IJSRD - International Journal for Scientific Research & Development| Vol. 3, Issue 10, 2015 | ISSN (online): 2321-0613

Role of Simulation in Deep Drawn Cylindrical Part

Kalpesh S. Patel[1] Jitendra B. Patel[2] Mehul K. Patel[3]
[1,2,3]Lecturers
[1,2,3]K D Polytechnic, Patan

Abstract— Simulation is widely used in forming industry due to its speed and lower cost and it has been proven to be effective in prediction of formability and spring back behavior. The purpose of finite element simulation in the sheet metal forming process is to minimize the time and cost in the design phase by predicting key outcomes such as the final shape of the part, the possibility of various defects and the flow of material. Such simulation is most useful and efficient when it is performed in the early stage of design by designers, rather than by analysis specialists after the detailed design is complete. The accuracy of such simulation depends on knowledge of material properties, boundary conditions and processing parameters. In the industry today, numerical sheet metal forming simulation is very important tool for reducing load time and improving part quality. In this paper finite element model for the deep-drawing of cylindrical cups is constructed and the simulation results are obtained by using different simulation parameters, i.e. punch velocity, coefficient of friction and blank holder force of the FE mesh-elements and these results are compared with experimental work.

Key words: Simulation, Blank holder force, co- efficient of friction

I. Introduction

In the finite element method, the actual continuum or body of matter like solid, liquid or gas is represented as an assemblage of subdivisions called finite elements. These elements are considered to be interconnected at specified joints, which are called nodes or nodal points. The nodes usually lie on the element boundaries where adjacent elements are considered to be connected. Since the actual variation of the field variable (like displacement, stress, temperature, pressure or velocity) inside the continuum is not known, we assume that the variation of the field variable inside a finite element can be approximated by a simple function. These approximating functions (also called interpolation models) are defined in terms of the values of the field variables at the nodes. When field equations (like equilibrium equations) for the whole continuum are written, the new unknowns will be the nodal values of the field variable.

A. What Is Finite Element Analysis?

FEA consists of a computer model of a material or design that is stressed and analysed for specific results. It is used in new product design and existing product refinement as well. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

Generally, there are two types of analysis that are used in industry 2-D modeling and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively.

B. Steps in Finite Element Analysis Using the Software

Finite element analysis consists of the three steps.

- Pre-Processing
- Solution
- Post-Processing

1) Pre-processing

Pre-Processing involves the preparation of data, such as nodal coordinates, connectivity, boundary condition, and loading and material properties.

2) The Solution

The Solution stage involves stiffness generation, stiffness modification, and solution of the equation, resulting the evolution of nodal variables.

3) Post-Processing

Post-Processing stage deals with the presentation of the results. Deformed configuration mode shape, temp. and stress distribution are computed and display at this stage.

C. Development of Finite Element Simulation

With the development of large powerful computers and the good understanding of the elements behaviour undergoing large inelastic deformation, FEM plays very important role in real 3-D engineering design as not only it can reduce dramatically the cost of design also it can layout clearly the complex physical phenomena helping engineers to better understand the deformation processes and to control the quality of the product.

Without properly designed die and process parameters, final product may be subjected to various defects. Complicated sheet metal parts are new subjected to simulation in order to design the process properly. Reliable method of simulation is then required to solve difficult problems involving geometric and material non-linearly as well as variable contact and friction interface conditions.

Expectation from the finite element simulation is powerful enough to predict all the forming defects and provide optimum stamping tools and conditions, we may completely eliminate the prototype tools form the design and manufacturing procedure, and also reduce number of trial and modification operations. Thus the process might be shortening dramatically.

Finite element codes developed specifically for sheet metal forming. Simulation may be classified by its formulation and time integration algorithm as classified four ways as listed below.

- Rigid-plastic or rigid-viscoelastic finite element method.
- Elasto-plastic finite element method-static implicit approach.
- Static explicit approach.

– Dynamic explicit approach.

A big advantage of dynamic equilibrium equation is that the stiffness matrix is not necessary to be constructed and solved, so that the solution of one time step can be obtained much faster than static approach.

D. Finite Element Procedure for Deep Drawing Process by Using Explicit Analysis LS-DYNA

1) Element Selection

The ANSYS element library consists of more than 200 different element formulations or types. Due to primary need in deep drawing process here SHELL 163 element is selected. An element type is identified by a name (8 characters maximum), such as SHELL163, consisting of a group label (SHELL163) and a unique, identifying number (163).

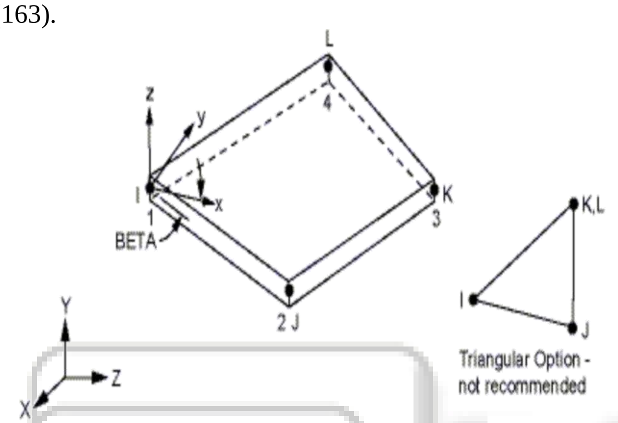

Fig. 1: Shell 163, 4 –Noded Quadrilateral Shell Element.

SHELL163 is a 4-node element with both bending and membrane capabilities as shown in figure 1. Both in-plane and normal loads are permitted. The element has 12 degrees of freedom at each node: translations, accelerations, and velocities in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes

E. Why Shell Element Is Selected?

Despite being robust for large deformations and saving extensive amounts of computer time, the one-point (reduced) integration solid and shell elements used in ANSYS LS-DYNA are prone to zero-energy modes. These modes, commonly referred to as hour glassing modes, are oscillatory in nature and tend to have periods that are much shorter than those of the overall structural response (i.e., they result in mathematical states that are not physically possible). They typically have no stiffness and give a zigzag appearance to a mesh) known as hourglass deformations as shown in figure 2. The occurrence of hourglass deformations in an analysis can invalidate the results and should always be minimized.

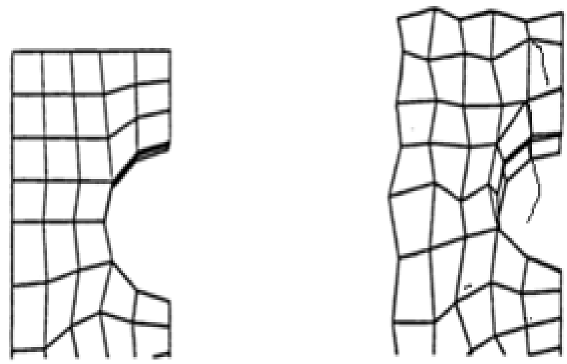

Fig. 2: Unreformed Mesh and Deformed Mesh with Hour Glassing Effect.

F. Real Constants

Data which are required for the calculation of the element matrix, but which cannot be determined from the node locations or material properties are input as "real constants." Typical real constants include area, thickness, inner diameter, outer diameter etc. In modeling areas are created by using line command and thickness is given in real content.

G. Material Model

Isotropic Elastic Model.

In isotropic material model the value of density (DENS), elastic modulus (EX) and Poisson’s ratio are required.

H. Boundary Condition

The properties of a system are externally imposed on it. The boundary conditions are applied to build analysis cases containing loads and restraint of the model. In order to get accurate result model is considered to be in equilibrium so much so that the load and moment should satisfies the condition of $\sum F=0$ & $\sum M=0$

Applying boundary conditions to the part geometry will mean that if the part is changed and model is updated, the boundary conditions will also be updated. In this work only y direction movement is allowed for punch & blank holder and the other direction movement is restricted and die movement of all direction are restricted.

Quarter Model is considered in order to reduce the analysis time because problem is symmetrical. In quarter model at one side $\sum x=0$ & on the other side $\sum z=0$. Figure 3 shows model with boundary condition.

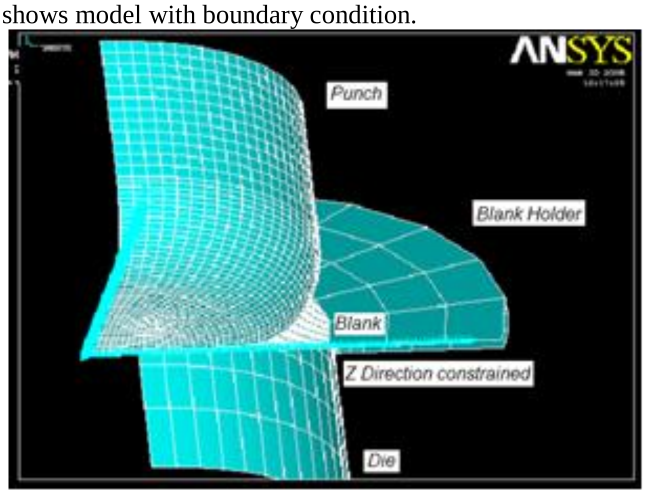

Fig. 3: Model with Boundary Condition.

I. Result and Discussion for FEA

In this paper different parameters are taken and find the effects on the quality of sheet metal part.

After completion of analysis the simulation gives the satisfactory result and it can apply in order check the effect of various parameters.

In simulation result product showing wrinkle at coefficient of friction value of 0.2 as shown in figure 4. Generally wrinkle is formed due to lesser grip between blank holder and die. In order to increase the grip high coefficient of friction is required.

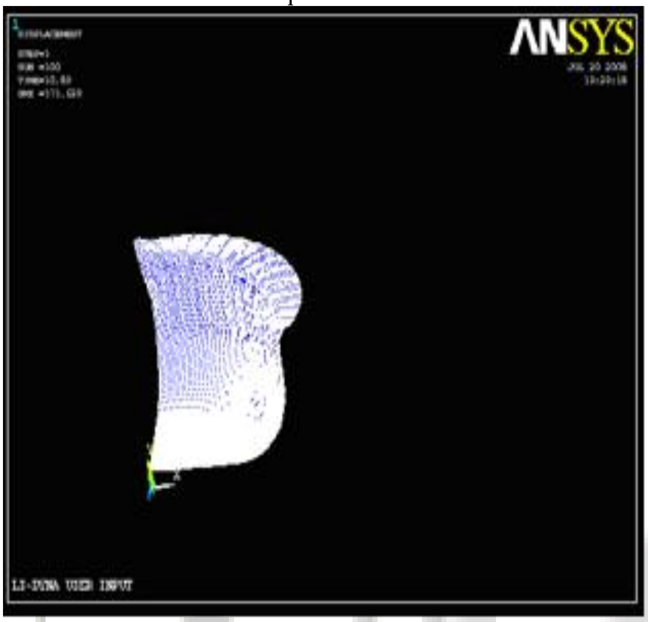

Fig. 4: Product Showing Wrinkle at Coefficient of Friction0 .2.

In same way product showing wrinkle at BHF 60 kN.If the value of blank holder force is increase at BHF 80 kN then product is wrinkle free which is the evidence that BHF play important role in quality of part. As shown in figure 5 products is wrinkle free at co efficient friction 0.25 and BHF80kN which is evidence of coefficient of friction and BHF play signification role in quality of product. Further increase in coefficient of friction may lead to tearing of sheet. In simulation final height of deep drawn cup is 181.95 mm and the experimental height of deep drawn cup is 180mm. Higher the punch speed in simulation produced the Dynamic effects in Explicit LS-DYNA. Higher the punch speed are detrimental. For this reason use suitable punch speed in order to avoid this problem.

In actual practice developing of new product has required some experiment trial for removing of wrinkle and tearing of sheet.i.e what amount of actual blank force, co-efficient of friction, punch speed, die gap are to be required and this process take some time and also wastage of material.But this process will be carried out in Ansys simulation it will take shorter time and give nearly same result of experiment work.

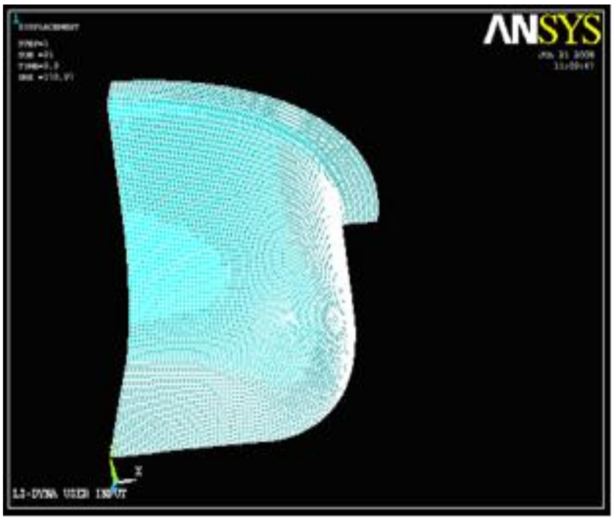

Fig. 5: Wrinkle Free Product at Coefficient of Friction 0.25

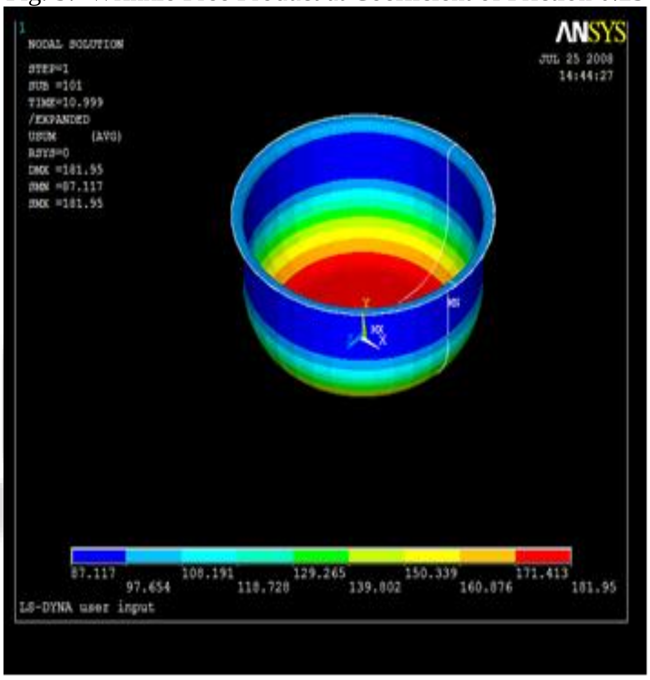

Fig. 6: Full Model of Deep Drawing at End of the Stroke.

Sr. No.	Simulation height of deep drawn cup h (mm)	Experimental deep drawn cup height h (mm)	Difference (mm)
1	181.95	180.00	1.95

Table 1: Comparison of FEA and Experimental Result.

II. CONCLUSIONS

In simulation, if the value of co-efficient friction is increased from 0.20 to 0.25 its shows lesser wrinkle in the products but value is further increase higher the punch drawing force required and it may lead to tearing of the sheet.

In same way at 60BHF product has wrinkle but at 80BHF product is wrinkle free.

In the simulation result final height of deep drawn cup is 181.95 mm and the experimental deep drawn cup height is 180.00 mm.

III. REFERENCES

[1] P.N.Rao "Manufacturing Technology, Foundry, Forming & Welding".
[2] Ansys User Manual 10.0.
[3] P.C.Sharma "A Text Book Of Production Engineering".
[4] R. Padmanabhana, , M.C. Oliveiraa, J.L. Alvesb, L.F. Menezesa "Influence Of Process Parameters On The Deep Drawing Of Stainless Steel" Science Direct.Com, Finite Element In Analyst & Design 43(2007).