IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

Spring Back Effect Analysis of Bracket Using Finite Element Analysis

Ravindra E. Gite[1], Kiran S. Phad[2],Dipak S. Bajaj[3]

Assistant Professor, Mechanical Engineering Department, AVCOE, Sangamner, India[1,2]

Associate Professor, Mechanical Engineering Department, AVCOE, Sangamner, India[3]

Abstract: The most sensitive feature of the sheet metal forming is the elastic recovery during unloading, called spring back. Spring-back is a critical phenomenon which is caused by the elastic redistribution of the internal stresses after the removal of deforming forces. The spring back is affected by the factors such as sheet thickness, material properties, tooling geometry etc. In this paper the finite element analysis followed by the experimental validation is done to evaluate the spring back effect. The model of the bracket is modelled using solid works & further analysis is carried out using ANSYS. This paper highlights the effective use of finite element analysis technique to analyse the spring back effect in the sheet metal forming operation so as to maintain the correct die valley angle by considering the spring back effect. Various parameters such as die valley angle, punch nose radius, depth of deformation etc. are considered to evaluate the spring back effect.

Keywords: Spring back, Sheet metal, Forming, Elastic Deformation.

I. INTRODUCTION

Metal forming processes are the manufacturing processes which are almost chip less. Generally these operations are mainly carried out with the help of press machines and press tools. These operations include deformation of metal work pieces to the desired size by applying pressure or force. Press machines and press tools facilitate mass production work. These are the fastest and most efficient way to form a sheet metal into finished products. Generally, metal forming is very suitable for mass production & not for small lot production. for example the press forming of sheet metal individual die is required for each desired shape, the production of the dies requires considerable time and cost. Yet, at the same time, the press sheet forming is not replaceable by any other machining method such as cutting in even small lot production. Forging, extrusion, rolling and bending are some of the important forming process. The forming process includes larger deformation of the structure either with temperature or without temperature application. But in the every mechanical process certain defects exists due to the inherent resistance property of the material for forming it to the required shape. One type of defect is spring back and is very important phenomenon which makes the object non suitable in case of rigid requirements. Spring back is mainly due to Bauchigner effect which allows the material to follow different paths during loading and unloading cycles. All forming process will take places in the plastic region. So to analyze or to estimate spring back, nonlinear material properties which will accommodate stress strain relations in the nonlinear regions are required.

Till today accurate spring back prediction was only available for pure bending via empirical handbook rules or simple analysis, and for a few other specialized two-dimensional geometries. Usually such results apply to very simple shapes with constant radii of curvature, and are based on well-studied materials. From the past decade the finite element methods are used for spring back effect prediction.

The prediction of spring back has challenging for a variety of reasons, including numerical sensitivity, physical sensitivity, and poorly characterized material behavior under reverse loading and unloading conditions. However, it has also been shown that accurate spring back prediction is achievable when these parameters are taken into account. Assuming that spring back can be predicted accurately, there still remains the problem of how to use such results to arrive at a suitable die design to produce a desired part shape. That is, the springback predictions allow ‘‘forward’’ analysis of forming and springback, while a ‘‘backward’’ analysis is needed to work from these results back toward an optimized die design.

1.1 The Spring-Back Principle

Fig.1.1 shows the spring-back principle. When forming sheet metal, the material property is generally divided into two zones, the elastic zone and the plastic zone. In the sheet metal forming process, the forming load increases until the elastic limit of material is exceed. The material state becomes the plastic deformation zone, hence, the sheet metal can be formed.

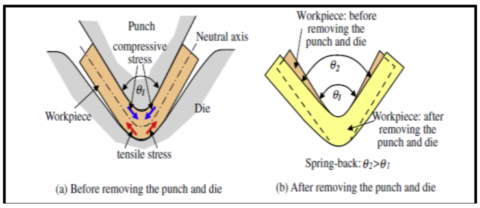


Fig. 1.1 Spring Back Principle

However, in the case of the bending process as shown in Fig.1.1 (a), the material which contacts the punch has compressive stress occur, whereas the material which contacts the die has tensile stress occur. As a result of the stress distribution, the elastic band in the work piece was generated and the material in this area tried to keep its original shape, hence, the material in the compression zone tried to enlarge and the material in the tension zone tried to shrink. The material tried to spring-back and the bended part slightly open as shown in Fig. 1.1 (b).

1.2 Mechanism of Spring-back Phenomenon

According to the plastic deformation theory, the material is generally divided into two zones, the elastic and the plastic zones. The elastic property tries to maintain the material in the initial shape, whereas the plastic property tries to retain the material in the deformed shape. In the sheet metal bending process, the bending load increases until the elastic limit of the material is exceeded and then the material state enters the plastic deformation zone.

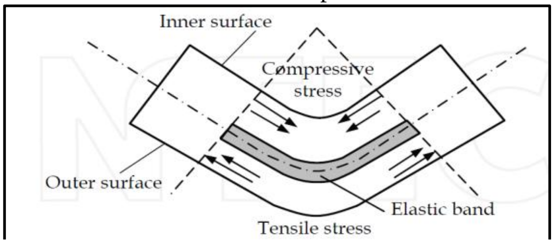


Fig. 1.2. Illustration of the ElasticBand in the Bend part

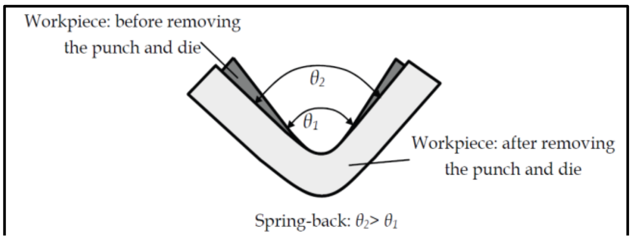


Fig. 1.3. Illustration of Spring-back Feature

The outer surface of the material generates the tensile stress, which propagates inward toward the neutral plane. Vice versa, the inner surface of the material generates the compressive stress and it propagates inward toward the neutral plane. Because of the stress distributions, this phenomenon causes the formation of a small elastic band around the neutral plane, as shown in Fig. 1.2. As the bending force is removed at the end of the bending stroke, the inner surface generates compressive stress tries to enlarge the workpiece and the outer surface generated tensile stress tries to shrink. In contrast, the elastic band remains in the bent parts trying to maintain its original shape, resulting in a partial recovery toward its initial shape. This elastic recovery is called “spring-back”. Thus, the workpiece tries to spring back and the bent part slightly opens out, as shown in Fig. 1.3.

1.3 Factors Affecting Spring Back Effect in Forming Operation

Here are some of the important factors which affects the spring back effect in forming operation of the sheet metal.

1.3.1 Punch Angle

Punch angle is the angle of deforming tool which is used to deform the sheet metal in forming operation. Depending on the required shape of the product, the punch may be having V- shape, U- shape or curvature shape. Punch angle affects the spring back effect of the sheet metal so correct punch angle should be maintain in the forming operation of the sheet metal.

1.3.2 Grain Direction of Sheet metal material

Grain direction of the sheet metal material plays an important role in the forming operation. Sheet metal material having uniform grain size and direction gives a better results as the deformation is uniform in the required direction.

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)
Amrutvahini College of Engineering, Sangamner
Vol. 3, Special Issue 1, March 2016

1.3.3 Die Opening

The forming operations are carried out on the press machines with the help of punch and die. Sheet metal is placed on the die and by applying force punch deforms the sheet metal. To obtain the desired shape of the component the dies are may be of V shape, U shape or curvature shape. Spring back of the sheet metal depends on the angle of the die so while design the die, the spring back effect of the sheet metal has to be considered.

1.3.4 Sheet Thickness

As the sheet thickness varies, the spring effect of the metal also varies. If the sheet metal thickness is less there will be more spring back and as the thickness of sheet metal increases the spring back of the metal decrease. For the same material of different thickness, the spring back effect is also different.

1.3.5 Punch Nose Radius

The total deformation of the sheet metal in the die depends on the punch nose radius. If the punch nose radius is increase the punch will not travel upto the end in the die valley region and this also affect the spring back of the sheet metal. So to reduce the spring effect of sheet metal the punch nose radius should maintain correctly.

1.3.6 Punch Height

Punch height is the distance travelled by the punch to deform the sheet metal. By calculating the required force for deformation, the punch height can be calculated. It also affects the spring back effect so it is necessary to calculate the correct punch height.

1.3.7 Pre Bend Condition of the Sheet Metal

The initially bend condition of the sheet metal also affect the spring back effect. The initially bend sheet metal have the already developed residual stresses which affect the spring back so to evaluate the correct spring back pre bend conditions of the sheet metal has to be considered.

1.4 Material Specification

Hot Rolled Structural Steel Sheet metal- HRSS Sheet

Specification- 5mm thick, Fe 410 as per IS-2062

Table 1. Chemical properties of HRSS sheet

Sr. No.	Chemical Properties	
1	Carbon (C)	0.2 max
2	Manganese (Mn)	1.50 max
3	Silicon (Si)	0.40 max
4	Phosphorus (P)	0.040 max
5	Sulphur (S)	0.040 max
6	Chromium (Cr)	0.040 max

Table 2. Mechanical properties of HRSS sheet

Sr.No	Mechanical Properties	
1	Ultimate tensile Strength	410 Mpa min
2	Yield Stress	250 Mpa min
3	Elongation	23% min
4	Hardness	70 HRBW

II. METHODS AND METHODOLOGY

2.1 Finite element method

The finite element method is a numerical method, which can be implement to solve many problems. The analysis which uses FEM is known as FEA. A general purpose FEA program consists of three modules; model set up, loading and a review the result.

Commercial FEA programs can handle very large number of nodes and nodal degrees of freedom provided a powerful hardware is made available. User's manual, theoretical manual, and verification problems manual, document a commercial FEA program.

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)
Amrutvahini College of Engineering, Sangamner
Vol. 3, Special Issue 1, March 2016

2.2 Conditions for the finite element study

2.2.1 Basic Assumptions

The work piece is assumed to be a continuous body that means the body does not contain any empty space or gaps. Next, the work piece is assumed to be isotropic and homogeneous. This assumption means the material properties do not vary with direction or orientation. A homogeneous material has identical properties at all points. The finite element analysis is simplified to a 2D plane strain problem. A reason is the ratio of the work piece width w to thickness t (w/t) is about 8. Thus, there is negligible or no strain along the width of the work piece. The two angles formed during the deformation are defined as the "DL" in the die-lip region and "V" in the valley region. These two angles are assumed to be independent of each other, meaning that deformation at the die-lip region does not affect deformation at the valley region.

2.2.2. Model set-up

Building a finite element model requires more time than any other part of the analysis. First, the user specifies a job name and analysis title. Then using the pre-processor, the element types are defined, element real constants, material properties, and the model geometry.

Fig. 2.1 shows the set-up of the FEA model. The die–tool set is composed of the die, punch and pressure pad. The Die and Punch are rigid tools for forming the desired "V" shape in the work piece. The Pressure Pad is used to press on the left end of the work piece during simulation.

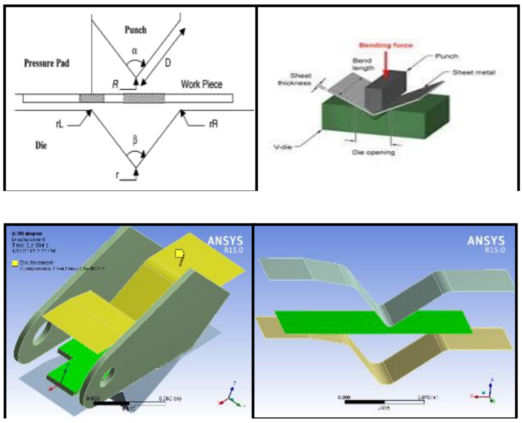


Fig.2.1 FEA Model set-up

Work piece model, punch & die models are generated in solidworks. The shaded regions shown in Fig.2.1 indicate the critical areas where mesh refinement is required. The non-shaded regions represent the coarser elements. The die–tool set is defined as the master surface. The work piece is defined as the slave. These definitions ensure that the work piece nodes do not penetrate the rigid bodies during simulation.

2.2.3 Loading

The main goal of a finite element analysis is to examine how a structure or component responds to certain loading conditions. Specifying the proper loading conditions is, therefore, a key step in the analysis. Loads can be applied on the model in a variety of ways in the ANSYS program. The word loads in ANSYS terminology includes boundary conditions and externally or internally applied forcing functions.

2.2.4. Parameters of the die–tool set

Definitions and relationships of the die–tool set parameters are shown below:

1. Punch angle (α); 88^0
2. Punch radius (R); 6
3. Punch "Leg" (D = 11t); 53
4. Die valley angle (β); 88^0
5. Die valley radius (r = R + t); 11
6. Die-lip radius (rL = rR = 1t); 5
7. Work piece thickness (t = 5 mm).

Die valley radius r = R+t, where t is the thickness of the work piece. If r > R+t, it will cause deeper penetration into the work piece. For this simulation, r will not be more than R+t as this may cause difficulty in achieving convergence. The die-lip radius, rL, is set to be at least one time the thickness of the work piece. This criterion is adopted because draw

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

effect is exhibited at the die-lip region when the punch travels downward to almost straighten the left leg of the V-bend. The parameters to vary for this simulation are punch angle α and punch radius R. As α and R are varied, the corresponding β and r are also changed. In this simulation, punch angle α is equal to die valley angle β.

2.2.5 Review the results. (Post Processing)
The post processors in the ANSYS program can help the user to obtaining the solution and others. Post processing means reviewing the results of an analysis. It is probably the most important step in the analysis.

2.3 Mesh design of work piece

The members are map mesh with the appropriate material properties. HRSS material properties are given for sheet metal and rigid material properties are given for die and punch members. Since the sheet metal is the point of interest, the body is meshed with deformable material properties. Shell 163 element with plane strain option is used for meshing. Shell 163 element has the properties of large deformation effects which is the essential requirement of the forming materials. A finer mesh is considered at the corner regions for better convergence. 1742 elements and 1836 nodes are used for meshing the blank. The punch has 622 elements and 679 nodes. For meshing die 444 elements and 495 nodes are used for the geometry.

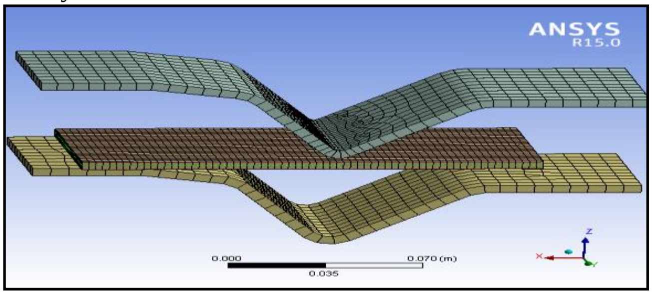


Fig 2.2 Map Meshed Geometry

Mesh design of the work piece is composed of two regions: refinement and coarse regions. Refinement region is required to obtain accurate spring-back angles for the die-lip and valley. However, fine mesh will also increase the computation time required to complete the simulation. Thus, only the critical regions considered are refined. Mesh refinement on the work piece is done at the die-lip and punch regions. These regions experience large deformations, high stresses and high strains.There are at least two different sized elements in the workpiece. Thus, multi-point constraint (MPC) or transition elements are used to prevent cracks caused by inconsistent elements.

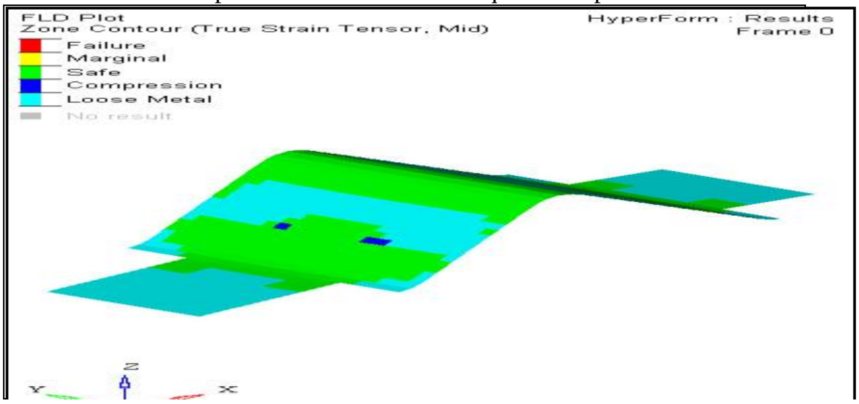
III. RESULTS AND DISCUSSION

3.1. Material Specifications

Material: HRSS sheet, Young's modulus=200Gpa, Density =7860 kg/m3, Yield stress=250Mpa, Plastic Modulus=3000Mpa.

3.2 Formability of sheet metal

The material selected for the component is hot rolled structural steel having thickness of 5mm. The tensile strength of the sheet metal is 410 N/mm^2. Before the development of model of sheet metal it is necessary to check the formability of the sheet metal so that the prediction will be correct. The fig.3.1 shows the formability limit of the sheet metal in which it can be observed that the formability limit is within the prescribed limit, it means the sheet metal can be form without the failure of the material and the part can be form in the required shape without failure.



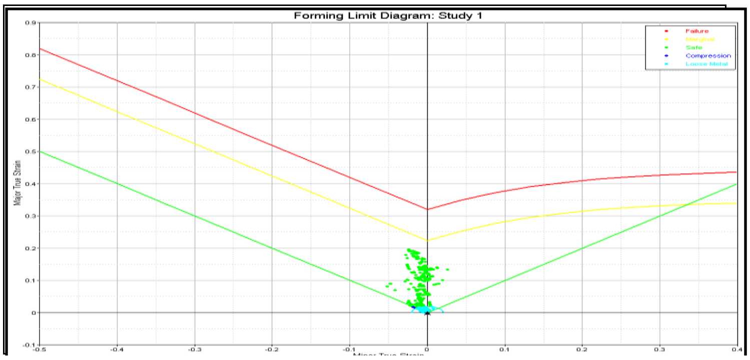


Fig.3.1 Forming limit diagram

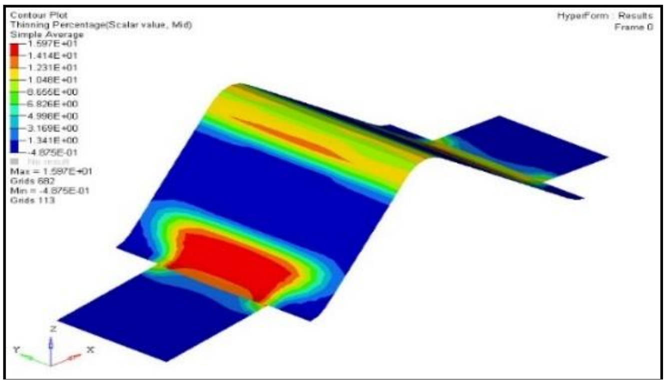
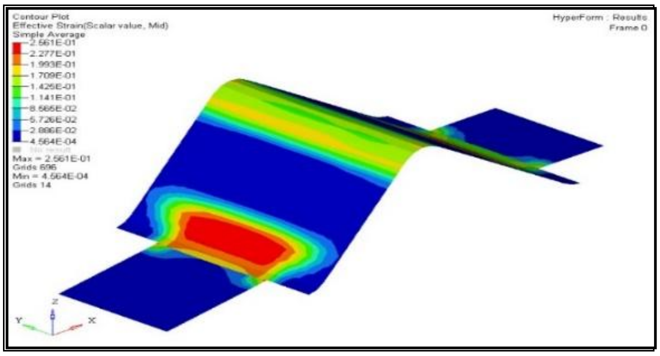


Fig.3.2 (a) Thinning percentage limit diagram



(b) Effective strain diagram

The fig.3.2 (a) shows thinning percentage of the sheet metal. The maximum thinning percentage of the sheet metal is 15.97 %.The fig.3.2 (b) shows effective strain of the sheet metal. The maximum effective strain of the sheet metal is 0.256. The maximum effective strain is observed in the area where the bending of the sheet metal takes place.

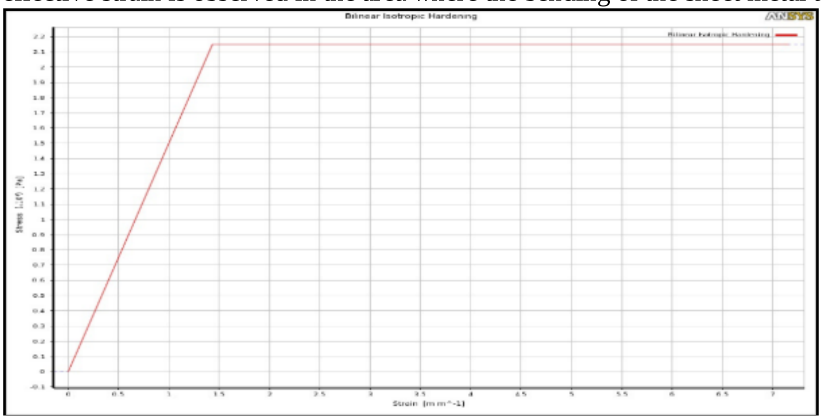


Fig3.3: Stress strain graph (Bilinear representation)

IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

The figure3.3 shows bilinear representation of the problem. The material is considered as elasto-plastic with strain hardening behaviour. Up-to yield point tangent modulus will be defined and later plastic or tangent modulus will be defined for the problem. Generally plastic modulus value is small compared to the Young's modulus value specified. The material will follow linear relation up to yield point and later follows tangent modulus for strain calculation. This is region is the source of residual stresses in the structure. Also this region influences the spring back phenomenon.

3.3. Analysis Results (Thickness 5 mm)

Analysis has been carried out after giving displacement to the punch elements. Here the area mesh of both punch and die are cleared to increase the speed of computation. This is possible after the contact pairs are created. So the punch target elements are given the required displacement load for bending process. The die target nodes are fixed in the position. Incremental procedure based on Newton Raphson method is applied to solve the problem in the nonlinear material and geometrical domain. The result for 5 mm thickness sheet metal is as follows.

3.3.1: Displacement Results for 5 mm plate thickness:

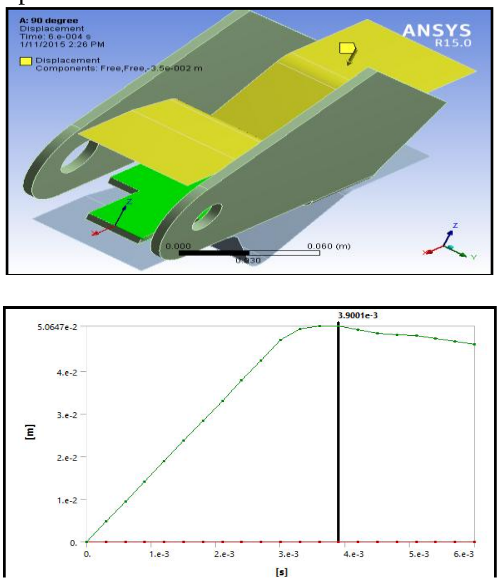


Fig.3.4 Displacement of component

The iteration process gives the required number of steps for proper convergence of the problem. In each step the punch will move smaller distance and the effect of this on sheet metal and die are shown. This iteration process helps to understand the process of bending process. Initially displacement pictures are captured. Later Stress pictures are captured. The iteration summary shows the development of the parameter of interest in the pictorial representation. Fig. 3.5 shows the displacement of punch and blank.

3.3.2: Contact Pressure Development**:**

The simulation helps in finding the load requirements for given punch movement. The maximum pressure developed is at the fillet and corner regions and in the final loading stage and reduces after load is removed. The simulation helps in the geometrical modelling of correct die. The contact pressure shows the regions of stress generation due to the shape of the die and punch

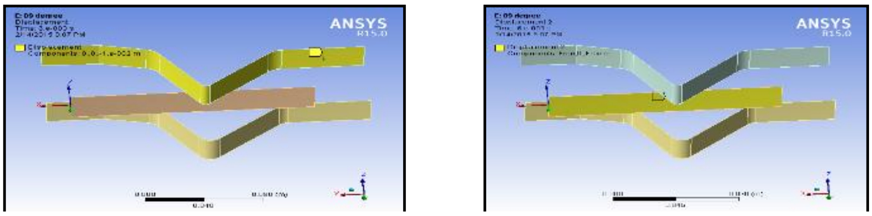


Fig. 3.5 Displacement of punch and blank

3.4 Spring-back angle calculation

After the punch has travelled the prescribed displacement, the simulation removes both punch and die so that the work piece can spring-back. Spring-back angles at the die-lip and valley regions are calculated using the Abaqus viewer. The viewer obtains the node coordinates after spring-back.

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology
2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)
Amrutvahini College of Engineering, Sangamner
Vol. 3, Special Issue 1, March 2016

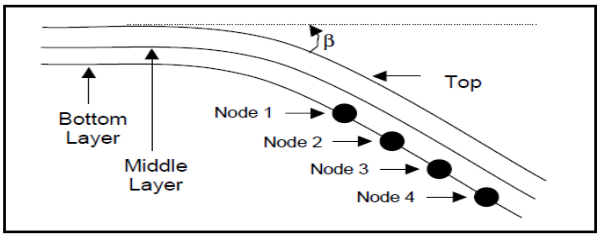


Fig.3.6 spring back calculation

Fig.3.6 shows an example of the die-lip region after spring-back. Coordinates of the 4 nodes are taken from the bottom, middle and top layers of the work piece. Thus, there are altogether 12 coordinates. The angle that the line connectingnodes 1 and 2 makes with the vertical is

$$\theta = \tan^{-1}\left(\frac{x_2 - x_1}{y_2 - y_1}\right)$$

For the bottom layer, the angles are calculated for lines that connect nodes 1–2, 1–3 and 1–4. Thus, three angles are obtained for the bottom layer. The average of these three angles is taken. The same method is used to compute the angle for the middle and top layers. Three average angles for the three different layers are obtained and averaged again to obtain an angle for the die-lip region. The spring-back angle, φ, is defined as:

$$\phi = 90^\circ - \tfrac{1}{2}\beta - \theta_{average}$$

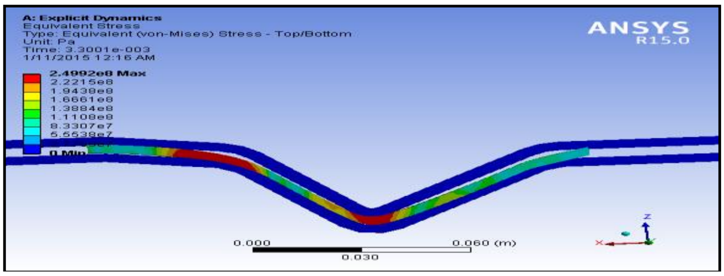


Fig 3.7 Final Vonmises stress during loading

The figure 3.7 shows final vonmises stress in the structure. Maximum stress is around 249.29 Mpa at the bottom bent corner of the sheet metal. This can be attributed to higher deformation at the bent region. The strain is directly proportional to stress. Minimum stresses are developed at the end portions of sheet metal which are not displaced from the original configuration. The status bar at the bottom shows the stress variation in the geometry along the sheet metal. Vonmises stress is considered for plastic condition as the vonmises theory of failure is the most used theory in the failure of ductile materials. Vonmises stress is the stress corresponding to the stored energy and also it is called as equivalent stress. Generally the structures are called yielded after it is crossing the yielding stress of the material.

3.4.2: Spring Back (Thickness 5mm) – Unloading Condition

The fig 3.8 shows residual stress retained in the structure after unloading process. Maximum residual stresses are observed at the corner fillet region and at the bottom corner region. Maximum stress value is around 232.293Mpa. Minimum residual stress observed is around 25.81Mpa

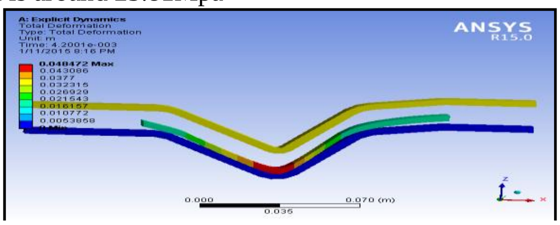


Fig 3.8. Residual stress in the sheet metal after unloading

Residual stress is an undesirable feature for the proper sheet metal bending process. It should be minimized for error free sheet metal products. Residual stress is the cause of crack formation and reduction in the life. It will reduce the tensile strength of the structures. So heat treatment is required to relieve this trapped stresses due to permanent deformation in the structure.

3.4.3 FEA results of spring back effect for die angle 88^0

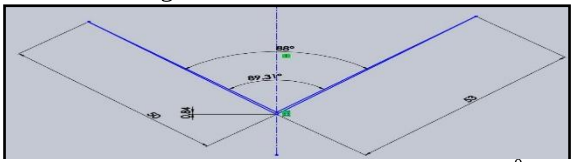


Fig.3.9 spring back when die valley angle 88^0

Calculation for spring back:

- Punch end Deformation: 50.647mm
- Retained Deformation: 49.812mm
- Spring back: 50.647-49.812=0.835 mm
- Spring back effect =1.648 %.

Table 8 Angle calculated by FEA and actual angle measured

Test sample	% spring back by FEA	Actual % spring back	Angle measured (Degree)	
			Angle by FEA	Actual angle
1	1.70	1.80	89.50	89.70
2	1.648	1.60	89.31	90.00
3	1.60	1.75	89.45	89.50
4	1.555	1.80	89.65	89.30
5	1.623	1.70	89.70	89.50

The fig. 3.9 shows the angle calculated by the finite element analysis. When the die and punch angle is kept 88^0, the forming angle obtained is an average of 89.70^0. The required forming angle for spring pivot bracket is $90^{\pm 1}$.

3.5 Results Comparison

Table 9 – Result comparison

Sr. No.	Die Valley Angle (Degree)	Loading Deformation (mm)	Unloading Deformation (mm)	Spring Back	Percentage Spring Back
1	91	50.625	46.304	4.32	8.53
2	90.5	50.625	46.505	4.12	8.13
3	90	50.625	46.709	3.92	7.73
4	89.5	50.625	48.131	2.49	4.91
5	89	50.625	49.340	1.285	2.53
6	88.5	50.647	49.665	0.982	1.93
7	88	50.647	49.812	0.835	1.648

The table 9 shows the tabulated result for different die valley angle. Here the die valley angle and punch angle are kept same for simulation. The die valley angle and corresponding spring back is shown in the table.

CONCLUSION

From the results obtained by finite element analysis and experimental validation of spring back effect of pivot bracket it can conclude that

1) The spring back effect (elastic recovery) of sheet metal varies with the die valley angle and punch nose radius.
2) With the increase of the die angles, spring back, percentage spring back, vonmises stress, residual stress, contact pressure and plastic strains are increasing.
3) To maintain the forming angle of the pivot bracket of 90^0, the die valley angle should be maintained at 88^0 so that the correct angle of pivot bracket can be maintained including the spring back effect.

REFERENCES

[1]. Prabhakar.A, Dr. Haneef Mohamed,.(2013)..Sheet metal forming analyses with spring-backdeformation on U-Bends in Isotropic platesInternational Journal of Innovative Research in Science, Engineering and Technology Vol. 2, ISSN: 2319-8753 pp. 4905-4913.

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

IARJSET

International Advanced Research Journal in Science, Engineering and Technology

2nd International Conference on Advances in Mechanical Engineering (ICAME-2016)

Amrutvahini College of Engineering, Sangamner

Vol. 3, Special Issue 1, March 2016

[2]. Baseri H, M. Bakhshi-Jooybari,(2011), "Modeling of spring-back in V-die bending process by using fuzzy learning back-propagation algorithm", Expert Systems with Applications, 38 pp 8894–8900.
[3]. Hakim S. Sultan Aljibori,": " Finite element Analysis of Sheet Metal Forming Process", European Journal of Scientific Research, ISSN 1450-216X Vol.33 No.1 (2009), pp.57-69
[4]. Fahd Fathi Ahmed Abd El Au(2003),"FiniteElement And Experimental Studies Of Springback In Sheet Metal Forming", National Library Of Canada.
[5]. Wei Gan, Wagoner R.H,(2004) "Die design method for sheet springback", International Journal of Mechanical Sciences 46 pp 1097–1113
[6]. Parsa M.H, Nasher al ahkami,(2012)" Investigating spring back phenomena in double curved sheet metals forming", Materials and Design 41 pp 326–337
[7]. Sutasn Thipprakmas a,b,*, Surasit Rojananan,(2008)," Investigation of spring-go phenomenon using finite element method", Materials and Design 29 pp 1526–1532
[8]. Sutasn Thipprakmas (2010)," Finite element analysis of punch height effect on V-bending angle", Materials and Design 31 pp 1593–1598
[9]. Chou I, Hung C. Finite element analysis and optimization on springback reduction. International Journal of Machine Tools & Manufacture 1999;39:pp 517–36.
[10]. Mohamed Faraj Alfaidi, Li Xiaoxing. Determination of spring back in sheet metal forming. The Annals Of ''Dunarea De Jos'' University Of Galati. 2009 ISSN pp 1221-4566.