy in the plane of the sheet is considered, the term *planar anisotropy* is used.

The level of the anisotropy is usually characterized by the *plastic anisotropy parameter* R*,* defined as the relation between plastic strain rates in the width and thickness directions, respectively, in a uniaxial tension test, i.e.

$$R = \frac{\dot{\varepsilon}_b^p}{\dot{\varepsilon}_t^p} \tag{5}$$

Normally, tension tests are performed on sheet strips cut from the blank in three different directions: 0^o, 45^o, and 90^o to the rolling direction. When only normal anisotropy is considered, an average value of the anisotropy parameters in three directions is used:

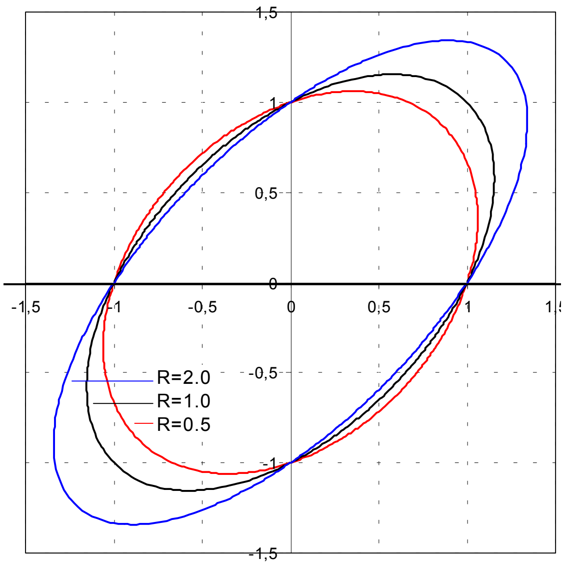
$$R = \frac{R_0 + 2R_{45} + R_{90}}{4} \tag{6}$$

Normally $1 < R < 2$ for steel, and $R < 1$ for aluminium. Below some of the most common yield surfaces for normal and planar anisotropy will be reviewed.

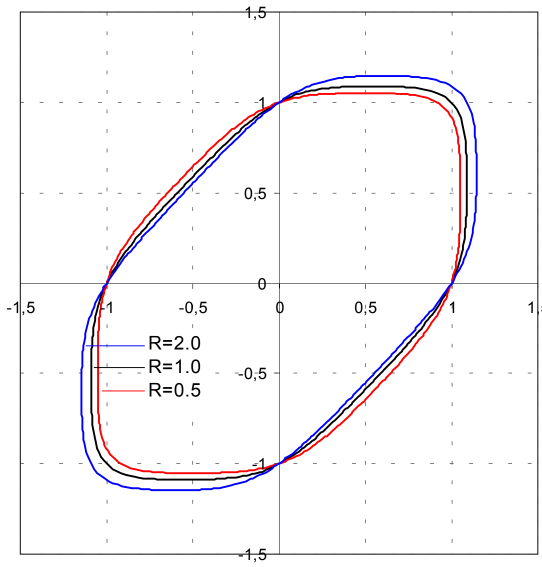
5.2 Yield criteria for normal anisotropy

In 1948 Hill[1] presented his classical quadratic yield function for three dimensional, orthogonal, anisotropic plasticity. Especially in its plane stress, normal anisotropic form, it is the, without comparison, most widely used yield criterion for sheet forming applications. The expression for the effective stress is given in Eq. (7) (compare Eq: (1))

$$\begin{aligned}\bar{\sigma}^2 &= \sigma_1^2 + \sigma_2^2 - \frac{2R}{R+1}\sigma_1\sigma_2 = \\ &= \sigma_x^2 + \sigma_y^2 - \frac{2R}{R+1}\sigma_x\sigma_y + 2\frac{2R+1}{R+1}\tau_{xy}^2\end{aligned} \tag{7}$$



(a)



(b)

Figure 3 (a) The Hill'48 yield function, and (b) Hosford's yield function for m=8

Hill's yield surfaces for different values of the anisotropy constant R are shown in Fig. 3a. Hill's yield condition has proved to yield good result for mild steel. However, for high strength steel qualities, and especially for aluminium, it fails in providing satisfactory results.

Another yield function, suggested by Hosford[20], has been shown to yield excellent fit to crystallography-based yield surfaces for values of the exponent m in the range 6-8 (see Eq. (8)). Hosford's yield surface for different values of R is displayed in Fig. 3b. The expression for the effective stress is shown in Eq. (8). Hosford's criterion, can be shown to reduce to Hill's quadratic yield function in the special case when R=2.

$$\begin{aligned}&\bar{\sigma}^m = a_1\left(|\sigma_1|^m + |\sigma_2|^m\right) + a_2|\sigma_1 - \sigma_2|^m \\ &a_1 = \frac{1}{1+R}; \qquad a_2 = \frac{R}{1+R};\end{aligned} \tag{8}$$

It has been observed in numerous experiments that the shapes of the yield surfaces for metallic materials can be found somewhere between two extremes, represented by the yield surfaces of Tresca and von Mises, respectively. It is interesting to note that Hosford's yield surface, for increasing value of the exponent m, approaches Tresca's yield surface.

5.3 Yield criteria for planar anisotropy

A great number of yield criteria for planar anisotropy have been presented. Here a couple of the most well known criteria will be presented.

Barlat and Lian[21] have proposed a yield criterion, which can be viewed as a generalization of Hosford's criterion for normal anisotropy to the more general planar anisotropic case. An advantage of this criterion is that the anisotropy properties can be represented by parameters obtained in simple standard tests. The effective stress in the Barlat-Lian criterion can be expressed as

$$2\bar{\sigma}^m = a\left|K_1 + K_2\right|^m + a\left|K_1 - K_2\right|^m + c\left|2K_2\right|^m \tag{9}$$

$$K_1 = \frac{\sigma_x + h\sigma_y}{2};$$

$$K_2 = \sqrt{\left(\frac{\sigma_x - h\sigma_y}{2}\right)^2 + p^2\tau_{xy}^2};$$

Note that this expression corresponds to Hosford's yield criterion with the normal stress in the y-direction weighted by a factor h, and the shear stress weighted by a factor p. The material constants a, c, and h can be expressed in terms of R-values in the 0^o- and 90^0-directions. The parameter p cannot be calculated directly, but has to be solved for iterativly from an equation involving the anisotropy parameter R in the 45^o direction. An extension of Eq. (9) to three dimensional, orthotropic plasticity has been proposed by Barlat et.al.[22].

Karafillis and Boyce[23] used the "mapped stress tensor" concept to derive a three dimensional, anisotropic material model. In their model they introduced a linear transformation tensor acting on the stresses σ_{ij} in the real material. The transformation tensor "weights" the different stress components of the anisotropic material. The weighted stresses can be considered to act on a corresponding, fictitious, isotropic material. For the case of an isotropic material, the transformed stress tensor will be equal to the deviatoric stress tensor acting on the real material. The transformed stress tensor is called the "isotropy plasticity equivalent (IPE) deviatoric stress tensor". The transformation can be written

$$\tilde{S}_{ij} = L_{ijkl}\,\sigma_{kl} \tag{10}$$

The isotropic yield function, corresponding to the stress state $\tilde{S}_{ij}$, is prescribed and the elements of the transformation tensor, describing the anisotropy of the material, are determined from a suitable set of experiments. In the present case the isotropic yield function is general enough to able to describe both the lower (Tresca) and the upper bounds, existing for isotropic yield functions.

6 PREDICTION OF FORMING DEFECTS

6.1 Introduction

Current methods and codes for sheet metal forming are quite successful in predicting parameters, which are related to the deformation of the sheet material, for instance strain distributions, thinning, and draw-in of the blank edge. The forces acting in the interfaces between the blank and the tools can usually also be predicted with satisfactory precision. However, some forming defects like rupture, springback, and surface deflections, can unfortunately not always be predicted with the desired level of accuracy.

Much of the modern research on sheet metal forming simulation is consequently devoted to these particular issues. It is, however, the object of the current presentation to describe the methods that are in practical use today to predict these defects.

6.2 Prediction of rupture

The risk for rupture in the material is usually evaluated by means of a so-called Forming Limit Diagram (FLD). This is an experimentally determined curve in the principal strain plane, showing combinations of principal strains leading to rupture. In these experiments rectangular sheets with different widths are stretched over a hemispherical punch until rupture occurs. Every single width of the sheet specimen corresponds to a unique linear strain path up to failure, and gives one point on the FLD.

The risk for failure is normally evaluated in the post-processing of the results from the simulation code, but the FLD can also be built in the material model as failure criterion. Critical zones in the formed part can be detected by visualising a “failure index”, defined as

$$c = \varepsilon_1/\mathrm{fl}(\varepsilon_2) \tag{11}$$

where $\mathrm{fl}(\varepsilon_2)$ is the forming limit curve viewed as a function of the minor principal strain. This index indicates rupture when $c \geq 1$. In Fig. 4 this index is visualised as colour fringes for a formed panel. As can be seen a critical area is detected and is marked by red colour.

If the strains in the middle surface of every element in the critical area are plotted in a principal strain diagram this results in a diagram like the one in Fig. 5. Some strain points in this diagram are situated above the forming limit curve, indicating material failure in the corresponding elements.

A lot of criticism can be raised against the use of FLDs as failure criteria. There is first of all a big uncertainty about the exact appearance of the forming limit curve itself, since this is highly dependent on the test procedure. Secondly, the FLD is created from linear strain paths, while the strain paths leading to failure in the actual forming operation very seldom are linear. It has in fact been shown that the limit strains are highly dependent on the strain path. This deficiency of the conventional failure evaluation procedure is especially evident for multistage forming processes.

Current research on methods for rupture prediction is therefore concentrated on finding procedures that can handle nonlinear or broken strain paths. Stress based forming limit concepts and damage mechanics models are example of attempts in that direction.

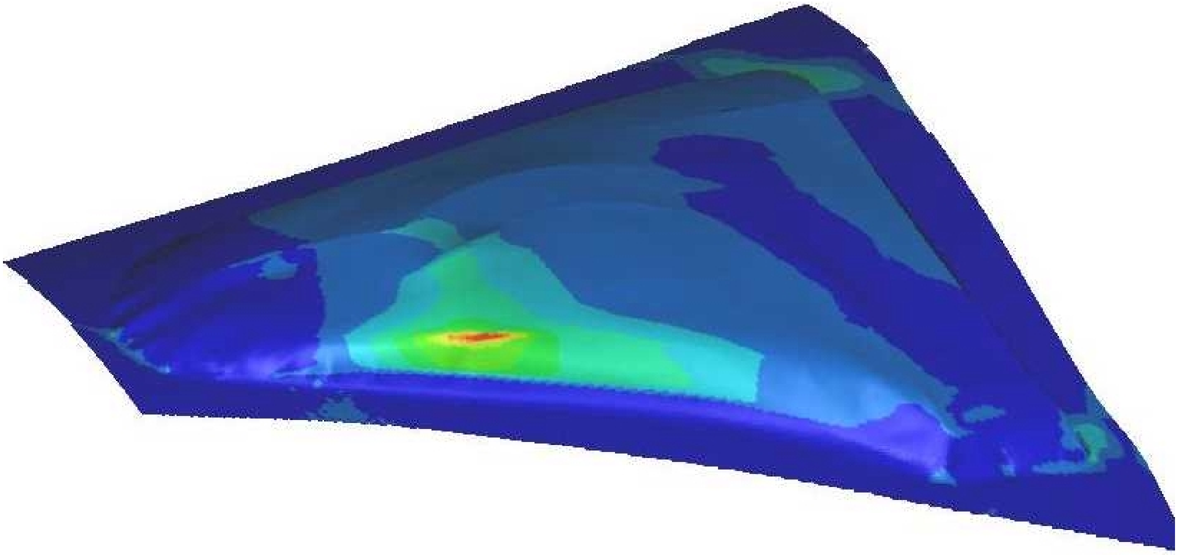


Figure 4 Colour fringes indicating ”failure index”. Red colour indicates rupture.

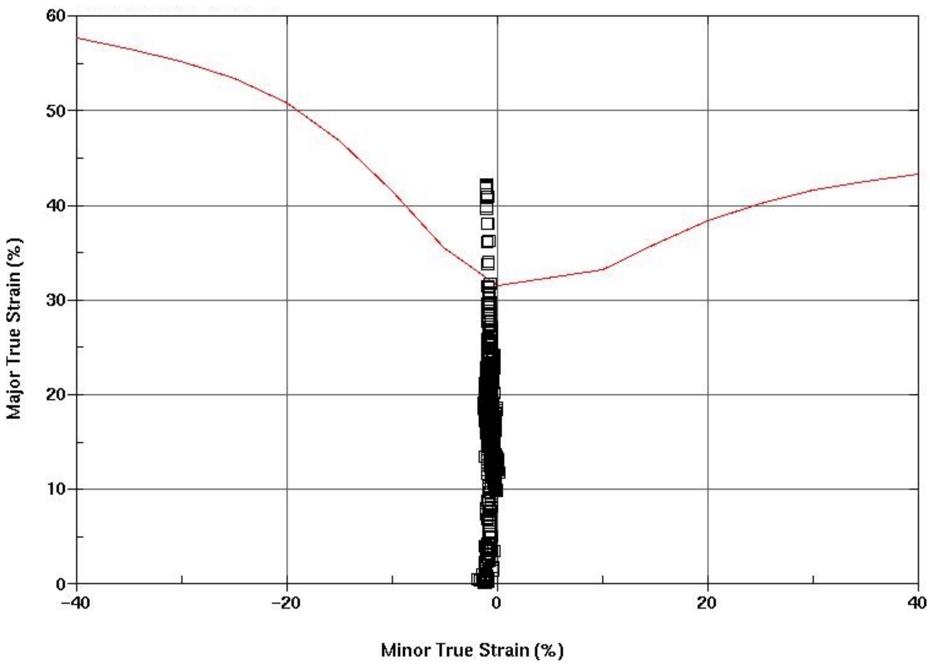


Figure 5 Forming limit diagram for the critical zone visualised in Fig. 4.

6.3 Springback

Springback analyses by means of dynamic, explicit codes must, in contrast to forming simulations, be performed in a real time scale. The normal procedure is then to apply boundary conditions so that rigid body motions are prevented, remove the tools instantaneously, apply a suitable amount of damping, and then let the work piece vibrate freely until a static equilibrium is reached. The drawback of such a procedure is that, first of all, it is very difficult to estimate what amount of damping should be applied without first doing a separate eigenfrequency analysis, and, secondly, that the time for reaching a static equilibrium many times can be several times longer than the time needed for the actual forming operation. For this reason the static-implicit method is preferred for springback analyses.

A condition for an accurate springback analysis is that the calculated stresses after the completed forming simulation are correct. It has, however, been shown that it is much more difficult to obtain accurate stresses than accurate strains.

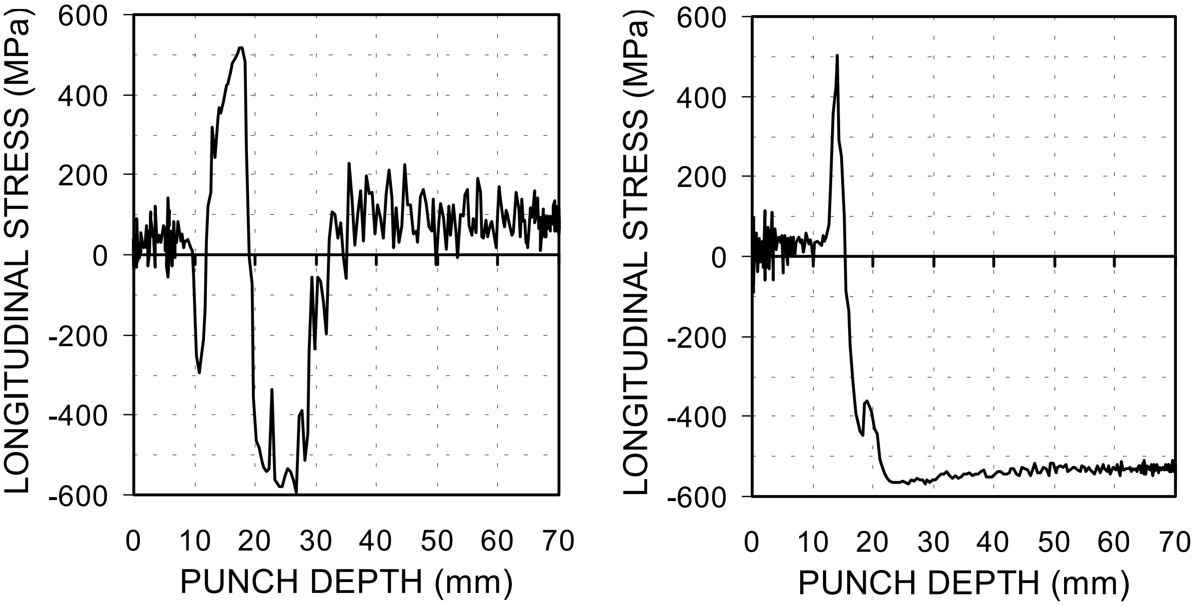
In connection to the NUMISHEET'93 conference a benchmark test was set up, which aimed at letting the participants determine the springback in a deep-drawn, U-shaped sheet strip, both experimentally and/or numerically. The numerical benchmark results were, however, very disappointing, showing a great scatter among the different participants. Most codes seemed to strongly underestimate the springback.

Later on the author and colleagues, Mattiasson et.al.[24], did reanalyse the present problem in order to find out the causes of the inaccurate springback predictions. Especially the influences of various model parameters on the resulting stress state after completed forming were studied. In Fig. 6 the longitudinal stress history during the forming operation in a point on the outer surface of the sheet strip is displayed. The influence of the mesh size in the sheet is studied, and results are shown for element sizes 3.0 mm and 0.5 mm. For the larger element size a pronounced relaxation of the stresses, after the point in question has left the draw radius, can be observed. The results for the finer mesh, which represent a converged solution with respect to the element size, do not show this stress relaxation. In Fig. 7 the geometry of work piece after springback is displayed for various element sizes.

The observed stress relaxation phenomenon could be explained by the small variations in strains in the vertical part of the work piece, which are caused by the basically flat elements in the sheet slipping over the draw radius.

The referred study showed, thus, that the main reason for the inaccurate springback results was the use of a too coarse mesh in the sheet. In fact, an extremely fine mesh was needed in order to get a converged solution. There were, however, a number of other factors that had substantial influence on the results. For instance, it was shown to be very important to include the Bauschinger effect in the modelling of the material hardening. Furthermore, the fictitious process time used in a dynamic-explicit method should be at least twice the time normally used, when an accurate solution for strains is of primary interest.

Even though the above observations have been considered, the agreement between measured and calculated springback for complex parts have in many cases been poor. This indicates the problem of springback is not yet fully understood, and that this should be a focused are for current research.



(a) (b)

Figure 6 Stress history at a point on the outer surface of a U-shaped deep drawn sheet strip (from Mattiasson et.al.[24]). (a) Element size 3.0 mm, (b) Element size 0.5 mm

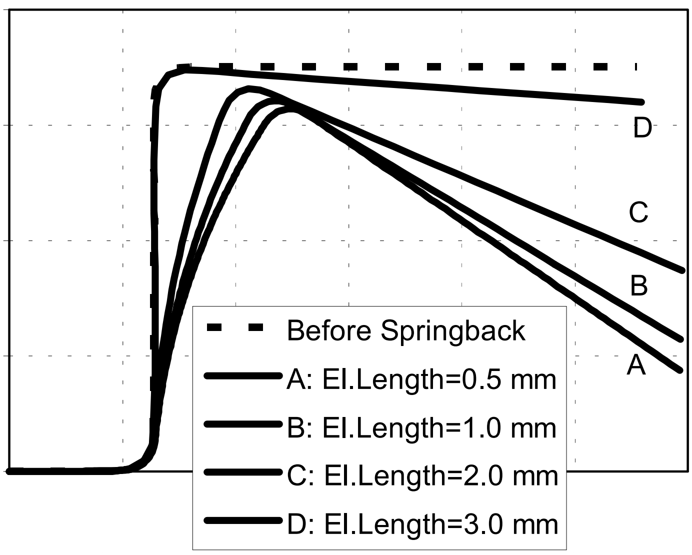


Figure 7 Geometry after springback for different element sizes (from Mattiasson et.al.[24])

7 SUMMARY

Sheet metal forming simulation is today used by routine by most car manufacturers and major tool makers. Simulations of varying complexity are performed in different phases of the forming process development. One-step codes are mainly used in the early product design stage to evaluate manufacturing feasibility. The advantage of one-step codes is the short turn around time. Computing time as well as the time needed for data preparation are considerably shorter than for incremental codes.

More thorough analyses are performed by means of incremental codes to support the die and process design. Today the software market for this type of codes is dominated by codes based on the dynamic-explicit method. The computing time for complex production parts is typically several hours.

There are some areas of sheet forming simulation for which there exist particular needs for further research and development. This concerns especially detection and evaluation of certain types of forming defects, such as rupture, springback and surface deflections.

REFERENCES

[1] R. Hill, "A theory of the yielding flow of anisotropic metals", *Proc. R. Soc. London*, **193A**, p281 (1948)

[2] E. Onate and C. Agelet de Saracibar, "Alternatives for finite element analysis of sheet metal forming problems", *NUMIFORM'92* (Eds.: Chenot, Wood and Zienkiewicz), Sophia Antipolis, A.A. Balkema (1992)

[3] N.-M. Wang, "A rigid-plastic rate sensitive finite element procedure for modeling sheet forming processes", In *Numerical Analysis of Forming Processes* (Eds.: Pitman, Zienkiewicz, Wood, and Alexander), John Wiley & Sons, p 117 (1984)

[4] C.H. Toh and S. Kobayashi, "Deformation analysis and blank design in square cup drawing", *Int. J. Mach. Tool Des. Res.*, **25,** p 15 (1985)

[5] Y.Geramain, K. Chung, and R.H. Wagoner, "A rigid-viscoplastic Finite Element program for sheet metal forming analysis", *Int. J. Mech. Sci.*, **31**, p 1 (1989)

[6] R. Sünkel, C. Pautsch, K. Roll, R. Toderke, F. Fuchs, and V. Steininger, "Numerical simulation of metal forming processes in industry using INDEED", *NUMISHEET'96* (Eds.: J.K. Lee and R.H. Wagoner), Dearborn, Michigan (1996)

[7] S.C. Tang and Y. Hu, "Quasi-static analysis of sheet metal forming processes on a parallel computer", *NUMIFORM'95* (Eds.: Shen and Dawson), Ithaca, N.Y., A.A. Balkema (1995)

[8] E. Nakamachi, "Sheet forming process characterization by static-explicit anisotropic elastic-plastic finite element simulation", *NUMISHEET'93* (Eds.: Makinouchi, Nakamachi, Onate and Wagoner), Isehara, Japan (1993)

[9] M. Kawka and A. Makinouchi, "Finite element simulation of sheet metal forming processes by simultaneous use of membrane, shell and solid elements", *NUMIFORM'92*

(Eds.: Chenot, Wood and Zienkiewicz), Sophia Antipolis, A.A. Balkema (1992)

[10] J.O. Hallquist, B. Wainscott, and K. Schweizerhof, "Improved simulation of thin sheet metal forming using LS-DYNA3D on parallel computers", *NUMISHEET'93* (Eds.: Makinouchi, Nakamachi, Onate and Wagoner), Isehara, Japan (1993)

[11] E. Haug et.al., "Numerical simulation of industrial sheet forming processes with PAM-STAMP", *4th Cars/Trucks Symposium*, Schliersee (1995)

[12] C.D. Mercer, J.D. Nagtegaal, and N. Rebelo, "Effective application of different solvers to forming simulations", *NUMIFORM'95* (Eds.: Shen and Dawson), Ithaca, N.Y., A.A. Balkema (1995)

[13] F. Aberlenc, J.-L. Babeau, and P. Jamet, "OPTRIS: The complete simulation of the sheet metal forming", Ibid.

[14] J.L. Batoz, Y.Q. Guo, and F. Mercier, "The inverse approach including bending effects for the analysis and design of sheet metal forming parts", Ibid.

[15] M. El Mouatassim, B. Thomas, J.-P. Jameux, and E. Di Pasquale, "An industrial finite element code for one-step simulation of sheet metal forming", Ibid.

[16] W Kubli and J. Reissner, "Optimization of sheet metal forming processes using the special-purpose program AUTOFORM", *NUMISHEET'93* (Eds.: Makinouchi, Nakamachi, Onate and Wagoner), Isehara, Japan (1993)

[17] N.-M. Wang and B. Budiansky, "Analysis of sheet metal stamping by finite element method", *J. Appl. Mech. Trans. ASME*, **45**, p73 (1978)

[18] S.C. Tang, R. Ilankamban, and P. Ling, "A finite element modeling of the stretch-draw forming process", *SAE-paper* 880527 (1988)

[19] A. Honecker and K. Mattiasson, "Finite element procedures for 3D sheet forming simulation", *NUMIFORM'89* (Eds.: Thompson, Wood, Zienkiewicz and Samuelsson), Fort Collins, A.A. Balkema (1989)

[20] W.F. Hosford, "Comments on anisotropic yield criteria", *Int. J. Mech. Sci.*, **27**, p423, 1985.

[21] F. Barlat and J. Lian, "Plastic behavior and stretchability of sheet metals. Part I: A yield function for orthotropic sheets under plane stress conditions", *Int. J. Plasticity*, Vol 5, pp 51-66 (1989)

[22] F. Barlat, D.J. Lege, and J.C. Brem, "A six-component yield function for anisotropic materials", *Int. J. Plasticity*, Vol 7, pp 693-712 (1991)

[23] A.P. Karafillis and M.C. Boyce, "A general anisotropic yield criterion using bounds and a transformation weighting tensor", J. *Mech. Phys. Solids*, Vol 41, 12, pp 1859-1886 (1993)

[24] K Mattiasson, A Strange, P Thilderkvist and A Samuelsson, "Simulation of springback in sheet metal forming", *NUMIFORM'95* (Eds.: Shen and Dawson), Ithaca, N.Y., A.A. Balkema (1995)