

FINITE ELEMENT ANALYSIS OF DEEP DRAWING AND SPRINGBACK

Kareem Najm Hussien

ABSTRACT

The paper studies the distribution of stresses and strains of deep drawing process until the failure. The process is modeled with its four basic parts: the punch, the die, the holder and the blank. As the study concentrates on the blank, the blank is modeled as an elastic body, where the other parts are modeled as rigid bodies. The study is performed on two sets: the first set with Von-Mises plastic criteria & the second set with Hill criteria to take the anisotropy into consideration. For both sets the coefficient of friction is set to constant value. The study focuses on the failure analysis by changing the punch stroke until the value of the blank diameter to monitor the thinning of the cup thickness. Due to the importance of the springback in metal forming which determines the residual stress inside the blank, a model of springback is added. The study is performed using the finite element code ANSYS 11. The effect of punch stroke length and springback on the deep drawing process can be tested on the behavior of the metal forming.

KEYWORDS: Finite Element, Deep Drawing, Spring-back, ANSYS.

الموجز

INTRODUCTION

Deep drawing is the metalworking process used for shaping flat sheets into cup-shaped articles such as bathtubs, shell cases, and automobile panels. This is done by placing a blank of appropriate size over a shaped die and pressing the metal into the die with a punch (**Figure 19**). Generally a clamping or hold-down pressure is required to press the blank against the die to prevent wrinkling.

This is best done by means of a blank holder or hold-down ring in a double-action press. Although the factors which control the deep-drawing process are quite evident, they interact in such a complex way that precise mathematical description of the process is not possible in simple terms. The greatest amount of experimental and analytical work has been done on the deep drawing of a flat-bottom cylindrical cup (Swift test) from a flat circular blank. The discussion of deep drawing which follows will be limited to this relatively simple situation.

The spring-back is referred to an event when the sheet metal starts vibrating after the punch is removed and might alter its final desired shape. Since it takes some time before the work-piece comes to a rest, almost all papers depend on the finite element methods as a numerical tool for simulating the deep drawing process. A good general treatment of this method is given by (S. S. Rao, 2004) but application of this method to metal forming processes is given by (Shiro, 1989). The degree of the application of the finite element depends on the nonlinearities (A. Ayari, et al 2009). The uses of the ductile material may the magnesium (T. S. Yang, 2008) or Aluminum at elevated temperature to predict the final geometry of the blank (G. Venkateswarlu, et al 2010). The influence of material parameters such as hardening exponent, yield stress and elastic modulus on the process are investigated (GAO En-zhi, et al, 2009).

The investigation of the geometrical tolerances between the punch and the blank or between the blank and the die is studied (Waleed, 2007).

Since the process involves many parameters, minimization of response time and maximization of the efficiency and quality is studied as well (Hakim S. Sultan Aljibori, 2009).

THEORITICAL BACKGROUND

Deep drawing is the metalworking process used for shaping flat sheets into cup-shaped articles (Dieter 1988). It is done by placing a blank over a shaped die and pressing the metal into die by a punch. A holder is required to press the blank against the die to prevent wrinkling. The process has complicity to be modeled directly. The four parts of the process are shown in **Figure 1**. The stress and deformation in a draw cup is show in **Figure 2**. The draw ability of a metal is measured by the ratio of the initial blank diameter D_0 to the cup diameter drawn from the blank D_p . For a given material there is a limiting draw ratio (LDR), representing the largest blank that can be drawn through a die without tearing. In our study the efficiency for frictional losses is $\eta = 0.7$ and $LDR=2$.

This means it is a difficult to draw a cup with a height greater than its diameter.

MODEL DESCRIPTION

The geometry, the material properties and the applied load is needed in our analysis. The geometry of the assembly is shown in the **Figure 1**. The finite element model is shown in the rest figures. A fine mesh of elements type (PLANE182) is generated on blank geometry (100 mm of diameter and 2 mm of thickness). This element has four nodes of two degree of freedom, two translations and one rotation. The punch, the holder and the die are assumed to be solid bodies. The gap between the punch and die is set equal to the initial thickness of blank sheet. The material properties of the blank sheet are:

Modulus of Elasticity	200 GPa
Tangent Modulus	1 Gaps
Yield Stress	250 MPa
Poisson Ratio	0.3

The load is applied as a prescribed displacement representing the punch stroke with holder force 5 KN. The friction between the parts was set as a Coulomb friction of 0.1 coefficients.

Two sets of calculation were performed.

The first set with no anisotropy while the second set has anisotropy according to Hill as 10% (ANSYS Manual, 2007). Appendix A

FAILURE CRITERIA

Two types of failures were followed through the modeling.

The depth of the cup created should not exceed the diameter of the blank sheet (**Dieter, 1988**) and the thinning should be kept below 20% of the original thickness (**T. Altan, 2000**).

RESULTS AND DISCUSSION

Figure 3 shows the effective Von-Mises stress for both sets of calculation, while **Figure 4** shows the effective elastic strain.

Figures 5 to 9 show the distribution of effective Von-Mises stresses for the first set of calculation for different stroke of punch, while **Figures 10 to 14** show the results of the second set. The Von-Mises stress after spring-back for stroke of 60 mm is shown in **Figure 15**. For this stroke the Von-Mises strain is shown in **Figure 16**. From the figure, the punch is in its original position after spring-back.

Taking the anisotropy into consideration, **Figure 17** shows the Von-Mises stress for 60 mm stroke and **Figure 18** shows the Von-Mises strain.

By zooming the graphical results, the thinning of the wall is evident.

It is clear that **Figure 9** for the first set of calculation is under failure mode which corresponds to stroke of 90 mm; **Figure 13** for the second set of calculation corresponds to 80 mm punch stroke.

CONCLUSIONS

The obtained results of simulation present a visual approach to predict the failure shape of the process according to the thinning of the thickness of the formed cup. The spring back is another visual approach to the accuracy of the modeling. By this approach a parametric design could be done and an optimized selection of parameters could be achieved.

REFERENCES

Achievement in Materials and Manufacturing Process, 2008

ANSYS 11 "Help Manual", 2007

F. Ayari, T. Lazghab and E. Bayraktar, " Parametric Finite Element Analysis of square cup deep drawing", Computational Materials Science and Surface Engineering, 2009

GAO En-zhi, LI Hong-wei, KOU Hong-chao, CHANG Hui, LI Jin-shan and ZHOU Lian, "Influences of material parameters on deep drawing of thin-walled hemispheric surface part", Nonferrous Met. Soc. China 19, 2009

George E. Dieter, "Mechanical Metallurgy", 3rd Edition, McGraw-Hill Company, 1988

G. Venkateswarlu, M. J. Davidson and G. R. N. Tagore, "Finite Element Simulation of Deep Drawing of Aluminum Alloy Sheets at Elevated Temperatures", ARPN Journal of Engineering and Applied Sciences, 2010

Hakim S. Sultan Aljibori, Abdel Magid Hamouda. "Finite Element Analysis of Sheet Metal Forming Process ", European Journal of Scientific Research, 2009

S.S. Rao, "The Finite Element Method in Engineering", Pergamon Press, Oxford, 4th Edition, 2004.

Shiro Kobayashi, Soo-Ik Oh and Taylan Altan, "Metal Forming and The Finite Element Method", Oxford University Press, 1989.

T. Altan, "Forming Improving Progressive Die Design by Simulation", Stamping Journal. 2000.

T. S. Yang “Finite element analysis of elliptic cup deep drawing of magnesium alloy sheet”, Journal of

Waleed Khalid Jawad, "Investigation of Contact Interface between the Punch and Blank in Deep Drawing Process", Eng. & Technology, Vol.25, No.3, 2007

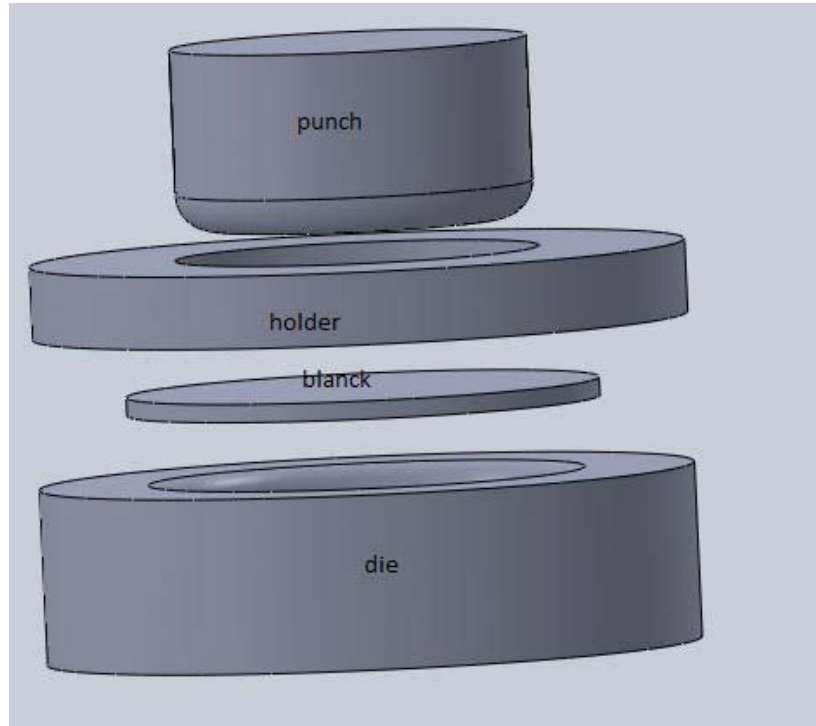


Figure 1 The four parts of deep drawing process

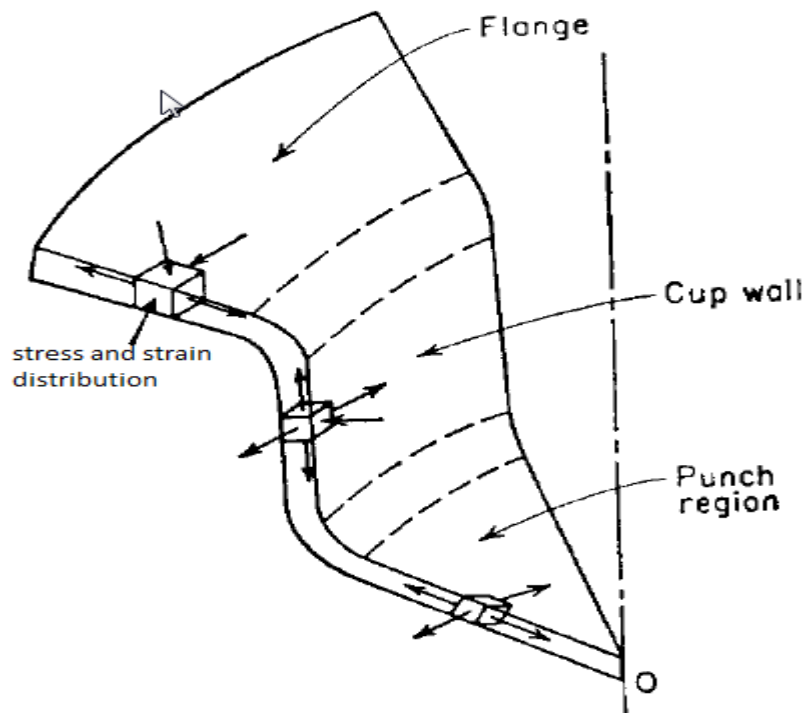


Figure 2 The stain and stress distribution inside the cup

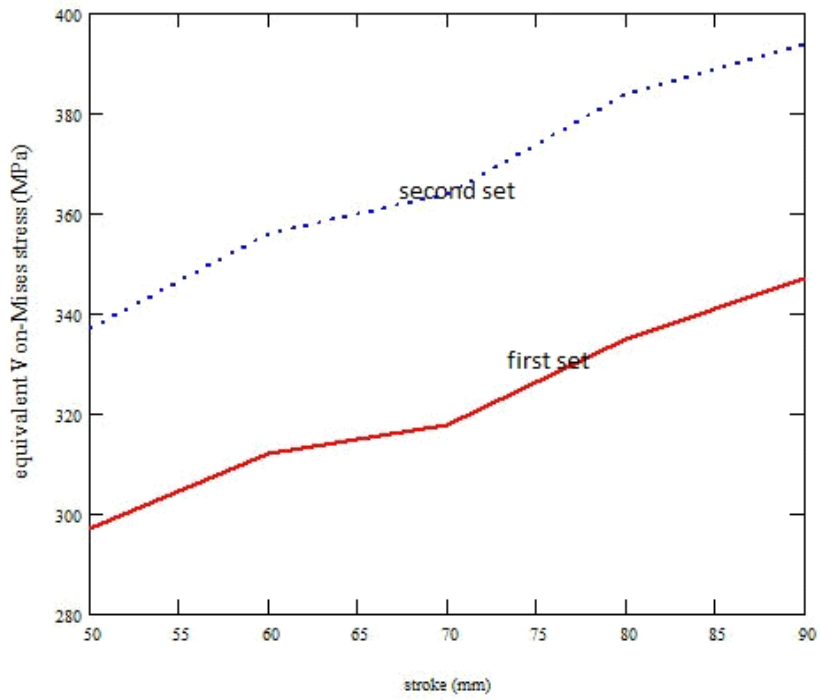


Figure 3 The equivalent Von-Mises stress against the length of punch stroke

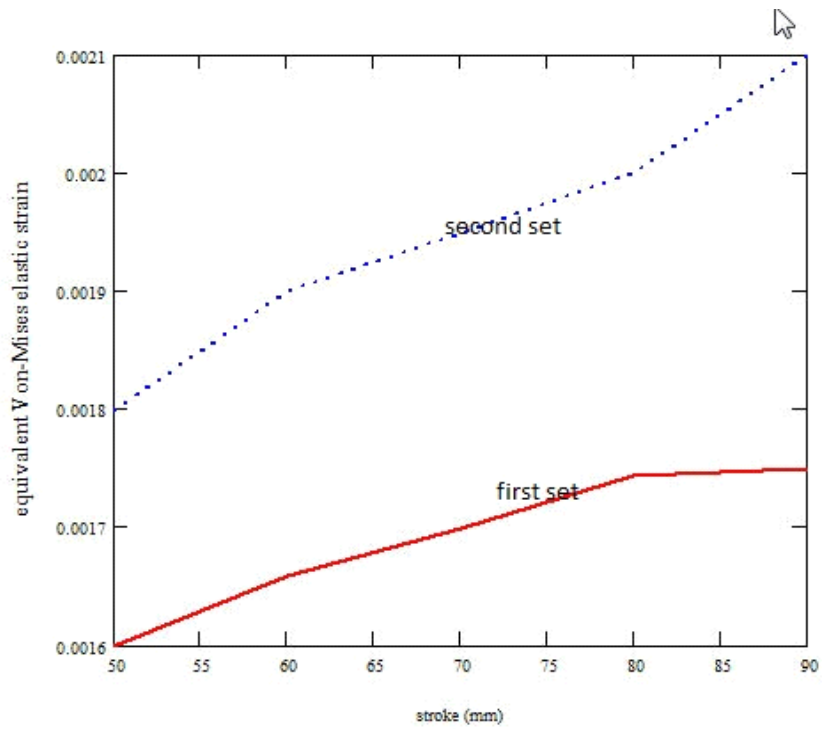


Figure 4 The equivalent Von-Mises strain against the length of punch stroke

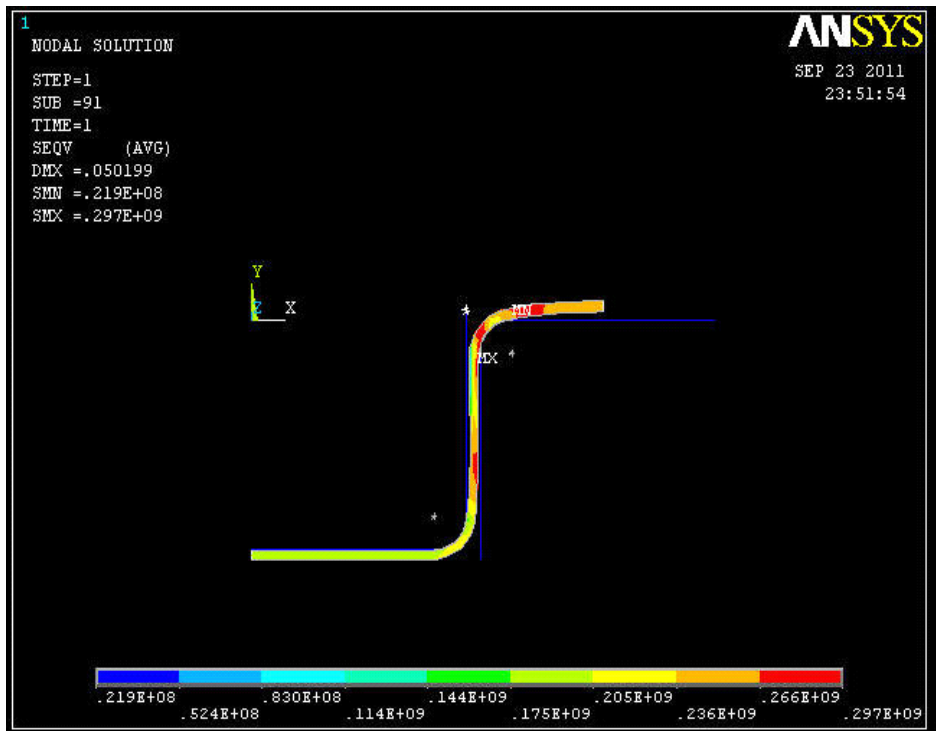


Figure 5 The Von-Mises stress for 50 mm stroke

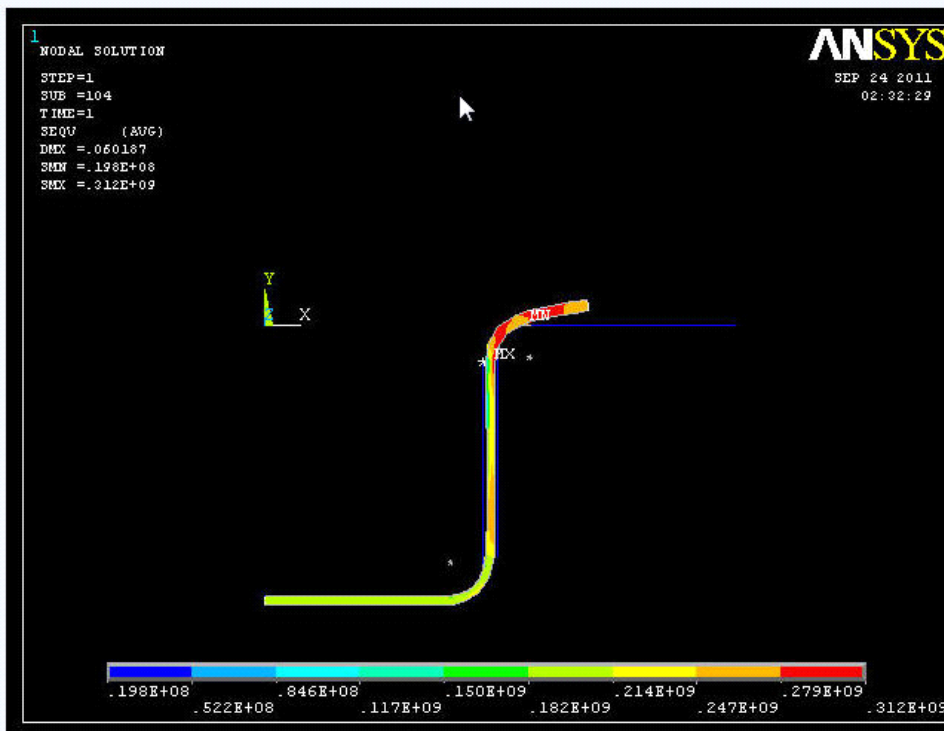


Figure 6 The Von-Mises stress for 60 mm stroke

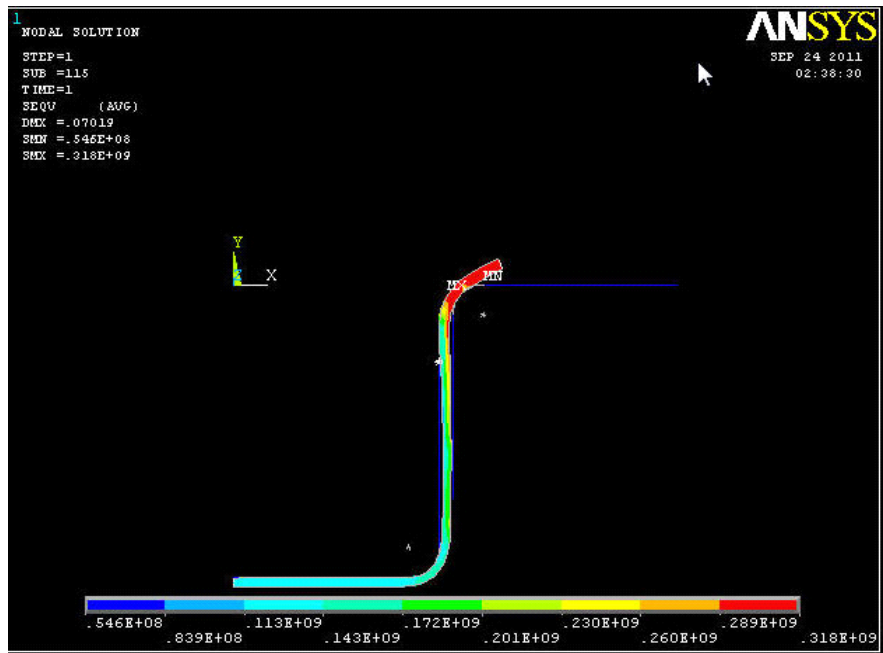


Figure 7 The Von-Mises stress for 70 mm stroke

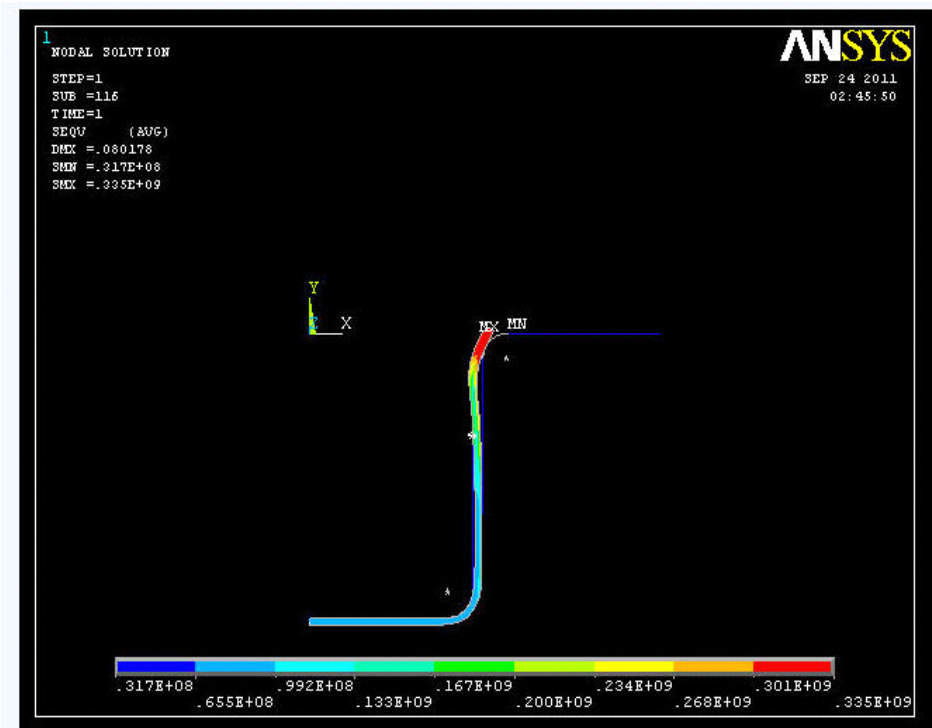


Figure 8 The Von-Mises stress for 80 mm stroke

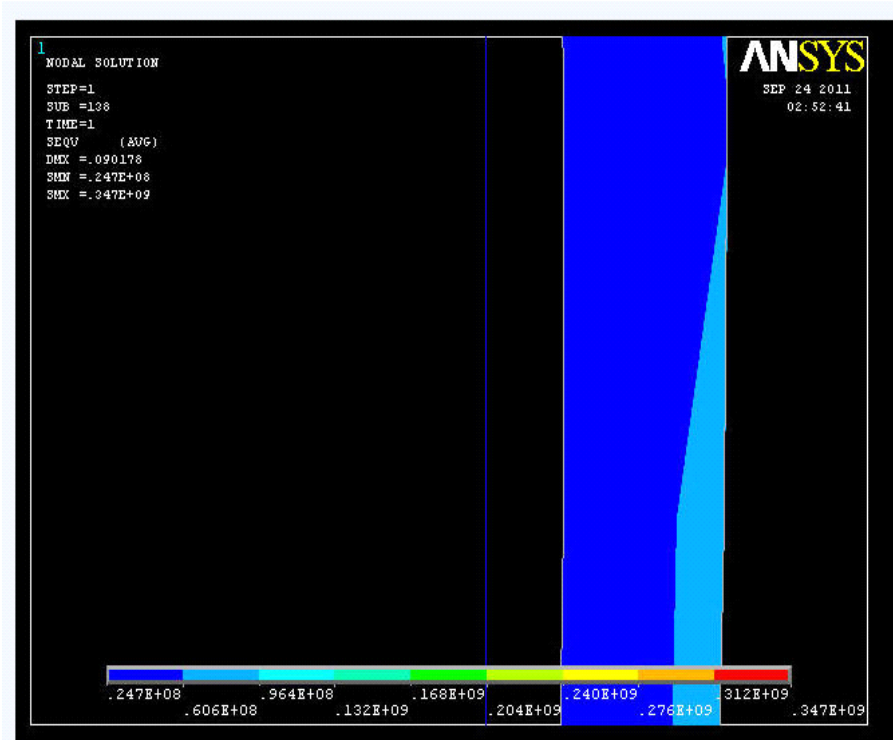


Figure 9 The Von-Mises stress for 90 mm stroke

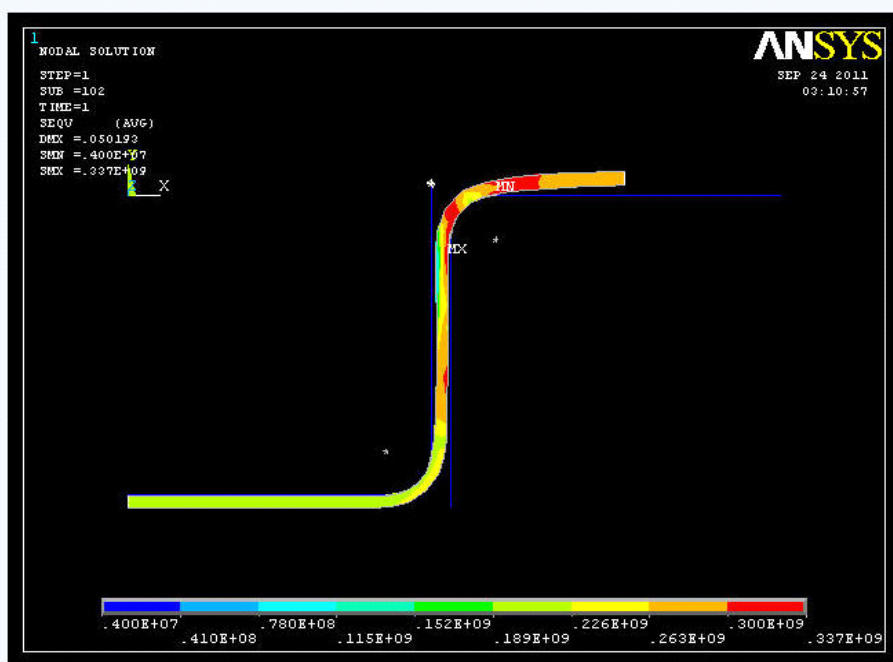


Figure 10 The Von-Mises stress for 50 mm stroke with anisotropy

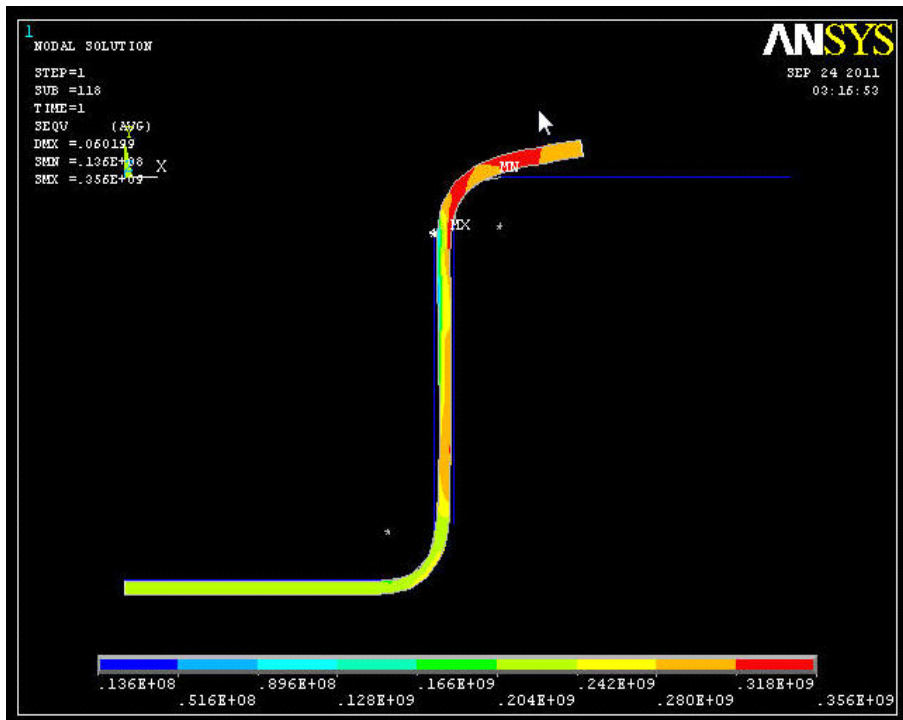


Figure 11 The Von-Mises stress for 60 mm stroke with anisotropy

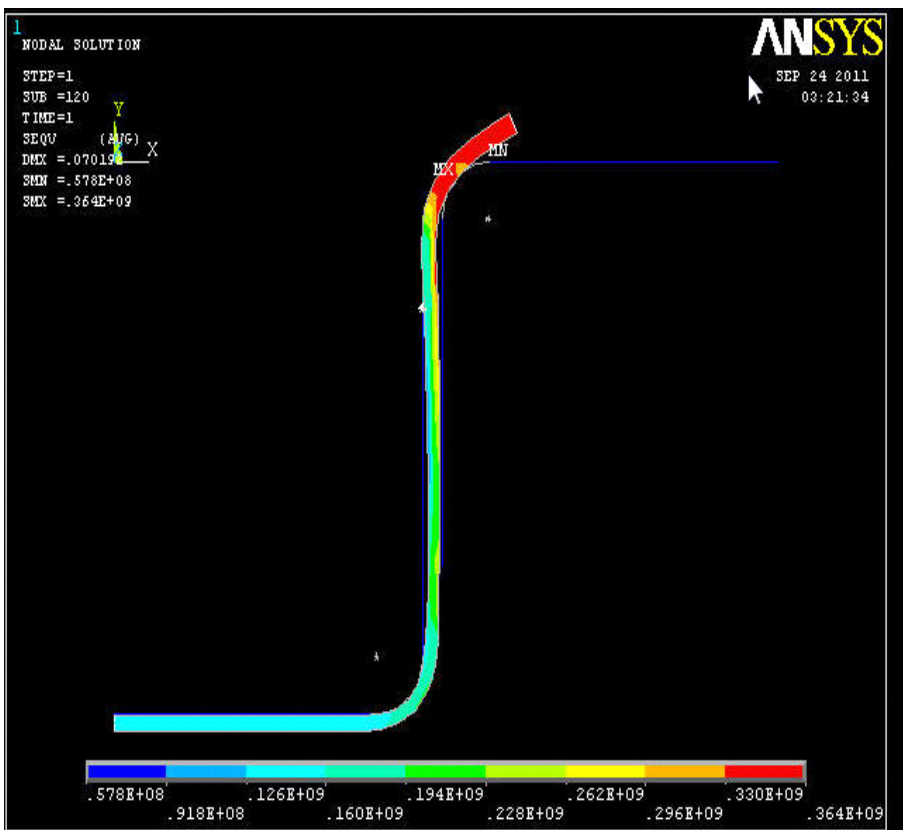


Figure 12 The Von-Mises stress for 70 mm stroke with anisotropy

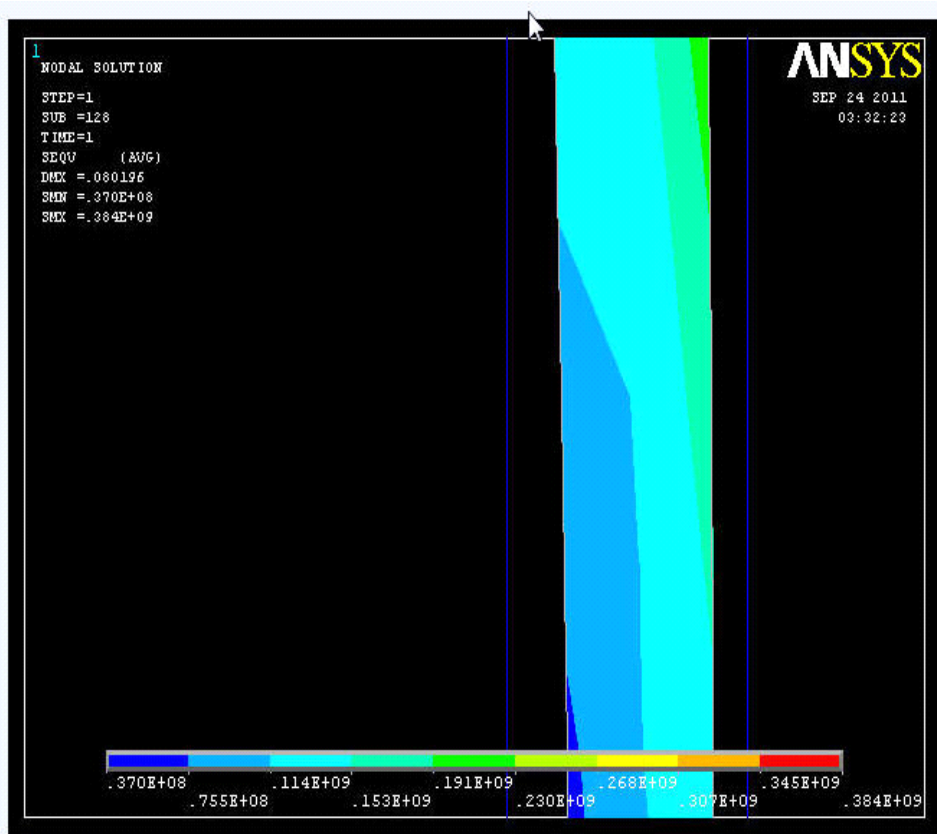


Figure 13 The Von-Mises stress for 80 mm stroke with anisotropy

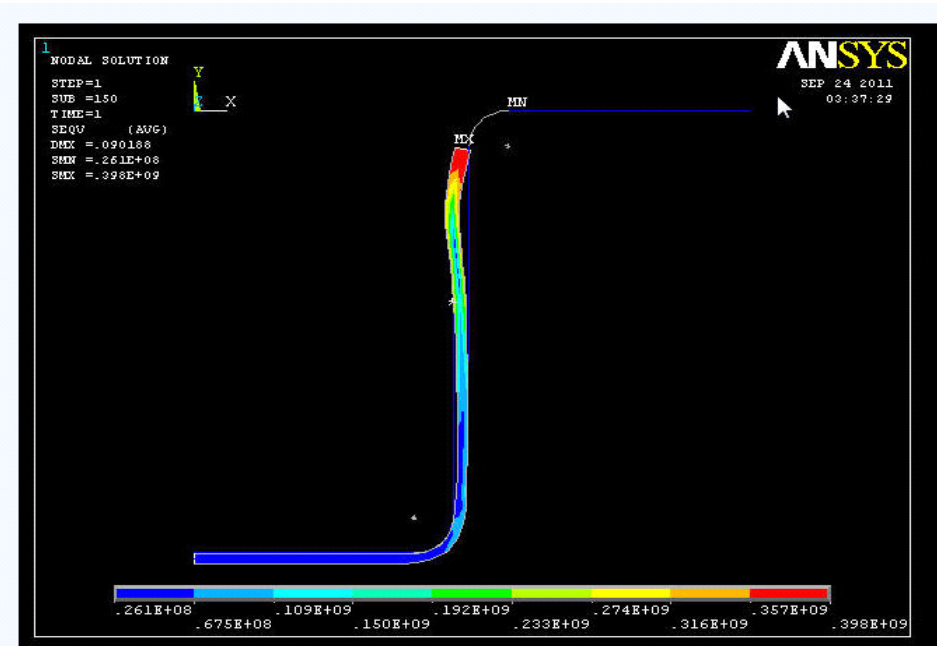


Figure 14 The Von-Mises stress for 90 mm stroke with anisotropy

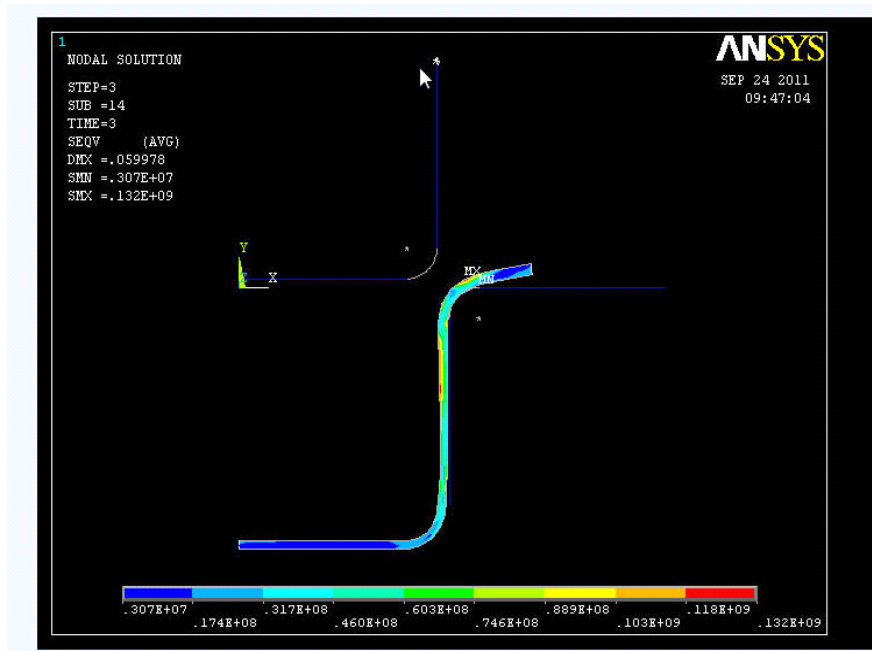


Figure 15 Von-Mises stress for 60 mm stroke after springback

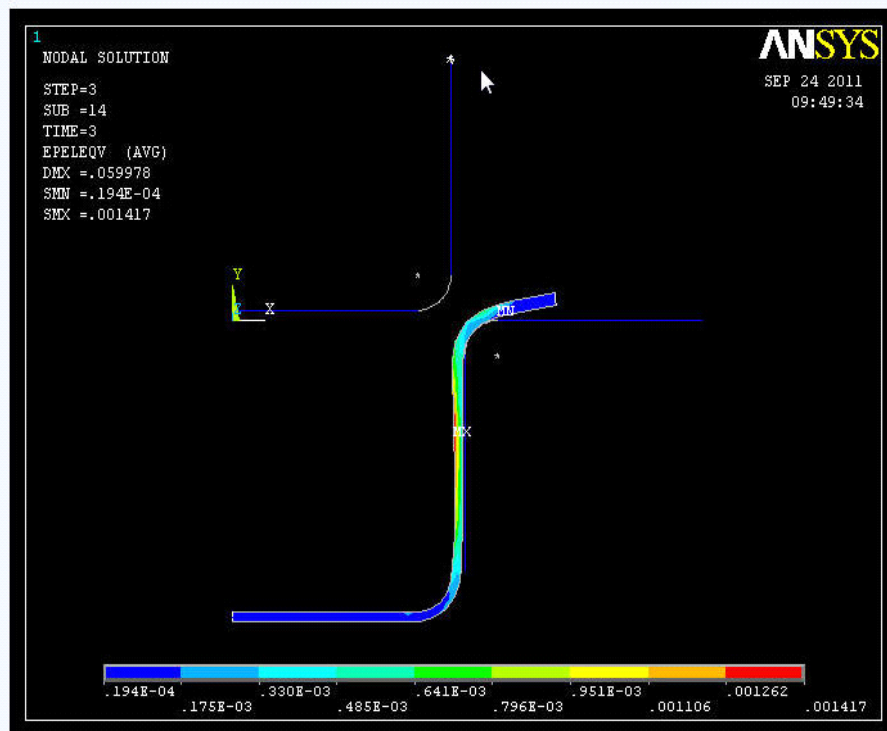


Figure 16 Von-Mises strain for 60 mm stroke after springback

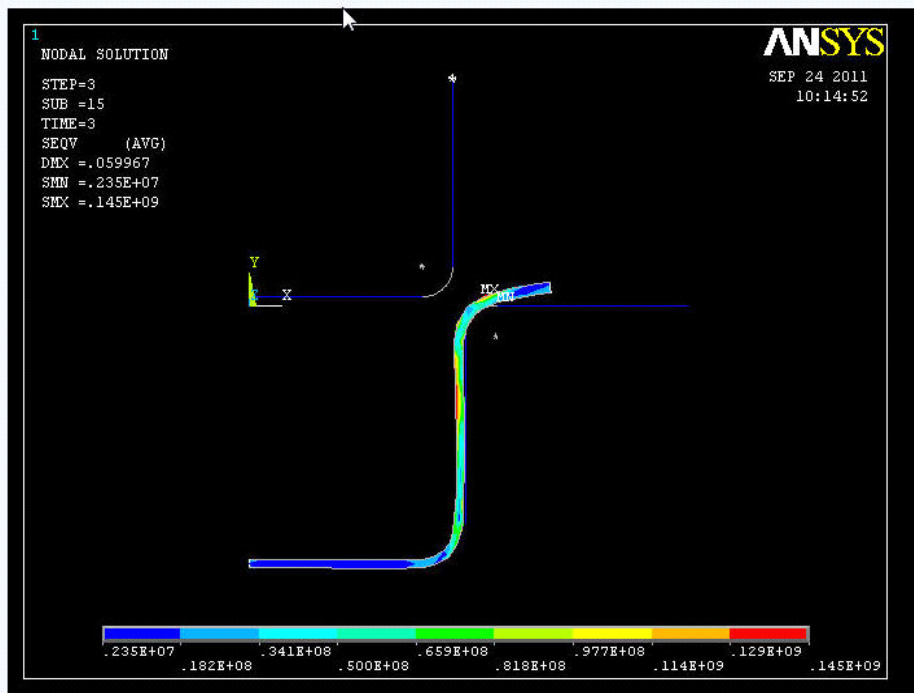


Figure 17 Von-Mises stress for 60 mm stroke after springback with anisotropy

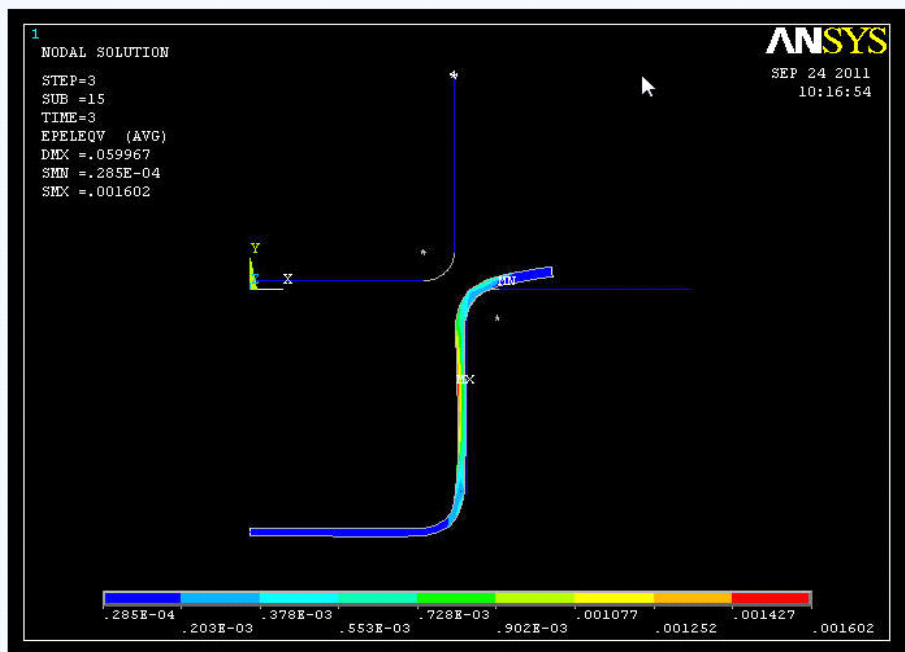


Figure 18 Von-Mises strain for 60 mm stroke after springback with anisotropy

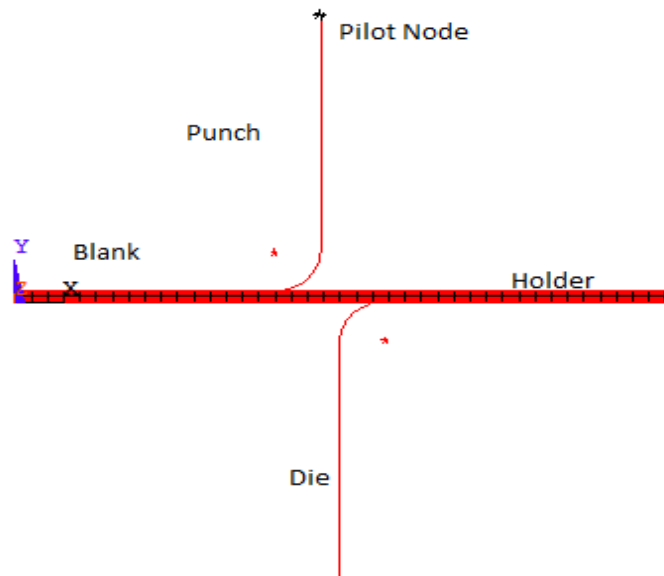


Figure 19 Model for the springback procedure

Appendix A

SPRINGBACK in ANSYS

As it is well known in the forming analysis, the springback phenomena should be taken into the consideration.

A procedure in ANSYS concerning this type of deep drawing analysis which has a prescribed displacement has the following steps:

- 1 solve the problem with displacement load as usual
- 2 find the reaction force at the pilot node
- 3 delete the displacement load and add the force load from 2
- 4 solve the problem one iteration only
- 5 apply zero force at pilot node
- 6 solve the problem as usual non-linear case

All the procedure is done inside the solution phase.

Hill's Table for Material Number 1

Temperature	0
rxx	1.1
ryy	1.1
rzz	1.1
rxxy	1.1
ryz	1.1
rxz	1.1