Review of Finite Element Simulations in Sheet Metal Forming Processes

Nileema B.Patil[1], Dr.Sudarshana Badhe[2]

[1]Assistant Professor, Department of Applied Physics, Lokmanya Tilak College of Engineering, Navi Mumbai-400708
[2]Faculty, Devagiri College of Science, Aurangabad

---***---

Abstract – *Sheet metal forming processes like bending, deep drawing process, and hydro forming involves forming of sheet metal in required shape from blank. Simulation of these processes is highly essential for many reasons like high cost involved in experimentation, elements of uncertainty involved trial and error manufacturing forming dies due to elastic recovery.*
This research paper aims to review attempts made by several researchers in simulation of these processes using various ways,.

Key Words: Finite Element Simulation, spring back, Elastic recovery, Deep Drawing, V-bending, hydro forming, design of experiments (DOE)

1. INTRODUCTION

Many sheet metals forming operation involves highly non linear deformation processes. In this process lot of elastic strain energy in blank. This energy stored is released when forming pressure is removed [1].

This is undesirable defect is popularly known as spring back. This is undesirable due to following reason:-

i. This defect tends blank towards original geometry of blank thus forfeiting purpose of forming.
ii. Prediction of springback is very difficult and hence poses lot of constraints on die designer.

Many researchers have made attempts to predict spring back using finite element simulation using different codes.

Many researchers have simulated several sheet metal forming processes like deep drawing, V bending ,hydro forming for optimization of various parameters associated with sheet metal forming.

This research paper takes review of various attempts made by several researchers in this regard.

2. FINITE ELEMENT SIMULATIONS FOR SHEET METAL BENDING

Panthi, S.K. et al.(2007) used analyzed elastic recovery in sheet metal bending with the help of finite element simulation. This study examined the effect of load on spring b back with varying thickness and die radius[2].

Bahloul R.et al. used finite element simulation for the prediction of punch load and stress distribution during the wiping-die bending process. Here numerical simulation was mofelled using elastic plastic theory coupled with Lemaitre's damage approach. They used ABAQUS for finite element simulation. The punch load and stress distribution was be predicted in view of optimization using response surface methodology (RSM) based on design of experiments [3].

Guo, Y.Q.et.al.(1990) studied the influence of friction forces under punch and blank holders. They studied optimization of the initial blank contour and deep drawing [4].

Papeleux, L.et.al.(2002) investigated the impact of various parameters on spring back in a 2D draw bending [5].

Lee C.H.et.al.(1998) commented on capability of Fiite element simulations for prediction of blank shapes and strain distributions in sheet metal forming. They developed algorithm which is applied to cylindrical cup drawing, square cup drawing and oil pan drawing. Their work demonstrates close accuracy of numerical results[6].

Karafillis, A.P. and Boyce, M.C.(1996) suggested need of appropriate design of tooling and binder shape together. They used finite element methodology to analyze this manufacturing process. The tooling for the purpose was numerically designed using the algorithm with input from finite element methodology. Tooling was then manufactured using CNC machines .This tooling was found to be producing accurate parts showing efficiency of finite element methodology[7].

Joshi, Patil and Satao (2014)M.K.N., Patil, B.T. and Satao, M.S. (2014) provided detail literature review about optimization aspects of deep drawing as well as use of finite element simulation in this area. This comprehensive review article clearly reveals importance of finite element simulation in sheet metal forming simulations[12].

Chung, K. and Shah, K.(1992) stated need of accurate description of anisotropic material behavior. They obtained finite element simulation results using Barlat's six-component anisotropic yield function. They used yield function in ABAQUS for modeling hydraulic bulge and cup drawing tests for a 2008-T4 aluminum alloy.FEM solutions showed closed proximity to experimental results showing suitability of Finite element methodology in sheet metal forming simulations[8].

Shinde, R.A., Patil, B.T. and Joshi, K.N.(2016) optimized various aspects of tube hydro forming process (without axial feed) using finite element simulations. Essentially tube hydro forming is the process of manufacturing light weight parts by passing pressurized fluid through tube. Modeling hydro forming process is complicated but essential for accurately producing parts. Shinde et al, developed 3-dimensional finite element model for tube hydro forming using Creo Parametric 2.0.They used Hyper Mesh for pre processing and used LS-DYNA explicit solver[9].

Following factors were considered in their study:-

i. Die corner radius,

ii. length of tube,

iii. thickness of tube

iv. internal pressure

They varied all these factors in three levels using finite element simulation .Their methodology of optimization involves following two major aspects:-

i. Taguchi's design of experiment

ii. Finite element simulations.

They confirmed their results using ANOVA.

Combined methodology used by Shinde et.al. throws light on hydro forming process showing significant effect of die corner radius, length of tube and thickness of tube on thickness reduction and also revealed that optimum combination of various parameters is function of length of tube i.e. for different length of tubes, different combination of factors exists for optimized results. Their results are based on finite element simulations and shows importance of FEA simulations in sheet metal forming simulations[9]

Joshi, Patil and Satao(2014) studied optimization of variation in wall thickness of deep drawn cup using combined methodology of design of experiments and finite element methodology. They called this methodology as virtual design of experiment. Their investigation involved the effect of die radius, sheet metal thickness and blank holder force on wall thickness variation in cup drawing using finite element simulation[10].

Joshi, Patil and Satao(2014) made geometry of tooling with the help of PRO-E and then imported using Hyperform pre-processor. They used LS DYNA as solver and Hyper view is used as post processor.Major factors considered in their study are given below-

i. Die Draw radius

ii. Sheet thickness

iii. Blank holder force

Above factors were varied in three levels and simulations were carried out using FEA methodology.

Full factorial design was used which essentially needed 27 simulations. Here L27 array was used. They also confirmed results using ANOVA and also studied interaction effects.

They found significant interaction between die draw radius and sheet thickness. They also found significant interaction between sheet thickness and blank holder force.

However according to their study interaction between die draw radius and blank holder force was insignificant.

They found optimal combination of various factors for minimum thickness variation. Their results clearly reveals that optimal combination of various parameters also varies with thickness of sheet indicating that perfect generic optimal combination of various parameters for minimum thickness does not exists[10].

This important study clearly reveals need of finite element simulations in deep drawing related field.

Chan, W. M. et.al.(2004) studied spring back in the V-bending process. They used finite element methodology for analyzing spring back. They varied following die and punch parameters :-

i. punch radius,

ii. punch angle

iii. die-lip radius

They studied effect of above parameters on spring back.

They used ABAQUS for finite element simulation. Their analysis showed reduction in spring back angle of the valley region with increment of punch radius and punch angle[11].

3. CONCLUSIONS

Above all discussion clearly reveals role of finite element simulations in sheet metal forming processes. Various sheet metal forming processes like V-bending,deep drawing and tube hydro forming can be accurately modeled using finite element methodology. This saves lot of time of expensive experimentation. Various designing processes related to tooling associated sheet metal forming becomes easier as well as accurate due to this methodology. Optimization of various performance parameters can be easily achieved with virtual experiments conducted with finite element simulation methodology.

Over all we can conclude that finite element simulation methodology has excellent potential in modeling sheet metal forming and can be used to model many sheet metal processes in near future and help sheet metal designers in many aspects.

REFERENCES

[1] Forcellese, A., Fratini, L., Gabrielli, F. and Micari, F., 1996. Computer aided engineering of the sheet bending process. *Journal of Materials Processing Technology*, *60*(1-4), pp.225-232.

[2] Panthi, S.K., Ramakrishnan, N., Pathak, K.K. and Chouhan, J.S., 2007. An analysis of springback in sheet metal bending using finite element method

(FEM). *Journal of Materials Processing Technology*, *186*(1), pp.120-124.

[3] Bahloul, R., Mkaddem, A., Dal Santo, P. and Potiron, A., 2006. Sheet metal bending optimisation using response surface method, numerical simulation and design of experiments. *International journal of mechanical sciences*, *48*(9), pp.991-1003.

[4] Guo, Y.Q., Batoz, J.L., Detraux, J.M. and Duroux, P., 1990. Finite element procedures for strain estimations of sheet metal forming parts. *International Journal for Numerical Methods in Engineering*, *30*(8), pp.1385-1401.

[5] Papeleux, L. and Ponthot, J.P., 2002. Finite element simulation of springback in sheet metal forming. *Journal of Materials Processing Technology*, *125*, pp.785-791.

[6] Lee, C.H. and Huh, H., 1998. Blank design and strain estimates for sheet metal forming processes by a finite element inverse approach with initial guess of linear deformation. *Journal of materials processing technology*, *82*(1), pp.145-155.

[7] Karafillis, A.P. and Boyce, M.C., 1996. Tooling and binder design for sheet metal forming processes compensating springback error. *International Journal of Machine Tools and Manufacture*, *36*(4), pp.503-526.

[8] Chung, K. and Shah, K., 1992. Finite element simulation of sheet metal forming for planar anisotropic metals. *International Journal of Plasticity*, *8*(4), pp.453-476.

[9] Shinde, R.A., Patil, B.T. and Joshi, K.N., 2016. Optimization of Tube Hydroforming Process (without Axial Feed) by Using FEA Simulations. *Procedia Technology*, *23*, pp.398-405.

[10] Joshi, M.K.N., Patil, B.T. and Satao, M.S., 2014. Optimization of Variation in Wall Thickness of a Deep Drawn Cup using Virtual Design of Experiments. *Optimization*, *4*(5).

[11] Chan, W. M., H. I. Chew, H. P. Lee, and B. T. Cheok. "Finite element analysis of spring-back of V-bending sheet metal forming processes." *Journal of materials processing technology* 148, no. 1 (2004): 15-24.

[12] Joshi, M.K.N., Patil, B.T. and Satao, M.S. 2014 Review of Optimization Aspects for Metal Forming Process. An International Journal (ESTIJ),ISSN: 2250-3498,Vol.4,no.2 pp 71-75

AUTHOR PROFILE

Mrs Nileema Patil is having M.Sc. degree in Physics. Currently she is working as Assistant Professor in Applied Physics department at Lokmanya Tilak College of Engineering; Navi Mumbai .She is having total teaching experience of 12 years

Dr.Sudarshana Badhe is having doctorate degree in physics. She is currently working as faculty at devagiri college of science, Aurangabad