IJRAME

ISSN (ONLINE): 2321-3051

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

Analysis of Sheet Metal Component using Numerical and F.E.M Method

[1]Nimbalkar D.H.

[1]Assistant Professor, College of Engineering, Osmanabad, Tuljapur road, Osmanabad,(M.S)India

E-mail: dhnimbalkar@gmail.com

Abstract

Manufacturing research and development is to determine the optimum means of producing sound products. The Process modeling for computer simulation has been major concern in modern metal forming technology. The Finite element simulation of sheet metal forming is increasingly applied to eliminate forming defects, predict and optimize process parameters and to predict stresses / strains in sheet metal blank to prevent blank failure. Determination of the effect of the process parameters on the final forming quality is very difficult in sheet metal forming process because forming process experience very complicated deformation. These process parameters have to be determined for the optimum forming condition before the process design. Conventional method to determine optimum process parameter is time consuming and costly. This work describes the mathematical modeling and theoretical analysis of sheet forming process. Further, F.E.M. simulation was carried out on deep drawn non-symmetric industrial component. HYPERMESH and HYPERFORM software's were used for meshing and to input data as a pre-processor. As a solver LS-DYNA-970 Software was used and results were viewed in LS-POST software

Keywords. Sheet metal forming, f.e.m, simulation, forming limit diagram.

1. Introduction

In the late 1970's and early 1980's the use of computer aided techniques i.e. (Computer Aided Engg. Design and Manufacturing) in the metal forming industry increases considerably. The trend seems to be toward ever wider application of this technology for process simulation and process design.

A goal in manufacturing research and development is to determine the optimum means of producing sound products [1]. The optimization criteria may vary depending on product requirements but establishing an appropriate criterion requires through understanding of manufacturing process.

In metal forming technology, proper design and control requires among other thing, the determination of deformation mechanics involved in the process. Without the knowledge of the influence of variables such as friction condition, material properties, and work piece geometry on the process mechanics, it would not be possible to design the dies and the equipment adequately, or to predict and prevent the occurrence of defect. Thus process modelling for computer simulation has been major concern in modern metal forming technology.

In past a no. of approximate method of analysis have been developed and applied to various forming process. The method most well known are the slab method the slip-line field method, the visco-plasticity method, upper and lower bound technique, Hills general method and more recently the finite element method (FEM) [2].

These methods have been useful in predicting forming loads, overall geometry changes of deforming work pieces, and qualitative modes of metal flow and in determine approximate optimum process condition. However accurate determination of the effects of various process parameters on the detailed metal flow became possible only recently when the finite element method was developed for the analysis. Since then finite element method has steadily increased importance in simulation of metal forming process.

The finite element technique, whose birth and boom in the 1960's was due to application of digital computers to structural analysis, has spread variety of engineering and physical science in the last decade [4].

The main advantage of the finite element method are 1) the capability of obtaining detailed solution or the mechanics in a deforming body namely, velocities, shapes strains, stress, temp. or contact presser distribution. 2) the fact that a computer code ,once written can be used for a large variety of problems by simply changing the input data [1].

$$(LDR) = \frac{d_0}{d_1}$$

2. Theoretical analysis

Sheet metal forming process broadly classified as deep drawing or stamping operation represents a wide spectrum of flow condition. The material properties that control formability vary with the specific sheet forming operation. At one end of the spectrum is the forming of flat bottom cylindrical cups by radial drawing. In this case one of the principle strains in the plane of sheet is positive and other is negative thee change of thickness being small. At the other end of spectrum are operations involving biaxial stretching of the sheet, where two of the principle strain are tensile and thinning is required. Many operations fall between these extremities. In forming many parts stretching may predominate in one region while drawing prevails in anther.

2.1Theoretical analysis of sheet metal by assuming deep drawing method

Material specifications EDD IS 513 1994 SS 4011- 2001

Mechanical properties.

1. Tensilestrength270-320MN/M2 2.Yield strength 210 MN/M2 3. Ratio 1.5 to 1.9
4. Elongation 36%

Limiting drawing ratio(LDR) $\therefore In(LDR) = \eta\sqrt{(\bar{R}+1)/2}$

Where $(LDR) = \frac{d_0}{d_1}$ limiting drawing ratio d0 : blank diameter d1: cup diameter.

η = Efficiency: depends on lubrication. Hold drawn pressure, sheet-thickness dies radius η = 0.74 to 0.79

$\bar{R}$ Represent material anisotropic value, $\bar{R}$ = 1.7 (Average)

$$\therefore In(LDR) = \eta\sqrt{\frac{(\bar{R}+1)}{2}} \quad = 0.75\sqrt{\frac{1.7+1}{2}} \quad = 0.75\sqrt{1.35}$$

= 0.87 *In(*LDR) = 0.87

$$\therefore \frac{d_0}{d_1} = 2.386$$

Limiting drawing ratio is more than 2.386 material will get fracture

Deep drawing

Average blank size = 450 mm   Limiting drawing ratio = blank dia/cup dia   = 480/205 =2.34

Theoretical limiting ratio = 2.386   Practical limiting ratio = 2.34

Result: For this problem practical limiting ratio which is 2.34 is approaching to theoretical limiting drawing ratio which is 2.386 so material will get fracture.

2.2 Mathematical modelling of sheet metal assuming biaxial stretching

1. Failure in stretching operations normally occurs by the development of a sharp localized neck on the surface
2. This expression shows at what condition localized necking will occur and material will get fracture

$\therefore$ Tensile strength of material = 320 Mpa = 320 × 106 N/m2

Now $\delta_u = k\left(\frac{n}{e}\right)^n$ δu = Tensile strength. k = strength coefficient; n strain – hardening exponent

320 × 106 N/m2 = $k = \left(\frac{0.21}{e}\right)^{0.21}$ $\therefore$ k = 176.224 × 106 N/m2

As the metal is anisotropic $\bar{R} = 1.7$ $\therefore p = \frac{\in 2}{\in 1} = \frac{d \in 2}{d \in 1} = -\frac{1.7}{1.7+1}$ = -0.6296   p = -0.6296

$\therefore$ Condition for necking becomes $\in^*{}_1 = \frac{n}{1+p}$

Where ε*1 critical strain   n = material finding coefficient   p = ratio of ε 2/ε1

Result: $\therefore \in^*{}_1 = \frac{0.25}{1+(-0.6296)}$

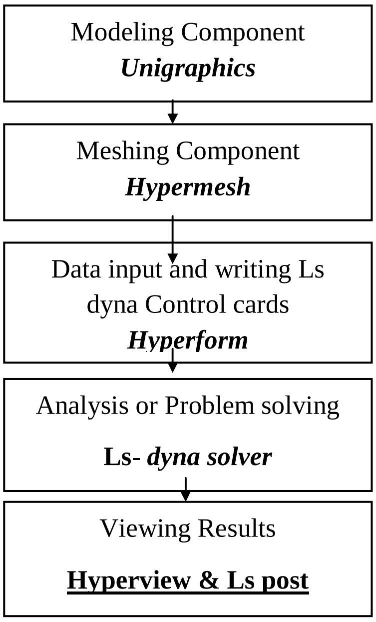
- Thus critical strain ε*1 should be less than 0.6749 so that fracture will not occurs
- If critical strain ε*1 is more than 0.6749 value local necking will start and material will start to fracture.

Concluding remark:

To describe the behaviour of sheet metal forming, properties i.e. work hardening and anisotropic plays major role. Sufficient hold down presser is needed to avoid wrinkling in sheet metal forming. Limiting drawing ratio (LDR) depends on tooling efficiency and plastic anisotropy. By mathematical analysis it came to know that present component reached the maximum L.D.R. and necking will start in component.

3. F.E.M. PROCEDURE:

Numerical simulation and modeling of bulk metal forming processes based on knowledge of underlying process physics and validated by experimental results is a powerful tool for optimizing process parameter.



Numerical simulation and modeling of bulk metal forming processes based on knowledge of underlying process physics and validated by experimental results is a powerful tool for optimizing process parameter. In may process development and design situation, simulation have replaced full scale process trials, reducing development time and cost compared with conventional experimental iterative methods FEM implementation is possible by following steps.

3.1 Modeling sheet metal component:

Sheet metal component is modeled using upigraphic-modeling software. All geometric parameter like shape, sheet thickness, and curvature are represented in computer model for visualization and further processing.

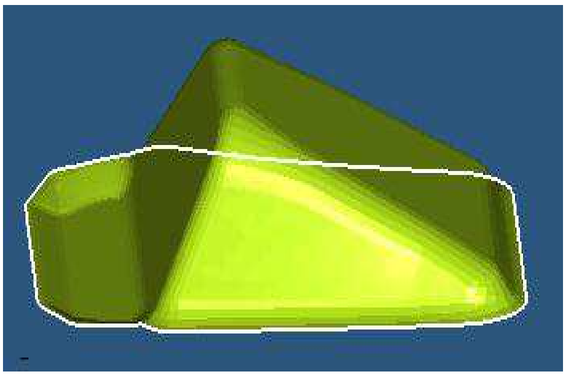


Figure 1: Model of Sheet Metal Component

3.2 Meshing the component:

In metal forming process the work piece generally undergoes large Plastic deformation relative motion between the deforming materials and die surface is significant. In the simulation of such processes the starting mesh must be well defined and can have the desired mesh density distribution. Mesh generation is time consuming and error prone process.

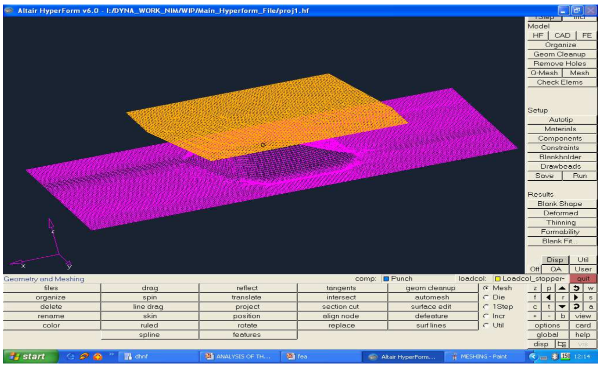


Figure2: Meshing of component with hypermesh

For this purpose commercial ***hypermesh*** software is used. IGES format of the modeled part in ***unigrophics*** is imported. By using mesh, create element, align node etc command. Even mesh is generated and care has been taken that the mesh confirms to geometry of part. It should take account boundary curvature and local thickness.For the present problem triangular shell element is used for meshing purpose. This gives good results for sheet metal component.

3.3 Data input and writing Ls dyna control cards.

Material properties and process parameter plays important role in sheet metal forming process. Final meshed blank shape is imported in the **hypeform** software. And from that blank shape automatically die and pouch and blank holder model is generated in the ***hyperform*** software. Following function is performed in **hyperform.**

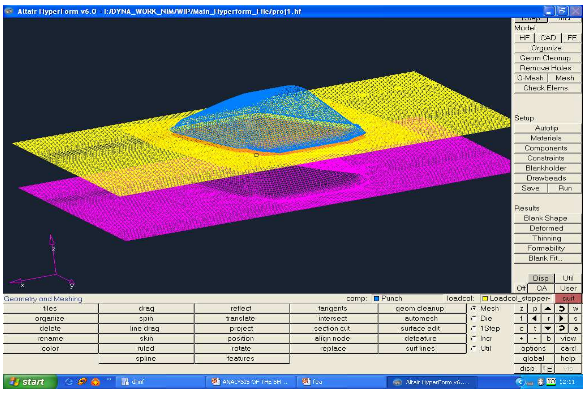


Figure 3: Automatic die and tool forming in hyperform

1) Material properties like 'young's modulus', density, stress – strain curve, strain hardening coefficient are assigned to material.
2) Punch velocity is assigned to punch.

3) Constraints are assigned to punch, die and blank holder. Die is kept fixed and motion of punch is allowed in only ‘Z’ direction. Blank holder is fixed and sheet blank is kept free.
4) Blank – holder load is assigned to blank holder.

File generated in **hyperform** is converted and exported in the format dyna.k. I.e. compatible format of **ls-Dyna.**

3.4 Solving the Problem.

F.E.M. based ***Ls-dyna*** is strong tool in predicting forming behavior of sheet metal. File exported and converted in dyna .k. format is imported in ***Ls- dyna Software***.The path to the solution of the finite clement problem consists of five specific steps.

A) Identification of the problem.
B) Definition of the element.
C) Establishment of the element.
D) Assemblage Element equation.
E) Numerical solution of the global equations.
This software generates results in “d3plot” file format

3.5 Viewing results.

It is necessary thing to get results in required and understandable format. For this purpose to viewing results ***hyper view*** and ***ls- post*** commercial software are used.

The ***Ls – Dyna*** software generates results is d3-plot file format, same file is imported in ***hyper view*** and ***Ls – post*** software to view results.

Results are available in different format such as stress strain, thickness change, forming limit diagram etc. For the present problem results are determine in terms of thickness variation and forming limit diagram, which provides necessary information regarding process decision-making.

4. Simulation of component

Simulation was carried out with punch speed 1m/s. Material model No. 24 was selected shell elements were used blank holding pressure 2×10^5 N/m^2 was assigned, model was imported in Ls-Dyna for analysis. It took near about 3 hours for analysis. Results were viewed in Ls-Post software.

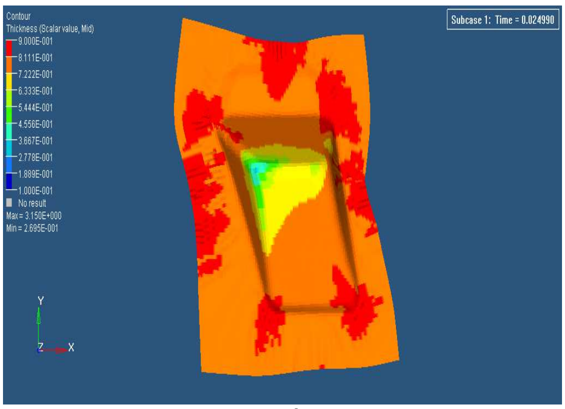


Figure4: Thickness Variation of Simulated component

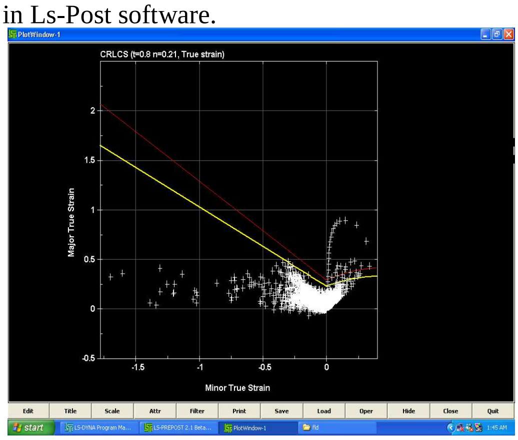


Figure 5: Forming Limit Diagram (F.L.D) of Component

Remark: From the above simulation it is clear that wrinkle were observed over the deform part from F.L.D. it can be concluded that some points are above fracture line so component would get fracture. Minimum thickness of bottom of cup is 2.69 mm.

5. Conclusion

In present analysis it shows that F.EM. Software results are fair agreement with numerical results. Somewhat variation in predicted behaviour and practical once is due to variation in process parameter like friction, sheet properties, like strain history rolling direction of sheet, direction of an isotropic ratio etc.

References

1. S. Kobayashis; "*Metal forming and the finite- element method*", McGraw-Hill, publication, New York (1980).
2. William F Hosford, Robert M.Caddell; "*Metal forming mechanics and metallurgy*", Prentice–Hall, USA (1978).
3. A. Makimuchi; "*Sheet metal forming simulation in industry*", Journal of Material processing Technology (1996) PP. 19-26.
4. G, Murali, M.Gopal , A Rajdurai, "*F.E.M. Simulation of Drawbead using F.E.M. method*", National conference on advance manufacturing technology VIT Vellare (2000) PP. 620-625
5. L.I.R. , Bohan , ML, Weinmannks, Chandra A, "*A study of the optimization of Sheet metal drawing with active drawbeads"*, Journal of Manufacturing Process (2000).
6. LS-Dyna user's manual, Theory Manual version 970.by Livermore software Technology corporation California (1998).
7. Dr.G.Chondra Mohan Reddy "*Finite Element Simulation in Manufacturing Engg"*, National Conference Advanced manufacturing, Hyderabad (2001) PP. 16-20.
8. Akira Teraka , Se Ho Him and Hoon huch "*Process Parameter Design in Sheet stamping Processes with Rigid Plastic Finite Element Analysis"*, Transachion of JSEES, Paper No 20000011.
9. 'Mohammad Habibi Parsa, Katsahiko Yamaguchi, Norio Takakura, "*Redrawing analysis of Aluminum- stainless – Steel Laminated Sheet using FEM Simulation and experiments*", International Journal of Mechanical Science 43 (2001) PP. 2331-2347
10. Tze Chi Hsu , chan – hung Chat, "*A finite element analysis of Sheet metal forming processes"*, Journal of Material processing Technology 54 (1995) PP. 70-75
11. Dong-kyun Min, Byung – Hee Jeon hyung – jonk him , nahsso him "*A study on process improvement of multi stage deep drawing by the finite – element method*", Journal of Material processing Technology 54 (1995) PP. 230-238.
12. H.L.Xing, Z.R. Wong "*Finite element analysis and design of thin sheet Soperplastic forming"*, Journal of Material processing Technology 68 (1997) PP.1-7.
13. L.G. Suhhomlinov , V.H. Engelsbern and V.N. Davydon , "*A finite element membrone Model for the analysis of Axsymmetric sheet metal forming process*", Int. J. Mech Sci Vol 34 (1992) PP. 179-193.
14. H.B.Sim, M.C. Boyce "*Finite Element analysis of a Real Time Stability control in sheet forming processes.*", Journal of Engg. Materials and Technology Vol 114,(1992), PP. 180-188.
15. E. Onate, O.C. Zicnkiewicz, "*A Viscous shell formulation for the analysis of thin sheet metal forming*", Int. S. Mech. Sci. Vol – 25 No: 5 (1983) PP. 205-335.
16. M.Gotoh and F Ishise "*A Finite Element analysis of Rigid Plastic deformation of the flange in a deep drawing process based n a fourth Degree yield function*", Int. S Mech Sci Vol- 20, (1978) PP. 423-435.