ISSN 2229-5518

A Review Paper on FEA Application for Sheet Metal forming analysis

Mr. Marlapalle Bapurao G.[1]

Student of PhD in Mechanical Engineering at JSPM, RSCOE, SPPU, Pune

Prof. Dr. Rahulkumar S. Hingole[2]

Professor in Mechanical Engineering at JSPM, RSCOE, SPPU, Pune

Address: Plot No.38, Shriniketan Colony, Behind Amarpreet Hotel, Jalna Road, Aurangabad-431001, India, email: mbapurao@gmail.com, contact no. 8421432497

Abstract: Finite element method (FEM) is used for simulating complex intricate shapes of industrial sheet forming operation. Effective physical parameters, as well as numerical solution, influence parameters of this phenomenon and its numerical prediction of results. In this review, the applications of FEM for analyzing various parameters such as blank Thickness, blank holding force, Material, coefficient of Friction on specimens of materials are discussed.The numerical results are found from literature survey to be in good agreement withthe experimental results and accurate thinning distributionshad been predict.The importance of sheet metal working process in modern technology is due to the ease with whichmetal may be formed into useful shapes by plastic stage deformation process in which the volume and mass ofmetal are conserved and metal is only displaced from one position to another.

Keywords: FEM, Numerical Solution, Simulation, Sheet Metal

1 Introduction

The sheet metal formingoperations process between the Male (Punch) and a female (Die). The procedure is of sheet metal deformation due to the relative movement between the punch tool and the sheet, an interaction that generates friction forcesoccurred between the elements. It is important to understand that the FEA tool is able to control the friction generated in the forming process in order to produce good quality products. Manufacturing defects can beanalyzed by using this tool and the defects occurred during manufacturing are crack, shrinkage, springback, surface defects and tool wear can be reduced by controlling the above defects in the process. It is generally believed that the friction between two surfaces in contact varies with velocity, applied load and type of lubricant, according to the Stribeck law. However, in a sheet stamping operation the friction cannot be considered as a static parameter due to the varying process conditions during the forming operation.

The FEA gives us Numerical solutions for the defects occurred while metal forming and the stress developed during manufacturing also we can analyzed based on this realistic solution we can make economical tools required for manufacturing sheet metal parts. The linear and non-linear dynamic analysis can be done on sheet metal part through this we can predict punch force, Blank Holding Pressure required and many other parameters. Based on this punch force and other

ISSN 2229-5518
parameters, how much to press machine is required for sheet metal forming operation.

2 Literature Review

Finite element analysis (FEA) is powerful tool to simulate complex draw of sheet metal forming operation; the accurate and reliable application of this technique to spring-back has not been widely demonstrated. The physical parameters can be predicted numerical values. In this paper, the effect of formed part thickness, blank holding force, formed surface grain structure, and the spring-back of specimens' part are discussed. The role that all the above parameters play in the spring-back is assessed through finite element simulations. Process conditions, such as Tool geometry, working temperature have an obvious effect on spring-back. Simulations are conducted with varying blank holding force, Materials, blank thickness, and coefficient of frictions to assess its role in spring-back of the formed part. In this paper, a spring-back prediction has compared their simulation results with experimental work. From this simulation study optimum blank has introduced. [1]

In sheet metal the drawings parameters like punch and dies radius, clearance, lubrication, blank holding force and its trajectories are studied. These tools need to designing drawing involves lots of trial and error steps. These trial and errors have reduced by simulation by using finite element tool. This simulation results give numerical results and approximate solutions. The given set of punch, die and working conditions has been applied to find an optimum blank holding force for reducing the wrinkles and at same time stresses induced in the forming part has been reduced. They are suggested Hyperform tool is feasible tool for forming problem. In this study Steel material grades selected for Punch and die block like HCHCr (High carbon High Chromium) & OHNS (Oil Hardening non Shrinkage) having EDD (Extra Deep Draw).They are recommended blank holding pressure (30Ton) for a defect-free component [2].

For any Deep drawing process most commonly used Metal Forming Process in the industries and based on this requirement this paper has different analytical, numerical, empirical and also experimental methods have been developed to analyze the manufacturing problems. In this paper, for deep drawing processes punch and die parameter geometries varying are analyzed by using finite element analysis (FEA) and simulations of a Deep drawing process. They have found optimum draw ratios in deep drawing [4].

Deep drawing process is one of the most commonly used Metal Forming Process within theindustrial field. Different analytical, numerical, empirical and experimental methods have been developed inorder to analyze it. In this paper deep drawing process with varying punch and die geometries are analyzed. This work reports on the stages of finite element analysis (FEA) and simulations of a Deep drawing process. The obtained result allows finding optimum draw ratios in deep drawing [4].

The single deep drawing operation with blank holder as shown in figure 1.

ISSN 2229-5518

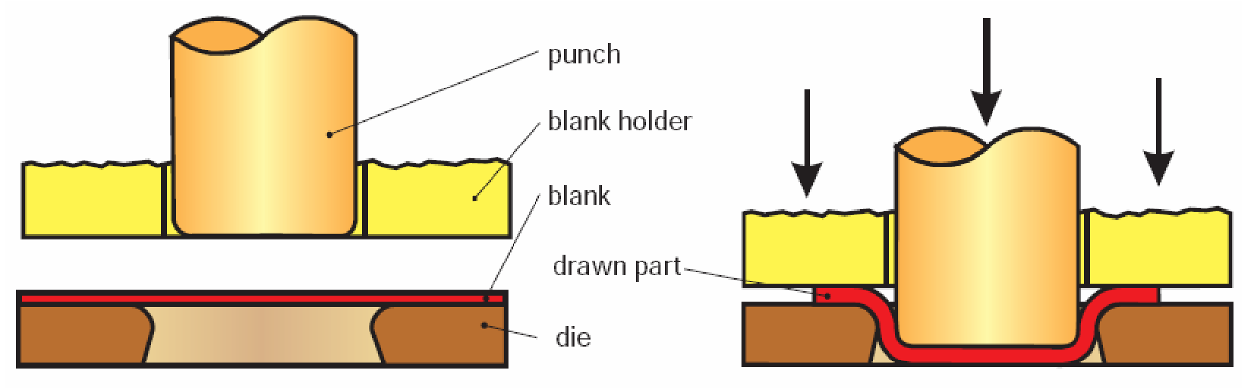

Figure 1 Single-draw deep drawing with blank holder (Schuler Metal Forming Handbook, 1998)

The diagrams of Keeler and Goodwin together give the values of ε1 and ε2 at fracture. This diagram currently is called the forming limit diagram; sometimes it is called as the Keeler – Goodwin Diagram shown below.

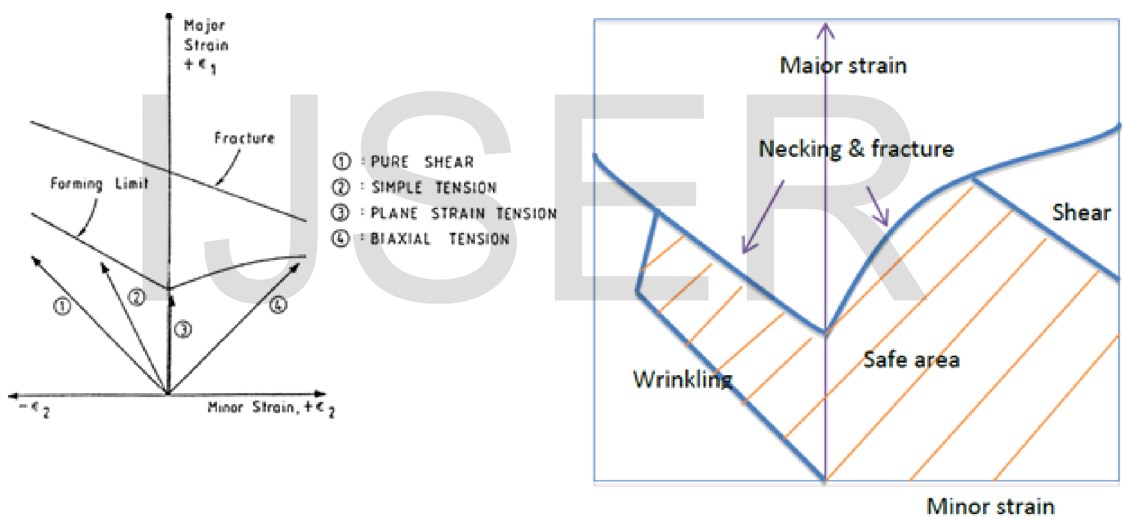

Figure 2 Keeler-Goodwin diagram or Forming limit diagram (FLD) [4,11]

The forming limit diagram (FLD) is a very effective graphical representation of optimizing sheet metal forming operations. The sheet is deformed by stretching sheet over a dome shaped die set. Strips having different widths they have taken for the test, therefore, it will induce uni-axial or biaxial stresses and due to these stresses the circles will deform into elliptic shapes. The change in strain has been expressed as the percentage change in length along two principal directions of the major and minor axes and the exact strains for failure as measured near necks or fracture. This change in strain can be plot on Keeler-Goodwin forming limit diagram and this diagram gives the limiting strains ratio for safe deformations. It is combination of strains along the axes represents the limiting curves in the Keeler-Goodwin diagram to prevent failure defect in the sheet metal parts. Below the curve represent safe deformations based on this we can go for further decisions. The safe zone in which no failure is there as shown as shaded region and the outside this zone there are different modes of failure represented at different strains combinations. In diagram shows the upper part of the curve is safe zone indicates necking and fracture region. The slope of the right hand side curve (i.e necking curve) can be decrease with an increasing the strain hardening exponent (n) value. Also, the slope of the neck curve can be reduced by changing the parameters like sheet thickness, chemical composition, and grain size structure. The safe region on diagram is narrowed down by biaxial stress level. Sheet thickness also has a major effect on FLD and by increasing sheet thickness it will increases the FLD and the FLD curve as shown in figure 3. [11]

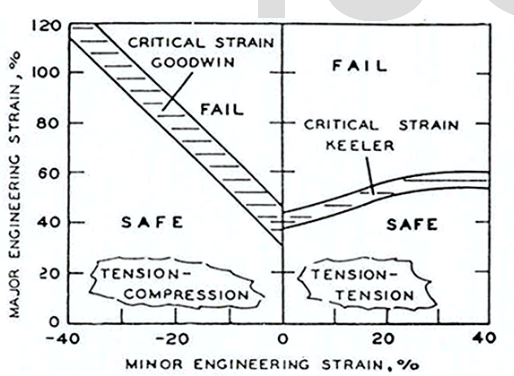

Figure 3 FLD Diagram [11]

The basic geometrical and material parameters are given for simple cup drawing operations for the process output. The following parameter has defining the tool geometry as shown in figure 4.

- Die radius (D), Punch radius (Pr), Blank radius (Br), Punch fillet radius (Rp), Die fillet radius (Rd), and Blank thickness (t)

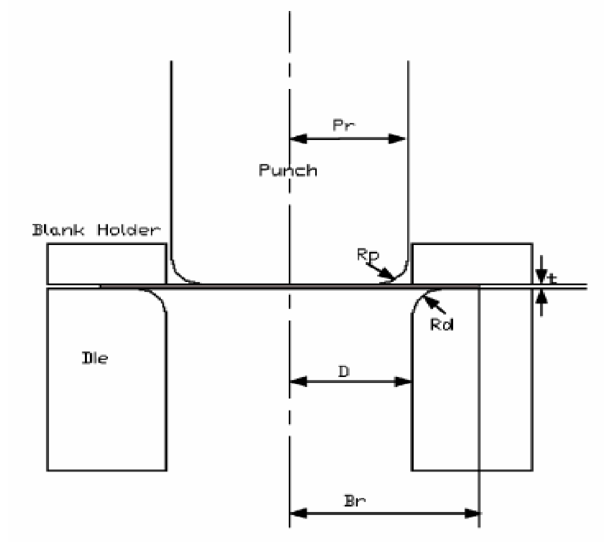

Figure 4 Geometrical and Material Parameters [5]

The physical parameters in deep drawing operations are studied and these parameters are blank holding force, Punch velocity, Lubrication, sheet blank material properties [5].

In this paper of wipe bending metal forming process springback effect has been studied. Spring back is the major defect in sheet metal forming operations, this defect occurs due to elastic recovery during unloading the punch from the die block. The purpose of this work is prediction and investigation of spring back and its causes of the spring back of sheet metal copper alloy. In this study for predicting spring back effect of wipe bending process has developed in finite element method code ANSYS APDL 14.0 platform. This simulation results investigate major affecting parameter is die radius on springback. From this analysis, they are suggested die shoulder should be small then we can reduce springback radius if die radius increase then the springback effect will be more. Also the punch nose radius has a significant effect on the springback effect and up to certain limit there is no major effect as compared to the die radius on springback effect. The die gap, clearance and sheet metal thickness also has a effect on springback and from this clearance parameter they are suggested to maintain minimum die gap, this gap could be the thickness of the sheet metal it means we can reduced the springback and tearing defects. They are also proposed to over bend operation to compensate the springback effect. This simulation results they are validated with a theoretical computation. Final it shows that good agreement with the simulated results. [6]

The purpose of this study is to determine the Forming Limit Stress Diagram (FLSD) of sheet steel of grade SPCE 270 by using numerical Finite Element Method (FEM). The failure criterion in sheet metal forming process for formability prediction and the conventional Forming Limit Diagram (FLD) has been used. For determining the stress based criterion or the Forming Limit Stress Curve (FLSC) and the FLD has initially determined by the Limiting Dome Height (LDH) test experiment. The LDH test has input for simulating in FEM and the FLD data as a failure criterion. The Calculated major and minor stresses have been used to draw the FLSC. After, this analysis validated these two FLD and FLSD criterion with an industrial case study and also they are investigated to FLD is insufficient strain path changes being manufactured in complex forming process part. Finally they are concluded that FLSD is a more precise tool to manufacture forming process or formability of sheet part as shown in figure 5. [7]

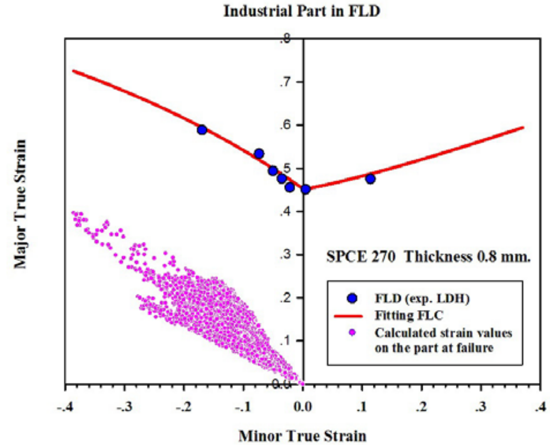

Figure 5 Verification of the FLD criterion with an industrial part made of SPCE 270 [7]

The necking and fracture behavior of a conventional HSLA steel and two AHSS for three different strain paths was determined using tensile, cup forming and stretch forming tests. While all materials showed similar tensile properties, significant differences in thinning and fracture behavior were observed in the cup and stretch forming tests. This has been related to differences in microstructure. FEA performed for both forming processes gave a good representation of the material behavior of all steel types for the cup drawing process. In contrast to that the FEA model failed to represent the forming behavior of all steels for the stretch forming process. This indicates that a more sophisticated material model might be necessary to represent the forming behavior of conventional as well as advanced high strength steels in stretch forming [8].

In this work, the mild steel used to draw a component in a single stage deep drawing process

ISSN 2229-5518

of thin walled, conical back plate of radial impeller of blowers and it is simulated in finite element tool. For modern manufacturing simulation study has been required to avoid the expensive and difficult experiments before actual production operations. Simulation study is used to determine stress distribution in the drawn and stretching component for a particular displacement. The study has conducted on ANSYS12.0. Two models test has been design and both models constructed of axisymmetric, quad 4 nodes, PLANE 42 elements used to simulate the drawing process for checking drawing quality of mild steel grade IS2062. The experimental work is carried out on two different flat plates having their thicknesses of 3 mm and 5 mm from this conical back plate is manufactured and this study will be more useful those who are working as a tool designer. [9]

In this work, forming limit diagram and the forming limit stress diagram obtained on Aluminum alloy3105 experimentally. For this analysis tool used is finite element code ABAQUS/Standard. The FLD and FLSD predicted for ductile fracture criteria and there is good agreement with simulation results and experimental results. [10]

Advantages and Disadvantages

The solution gives us technical support for taking manufacturing decisions through this we can improve productivity and quality of the product. The day by day the customer needs and wants are increased andalso due to the competition of the manufacturing as well as selling price of the products in the market are reducing continuously in the market through this study we can satisfy our customers by selling our product in a affordable price to the customers.

This tool gives us a numerical or approximate solution for our product and for getting required solution their number of iteration will be increased.

Conclusion

A Finite element method or Analysishas been used for the Linear and Non-Linear Static and Dynamic forces predictions like failure criterion, Limiting Dome Height (LDH), Conventional Forming Limit Diagram (FLD), Vonmises Stresses, Flow curve of selected material, Formability and Forming Limit Stress Diagram (FLSD)for sheet metal forming. To obtain the FLSD and FLD FE simulations of the limit dome height testing have performed and maximum and minimum in-plane stresses for the critical area of tested samples can be evaluated. The FLSD criterion was verified with experimentally by selecting an automotive part on any material. It will be concluded that the conventional forming limit diagram FLD is insufficient for predicting material failure in forming processes with accuracy, whereas the forming limit stress diagram FLSD can reveal formability limit of material more precisely solution for Manufacturing decision. Prediction, the FLSD calculated by FE simulations coupled with Hills yield anisotropic yield criterion provides a better prediction than FE simulations using isotropic hills yield criteria model. By applying the FLD and FLSD criterion to the pressed automotive part it was observed that the stress based failure criterion FLSD can reproduce the local state at crack initiation more realistically than the strain based failure criterion FLD. The accuracy of the FLSD depends strongly on the yield function applied in the prediction.

This solution will be useful in Industry for enhancing their manufacturing decisions and process modification or process improvement for reducing manufacturing cost or process cost through which we canimprove productivity. The Industrial management is continuously searching innovative solution for manufacturing of automotive sheet metal parts. Through this idea they can take economical decision for their business. For the interchangeability of their parts in various countries for assembly and through this they are expecting close tolerance. This tool gives us a realistic solution or support for designing and manufacturing of Punch, Die, Blank Size, Cycle time, etc based on this we can take further decision for Manufacturing of Sheet metal parts.

References

1. Dipak C. Talele, Dr. Dheeraj S. Deshmukh, Mahesh V. Kulkarni, “Simulation & Parametric Finite Element Analysis for Sheet Metal FormingProcess”, Pratibha: International Journal Of Science, Spirituality, Business And Technology (IJSSBT), Vol. 2, No. 2, May 2014,ISSN 2277—7261.
2. Sachin Ramdas Jadhav, Sunil Hiraman More, Swapnil S. Kulkarni, “Die Design For Formed Component Using Inputs From Fea For Determining The Most Suited Values For The Design/ Process Parameter”, International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974.
3. MohamadJazmir, “Experimental And Finite Element Evaluation Of Bending For Stainless Steel,”Universiti Malaysia Pahang, 2012.
4. K Mohan Krishna1, M Kumara Swamy“Prediction of Draw Ratio in Deep Drawing through Software Simulations”, International Refereed Journal of Engineering and Science (IRJES) ISSN 2319-183X, 2319-1821 Volume 3, Issue 12 (December 2014), PP.24-29.
5. Michael P.Pereira, MatthiasWeiss, BernardF.Rolfe, TimB.Hilditch, “Investigation Of The Deep Drawability Of Steel And Aluminum Sheets By Finite Element Simulation ”, Metallurgical And Materials Engineering, 2005
6. AmulBiradar, M.D Deshpande, “Finite Element Analysis Of Springback Of A Sheet Metal In Wipe Bending Process”, International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064.
7. SansotPanich, VitoonUthaisangsuk, JittichaiJuntaratin, “Determination of Forming Limit Stress Diagram for Formability Prediction of SPCE 270 Steel Sheet”, Journal of Metals, Materials and Minerals, Vol.21 No.1 pp.19-27, 2011.

8. C. Nikhare, P. V. Marcondes, "Experimental And Numerical Evaluation Of Forming And Fracture Behaviour Of High Strength Steel", 1School of Engineering and Technology, Faculty of Science and Technology,*Deakin University, Waurn Ponds 3217, Australia*.
9. ShishirAnwekar, Abhishek Jain, " Finite Element Simulation of Single Stage Deep Drawing Process for Determining Stress Distribution in Drawn Conical Component " International Journal Of Computational Engineering Research (ijceronline.com) Vol. 2 Issue. 8.
10. Mehdi Safari, Seyed Jamal Hosseinipour, "Experimental and Numerical Analysis of FormingLimit Diagram (FLD) and Forming Limit StressDiagram (FLSD) " Materials Sciences and Applications, 2011, 2, 497-503
11. R. Chandramouli Associate Dean-Research Sastra University, Thanjavur-613401,NPTEL - Mechanical Engineering – Forming
12. R.S.Hingole&V.M.Nadedkar, "A study of spring back effect in bending component", International Conference International Deep Drawing Research Group IIT Bombay, pp 25-30(IDDRG 2012).
13. R.S.Hingole&V.M.Nadedkar, "A case study of spring back effect in Automobile sheet metal component", International Conference International Deep Drawing Research, CSM (Colorado School of Mines), Denver USA (IDDRG 2009).
14. Schuler, Hydroforming Handbook: http://www.schulergroup.com/major/us/index.html, 1988