



Formability Analysis for Cylindrical Cup Using Fea Tool Hyper Form

KEYWORDS

crumb rubber, utilization, compressive strength, low cost, sustainable

Shailendra Kharkate

Amit Kumar Gupta

Mechanical Engineer, Indo German Tool room, Indore(M.P.)

Asst.Prof. Mechanical,IET-DAVV,Indore,RH-12,Scheme No.54 Sector-A,Vijay Nagar,Indore(M.P.)

ABSTRACT

Majority of automobile and appliances components are made by deep drawing sheet metal process. So these growing need demands a new design methodology based on metal forming simulation. With the help of metal forming simulation we can identify the problem areas and solutions can be validated in computers without any expansive shop floor operations prior to any tool construction. Metal forming simulation is also helpful at the product and tool design stage to decide various parameters. Optimization of process parameters in sheet metal forming is an important task to reduce tryout and manufacturing cost. To determine the optimum values of the process parameters, it is essential to find their influence on the deformation behavior of the sheet metal. This paper describes the methodology by which the influence of various parameters on deformation behavior of the sheet metal can be predicted and the data of post processed result can be used during product design and tool design. The virtual FE simulation can predict various defects during deep drawing process at the product design stage. The study performed in this paper to predict the effect of parameters on formability of a trapezoidal cup using Altair Hyper Form radioss predictive tool. The formability analysis is performed for various cases of different values of parameters blank holding force and friction coefficient and various virtual tryout set is developed and thickness variation are analyzed. The die, punch, binder and blank are the main components developed as virtual tryout set.

INTRODUCTION

The majority of automotive and appliances parts are produced by means of the sheet metal forming technology. With the increasing popularity of FE simulations are performed repeatedly in the design feasibility studies of production tooling and die designs. Sheet metal forming is a technique by which most body parts are produced in automobile industries. One of the most important formative processes of sheet metal parts in manufacturing industries is deep drawing (DD). The DD process is a technique/tool which is often applied to fabricate hollow sheet metal parts with high drawing ratios or complicated shapes. In sheet metal forming, a blank sheet is subjected to plastic deformation using forming tools to conform to a designed shape. During this process, the blank sheet is likely to develop defects if the process parameters are not selected properly. Therefore, it is important to optimize the process parameters to avoid defects in the parts and to minimize production cost. Optimization of the process parameters such as die radius, blank holder force, friction coefficient, etc., can be accomplished based on their degree of importance on the sheet metal forming characteristics [1].

Deep drawing is a process for shaping flat sheets into cup-shaped articles without fracture or excessive localized thinning. The design and control of a deep drawing process depends not only on the work piece material, but also on the condition at the tool work piece interface, the mechanics of plastic deformation and the equipment used. The most used numerical method for numerical simulation of the forming process is finite elements method (FEM). The numerical simulations included the evaluation of the influence of various factors on the production process, the analysis of various test geometry, as well as the evaluation of loads on the production process. The FE analysis software is regularly employed in the design assessment of stamping tooling and dies in automotive industries, and the process simulation approach has been established as a practical methodology in the part formability and stamping failure predictions. With the enhanced insight into the stamping process, potential problems in the process tooling and die tryout phases may be eliminated in the virtual environment, and both the costs and lead-times can be considerably reduced. One of the objectives of the

FEM simulations was to determine test geometry both for the FEM model and laboratory tests. In determination of test geometry, the following viewpoints are considered:

- In order to simplify the tooling system and the test procedure, the geometry should be as simple as possible.
- Its shape should allow the easy strain measurements at any point of the surface.
- During the forming process the friction effect should be minimum.
- Both parts of the FLD (draw and stretch regions) should be covered by the deformation values.
- The results are presented in following sections.

Failure of sheet metal parts during deep drawing processes usually takes place in the form of wrinkling and/or necking. Wrinkling normally occurs at the flange and is generated by excessive compressive stresses that cause the sheet to buckle locally. On the other hand, fracture or necking occurs in a drawn part which is under excessive tensile stresses. Wrinkling and tearing rupture thus define the deep drawing process limits [3-5]. For a given problem, many variables affect the failure of a stamping. These include material properties, die design, and process parameters such as friction conditions, the drawing ratio as well as the blank- holder force (BHF), and the careful control of these parameters can delay the failure of the part. Among these process and design variables, one in particular, the blank-holder force (BHF) scheme, has been shown to greatly influence the growth and development of part defects. Studies have shown that a deep drawn part's quality is affected significantly by the flow of metal into the die cavity. The force exerted by the blank holder on the sheet supplies a restraining force which controls the metal flow. This restraining action is largely applied through friction. Excessive flow may lead to wrinkles within the part, while an insufficient flow can result in tearing.

In this paper the optimization of parameters in deep drawing (DD) of cup is performed with the help of predictive tool Altair Hyper Form Radioss. To determine the optimum values of the process parameters, it is essential to determine their

influence on the deformation behavior of the sheet metal. The most important process parameters for affecting thickness distribution of a cylindrical cup, namely Blank holding Force and friction coefficient, were determined.

PROCESS METHODOLOGY (DETAILS WITH FIGURES)

The formability of blank sheet depends on the process parameters such as pressure, punch speed, friction coefficient, and blank holder force. Fracture and wrinkle are the major modes of failure in sheet metal parts. Hence, using proper process parameters are essential to restrict wrinkling tendency and avoid tearing. One of the quality criterions in sheet metal formed parts is thickness distribution. In this study, a cylindrical cup with CRCA steel and blank thick of 1 mm is simulated by using Altair's Hyper Form radios to study the effect of these parameters on failure modes and thickness distribution.

Table:-1 THE VALUES OF PROCESS PARAMETERS FOR DIFFERENT EXPERIMENTS

| S.NO. | BLANKHOLDER FORCE (KN) | PUNCH SPEED (mm/s) | Friction coefficient |
|-------|------------------------|--------------------|----------------------|
| a. | 3 | 5000 | 0.025 |
| b. | 10 | 5000 | 0.050 |
| c. | 15 | 5000 | 0.100 |

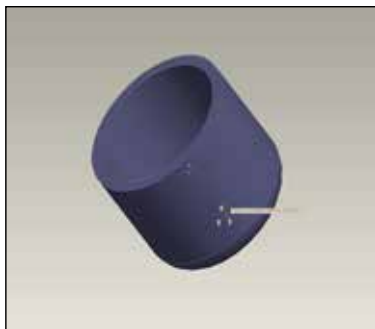


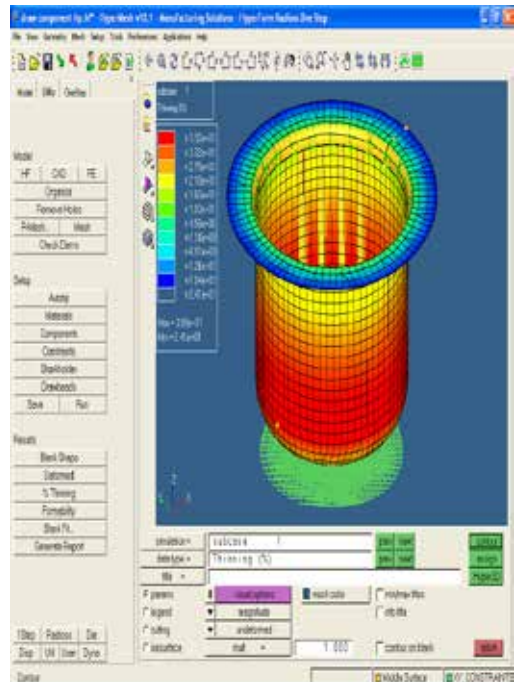
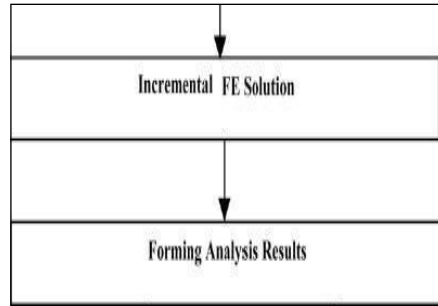
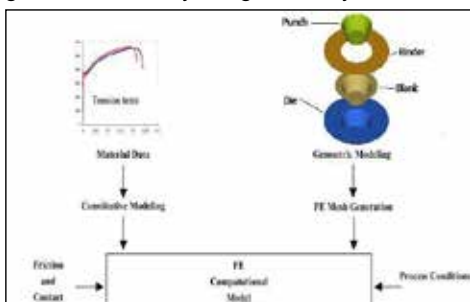
Fig.1:- Cylindrical Cup

PREPARING THE MODEL:

The component to be formed is taken into Hyper Form as an IGES data and die design was carried out for the component using the available features in Hyper Form.

PREPROCESSING:

Using the available commands in Hyper Form a mesh was created die and blank. For creating the mesh for other parts like punch, binder automatic tool build option was used. To successfully form the component several iteration was performed. To conduct the study for different coefficient friction, blank holding force values as shown in table are considered and different study performed with Hyper Form Radioss. The basic steps for conducting a representative FE simulation of a sheet metal forming process including formability are given schematically in fig. and may be outlined as follows



RESULTS AND DISCUSSION

Effect of the Blank holding force on the wrinkling of cup Flange To research the effect of the blank holder force on wrinkling of cup bottom, this study was carried out with some blank holder force values as 3, 10, 15 KN while other parameters have constant values as in table 1. The results3KN and 5 KN, the heavy wrinkles appears, with the values of 10 KN the effect of wrinkles are reduced and the blank holder force equal to 10 KN and 15 KN the wrinkling on bottom of product does not occur. As shown in Fig.2. The result shows that the blank holder force plays as important role in cup deep drawing for affecting the Wrinkling of bottom of the cup, but it do not influent much on the thickness distribution of cup.

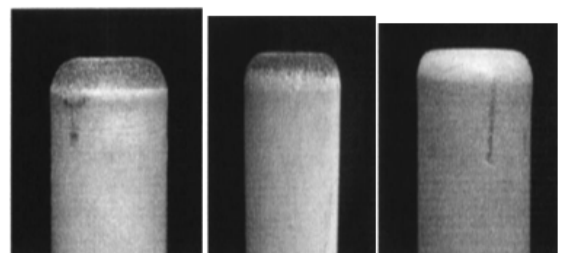


Fig.2: (A) 3KN (B) 10 KN (C) 15KN

Fig.2:- Effect of the Blank holding force on the wrinkling of cup bottom

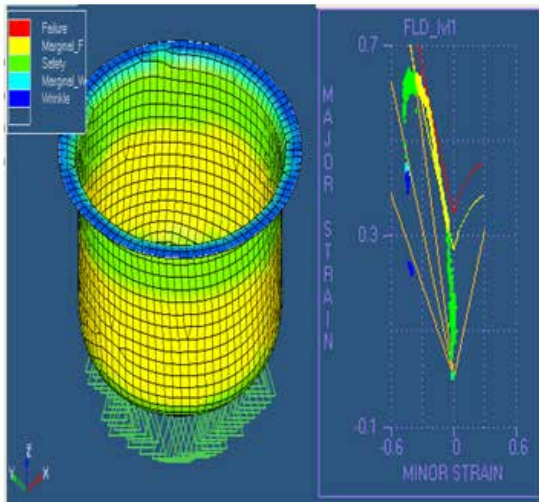


Fig.3:- Forming Limit Diagram of Cylindrical Cup

FUTURE PLANS

For such component the defects can also be controlled by selecting optimized punch speed. Further the research and study can be extended to predict the effect of punch speed on formability of cup drawing and on thickness distribution. Also with the help of DOE techniques the above parameters can be optimized. For DOE study Altair's Hyper Study tool is very much suitable and applicable.

CONCLUSIONS

Metal forming, product design & Die design industry can be largely benefited to carry the virtual forming simulation and thus reduce the manual tryouts which involves time and money. Simulation technique can be used effectively to optimize the die design and process parameters. Using Hyper Form and available CAE technology any modification required to modify the die or the component can be carried out in the software and multiple iterations can be performed and accordingly the design can be finalized.

REFERENCE

- [1] Metal Working, "Sheet Metal Forming", ASM Handbook, Volume 14B, (2006), U.S.A. | [2] Djavaanroodi F., A. Derogar, "Experimental and numerical evaluation of forming limit diagram for Ti6Al4V titanium and Al6061-T6 | aluminum alloys sheets", Journal of Materials & Design, pp 4866-4875 (2010). | [3] Sansot P., U. Vitoon, J. Jittichai, S. Surasak, "Determination of Forming Limit Stress Diagram for Formability Prediction of SPCE | 270 Steel Sheet", Journal of Metals, Materials and Minerals, Volume 21, pp 19-21, (2011). | [4] Damoulis G. L., G. Edson, B. G. Ferreira, "Analysis of the Industrial Sheet Metal Forming Process using the Forming Limit Diagram (FLD) through Computer Simulations as Integrated Tool in Car Body Development", International Journal of Mechanical | Sciences, pp 210-217,(2002). | [5] ASTM Handbook, "Standard test methods and Definitions for Mechanical Testing of Sheet Product", ASTM A370, (2011), U.S.A. |