International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Development in Simulation of Sheet Metal Forming

***Dr. A.K. Madan*[1], *Pranjay Vardhan*[2], *Pratyashu Shekhar*[3]**

[1](Deptt. of Mechanical Engineering, Delhi Technological University)
[2](Deptt. of Mechanical Engineering, Delhi Technological University)
[3](Deptt. of Mechanical Engineering, Delhi Technological University)

ABSTRACT

The present-day industrial forming simulation is the outcome of a two-decade research process. The ongoing optimization of material properties was a major focus of this procedure. The discussion will provide an overview of the simulation technology issues that have arisen as a result of the need to better the bridging of material data and computational technology. In addition, an integrated technological roadmap is examined from the standpoint of simulation requirements, material characterisation needs, simulation technology advancements, and the consequences of those requirements on the accuracy of actual applications in the automotive world. To underline the paper's ideas, the presentation will include industrial examples.

1. Introduction

Sheet metal forming is simulated on a computer using special software in metal forming simulation. Simulation allows for the early detection of flaws and difficulties in the form of wrinkles or cracks in components on the computer. It is thus unnecessary to create real tools in order to conduct realistic tests. Since it is used to create and optimize every sheet metal item in the automotive industry, forming simulation has become well-established.

A prototype of the actual metal forming process is required to demonstrate the process. The finite element method based on implicit or explicit incremental procedures is used to calculate this in the software. The model's parameters must accurately reflect the actual process.

2. Industrial requirements to simulation of sheet metal forming

One of the most important aspects of using a simulation tool in industry is its user-friendliness. The industry anticipates that using the code will be so straightforward that no additional finite element expert will be required. Furthermore, the simulation tool should be accessible where it is needed, that is, in the design office rather than just in the company's computational department.

A CAD/FEM/CAD interface is also required. The ideal need is for the tool's CAD model to be quickly converted to the finite element code, and for any geometrical changes made during the finite element simulation process to be simply transferred back to the CAD model and then to the CAM system.

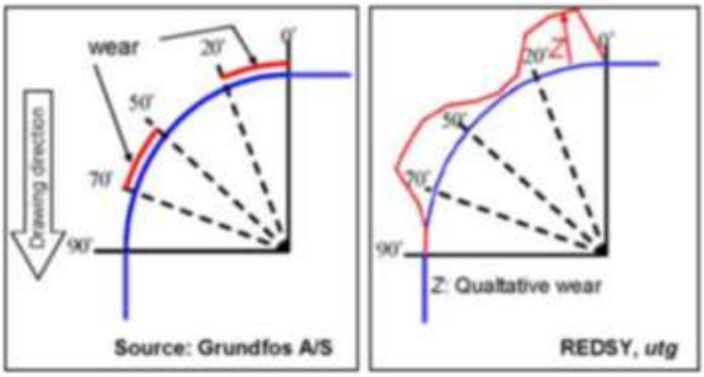
Another key consideration is simulation efficiency. Different computational times are desired depending on the design and development stage at which the simulation tool is employed. In the early design phases, reaction times of less than one hour are essential; however, in the major design stage or problem solving stages, overnight answers are acceptable. However, because the goal is to optimize the process, the pressure to have response times shorter than 2 hours is increasing even in the key design stages.

Material conduct is also significant. The material model required for sheet metal simulation must include initial anisotropy owing to the sheet's rolling process. Induced anisotropy in the form of kinematic hardening models is now necessary due to an increase in the desired accuracy.

3. Advanced Wear Simulation

The rigorous damage and loss of material, occurring on the outer surface of the contact as a result of the motion relative to the working parts adjacent to the surface is defined as wear. Usage of high-strength steels in numerous applications proves the necessity of predicting the tool wear higher as that of the past. One of the most important problems in the production of sheet metal is that of the wear of the draw-die. On the basis of the finite element analysis, estimations have been made about the wear.

For use in conjunction with sheet metal forming simulation (under the classical wear equation for finite element) The Archard wear equation indicates that the wear volume (W) is proportional to the product of the impact load (P) and the sliding distance (L), but inversely proportional to the wearing material's hardness value (H).



(a) Experiment (b) Simulation

The normal pressure on the elemental surface(p) and the elemental sliding velocity(v) in the equation represent the wear volume change of an element(W) over time(t).

For wear simulation, an algorithm which was developed on the basis of classical wear equations for finite elements proceeds by firstly analyzing the initial tool geometry, followed by forming simulation(PAM-STAMP). After that the wear simulation is run on the tool's outer surface in contact with the object, finally leaving us with the predicted worn tool geometry.

Classical wear simulation helps in finding out the important areas of critical abrasion in figure. Experiments referred from the study of determination of tool wear, the qualitative wear from simulation is likely to be compared with.

The classical method is not ideal for the prediction of the quantities of wear as it does not consider the tool geometry changes by wear. Therefore, Geometry-update-scheme(GUS) was developed as it considers the changes of tool geometry by the increasing number of punch strokes. To describe these changes, GUS uses an iterative scheme where every iteration is made up of forming and wearing simulation. GUS was verified through simulation and experimentation with a rectangular cup drafting method. After 200 punch strokes, the tool geometry was assessed using the GOM-ATOS@ system.

Thus, the experimental data agrees well with the strategy employing Geometry-Update-Scheme (GUS). GUS can be used to anticipate a tool's life cycle in sheet metal forming and to develop the best tool geometry for minimizing wear.

4. Technological Developments

Finite element based analysis for all types of processes, such as casting, forming, and cutting, has become possible because of recent technology breakthroughs. Simulation, often known as Computer Aided Engineering (CAE), is the most common type of FE-based analysis. Simulation, by definition, is a software tool for both creating and testing the possibilities of a certain process. Many design software programmes, such as LS-Dyna, AutoForm, and HyperForm, have evolved through time to provide the greatest functionality for the forming business. This simulation-based manufacturing has greatly reduced the time-cost factors, and the money that would have been squandered on trial and error is now being used to enhance simulation technology. Simulation has decreased tool development and manufacturing time by around 50% in recent years, and a further 30% reduction over the next few years looks feasible . For a specific operation, material attributes, geometrical parameters, contour definition, physical parameters, and output parameters can now be precisely defined. On the contrary, not everything with simulation-based manufacturability is acceptable. When we examine the definitions of numerous sheet metal forming features between simulation models and physical reality, we observe that key factors are incorrectly described .

4.1. Drawing

Drawing is the process of making cups, shells, boxes, and other similar items out of sheet metal blanks. The technique entails the downward movement of a punch with a predetermined travel speed and stroke to draw a metal blank placed on the die. Box or shallow drawing refers to a cup that is little more than half its diameter deep. When the cup depth is more than half the diameter, the drawing is called cup or deep drawing . Figure 1 depicts a typical deep drawing procedure for making a cup.

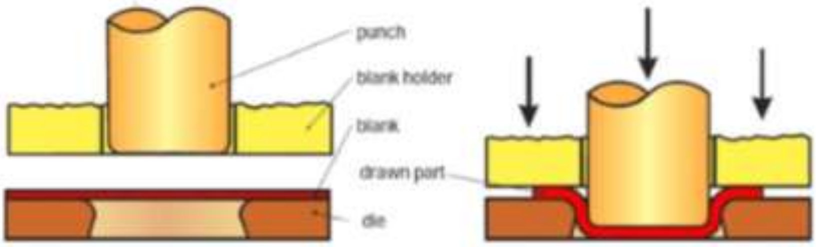
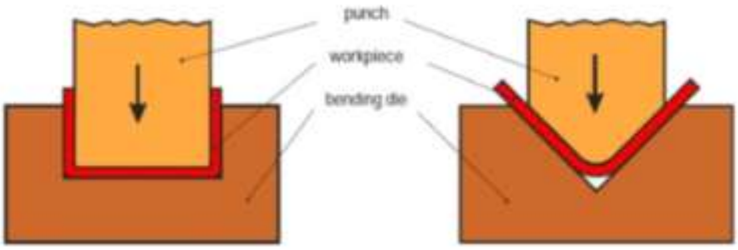


Fig 1: Deep Drawing Process

4.2. Bending

Bending is the process of shaping metal around a straight axis that runs the length of the material. Figure 2 shows a typical tool setup for U-bending and V-bending. The result is a level surface that is at an angle to the workpiece's original plane. With the interior surface of the curve in compression and the external surface in tension, metal flow is equal across the bend axis. However, when the load is removed, the overall strain on the work decreases due to elastic recovery, resulting in a form disparity. Springback is the term for this. When it comes to V and Wipe bending, springback is the most discussed topic. The springback effect while bending is influenced by a number of factors.



U-Die

V-Die

Fig 2: Die Bending Operation

4.3. Blanking

Blanking and punching, unlike the previous two shaping procedures, are two sheet metal cutting operations. The stock left on the die after punching is scrap in blanking, as seen in Fig 3, whereas the cut part is scrap in punching. The production of a washer is one of the most common examples of both punching and blanking procedures together.

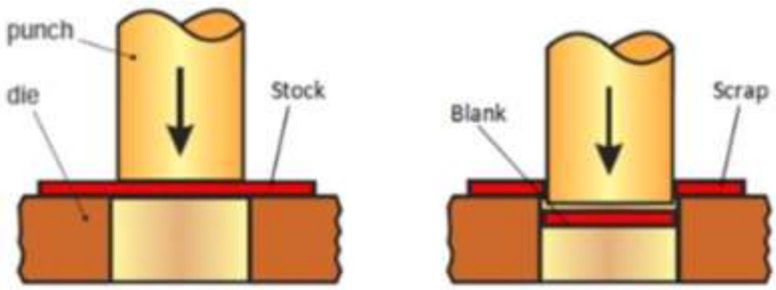


Fig 3: Blanking Process

4.4. Stretch Forming

Stretch forming is a type of tensile forming that is used to make impressions on curved or flat sheets, with the impressions being enlarged due to the thinner sheet. The thickness of the sheet is lowered from S0 to S1 when it is stretched between the two collets in Fig 4, resulting in a convex impression on the sheet.

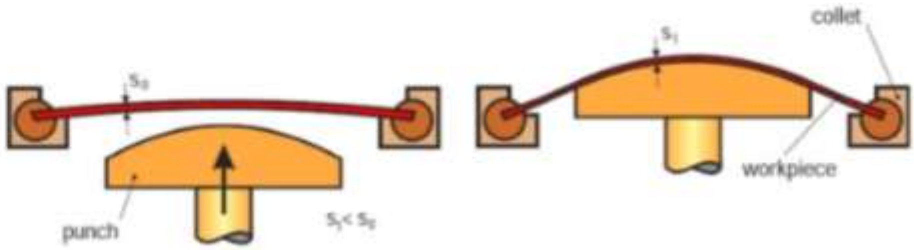


Fig 4: Stretch Forming Process

5. Development of forming simulation

Before the upbringing of the ages of computers the simulation method used to work in consideration of pure logic and of that of experimental results. The determination of basic results such as forming forces, material flow and failure phenomena all were derived using experimental results only. For the pre-calculation of forming forces and rough calculation of stresses, the theoretical modeling techniques relied heavily on elementary plasticity theory. Slip line theory, the upper boundary method, and the weighted residual method were also used to a lesser extent.Given the challenges of the engineering in the forming zone, theoretical simulation methodologies were affiliated with broad assumptions and approximations, severely limiting the expressiveness of the parameters associated.

After the revolution in computers the development of simulation softwares helped the researchers to finally apply available approaches of higher plasticity theory. The derived numerical methods were reformulated so that they could be simply executed by the software. It began with finite difference methods at the time of theoretical approach which continued with error compensations procedure and at last the development of finite element method(FEM), that we still use today, was developed.

6. Refinement of materials models

Tresca and von Mises' rudimentary materials models were quickly shown to be insufficient for characterizing material behavior (normally, only the von Mises model is used in simulation). As a result, the models were expanded to include factors like anisotropy, kinematic hardening, and so on. Tensile testing is no longer adequate for determining the required material properties. There are a few tests which are very difficult to be conducted due to various factors but are equally important and in need, such as hydraulic cupping testing, cross-tensile testing, torsion testing, etc. Until now, the origin of all models has been the normal rule and as per that during the plastic deformation, the yield surface is placed vertically to the deformation gradient. Material behavior is found to be incompressible during the plastic deformation.

The structure of transformation-induced plasticity steels changes during forming because austenite is transformed into martensite, changing the volume and structure, and thus the hardening of the material also varies depending just on stress throughout forming. In present times, there is no material model yet found which can justifiably explain the behavior of material with unobjectionable accuracy. The possible critical point for the material behavior of the model is the numerical approximation of the relationship between the FE system. Today's practice frequently consists of estimating unknown parameters, transferring known parameters from a homogeneous material, or selecting a structural analysis that unconstitutionally simplifies the normal relationships. Almost in every case, we begin with an incorrect modeling of material behavior.

It's these factors why an adequate explanation of material behavior and tribological factors must be chosen: to optimize the accuracy of the result in order to identify the causes under consideration and the (multi-)physical impacts at work. The determining the conditions necessary with an acceptable expenditure of development and innovation is a necessary precondition for the practical application of one of the most complex systems of laws.

7. Conclusion

It is impressive that simulation can satisfy such a broad set of requirements in today's circumstances, where simulations are widely used in production and manufacturing industries for a variety of reasons, including wrinkle prediction, surface deflection projection, breaking limit condition study, blank geometry determination, mechanical properties prediction, sheet thickness and residual stress evaluation, and so on. Simulations can also greatly improve the predictions of a tool's life cycle in sheet metal forming and the optimum utilization of the geometrical parameters in terms of wear minimization. Today's commercial finite-element software programmes for the simulation of sheet-metal manufacturing methods are still being developed, allowing for a numerical analysis of complicated, multi-physical metal-forming processes. Because typical material values are omitted, the genuine material behavior of sheet metal cannot be effectively represented. As a result, if indeed the quality of the outcome ought to be enhanced, a rational description of material behavior & tribological aspects should be refined in relation with the situation at hand as well as the (multi-) physical influences acting on this. The ability to establish the parameters required with an acceptable research outlay is a necessary precondition to effective application of its most sophisticated set of principles.

8. References

1.1. H. Hoffmann, C. Hwang, K. Ersoyl "Advanced Wear Simulation in Sheet Metal Forming" Institute of Metal Forming and Casting, Technische Universitat Munchen, Garching, Germany

1.2. M.R. Jensen, F.F. Damborg, K.B. Nielsen, J.Danckert, Applying the finite-element method for determination of tool wear in conventional deep-drawing, J. Mat. Proc. Technol

1.3. A. Makinouchi "Sheet metal forming simulation in industry" The Institute of Physical and Chemical Research - RIKEN, 2-1 Hirosawa, Wako, 351-01 Japan

1.4. Prof. Dr.-Ing. Karl Roll "Simulation of Sheet Metal Forming – Necessary Developments in the Future" LS-DYNA Anwenderforum, Bamberg

1.5. A.G. Mamalis, D.E. Manolakos, A.K. Baldoukas "Simulation of sheet metal forming using explicit finite-element techniques: effect of material and forming characteristicsPart 1. Deep-drawing of cylindrical cups" Journal of Materials Processing Technology 72

1.6. W.L. Xu∗, C.H. Ma, C.H. Li, W.J. Feng "Sensitive factors in springback simulation for sheet metal forming" Research and Development Center, Journal of Materials Processing Technology 151

1.7. A. Erman Tekkaya "State-of-the-art of simulation of sheet metal forming" Department of Mechanical Engineering, Middle East Technical University, 06531 Ankara, Turkey

1.8. FE-simulation of 3-D sheet metal forming process in automotive industry, VDI Verlag