(IJRST) 2016, Vol. No. 6, Issue No. I, Jan-Mar e-ISSN: 2249-0604; p-ISSN: 2454-180X

SPRING BACK ANALYSIS OF PERFORATED STEEL SHEET METAL USING FINITE ELEMENT METHOD

***Sagar L Uniyal, *Anit Jain, *Lalitkumar R Soni, *G. Venkatachalam**

**School of Mechanical and Building Science,VIT University, Vellore, India.*

ABSTRACT

Sheets are deformed to required shapes by plastic bending with the help of punch and die set in sheet metal working. However, after withdrawing the applied load, the contour assumes a different shape than that of die because of release of elastic stress in the metal. This elastic strain recovery is commonly called as springback. Springback effect is a major cause of concern for metal forming industries which leads to inaccuracies in final product produced and eventually leads to problems in assembly. This paper deals with spring back analysis of perforated steel sheet metal having circular holes arranged in square pattern using finite element analysis. The modelling of tools and blank are carried out in SolidWorks, meshing is carried out in HyperMesh and the analysis is carried out in Ansys Workbench to study the springback effect. An attempt is made to study the effect of various parameters like percentage of open area, hole size and blank thickness on springback of formed perforated steel.

Index Terms: Springback, sheet metal forming, FEM, Ansys workbench, Taguchi, blank.

INTRODUCTION

Metal forming process is one of the most widely used manufacturing processes. It is the process of plastically deforming the raw material into required shape and size. Drawing is the process in which a sheet is plastically deformed to the shape of die. The operation of producing many number of small size holes in the sheet is called perforating. The holes are of varied shape and size arranged in different pattern. These perforations reduces the weight of the component and finds its application in automotive industry, HVAC, Furnishing, interior design etc.

During forming, at the inside surface, compressive stress and outside surface tensile stress is induced. In the tensile zone even though the stress induced is greater than yield and less than ultimate, but there is some amount of elastic deformation is present. This elastic deformation will get recovered after withdrawing the load applied. Due to this, the radius of curvature of deformed component will be increasing and this is called as springback in forming operation.

PROBLEM DEFINATION

The precision in the dimension of the formed part is a major concern in sheet metal forming process due to springback effect. The springback is normally measured in terms of change in radius of curvature due to elastic recovery. In large metal sheets even a small springback effect can cause a large change in displacement leading to fabrication problems. The formed sheet metal and tool design should accurately incorporate the elastic strain recovery. The aim of this paper is to introduce a finite element approach to predict springback deformations in sheet metal forming and study the effect of various parameters like percentage of open area, hole size and blank thickness.

METHODOLOGY

With the help of metal forming simulations we can predict the springback which can lead to satisfactory tool design and select proper process parameters. This can reduce the cost involved during shop floor operation prior to metal forming operation. The modelling can be done in any of the CAD software packages. The sheets were modelled in SolidWorks. The Meshing was done in Hypermesh. The meshed tool and blank was imported to Ansys Workbench in IGES Format. The non linear material property was assigned to steel during analysis and with 3 step loading condition the springback was predicted for each steel sheets. The parametric analysis was done using ANOVA (included is MINITAB package) to find out level of influence of each parameter on springback. The methodology followed is shown in fig 1

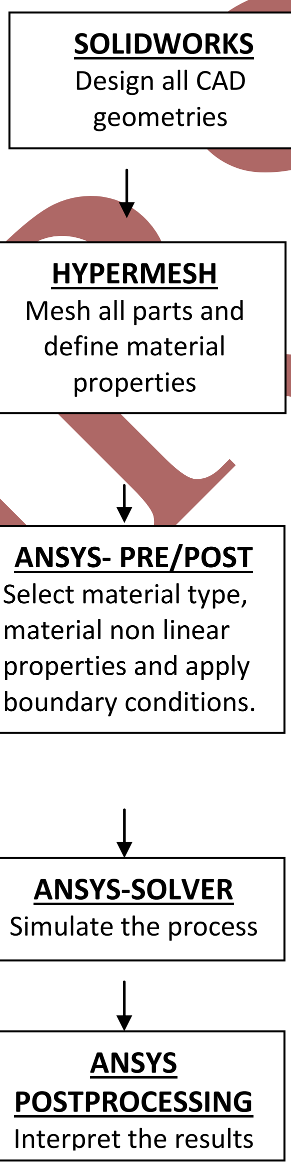


Fig 1 Methodology flow chart

(IJRST) 2016, Vol. No. 6, Issue No. I, Jan-Mar e-ISSN: 2249-0604; p-ISSN: 2454-180X

CAD GEOMETRY

The assembly had been modelled in SolidWorks. It consists of binder, punch, sheet and die. The sheet material is taken as steel where as tool material is tungsten carbide. The sheet size is taken as 100mm by 100 mm and thickness, size of hole and percentage of open areas are varied with a constant ligament ratio of 0.3.

HYPERMESH

The assembly from SolidWorks is then imported to Hypermesh in IGES format for meshing. The mesh element is taken as R-Tria for curved surfaces and quad element for flat surfaces and geometric clean up is done. The meshed assembly is then exported in IGES format to Ansys workbench for pre-processing and post processing. The meshed assembly is as shown in Fig 2

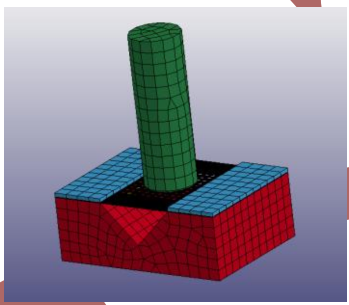


Fig 2- Set up of meshed assembly in Hypermesh

The meshed assembly is then saved as IGES file. This IGES file is imported in ANSYS. All other constraints and parameters are applied over assembly in ANSYS.

PRE-PROCESSING

In ANSYS material assign to die, binder and punch is tungsten carbide. Blank material is structural steel. Linear and non linear analysis is done on the assembly.

POST-PROCESSING

A Total of 9 combinations are simulated in ANSYS by considering factors mentioned in TAGUCHI'S method. Table 1 shows the various lists of combinations.

(IJRST) 2016, Vol. No. 6, Issue No. I, Jan-Mar e-ISSN: 2249-0604; p-ISSN: 2454-180X

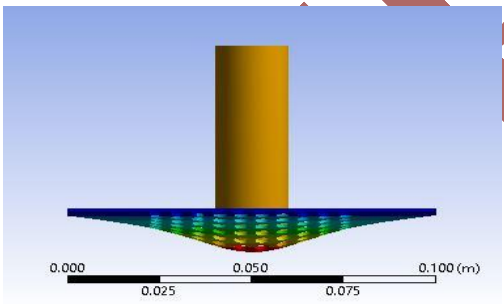
Table 1-List of combinations

Sheet no	Hole diameter	Open area %	Blank thickness
L1	S1	O1	T1
L2	S2	O1	T2
L3	S3	O1	T3
L4	S1	O2	T2
L5	S2	O2	T3
L6	S3	O2	T1
L7	S1	O3	T3
L8	S2	O3	T1
L9	S3	O3	T2

S1 = 4mm O1 = 15% T1 = 0.5mm
S2 = 8mm O2 = 20% T2 = 1.0mm
S3 = 12mm O3 = 25% T3 = 1.5mm

RESULTS

Fig 3a shows the simulation when the displacement is applied to the punch and Fig 3b shows the simulation when the displacement is withdrawn for sheet no L4.



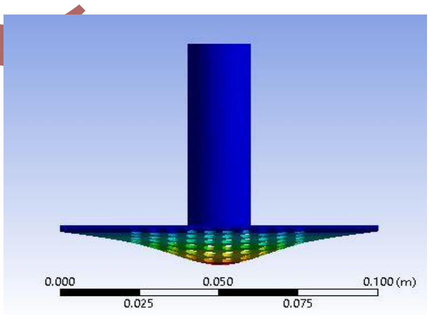


Fig 3a- displacement is applied to the punch

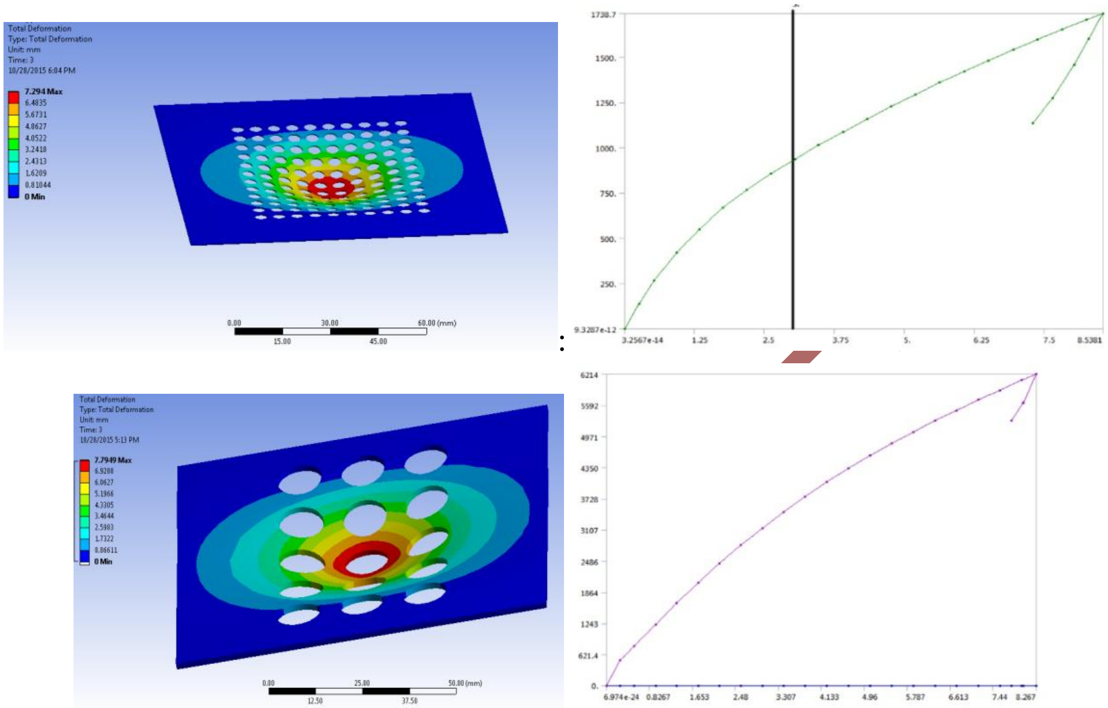


Fig 4- Maximum displacement and springback for Sheet L1

Figures 4-7 shows the sample simulations with respective springback graph

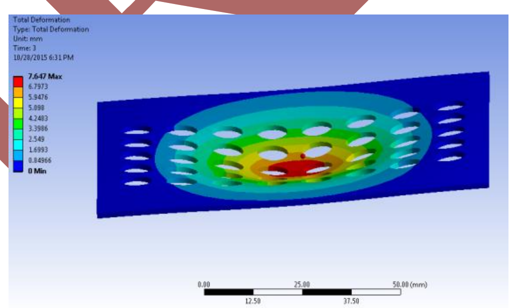


Fig 5- Maximum displacement and springback for Sheet L3

Sheet no	Hole diameter	Open area %	Blank thickness	Spring Back (mm)

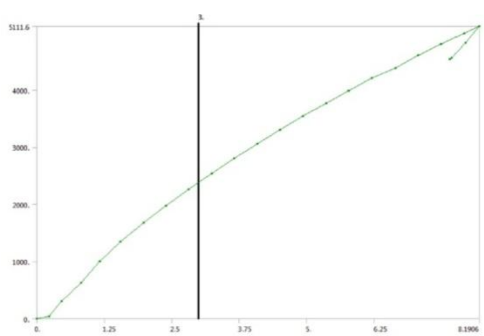


Fig 6- Maximum displacement and springback for Sheet L5

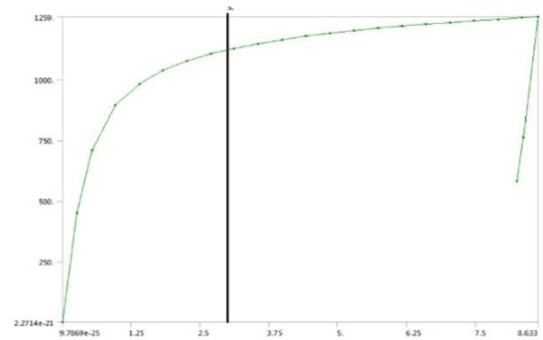


Fig 7- Maximum displacement and springback for Sheet L5

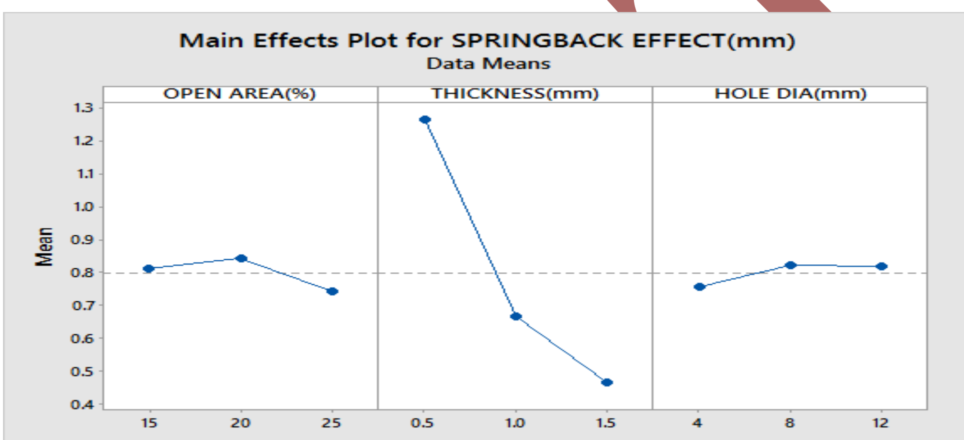


Fig 9-Main effect plot for springback

Table 2-Result Table

L1	S1	O1	T1	1.2441
L2	S2	O1	T2	0.7161
L3	S3	O1	T3	0.4721
L4	S1	O2	T4	0.6428
L5	S2	O2	T5	0.5436
L6	S3	O2	T6	1.3415
L7	S1	O3	T7	0.3811
L8	S2	O3	T8	1.2125
L9	S3	O3	T9	0.6413

S1 = 4mm O1 = 15% T1 = 0.5mm
S2 = 8mm O2 = 20% T2 = 1.0mm
S3 = 12mm O3 = 25% T3 = 1.5mm

CONCLUSION

Following are the conclusion for the springback effect on perforated steel sheets:

- The springback effect value increases with increase in size of the hole.
- The springback effect value decreases with increase in thickness of the blank.
- The springback effect value increases with increase in percentage of open area and then decreases.

ACKNOWLEDGMENT

The authors would like to express our deep gratitude to our project guide Dr. G. Venkatachalam, Associate Professor, SMBS, VIT University, Vellore, for his patience guidance, enthusiastic encouragement and useful critiques of this project work. We also take this opportunity to express our profound gratitude to all the non teaching staff for their help and guidance.

REFERENCES

[1] A.G. Gardiner, The spring back of metals, Trans. ASME (1957) 1–9.

[2] Dr. G. Venkatachalam, Influence of hole size, hole shape and hole pattern on spring- back effect on perforated sheet metals using FEM, International Journal of Engineering Science and Technology (IJEST).

[3] Narkeeran Narasimhan, Michael Lovell, Predicting spring back in sheet metal forming, ELSEVIER, Volume 36. April 2012, Pages 331–332

[4] K.P. Li, W.P. Carden, R.H. Wagoner, Simulation of springback, International Journal of Mechanical Sciences, Volume 44, Issue 1, January 2002, Pages 103–110.

[5] Claudiu D, "Transient Structural FEA of bending a lamella with plasticity and spring back effect", Finite element analysis, Ansys workbench tutorial.