Experimental study of spring back effect of aluminum sheet metal with various thickness

Veerendra Singh Lodhi
ME IVth sem student
Jabalpur Engg. College Jabalpur (M P)
Prof. A.K. JAIN
Associate professor ,Jabalpur Engg. College Jabalpur (M P)

Abstract

The present investigation was carried out to determine the spring back and thinning effect of aluminum sheet metal during L-bending operation. Number of specimens with thickness varying from 0.5 mm to 3.5 mm were prepared. The experiments were conducted for different clearances between punch and die. It is observed that, beyond a particular clearance for each thickness of the sheet metal, the spring back and thinning effects were linearly increasing. However, below the critical clearance, scratches on the surface of the sheet metal were seen due to wear. The scratches were analyzed through Scanning Electron micrographs. As the clearance between punch and die reduces further, more wear on the punching surface was observed. And, as the clearance increases it leads to increase the spring backeffect and fracture propagation.

Key Word- AL sheet metal , applied force, lever , guide way

1 .Introduction-

Sheet metal is metal formed by an industrial process into thin, flat pieces. It is one of the fundamental forms used in metalworking and it can be cut and bent into a variety of shapes. Countless everyday objects are constructed with sheet metal. Thicknesses can vary significantly; extremely thin thicknesses are considered foil or leaf, and pieces thicker than 6 mm (0.25 in) are considered plate. Sheet metal is available in flat pieces or coiled strips. The coils are formed by running a continuous sheet of metal through a roll slitter.

The thickness of sheet metal is commonly specified by a traditional, non-linear measure known as its gauge. The larger the gauge number, the thinner the metal. Commonly used steel sheet metal ranges from 30 gauge to about 8 gauge. Gauge differs between ferrous (iron based) metals and nonferrous metals such as aluminum or copper; copper thickness, for example is measured in ounces (and represents the thickness of 1 ounce of copper rolled out to an area of 1 square foot).

There are many different metals that can be made into sheet metal, such as aluminum, brass, copper, steel, tin, nickel and titanium. For decorative uses, important sheet metals include silver, gold, and platinum (platinum sheet metal is also utilized as a catalyst.)

2. Press brake forming

This is a form of bending used to produce long, thin sheet metal parts. The machine that bends the metal is called a press brake. The lower part of the press contains a V-shaped groove called the die. The upper part of the press contains a punch that presses the sheet metal down into the v-shaped die, causing it to bend. There are several techniques used, but the most common modern method is "air bending". Here, the die has a sharper angle than the required bend (typically 85 degrees for a 90 degree bend) and the upper tool is precisely controlled in its stroke to push the metal down the required amount to bend it through 90 degrees.

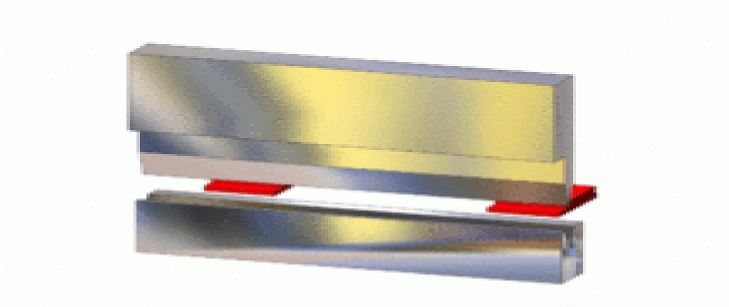

Figure. 1. Press brake forming

Figure. 2. Press forming with angle

Typically, a general purpose machine has an available bending force of around 25 tonnes per metre of length. The opening width of the lower die is typically 8 to 10 times the thickness of the metal to be bent (for example, 5 mm material could be bent in a 40 mm die). The inner radius of the bend formed in the metal is determined not by the radius of the upper tool, but by the lower die width. Typically, the inner radius is equal to 1/6 of the V-width used in the forming process.

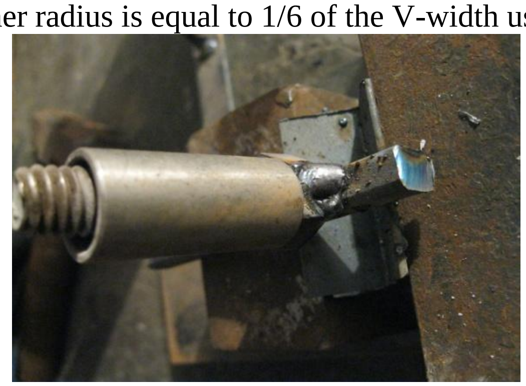

Figure.3. Press forming with die

The press usually has some sort of back gauge to position depth of the bend along the workpiece. The backgauge can be computer controlled to allow the operator to make a series of bends in a component to a high degree of accuracy. Simple machines control only the backstop, more advanced machines control the position and angle of the stop, its height and the position of the two reference pegs used to locate the material. The machine can also record the exact position and pressure required for each bending operation to allow the operator to achieve a perfect 90 degree bend across a variety of operations on the part.

Figure.4. digital reading of sheet metal

3.Punching

Punching is performed by placing the sheet of metal stock between a punch and a die mounted in a press. The punch and die are made of hardened steel and are the same shape. The punch just barely fits into the die. The press pushes the punch against and into the die with enough force to cut a hole in the stock. In some cases the punch and die "nest" together to create a depression in the stock. In progressive stamping a coil of stock is fed into a long die/punch set with many stages. Multiple simple shaped holes may be produced in one stage, but complex holes are created in multiple stages. In the final stage, the part is punched free from the "web".
A typical CNC turret punch has a choice of up to 60 tools in a "turret" that can be rotated to bring any tool to the punching position. A simple shape (e.g., a square, circle, or hexagon) is cut directly from the sheet. A complex shape can be cut out by making many square or rounded cuts around the perimeter. A punch is less flexible than a laser for cutting compound shapes, but faster for repetitive shapes (for example, the grille of an air-conditioning unit). A CNC punch can achieve 600 strokes per minute.
A typical component (such as the side of a computer case) can be cut to high precision from a blank sheet in under 15 seconds by either a press or a laser CNC machine.

4.Bending (metalworking)

Bending is a manufacturing process that produces a V-shape, U-shape, or channel shape along a straight axis in ductile materials, most commonly sheet metal. Commonly used equipment include box and pan brakes, brake presses, and other specialized machine presses. Typical products that are made like this are boxes such as electrical enclosures and rectangular ductwork.

5.Process

In press brake forming, a work piece is positioned over the die block and the die block presses the sheet to form a shape.[1] Usually bending has to overcome both tensile stresses and compressive stresses. When bending is done, the residual stresses cause the material to spring back towards its original position, so the sheet must be over-bent to achieve the proper bend angle. The amount of spring back is dependent on the material, and the type of forming. When

sheet metal is bent, it stretches in length. The bend deduction is the amount the sheet metal will stretch when bent as measured from the outside edges of the bend. The bend radius refers to the inside radius. The formed bend radius is dependent upon the dies used, the material properties, and the material thickness.
The U-punch forms a U-shape with a single punch.

6.Types

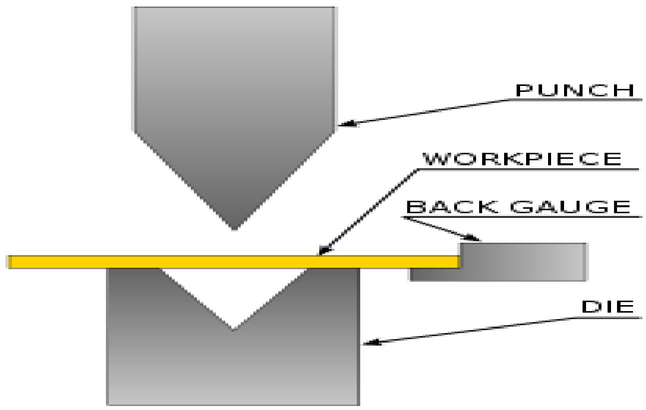

Figure.5. A schematic of air bending with a back gauge.

There are three basic types of bending on a press brake, each is defined by the relationship of the end tool position to the thickness of the material. These three are Air Bending, Bottoming and Coining. The configuration of the tools for these three types of bending are nearly identical. A die with a long rail form tool with a radiused tip that locates the inside profile of the bend is called a punch. Punches are usually attached to the ram of the machine by clamps and move to produce the bending force. A die with a long rail form tool that has concave or V shaped lengthwise channel that locate the outside profile of the form is called a die. Dies are usually stationary and located under the material on the bed of the machine. Note that some locations do not differentiate between the two different kinds of dies (punches and dies.) The other types of bending listed use specially designed tools or machines to perform the work.

7.Air bending

This bending method forms material by pressing a punch (also called the upper or top die) into the material, forcing it into a bottom V-die, which is mounted on the press. The punch forms the bend so that the distance between the punch and the side wall of the V is greater than the material thickness (T).
Either a V-shaped or square opening may be used in the bottom die (dies are frequently referred to as tools or tooling). A set of top and bottom dies are made for each product or part produced on the press. Because it requires less bend force, air bending tends to use smaller tools than other methods.

8.Bottoming

In bottoming, the sheet is forced against the V opening in the bottom tool. U-shaped openings cannot be used. Space is left between the sheet and the bottom of the V opening. The optimum width of the V opening is 6 T (T stands for material thickness) for sheets about 3 mm thick, up to about 12 T for 12 mm thick sheets. The bending radius must be at least 0.8 T to 2 T for sheet steel. Larger bend radius require about the same force as larger radii in air bending, however, smaller radii require greater force—up to five times as much—than air bending. Advantages of bottoming include greater accuracy and less springback. A disadvantage is that a different tool set is needed for each bend angle, sheet thickness, and material. In general, air bending is the preferred technique.

9.Three-point bending

Three-point bending is a newer process that uses a die with an adjustable-height bottom tool, moved by a servo motor. The height can be set within 0.01 mm. Adjustments between the ram and the upper tool are made using a hydraulic cushion, which accommodates deviations in sheet thickness. Three-point bending can achieve bend angles with 0.25 deg. precision. While three-point bending permits high flexibility and precision, it also entails high costs and there are fewer tools readily available. It is being used mostly in high-value niche markets.

10.Folding

In folding, clamping beams hold the longer side of the sheet. The beam rises and folds the sheet around a bend profile. The bend beam can move the sheet up or down, permitting the fabricating of parts with positive and negative bend angles. The resulting bend angle is influenced by the folding angle of the beam, tool geometry, and material properties. Large sheets can be handled in this process, making the operation easily automated. There is little risk of surface damage to the sheet.

11.Wiping

In wiping, the longest end of the sheet is clamped, then the tool moves up and down, bending the sheet around the bend profile. Though faster than folding, wiping has a higher risk of producing scratches or otherwise damaging the sheet, because the tool is moving over the sheet surface. The risk increases if sharp angles are being produced. Wiping on press brakes involves special tools.

This method will typically bottom or coin the material to set the edge to help overcome springback. In this bending method, the radius of the bottom die determines the final bend radius.

12.Rotary bending

Rotary bending is similar to wiping but the top die is made of a freely rotating cylinder with the final formed shape cut into it and a matching bottom die. On contact with the sheet, the roll contacts on two points and it rotates as the forming process bends the sheet. This bending method is typically considered a "non-marking" forming process suitable to pre-painted or easily marred surfaces. This bending process can produce angles greater than 90° in a single hit on standard press brakes or flat presses.

13.Calculations

Many variations of these formulas exist and are readily available online. These variations may often seem to be at odds with one another, but they are invariably the same formulas simplified or combined. What is presented here are the unsimplified formulas. All formulas use the following keys:

- BA = bend allowance

- BD = bend deduction
- R = inside bend radius
- K = K-Factor, which is t / T
- T = material thickness
- t = distance from inside face to the neutral line
- A = bend angle in degrees (the angle through which the material is bent)

The neutral line (also called the neutral axis) is an imaginary line that can be drawn through the cross-section of the workpiece that represents the lack of any internal forces. Its location in the material is a function of the forces used to form the part and the material yield and tensile strengths. In the bend region, the material between the neutral line and the inside radius will be under compression during the bend. The material between the neutral line and the outside radius will be under tension during the bend.

Both bend deduction and bend allowance represent the difference between the neutral line or unbent flat pattern (the required length of the material prior to bending) and the formed bend. Subtracting them from the combined length of both flanges gives the flat pattern length. The question of which formula to use is determined by the dimensioning method used to define the flanges as shown in the two diagrams below.

14.Bend allowance

The bend allowance (BA) is the length of the arc of the neutral line between the tangent points of a bend in any material. Adding the length of each flange taken between the center of the radius to the BA gives the Flat Pattern length. This bend allowance formula is used to determine the flat pattern length when a bend is dimensioned from 1) the center of the radius, 2) a tangent point of the radius or 3) the outside tangent point of the radius on an acute angle bend.

The BA can be calculated using the following formula:

$$BA = A\left(\frac{\pi}{180}\right)(R + K \times T)$$

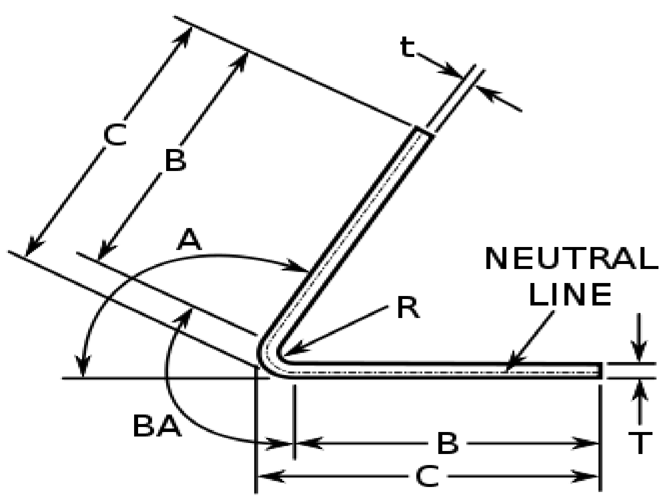

Figure.6. A schematic Diagram of bending

Diagram showing standard dimensioning scheme when using Bend Allowance formulas. Note that when dimensions "C" are specified, dimension B = C - R - T

Example

Angle	90
PI	3.142
Radius	10.0
K-Factor	0.33
Thickness	2 mm

Bend deduction

The outside set back (OSSB) is the length from the tangent point of the radius to the apex of the outside of the bend. The bend deduction (BD) is twice the outside setback minus the bend allowance. BD is calculated using the following formula:

$$BD = 2(R + T)\tan\left(\frac{A}{2}\right) - BA$$

Diagram showing standard dimensioning scheme when using Bend Deduction formulas.
The above formula works only for right angles. For bend angles 90 degrees or greater the following formula works, where A is the angle in radians (=degrees*π/180)

$$\mathrm{BD} = \mathrm{R}(\mathrm{A} - 2) + \mathrm{T}(\mathrm{kA} - 2)$$

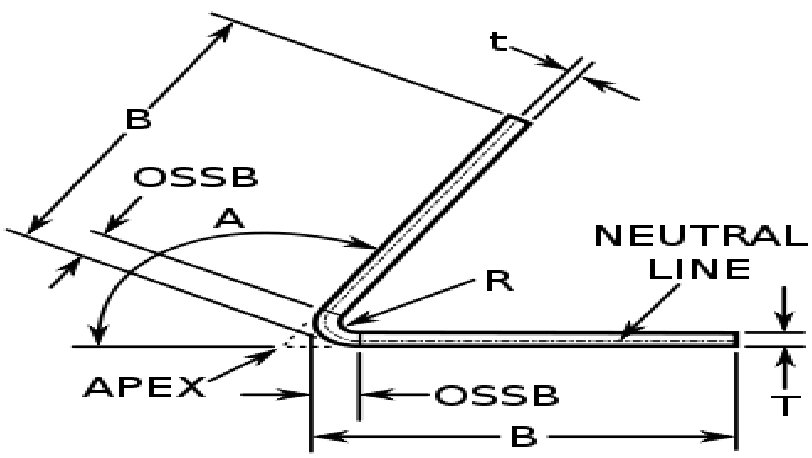

Figure.7. A schematic Diagram of Bend Deduction

K-factor

K-factor is a ratio of location of the neutral line to the material thickness as defined by t/T where t = location of the neutral line and T = material thickness. The K-Factor formulation does not take the forming stresses into account but is simply a geometric calculation of the location of the neutral line after the forces are applied and is thus the roll-up of all the unknown (error) factors for a given setup. The K-factor depends on many factors including the material, the type of bending operation (coining, bottoming, air-bending, etc.) the tools, etc. and is typically between 0.3 to 0.5.

Table .1

Sr. No.	Sheet thickness in mm	Applied load in N
1	1.00	10
2	1.25	20
3	1.50	30
4	1.75	40
5	2.00	50

Table .2 Spring back for 1 mm AL sheet metal

Sr. No.	Clearance in mm	Spring back in degree
1	1.0	2.3
2	1.2	4.0
3	1.4	5.2
4	1.6	5.4
5	1.8	5.6
6	2.0	5.8

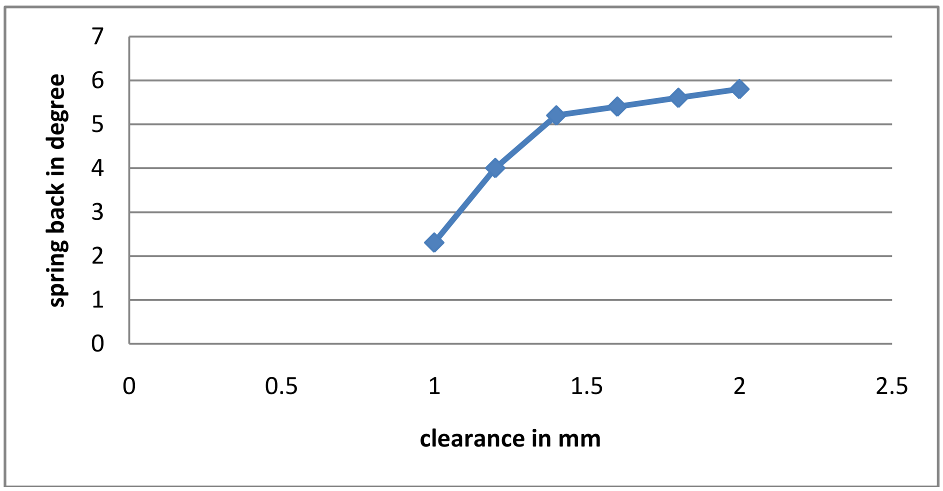

Figure.8. Graph of clearance & spring back

Table .3 Spring back for 2 mm AL sheet metal

Sr. No.	Clearance in mm	Spring back in degree
1	2.1	3.2
2	2.4	3.9
3	2.7	4.2
4	2.8	4.5
5	2.9	4.8
6	3.0	4.9

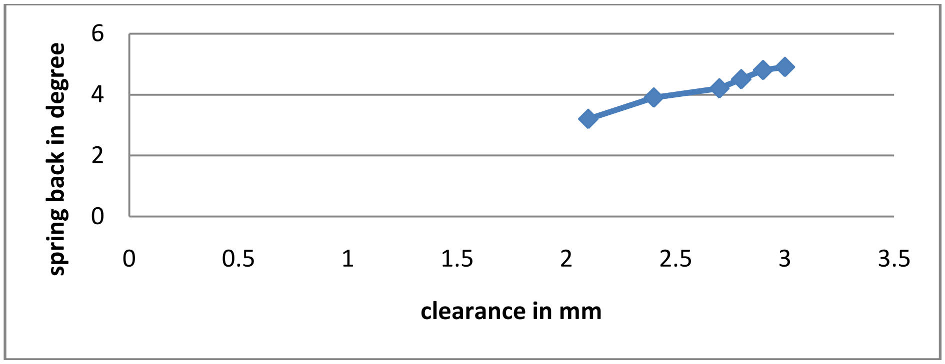

Figure.9. Graph of clearance & spring back

Table .4 Spring back for 3 mm AL sheet metal

Sr. No.	Clearance in mm	Spring back in degree
1	3.0	3.1
2	3.2	3.7
3	3.4	4.1
4	3.6	4.3
5	3.8	4.6
6	4.0	4.7

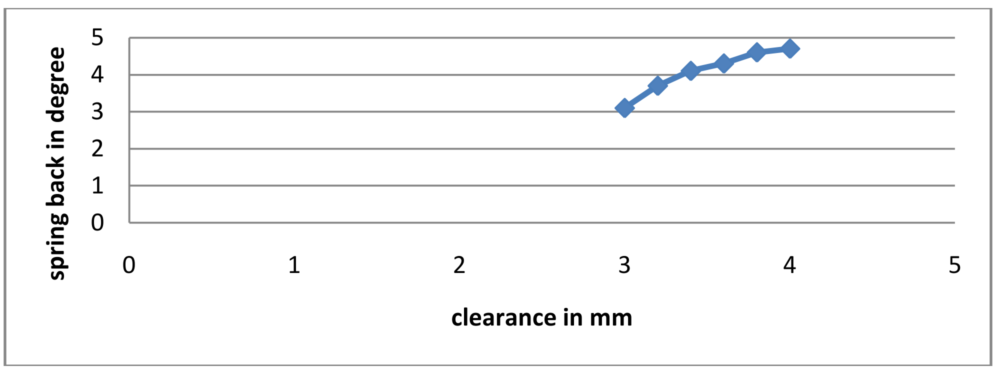

Figure.10. Graph of clearance & spring back

Table .5 Spring back for 1 mm AL sheet metal

Sr. No.	Clearance in mm	Thinning in mm
1	1.0	0.15
2	1.2	0.14
3	1.4	0.13
4	1.6	0.14
5	1.8	0.16
6	2.0	0.17

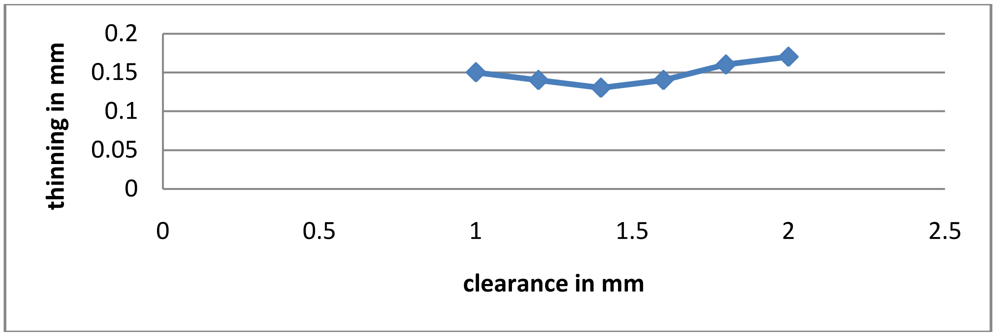

Figure.11. Graph of clearance & Thinning

Table .5 Spring back for 2 mm AL sheet metal

Sr. No.	Clearance in mm	Thinning in mm
1	2.1	0.49
2	2.4	0.45
3	2.7	0.51
4	2.8	0.53
5	2.9	0.54
6	3.0	0.56

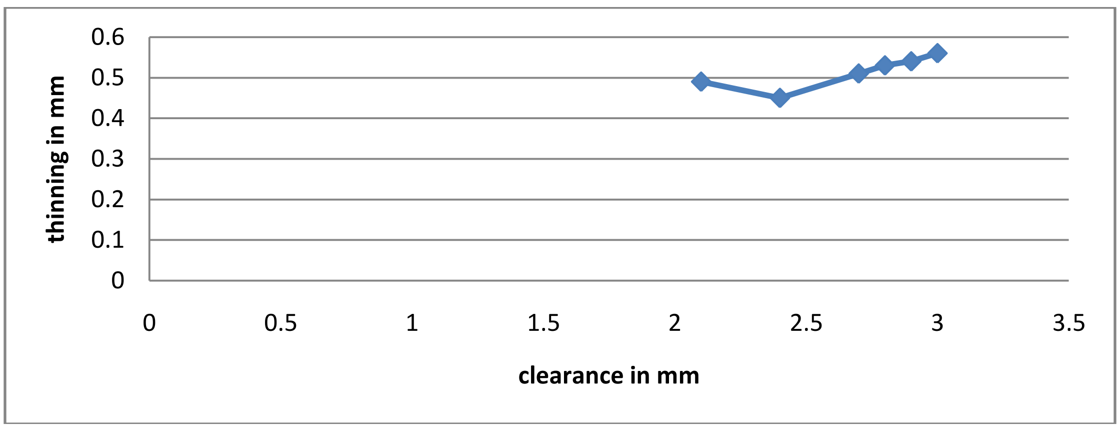

Figure.12. Graph of clearance & Thinning

Table .7 Spring back for 3 mm AL sheet metal

Sr. No.	Clearance in mm	Thinning in mm
1	3.1	0.58
2	3.2	0.55
3	3.4	0.53
4	3.6	0.59
5	3.8	0.65
6	4.0	0.66

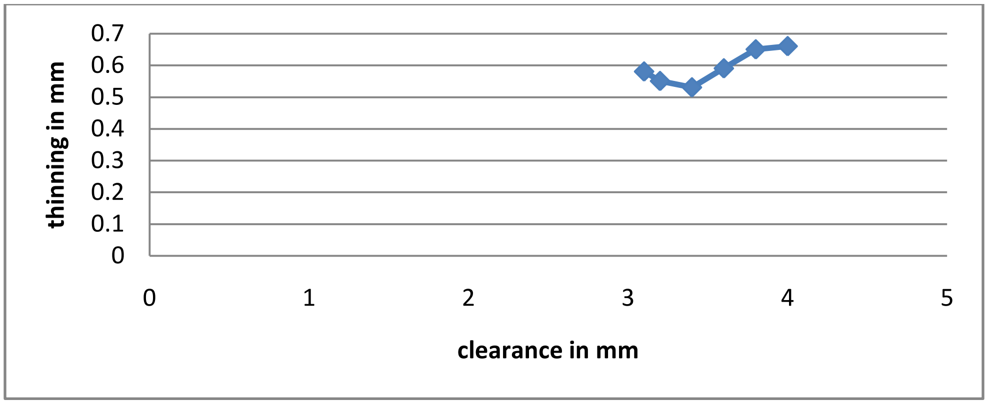

Figure.13. Graph of clearance & Thinning

Conclusions

Investigation was carried out to determine spring back and thinning effect of aluminum sheet metal during L-bending operation. Number of specimens with various thicknesses were prepared and the experiments were carried out for different clearances. However, below the critical clearance it is observed through SEM photographs that scratches on the surface of the sheet metal will occur due to wear. Further it is observe that as the clearance reduces, the wear rate increases on the punching surface. There was an increase in the spring back effect as well as the fracture propagation as the clearance between punch and die was increased.

Reference –

[1] Coles Richard, Kirwan MarkJ. Food and beverage packaging technology. London: Blackwell Publishing; 2011. p. 344.
[2] Colgan M, Monaghan J. Deep drawing process: analysis and experiment. J Mater Process Technol 2003;132:35–41.
[3] Ibrahim Demirci H, Yas_ar M, Demiray K. The theoretical and experimental investigation of blank holder forces plate effect in deep drawing process of Al 1050 material. Mater Des 2008;29(2):526–32.
[4] Jawad Waleed Khalid. Investigation of contact interface between the punch and blank in deep drawing process. J Eng Technol 2007;25(3):370–82.
[5] Vladimirov IvayloN, Pietryga MichaelP, Reese Stefanie. Anisotropic finiteelastoplasticitywith nonlinear kinematic and isotropic hardening and application to sheet metal forming. Int J Plasticity 2010;26:659–87.
[6] Fereshteh-Saniee F, Montazeran MH. A comparative estimation of the forming load in the deep drawing process. J Mater Process Technol 2003;140:555–61.
[7] Lange Kurt. Handbook of metal forming. Society of Manufacturing Engineers; 1985.
[8] Meguid SA, Refaat MH. Finite element analysis of the deep drawing process using variational inequalities. Finite Elements Anal Des 1997;28:51–67.
[9] Suchy I. Handbook of die design. 2nd ed. McGraw-Hill Publishing; 2006. p. 353–400.
[10] Garcia-Romeu ML, Ciurana J, Ferrer I. J Mater Process Technol 2007;191(1– 3):174–7.
[11] Gan W, Wagoner RH. Int J Mech Sci 2004;46(7):1097–113.
[12] Hsu Tze-Chi, Chu Chan Hung. A finite element analysis of sheet metal forming process. J Mater Process Technol 1995;54:70–5.
[13] Vegter H, Van den Boogaurd AH. A Plane stress yield function for anisotropic sheet material by interpolation of biaxial stress status. Int J Plast 2005:24.

[14] Hill R. The mathematical theory of plasticity. London: Oxford University Press; 1950.
[15] Yoon Jeong Whan, Chung Kwansoo, Lee Myoneng-Gyu, Kim Daeyong, Kim Chongmin, Wenner Michael L, Fredric Barlat C. Spring back evaluation of automotive sheets based on isotropic-kinematic hardening laws and non quadratic anisotropic yield functions, part I, II, III. Int J Plast 2005;21:861–82.