Simulation of Sheet Metal Forming – Necessary Developments in the Future[1]

Prof. Dr.-Ing. Karl Roll

Daimler AG; Werk Sindelfingen, PWT/VAS, Germany

Email: karl.roll@daimler.com

Abstract:
The industrial forming simulation used today is the result of a development process during the last two decades. A main focus of this process was the continuous optimization to characterize the material properties. The presentation will give an overview of the simulation technology challenges arising from the need to improve bridging of the gaps between material data and computational technology. Furthermore, an integrated technology pathway from the perspective of simulation requirements, material characterization needs, simulation technology innovations, and impacts of those requirements on the accuracy of practical applications in the automotive world is discussed. The presentation will feature industrial examples to reinforce the topics of the paper.

Keywords:

Forming simulation, materials characteristics, materials models.

[1] Keynote Paper NUMISHEET 2008; Interlaken, 1.-5. September 2008

1 INTRODUCTION

Until a few years ago, the design of metal-forming tools was mostly based on knowledge gained through experience, and designing the optimum tool often required a protracted and expensive trial and error process. Today, even in the early design phases, simulations of sheet-metal parts forming processes are performed using finite element methods. The most important goals in this process are verification of the manufacturability of the sheet-metal parts and obtaining vital information on optimum tool design.

Today, simulations in the automobile industry are as a rule restricted to the sheet-metal forming process, which is where the greatest changes in shape take place. For a typical sheet-metal body in the course of fabrication, the first forming step is the deep drawing process. Many commercial finite element programs are today capable of reliably forecasting possible crack and tear formation and the sheet thickness and form changes of conventional sheet-metal materials. In the past ten years, a number of studies have been made of sheet-metal forming process simulation [1,2,3], based on a very broad range of approaches, where, in addition to implicit and explicit processes, what are known as “one-step procedures” (often also known as inverse procedures) are also on the market.

Different metal-forming simulation programs are used, depending on the chronological position of the operation in the production process. They differ primarily not only in terms of computing time and user friendliness, but also in the theories underlying them and in the quality of the results. As a rule, it is inverse, explicit, and implicit simulation procedures that are used. Today, sheet-metal forming simulation is used as follows [4]:

- Inverse programs are used to gain a rapid overview and a rough verification of the manufacturability, and for pre-optimization.
- Depending on the formed sheet-metal part, both implicit finite element programs with membrane or shell elements and explicit programs with shell elements are used to optimize the tool and the process.

The use of sheet-metal forming simulation is no longer restricted to verification of manufacturability, although this continues to be the most important result, and simulation is increasingly used to optimize the first forming stage. Up to 30 computing runs are made for complex body parts, with the formed metal parts being optimized to counteract two failure phenomena: tearing and wrinkle formation. In exceptional cases, distribution of sheet thickness is improved.

Factor	Reality	Simulation
Production stroke rate	Not constant	Not in the model
Machine	Elastic	Not in the model
Tool	Elastic	Rigid
Characteristics of the direction of draw	Not constant	Not mapped
Coefficient of friction	Not constant	Constant
Temperature	Not constant	Not in the model
Topology of blank holder surface	Not constant	Not in the model
Material	Complex	Simple models
Material characteristics	Not constant	(Not) constant

Table 1: Comparison in reality and in the simulation model

It should be noted that large savings have already been achieved by the introduction of process simulation in designing shaped sheet-metal parts. These savings are conferred mainly by the more rapid development of tools and by a dramatic shortening of the trial and error process for series production tools. The whole process of laying out the forming method and the designing and producing sheet-metal tools can be abbreviated, while development time is similarly shortened. In recent years, tool development and production time has been reduced by about 50% by simulation, and a further 30% reduction over the next few years appears realistic. The simulation of forming has today reached a stage where its results can be fed directly into the press tool digital planning and validation process. Thus today, starting from the design model, through a number of process steps as far as the actual design of the press tool, already computer-assisted, the production of a component in the line of presses can be simulated before a first prototype is built.

2 Necessary developments in simulation engineering

When we compare the physical reality of a sheet-metal forming process with the simulation models used today, we recognize that significant factors in the forming process are incorrectly described in the simulation model, or even not at all [5].
Table 1 gives an overview of the reality and of simulation models in use today. Approximation of the simulation model is sufficient to yield good forming simulation results. However, for a more accurate calculation of springback, the model must be amplified at various points.
To date, the effects of the forming process, tool surface quality, and temperature have not been specified in detail. In the numerical computation of springback-related form changes using sheet-metal forming simulation, numerical factors as well as geometrical, materials, and technological characteristics affect the calculation results. For example, the material model used and the hardening rule for describing metal behavior, the finite elements used to describe the deformable sheet, or the friction model used all have great influence. Numerical values, such as the time discretizing chosen, also have an effect.

3 Development of forming simulation

The historical development of methods and procedures for simulation of metal-forming processes can be divided into two temporal segments. In the segment before the economic exploitation of the computer (before 1960, roughly), simulation procedures were predominantly empirical. The fundamentals for the determination of forming forces, material flow, and failure phenomena were created by systematic experimental investigations of parameters (partly based on similitude mechanics). The theoretical simulation methods were based chiefly on the elementary plasticity theory (although the fundamentals of the higher plasticity theory had been available since about 1930) for the pre-calculation of forming forces and rough calculation of stresses. Also applied to a lesser extent were the procedures of slip-line theory, the upper boundary method, and the weighted residual method. [6].
It was decisive for this approach that it dealt with simple, closed analytical calculation rules, or perhaps also that these theories could be applied using graphic measures (slip-line theory). Given the complexity of the mechanics in the forming zone, theoretical simulation procedures were associated with broad assumptions and simplifications, which in turn severely weakened the expressiveness of the predetermined values. The materials models underlying these models are those of Tresca and von Mises. The characteristic values required to describe yield surfaces are determined from single-axis tensile testing.
The arrival of computers sparked a dramatic revolution in the theoretical simulation of forming processes. Now, the available approaches of higher plasticity theory could be applied. These approaches were reformulated so that the numerical methods derived from them could be simply executed by computer. Development took off with finite differences methods, continued with error compensation procedure, and finally concluded around 1970 with the application of the finite element method (FEM) as the standard numerical procedure today. It should, however, be emphasized that the last-mentioned procedures are only “numerical tools” for application of the plasticity theory approaches [7].

4 Refinement of materials models

It was very soon evident that the simple materials models of Tresca and von Mises were inadequate for describing materials behavior (normally, only the von Mises model is used in simulation). The models were therefore extended in order to describe effects such as anisotropy, kinematic hardening, and so on. Tensile testing is no longer sufficient to determine the required materials characteristics. Other tests, some of them very demanding, such as cross-tensile testing, hydraulic cupping testing, torsion testing, and so forth, are required. Some of the characteristic values required in certain material models can be determined only very imprecisely or even not at all (for example, tests in which only shear stress occurs). Until now, the basis of all expanded models has been the normal rule, that is, during plastic deformation, the deformation gradient is normal (vertical) to the yield surface and the convexity of the yield surface. The material behaves incompressible during plastic deformation.
These assumptions no longer appear to have unlimited validity for high-strength and super high-strength steels, for instance. With TRIP- (transformation-induced plasticity) steels, there is an alteration in structure during forming, since austenite is transformed into martensite, changing the volume and the structure, and hence the hardening of the material also depends on the stress condition during forming [8]. Today, there is still no material model that describes this material behavior with satisfactory accuracy. Initial attempts have been made, but not all observed effects can be described [9].
A critical point for the possibility of modeling material behavior is the numerical approximation of real relationships within the FE system. In this case, not much is known about which parameters are im-

portant or unimportant, and which directly affect the forces generated and thus the overall deformation behavior of the metal sheet.
Current practice today often consists of an estimation of unknown parameters, the transfer of parameters known from a similar material, or the choice of a theoretical description that impermissibly oversimplifies the real relationships. In almost all cases today, we start out with a modeling of material behavior that is not fully correct. Depending on the analytical result desired, these modeling errors can significantly affect the solution – the extremes here being the sensitive springback calculation, and the relatively robust determination of changes in sheet thickness. These are the reasons that an adequate description of material behavior and tribological factors must be chosen: to improve the quality of the result, relative to the problem under analysis and the (multi-) physical effects operating in it. An essential precondition for the practical use of this most complex set of laws is the possibility of being able to determine the parameters required with a reasonable expenditure of experimental effort. What follows, therefore, is a discussion of the effect of some important material characteristic values on the result of a forming simulation.

4.1 Effect of the Module of Elasticity

Springback depends primarily on the E-module and yield stress of the material. Springback increases as yield stress increases (example: highstrength steel) and as elasticity module decreases (example: aluminum). The E-module changes during forming. This dependency of the E-module on plastic elongation has been experimentally determined for few materials so far, and cannot be described in the programs commercially available today.
The change in E-module during plastic forming is significant only for the simulation of springback [10]. The actual implications are so far still unclear. With steel, for example, we always simulate using 210 GPa, and with aluminum, 72 GPa, and yet the results fit the measured values with satisfactory accuracy.
Another current problem in forming simulation is approximation of flow stress. The standard tensile tests used today can capture the relationship between elongation and flow stress up to uniform elongation (about 20%, depending on the particular material). In actual forming processes however, and particularly with deep drawing, very much higher degrees of forming occur. Different procedures are used to extrapolate the experimentally determined flow curves in these elongation regions [11, 12]. Clearly, a falsely extrapolated flow curve can strongly affect the simulation result of the forming process and thus the accuracy of the springback [13].
Selection of the correct extrapolation procedure depends on the material. For higher degrees of forming such as uniform elongation, it is therefore necessary to have materials test procedures with which flow curves can be experimentally determined. These test methods may include the torsion test, the hydraulic bulge test, or special tensile tests.

4.2 Influence of anisotropy

Sheet metal behaves anisotropically as a result of the rolling mill production process. This anisotropy must be described if an accurate simulation is to be obtained. The first work describing anisotropy was performed by Hill in 1948. That description has since become known as the Hill48 Model. Since then, it has been modified many times (e.g., Hill90, Barlat91, Barlat96, Banabic2000, and so forth) [14].
Not all the anisotropy models known in the literature have been implemented in commercial programs. Usually, to describe anisotropy, the vertical anisotropic value "*r*" is measured. This value describes the relationship between forming in the direction of thickness and forming in the plane of the sheet. An *r* value of 1 signifies isotropic behavior, at *r* values > 1 forming takes place predominantly in the plane (which is good for deep drawing), and at *r* values < 1 forming primarily occurs through a reduction of sheet thickness. Normally, the *r* value is experimentally determined by tensile testing in which the change in width is measured by the tensile test rod. Today, we have a number of procedures for determining the *r* value.
Anisotropy also changes as a function of plastic elongation. Today, a mean value is normally used, since this dependency of anisotropy on plastic elongation has not yet been implemented in any commercial simulation program. The effect of plastic elongation on the distribution of anisotropy is appreciable. For more accurate calculations, this effect must therefore be taken into account.

4.3 Effect of yield loci and the hardening model

To some degree, the models disclosed in the literature enable a very good description to be made of the anisotropic behavior of materials, but "correct" models for the material must be determined by additional testing.

The application of appropriate material and frictional laws is of central importance for the correct simulation of springback [13]. Here, both the choice of the correct yield locus and consideration of kinematic hardening, that is, the displacement of the yield surfaces, are critical. Only the correct yield locus, and the correct ratio of isotropic and kinematic hardening in the applied materials law, enable the load change to be represented correctly, as occurs, for example, in drawing across small draw radii. At present, materials constants for simple models are determined by tensile testing. Since bending phenomena cannot be correctly captured with these simple materials models, the constants for more complex models must be specified.
The theoretical differences of the different hardening models are very large, but what does this mean for the forming simulation? It must be clearly stated that the differences are important only in some circumstances. On the one hand, the very first thing is that the material must display a markedly anisotropic hardening, and this effect is greatest in steels exhibiting induced plasticity, but is not limited to them. As a rule, various structural changes and dislocation nodes that are stress-dependent occur in these steels during forming. On the other hand, there must be different stress conditions and high enough plastic strains present in the formed part to allow the differences to have an effect. The different yield curves move away from each other as the plastic strain increases. Furthermore, the effect of the isotropic-kinematic hardening is also more marked with higher plastic strains [15, 16, 17].
The results given in [18, 19] are based on a study in which the anisotropic hardening model was used. In addition to the pure tension test, data taken from the hydraulic bulge test, stacked compression test, compression test in the plane of the sheet material, and an experiment under shear load were used to compare the different yield properties of the material in the stress combinations. All computed results were compared with those from actual experiments.
To verify the simulations, forming tests were carried out on a very highstrength steel with induced plasticity. An experimental forming tool that is also used in Daimler AG’s material release process was used here. The tool offers e.g. the possibility to measure the actual stamping force during forming. By applying a regular point grid, it is possible to ascertain the plastic strains on the surface of the finished part. The GOM company’s ARGUS system was used for the tests. A compromise had to be reached to allow for adjustment to the simulation. The simulation with membrane or shell elements runs all calculations in the middle of the sheet, the neutral fiber, or membrane plane.
Deep-drawing parts with a drawing depth of 40 mm were produced and then simulated with version 9.71 of LS-DYNA Version 3.34 of the user subroutine MFGenYld was used [20]. The yield locus was always described using the Hill model, hardening was assumed to be once isotropic, which is standard procedure, and once anisotropic.
The stamping force and the major, minor, and equivalent plastic strain were used as the comparative sizes between measurement and simulation. The stamping force and the major and minor plastic strain are directly measured sizes, whereas the equivalent plastic strain is calculated from the two other plastic strains.
As well as plastic strains, one criterion for the quality of result of a forming simulation is the forming force applied to the stamping machine. This force was measured during the forming tests. As mentioned in the section above, the differences between isotropic and anisotropic hardening show only at higher plastic strains, which can be identified by the fact that the force drops noticeably with effect from a drawing depth of 15 mm. It can be seen that the anisotropic calculation reflects the force maximum best of all, but none of the simulations reflect the exact force progression.
The distributions of the major and minor plastic strain are compared. The result is not clear. As regards the major plastic strain, the areas around the die radius and the curved frame areas are better illustrated by the anisotropic calculation. At the base and stamping radius, the isotropic calculation is much more like the results obtained from measurement. The representation of the minor plastic strain does not show any significant differences; the hardening modeling does not appear to have any influence here. The equivalent plastic strain shows, as a size calculated from the major and minor plastic strain, a similar picture to the major plastic strain, and is therefore not shown here. Part base and stamping radius are described more precisely as isotropic, and frame and die radius as anisotropic.
The conventional simulation of deep drawing of modern highstrength steels predicts only inadequately in some cases the plastic strains in the finished part. To improve the result at these locations, the consideration of the anisotropic hardening of the material represents a very promising option. In many areas, a more precise prediction is obtained as a result, but in some places the result does not improve at all and even gets worse. There are many reasons for this. The treatment of the experimental data is extensive and is mostly based on non-standardized tests to determine them. Furthermore, only a small number of experiences with this method of description of steel hardening were gathered, thus making the verification of the compiled material cards difficult, since the earlier verification calculations are no longer adequate.

There is still a great deal of potential in the processing of the experimental data, as there is in the actual determination of the input values. In the past ,the fundamental input value for the forming simulation was the uniaxial tension test. This test has been standardized for decades, and in the meantime the result is largely independent of the testing center. In contrast, modeling of the anisotropic hardening requires some experiments that are still virgin territory experimentally and are far removed from standardization. However, this is how the unfortunate position can arise that the results for one and the same material can vary from test lab to test lab. The experiments also frequently represent an approximation to the stress condition that one would like to depict. This means, for example, in the hydraulic bulge test, the currently preferred experiment to record a yield curve under equal biaxial stress, that the biaxial tension stress is always overlaid with a bending stress that must be worked out with the help of theoretical assumptions, which, in turn, are of course a simplification. It is also difficult to select a suitable test for recording a yield curve under pure shear. There are indeed a range of different tests, but in almost all of them the shear stress is overlaid with a not inconsiderable tension stress component, even for small plastic strains.
The behavior of the model under load changes is also not satisfactory, and experimental calculations to take account of the kinematic hardening show that this is not always correctly reflected. The anisotropic effects have also been partly overestimated, and this shows, for example, that an excessively large forming is calculated in areas with tension-compression loads in which the plastic strain is too low in the isotropic calculation. These problems stem primarily from the lack of experience with simulating sheet-forming processes that take these effects into account. Further studies will be required to determine beyond doubt where the problem finally lies, whether it is the input data, the application of the model, or perhaps the model itself. Overall, it can be said that the method presented here has great potential, but to exploit that potential will require some more effort and extensive development work. The current position does not yet allow for productive use in the industrial, daily forming simulation, where it is vital to calculate many parts in a short time. However, if the concept is pursued, a powerful and efficient facility will then be available for simulating modern, very highstrength steels with the same high result quality that is currently achieved with mild steels.

5 Effect of friction

Friction is another important process parameter. In simulation today, it is assumed to be constant, and is described by a coefficient of friction, μ. In reality, it depends on [21].

- the surface fineness of the sheet metal material,
- the surface fineness of the tool surface,
- the actual local surface pressure,
- the actual relative speed between the sheet segment concerned and the tool surface, and
- the temperature in the active interface.

Today, there are still major uncertainties in the description of the friction-contact situation. Simulations calculations show that a small change in the friction-contact situation can have a significant effect on springback. In the past, highly developed friction contact models have been developed. The parameters of these models can be determined only with major effort. As tools are fine-tuned, the contact area is especially affected, producing significant changes in contact forces, and thus in frictional forces. The changed contact surfaces resulting from the refinement of the tools are generally not considered further in the simulation. The contact situation, which is entered as an input value in the friction-contact model used, can therefore be only insufficiently specified. The errors made here may exceed the errors of the friction-contact model by several orders of magnitude. The frictional surfaces assumed in the programs no longer correspond to reality, since the tools in the simulation models are assumed not to have changed. A further weakness of the simulation programs is the punctiform contact. In the simulation programs, there is contact at nodal points only, and the frictional stresses calculated from the contact forces also act at the nodal points only. Therefore, the calculated distribution of frictional shearing stresses in the active interface does not reproduce the actual distribution of the frictional stresses affecting the frictional surfaces. New contact models must therefore be developed to improve the contact situation, and the first beginnings have been made. For a better description of friction in the simulation programs, consideration must be given to the elastic properties of the tool, since only then can an accurate calculation of contact stresses in the active interface be made [22, 23].

6 Failure models

In sheet-metal forming simulations, the forming limit diagram (FLD) is generally used as a basis for predicting crack failures. The forming limit diagram is experimentally established for each material through bulge testing, where the main elongations when cracks appear are measured and plotted on a diagram. Changes in the geometry of the sheet metal generate different combinations of elongations, so that forming limit diagrams can be determined for every material. In the simulation, the elongations are now compared with experimentally determined values, and, if the limiting curve is exceeded, it is assumed that failure from cracking will take place, and so the tool or the forming method must then be modified so that all the elongations that occur will lie below the limiting curve, and the material will not fail during forming. The diagram experimentally determined in the manner described above is correct for linear elongation paths only; despite these restrictions, evaluations based on the diagram can give usable results for conventional deep-draw steels, and, with some limitations, for aluminum alloys as well.

With high and very highstrength steels, however, there are wide differences between experiments and simulations. With increasing requirements on crashworthiness, and lightweight car body structures being a central issue in future automotive development, the use of highstrength steel qualities has become wide-spread in modern cars. Since these materials often show significantly lower ductility than conventional steels, it is of great importance to precisely predict failure under crash loading conditions. Hence, constitutive models in crashworthiness applications – as for instance the Gurson model which is applied widely at Daimler AG – need to be initialized with correctly determined internal variables mapped from a corresponding sheet metal forming simulation. [24]

Here, two principle ways could theoretically be used: on the one hand, different understanding of damage and failure in crashworthiness and sheet-metal forming applications may be unified by a generalized incremental stress state dependent damage model (GISSMO). This approach can be considered as an attempt to replace the currently used FLD for the failure description in forming simulations. Furthermore, an advantage would be the inherent ability to account for load-path dependent failure behavior.

On the other hand, the already applied Gurson model in crash simulations may be fed by an estimation of the internal damage value from the forming simulation. The idea here would be to perform the forming simulation with a state-of-the-art anisotropic material model such as, e.g., the Barlat model, with a simultaneously executed estimation of Gurson's damage evolution law.

An appropriate correction of the diagram makes it possible to consider non-linear elongation paths that occur during forming, but the material fails earlier in the simulation than it does experimentally [17].

For a more accurate analysis of material failure, models are required that can identify failures resulting from a criterion that is dependent on the elongation path. A stress-based criterion independent of strain path would be desirable. The limiting stress curves determined from the forming limit diagrams are in no case path-dependent.

The described possibilities for determination and transfer of local pre-damage data from forming to crash simulations are promising potentials to make crack prediction in crash simulations more accurate in the future. Both options proposed, for the combination of a material model for forming simulations (such as Barlat89), with a crash material can be improved by implementing the described extensions to the damage models. As some unintended, but very welcome "side-effect," the damage models also show promising results in predicting ductile failure in forming simulations. The use of these damage models could therefore also lead to an improved failure prediction in forming simulations[24].

7 Influence of Temperature

In the automotive industry, there is a general tendency to choose steels with enhanced strength for structural parts. This trend results from increased lightweight design efforts to satisfy fleet consumption restrictions. Hot-forming and quenching of highstrength sheet steel offers the possibility of improving the component strength and reducing the weight of structural car body parts. The simulation of such processes is different from conventional sheet-metal forming simulations, since the fast cooling of the hot blank during the forming process necessitates a thermomechanically coupled simulation. Thus, a convenient modeling approach must account for both the fast cooling of the blank and the heating of the tool surface in contact with it. The simulation of hot-forming processes has a noticeably higher complexity in relation to conventional sheet-metal forming processes. Therefore, a complex thermomechanically coupled process simulation must be used instead of a conventional forming simulation. In particular, the calculation of the temperature distribution in the sheet metal is important for a sufficiently exact simulation of hot-forming processes. Owing to the high temperature gradient between the component and the tool, the sheet metal cools down quickly at the contact points during the forming operation, leading to an inhomogeneous temperature distribution. Therefore, the elastoplastic material behavior in the simulation over a broad temperature range must be described. In order to be able to calculate the sheet metal temperatures during the total process cycle with sufficient accuracy, it is necessary to consider the heat transfer between the sheet metal and the tool in the simulation. The heat transfer between the sheet metal and tool is determined by the heat transfer coefficient as

well as by the temperature gradient between the sheet surface and tool surface. Therefore, the heating up of the tool surfaces at the contact areas should also be considered in the simulation [25].
In particular, the modeling of the tools differs substantially from conventional sheet-metal forming simulation. Different modeling lugs were discussed, and an efficient method for the modeling of the tools with shell elements with consideration of the tool heating up was presented. The applicability of the method was given at a practical example of use. Additionally, the method using the example of the production process of a B column was applied and compared with measured results [26]. Apart from application with typical hot-forming processes, the thermomechanical coupling becomes increasingly interesting, also for the simulation of cold-forming processes. The increasing trend towards steel materials with higher yield strengths and tensile strengths results in a higher heating-up of the sheet metal due to the higher flow stresses, also during cold forming. The heating-up takes place because of the plastic work dissipated in the forming process. The resulting higher surface pressures at the contact surfaces lead to higher frictional energies, which likewise contribute to the component heating up. An estimation of these effects is possible with a thermomechanically coupled forming simulation [27, 28].

8 Elastic Tools

Today, a standardized simulation of the sheet-metal forming process is already used in the tool engineering and design stages to ensure defect-free production of the sheet-metal component.
It is becoming more and more important to simulate springback and its computer-assisted compensation. The results that have been achieved show that simulation is indeed capable of computing, down to a fairly narrow bandwidth, the geometric changes caused by the springback in a manner that is qualitatively correct and quantitatively precise. The numerically determined deformations provide a useful indication for compensating springback through the use of adaptive tool design, and/or by modifying the process parameters used in the metal forming.
In springback simulation in particular, it has become evident that the currently used simulation models, which employ rigid tools to simulate the springback of highstrength and ultra-highstrength steels, will soon reach the limits of their accuracy. It has therefore become necessary to upgrade the simulation models to incorporate the elastic properties of the tools (and subsequently even those of the machine) for the simulation of sheet-metal forming.
Reducing the stiffness of the forming tools also adds new challenges to the modeled forming system: now, elastic deformations of the press equipment, i.e. the stiffness of the frame and table, also need to be taken into account. One possible approach, of course, is to model the entire press equipment by finite elements to study the elastic response. It must be taken into account however, that the modeling outlay may be very high, and usual engineering assumptions might already greatly influence the results. Hence, it might be much easier to carry out a limited number of real-world stiffness measurements of the machine that can, in turn, be used as boundary conditions, i.e. via elastic translational and rotational springs. Here, worn-out connections and unwanted clearance in the press equipment are automatically included in a subsequently defined finite element model.
Some studies [22, 29, 30] investigated the influence of the elastic tool on the results of the deep drawing simulation. For this, the blankholder device was discretized with elastic volume elements. By using elastic tools, it is possible to incorporate the local application of forces into the blankholder forces that are used in the simulation model. These local load applications were included in the tested model variants under analysis, and their effects were analyzed. A primary objective was to simplify the volume model of elastic tools through the use of substructures. The main goal was to find a real-world application for this technology. We compared the equivalent models (simplified with substructures) for the deep drawing simulation with the elastic volume models for the various tools. In the computations, the various model variants were studied for three test tools. The results show that there are characteristic differences between the models that use rigid bodies and those that use elastic tool parts. When local load applications were added to the blankholder forces, significant differences were observed between the model variants with elastic blankholder devices and the rigid body model. The condensation of the tool stiffness onto the effective areas of the substructures could be accomplished only to a limited extent. As far as quality is concerned, calculations with these equivalent models show the same deformation behavior of the blankholder structures that is found in the elastic volume models. Minor deviations in the stiffness were found between the two variants. In the simulation of the deep drawing, no visible difference was observed in the sheet-metal drawing for the tools tested. Substructures therefore make it possible to take elastic tool properties into account in deep drawing simulation in a simplified manner.
The generation of equivalent models for larger tools necessitates improved interfaces in the computation programs used. In order to assess the efficiency of the equivalent models, we recommend that a validation be conducted by means of real experiments [31].

9 Summary

Continuing development of today's commercial finite-element software packages for the simulation of sheet-metal forming processes enables a numerical analysis to be made of complex, multi-physical metal-forming processes. The targeted results, however, do not always correspond to what is technically feasible, because while the theoretical descriptions in the form of material laws required for improved quality have to date been only partly implemented in commercial FE programs, the real material behavior of the sheet-metal material cannot be adequately represented, because characteristic material values are missing. For these reasons, if the quality of the result is to be improved, an adequate description of material behavior and tribological factors must be further refined relative to the problem under analysis and the (multi-) physical effects acting upon it. An essential precondition for the practical use of this most complex set of laws is the possibility of being able to determine the parameters required with a reasonable experimental outlay.

10 References

[1] Proceedings of the 5th International Conference, NUMISHEET 2002, ed. Yang, D. Y., Oh, S. I., Huh, H., Kim, Y. H., 2002.

[2] Proceedings of the 6th International Conference, NUMISHEET 2005, ed. Smith, L. M., Pourboghrat, F., Yoon, J-W., Stoughton, T. B., 2005.

[3] Roll, K.: Advanced Simulation Techniques – Exceeding Reality? Proceedings, Materials & Technology Conference, September 16–20, Detroit, 2007.

[4] Roll, K.: Stand der Simulation umformtechnischer Prozesse in der Automobilindustrie [Status of simulation of forming processes in the automobile industry] . In proceedings of the symposium "Weitab vom Hookeschen Gesetz" ["Far away from Hooke's Law"], Akademie der Wissenschaften und Literatur, Mainz 2004.

[5] Roll, K.; Bogon, P.: Rückfederung als Genauigkeitshindernis – notwendige Weiterentwicklungen der Simulationstechnik [Springback as an obstacle to accuracy – necessary developments in simulation technology]. In: Proceedings of the 10th SFU [Saxony Conference on Forming Technology] 2003, Chemnitz, pp. 89–108.

[6] Steck, E.: Plastizitätstheoretische Verfahren in der Umformtechnik [Plasticity theoretical procedures in forming technology]. Colloquium on forming technology on the occasion of the 60th birthday of Prof. Lange, Forming Technology Institute, University of Stuttgart, 1979.

[7] Roll, K., Altan, T. Tekkaya, A. E., Hermann, M: Virtuelle Umformtechnik [Virtual Forming Technology]. Colloquium on forming technology on the occasion of the 80th birthday of Prof. Lange, Forming Technology Institute, University of Stuttgart, 1999.

[8] Delannay, L., Doghri, I., Lani, F., T. van Rompaey, Jacques, P. J., T. Pardoen, T.: Multiscale constitutive model for the simulation of forming operations on TRIP-assisted multiphase steel. Proceedings, ESAFORM Conference, 2004.

[9] Hänsel, A. H. C., Hora, P., Reissner, J.: Model for the kinetics of strain-induced martensitic phase transformation at non-isothermal conditions for the simulation of sheet metal forming processes with metastable austenitic steels. In: Simulation of Materials Processing, Huétink and Baaijens (eds.), Balkema, Rotterdam, 1998.

[10] Roll, K.: Stand der Blechumformsimulation heute – notwendige Weiterentwicklungen [State of the art in forming simulation today – necessary developments]; 1st Industry Colloquium Fertigen in Feinblech ["Thin sheet fabrication"], pp. 129–132, TU Clausthal, 1998.

[11] Swift, H.W.: Plastic instability under plane stress. J. Mech. Phys. Solids, Vol. 1, pp. 1–18, 1952.

[12] Voce, E.: The relationship between stress and strain for homogeneous deformation. J. Inst. Met., pp. 537–562, 1948.

[13] Rohleder, M: Simulation rückfederungsbedingter Formabweichungen im Produktentstehungsprozess von Blechformteilen [Simulation of springback related shape deviations in the process of producing sheet-metal parts]; doctoral dissertation, University of Dortmund, 2001.

[14] Barlat, F.: Constitutive modeling of metals. Proceedings, ESAFORM – Conference, Cluj-Napoca, 2005.

[15] Gerlach, J., Kessler, L.: The impact of material testing strategies on the determination and calibration of different FEM material models. Proceedings of IDDRG 2006, June 19–21, Porto, pp. 113–120.

[16] Krasovsky, A.: Verbesserte Vorhersage der Rückfederung bei der Blechumformung durch weiterentwickelte Werkstoffmodelle [Improvements in the prediction of springback in sheet metal forming by using sophisticated materials models], dissertation, Fraunhofer Institute for Materials Engineering (IWM), 2005.

[17] Roll, K., Faust, A.: Forming simulation of high-strength steels with direction-specific hardening modelling. Proceedings, AutoMetForm, Wroclaw, May 2008.

[18] Roll, K., Faust, A., Kessler, L.: Deep drawing simulation of high and ultrahigh strength steels under consideration of anisotropic Hardening. In: Proceedings, NumiForm, Porto, 2007.

[19] Faust, A., Roll, K.: Numerical springback evaluation for modern high strength steels using advanced material modelling. In: Proceedings of the 7th International Conference, NUMISHEET 2008.

[20] Gese, H., Oberhofer, G., H., Dell: MF GenYld + CrachFEM – A Modular Material and Failure Model for Structural Materials to be Used in Metal Forming and Crash Simulations. NAFEMS Seminar: „Materials Modeling - FE Simulations of the Behavior of Modern Industrial Materials Including their Failure“. Niedernhausen near Wiesbaden, Germany, December 5-6, 2006.

[21] Steinicke, M.: Modifiziertes Reibgesetz für die finite Elemente Simulation des Tiefziehens [Modified frictional law for finite element simulation of deep drawing], dissertation, University of Stuttgart, 2003.

[22] Roll, K., Wiegand, K.: An option for including tool elasticity in the simulation of sheet metal forming. Proceedings, “New Developments in Sheet Metal Forming.” Institute for Metal Forming Technology, University of Stuttgart, 2006.

[23] Keum, Y. T., Ahn, I. H., Lee, I. K., Song, H. M., Kwon, S. O., Park, J. S.: Simulation of stamping process of automotive panels considering die deformation. In: Proceedings of the 6th International Conference, NUMISHEET 2005.

[24] Neukamm, F., Feucht, M., Haufe, A., Roll. K.: A generalized incremental stress state dependent damage model for forming and crashworthiness simulations. In: Proceedings of the 7th International Conference, NUMISHEET, 2008.

[25] Lorenz, D., Roll, K.: Simulation of hot stamping and quenching of boron alloyed steel. In: Proceedings of the ESAFORM Conference, Trondheim, 2004.

[26] Lorenz, D., Roll, K.: Coupled thermal and mechanical simulation of press hardening processes. In: Proceedings, Dyna-User Conference, DYNAMORE, 2005.

[27] Lorenz, D., Roll, K.: Thermisch-mechanisch gekoppelte Blechumformsimulationen mit LS-DYNA. [Thermomechanically coupled sheet metal forming simulation with LS-DYNA] In: Proceedings, Dyna-User Conference, DYNAMORE, 2006.

[28] Hein, P., Kefferstein, R., Dahan, P.: Presshärten von USIBOR 1500 P: Simulationsbasierte Bauteil- und Prozessanalyse. [Hot Stamping of USIBOR 1500 P: Part and Process Analysis Based on Numerical Simulation] Proceedings, “New Developments in Sheet Metal Forming.” Institute for Metal Forming Technology, University of Stuttgart, 2006.

[29] Lingbeck, R. A.: Compensation of deep drawing tools for springback and tool deformation. In: Proceedings of IDDRG Conference, Olofstöm, 2008.

[30] Lingbeck, R A.: Tool and blank interaction in the cross-die forming process. In: Proceedings, ESAFORM Conference, Lyon, 2008.

[31] Haufe, A., Bogon, P., Roll, K.: Sheet metal forming simulation with elastic tools in LS-DYNA. In: Proceedings of the 7th International Conference, NUMISHEET, 2008.

