Spring Back Analysis of High Strength Steels

Vasundhara Sutar, PG Student, Department of Mechanical Engineering, AISSMS COE, PUNE,
C.S.Dharankar, Assistant Professor, Department of Mechanical Engineering, AISSMS COE, PUNE,
B.Thirupathi Raju, Senior Manager, ERC TATA MOTORS, PIMPRI, PUNE-18

***Abstract*— In sheet metal forming, the spring back is the geometric change made to a part at the end of the forming process. This happens when the part has been released from the forces of the forming tool and due to uneven force distribution along the cross section. After the completion of sheet metal forming, deep-drawn and stretch-drawn parts undergo spring back and hence the dimensional accuracy of a finished part is affected. The final form of a part is changed by spring back, which makes it difficult to produce the part. As a result, the manufacturing industry faces some practical problems like, prediction of the final part geometry after spring back and appropriate tools must be designed to compensate for these effects. During the development of tools, spring back is compensated by software in order to remove the component from the tool to get the required dimensions. As a result, intense tryout loops that occur at a very late stage in the development of the tool can be reduced to a minimum. Simulation software can not only detect spring back early on, it can also compensate for it. Hence we can improve the tooling processes and the manufacturing costs can be decreased significantly. Spring back compensation thus minimizes the risk of costly changes to tools or processes later on. In this paper, the spring back analysis of different sheet metal material necessary steps for the compensation of the spring back angle has been studied.**

***Index Terms*— Sheet metal, CAD, FEA, Spring Back, forming**

I. INTRODUCTION

SPRING back refers to elastic recovery of non uniform stress distribution in a deformed part after the removal of bending forces. Finite element simulation of sheet metal forming is an efficient tool which issued in industrial practice especially in Stamping die Production, to eject the geometrical defects caused due to elastic spring back. This phenomenon results in a change of the actual designed part geometry from that desired in the design phase and hence can cause significant problems during assembly. In order to keep the product development time and manufacturing Costs low, finite element analysis (FEA) aims to provide reliable information required for the desired modification of tool and product geometry, before the tool manufacturing. Therefore, the accurate information obtained from the numerical simulation of spring back is essential for the development Engineers.

In forming process, due to the plastic-elastic characteristic of metals it is typical that any deformation of sheet metal consists of both elastic as well as plastic deformation. On the removal of the metal work piece from the tool or deformation implements that the elastic deformation regains its original shape and only the plastic deformation persists. When a metal forming tool is planned and constructed to deform a work piece, the shape taken by the tool will be a combination of elastic and plastic deformation, on the release of the elastic deformation, spring back is often observed at the end of a metal forming process. This spring back has to be compensated or corrected to get an accurate result.

In this paper, the spring back analysis (elasticity of sheet metal after forming) of sheet metal components made up of High Strength Steels (HSS) is studied. A sheet metal component from automobile is chosen for analysis. The analysis of Bracket Shock Absorber mounting is carried out. Using Hyper form simulation, spring back analysis of sheet metal formed component by applying different materials for analysis is performed. Different techniques are applied to the tool geometry to minimize the spring back. The actual tool profile in the analysis is represented by computer-aided design (CAD) model of the tool geometry. Springs back are checked in the finite element analysis (FEA).

II. LITERATURE REVIEW

M.Tisza , Z. Lukács[1] devised a new experimental technique .The device is used in tension as well as in compression cyclic loading to determine the important parameters needed for modelling the spring back behavior. They performed various numerical simulations with the experimentally measured γ, χ, and K and E values to study the effect of these material parameters that will affect the spring back behavior of the material grades.

S. Gawade and V.M.Nandedkar [2] presented **a** study of the factors affecting the spring back such as sheet thickness, material properties along with the tooling geometry etc. The paper also reviews the various parameters affecting spring back like ratio of die radius to the sheet thickness, blank holder force, coefficient of friction etc. predict the effect of these parameters on formability of a trapezoidal cup using Altair Hyper Form radios predictive tool.

Vorkov*, R.Aerens, D.Vandepitte, Joost R. Duflou[3]studied the spring back analysis of different types of HSS using the FEM element modeling using two standard types of elements; shell elements and solid elements. The calculations were used in commercial finite element software, Abaqus. They implemented a model to predict precise spring back and found out the accuracy in the function of the number of elements through the thickness. They also measured the deflections in the sheet during and after bending to study the effect of spring back on the sheet metal.

A. Lokhande, V.M Nandedkar.[4], studied the variations in the mechanical and dimensional properties of the incoming material, lubrication and other forming process parameters that cause spring back variation. they examined the behavior of sheet metals of different thickness' subjected to different blank holding forces and also studied their spring back behavior.

.

A. CAD Model

CAD model of existing chassis has been prepared in CATIA V5 as shown in figure 2. Cad model data is migrated into hyper form, for meshing

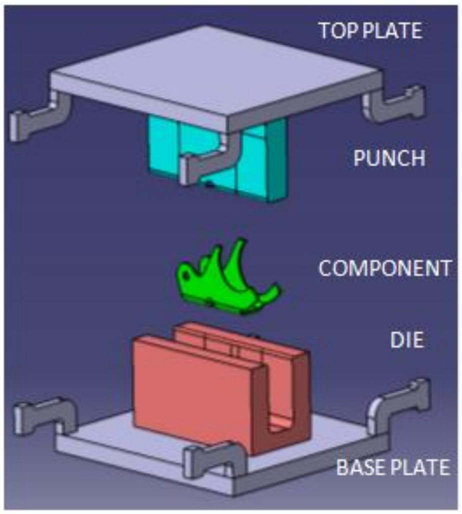

Fig.1. CAD Model

This tool model will be migrated into IGES format and meshing of punch,die and blank will be done by choosing appropriate mesh size,followed by defining the suitable parameters and simulated in HYPERFORM 12.0 for various Sheet metal materials and thickness The finite element analysis is carried out on HSS and materials, Like BSK 46 and E34 . From the analysis max spring back value and angles are evaluated. Table II shows the comparative analysis of properties of E34 and BSK46.

TABLE I
TOTAL WEIGHT OF THE TOOL

Sr No	DESCRIPTION	WEIGHT(T)	SIZE(mm)
1	PUNCH	0.024	300*170*63
2	DIE	0.039	300*180*60
3	TOP PLATE	0.049	450*400*32
4	BASE PLATE	0.049	450*400*32
	TOTAL WEIGHT	0.161	

TABLE II
MATERIALS APPLIED AND THERE PROPERTIES

S.No.	Material	Young's Modulus E	Poisson's Ratio υ	Density ρ(kg/m3)	Ultimate Tensile Stress σ_{uts} (GPa)
1.	E34	210 GPa	0.3	7.8 x 10^{-6}	0.4-0.5
2.	BSK 46	200 GPa	0.33	7.8 x 10^{-6}	0.5-0.62

TABLE III
LOAD CONDITIONS APPLIED

Draw Force	68 T
Tryout press	300T
Blank Size	300*275*5Thk

B. Analysis Results;

The analysis of the component is carried out using Hyperform. Altair Hyper Form is a comprehensive finite-element-based sheet metal forming simulation framework. Its unique process-oriented environment captures the forming process with a suite of highly tailored and configurable analysis and simulation tools. Hyper Form delivers a cost-effective solution that allows users to develop an optimal manufacturing process. The analysis of the automotive component by using HSS materials like E34 and BSK 46 is shown respectively and their formability and thinning analysis and has also been carried out. The component spring back analysis is shown in order to check their spring back angle. Given below is the analysis and comparison of two different types of High Strength Steels used in the automobile industry.

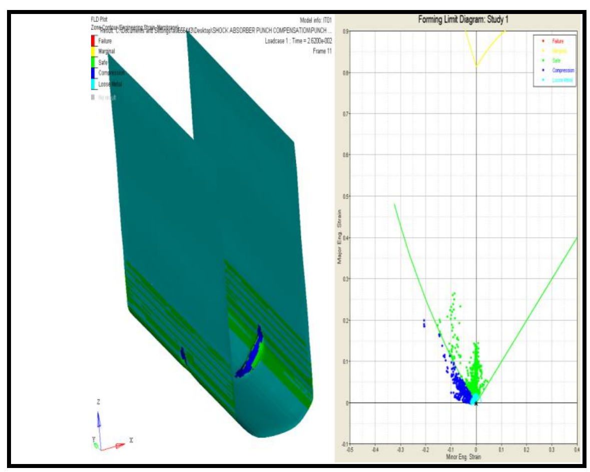

Fig .1. Formability analysis(E34)

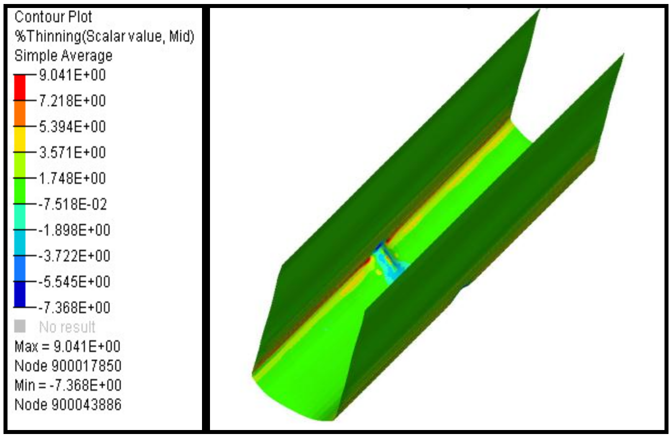

Fig. 2. Thinning analysis(E34)

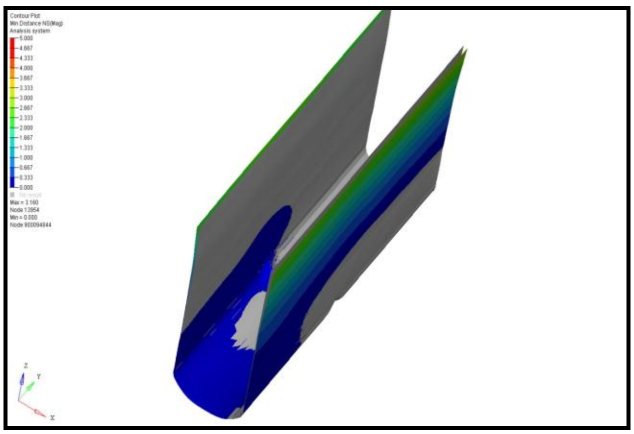

Fig. 3. Spring back analysis(E34)

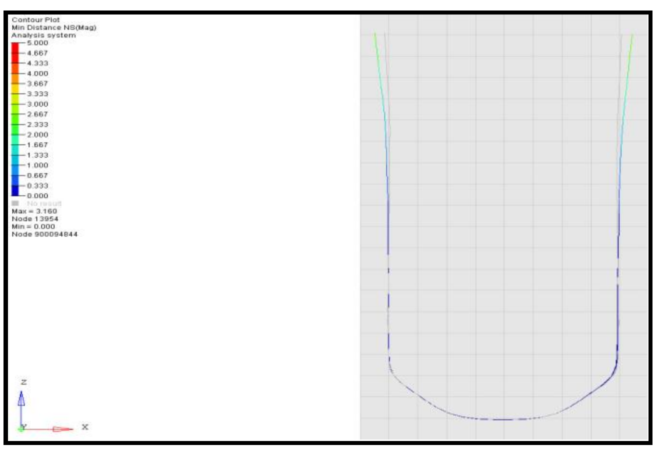

Fig. 4. Spring back result(E34)

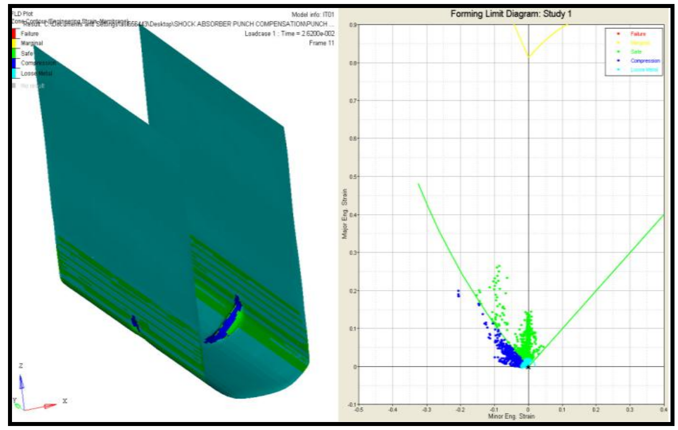

Fig. 5. Formability analysis (BSK 46)

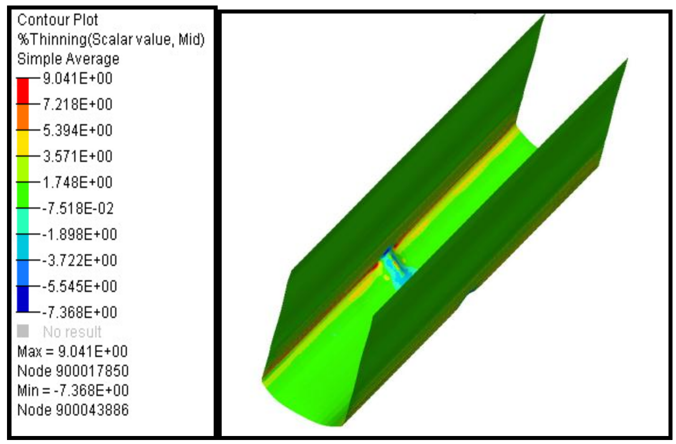

Fig .6. Thinning analysis (BSK46)

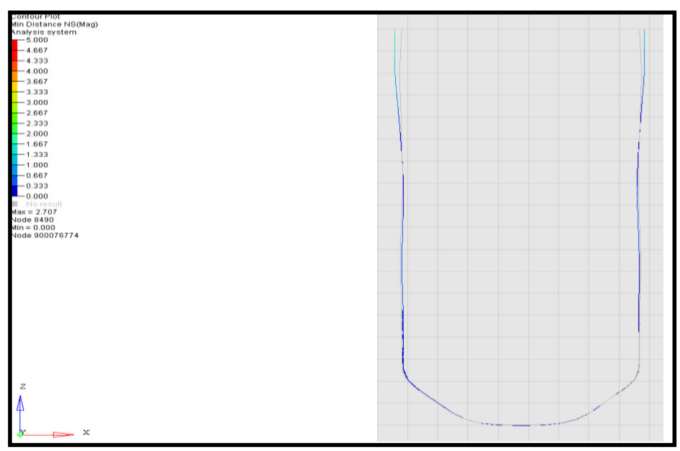

Fig. 7. Spring back analysis (BSK46)

C. Fabrication Work:

Fig. 8. Punch and die block before machining

Fig. 9. Die block after CNC machining

Fig. 10. Punch block after CNC machining

III. RESULTS AND DISCUSSIONS

From the analysis shown above, the comparison and results of the analysis is presented in the table given below. The spring back angle of the one of the automotive component is calculated by using HSS materials and then selected the material with least spring back angle for manufacturing.

TABLE III
ANNALYSIS RESULT TABLE FOR SAME RADIUS

Serial No.	CASE-1	CASE-2
Material	**E34**	**BSK46**
Radius	**R=8**	**R=8**
Thickness	**5mm**	**5mm**
Max Spring back value	**3mm**	**2mm**
Max Spring back angle	**4°**	**2°**

TABLE IV
ANALYSIS RESULT TABLE FOR VARYING RADIUS

Serial No.	CASE - 1	CASE - 2
Material	**BSK46**	**BSK46**
Radius	**R=8**	**R=5**
Thickness	**5mm**	**5mm**
Max Spring back value	**2mm**	**1mm**
Max Spring back angle	**2°**	**0.5°**

From the above analysis, it can be seen that the spring back angle is greater in case of E34 than BSK46. Hence, the material BSK46 is used for manufacturing. In order to further minimize the spring back effect, the compensation in radius is done by reducing the punch radius to get required results. Reduction in radius is one of the commercial effects on spring back for producing components.

IV. CONCLUSION

In this paper, the spring back behavior of the automotive component by applying two different HSS materials is studied. FEA Study has been done with help of hyper form 12.0. Experimental study on spring back has been done by considering HSS materials like, E34, BSK46 having component thickness 5mm. Spring back compensation is also carried out . Maximum Spring back value has been calculated by FEA for the material E34 is 4° and 2° for the material BSK46. The compensation method to reduce spring back in automotive component is carried out and it is found that the spring back angle can be further reduced by compensation method. Hence the component is manufactured with the material having the least spring back angle.

ACKNOWLEDGMENT

I would like to thank TATA MOTORS, PIMPRI, PUNE for giving me an opportunity to work in their organization for this project work.

REFERENCES

[1] M. Tisza, Z. Lukács. (2014). Springback Analysis of High Strength dual phase steels. *Procedia Engineering*.81(2014)975-980.

[2] S. Gawade , V. M. Nandedkar. Spring back in Sheet Metal Bending-A Review**.** *IOSR Journal of Mechanical and Civil Engineering.(IOSR-JMCE)*ISSN:2278-1684,PP:53-56E.

[3] V. Vorkav, R.Aerens, D.Vandepitte, J.R.Duflou.(2014). Springback prediction of high strength steels in large radius air bending using finite element modeling approach. *Procedia Engineering* .81(2014) 1005 – 1010.

[4] M. K. Choi, H. Huh. (2014). Effect of punch speed on amount of springback in U-bending process of auto-body steelsheets.*Procedia Engineering*.81(2014)963–968.

[5] A.M. Lokhande, V. M. Nandedkar. (2014).Effects of Process Parameters and Investigation Of Spring back Using Finite Element Analysis. *International Journal of Recent Development in Engineering and Technology*. Volume 3, Issue 1, (ISSN 2347-6435)

[6] Chikalthankar, G D Belurkar, V M Nandedkar.(2014).Factors Affecting on Spring back in Sheet Metal Bending. *International Journal of Engineering and Advanced Technology* (IJEAT). Volume-3.April 2014. ISSN: 2249 – 8958