International Journal of Technology Research and Management
ISSN (Online): 2348-9006
Vol 3 Issue 12 December 2016

Analysis of Sheet Metal Forming Process

P. K. Sharma[1] and Nitin Tiwari[2]
Department of mechanical engineering, NRI-IST, Bhopal (M.P.)[1, 2]
Nitintiwari1990@gmail.com[1]

Abstract: *This paper presents Analysis of sheet metal forming process which is widely used in automobile industries. Several different processing methods have been implemented in the industries to attain its repeatability and productivity. Sheet metal forming process includes nonlinearity, complex material behaviour and machining technology. As on today FEM is useful for determining stresses and additional failure criterion for the formability prediction in the sheet metal forming process. Material optimization by saving in the raw material size for sheet metal is focused. Computer Aided design model is settled by scanning the die in coordinate majoring machine is been represented. FEM, Probabilistic design, numerical analysis, forming limit stress diagram, press developing analysis, material optimization are some of the criterion being used for the sheet metal forming analysis.*

Keywords: *sheet metal forming analysis, FEM, forming limit diagram, sheet metal forming process.*

1. INTRODUCTION

Sheet metal forming process have been used for manufacturing of various metal products like many automobile, aerospace and other industries due to productivity and high strength as compared to light weight products. Sheet metals are having various applications in automobile industries for manufacturing automotive parts and hence it is required to study the formability of these sheet metals.

The sheet metal forming processes are influenced by some of the variables such as material property, blank holding pressure, press speed etc. forming limit diagrams are used to find these restrictions as it represents tolerable limit of strain with two principal surface strains. Finite element methods, numerical analysis, press forming analysis are the criterion used for the sheet metal forming analysis.

2. TYPES OF METAL FORMING

Sheet metal forming processes are those in which force is applied to a sheet metal piece to change its geometry without removing any material. The forces applied on sheet metal piece beyond its yield strength, causing the material to plastically deform, but without failure. By doing so, the sheet can be bent or stretched into a variety of complicated shapes. Sheet metal forming processes like Bending, Roll forming, Spinning, Deep Drawing, Stretch forming etc.

Bending Process

Bending is a metal forming process in which a force is applied to a sheet metal piece, causing it to bend at an angle and form the desired shape. A bending operation causes deformation along one axis, but a sequence of some different operations can be performed to develop a complex part. Bent parts can be little small, such as a bracket, or up to 20 feet in length, such as a large enclosure or chassis. A bend can be characterized by some different parameters, shown in the image below.

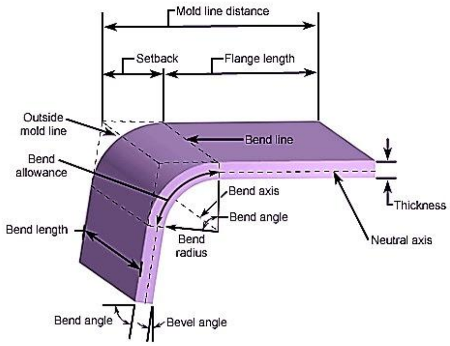


Figure 1:

Bending Diagram

a) Bend line - The straight line on the surface of the sheet, on either side of the bend, that defines the end of the level flange and the start of the bend.

b) Outside mold line - The straight line where the exterior surfaces of the two flanges would assemble, were they to continue. This line defines the edge of a mold that would bound the bent sheet metal.

c) Flange length - The length of the two flanges, extending from the edge of the sheet to the bend line.

d) Mold line distance - The distance from any one end of the sheet to the outside mold line.

e) Setback - The distance from bend line to the outside mold line. It is also state that the difference between the mold line distance and the flange length.

f) Bend axis - The straight line defines the center around which the sheet metal is bent.

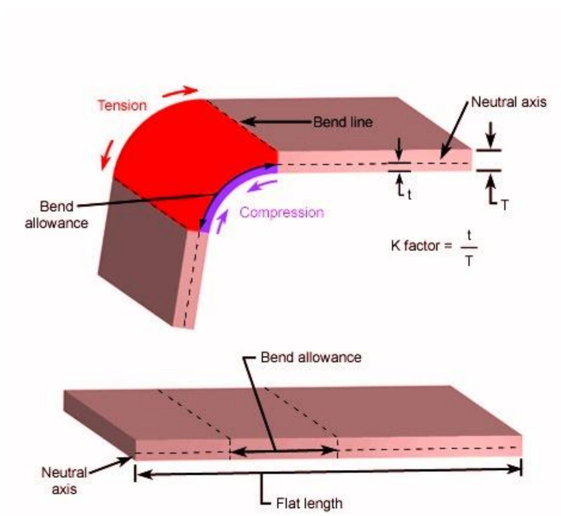
g) Bend length - The length of the bend is measured along the bend axis.

h) Bend radius - The distance between the bend axis and the inside surface of the material,. It is also called as inside bend radius. The outside bend radius is defined as summation of inside radius and sheet thickness.

i) Bend angle - The angle of the bend, measured between the bent flange and its initial position, or it is also state as the included angle between perpendicular lines drawn from the bend lines.

j) Bevel angle - The flattering angle to the bend angle.

The bending results in both tension and compression in the sheet metal. The outside portion of the sheet will undertake tension and stretch to a greater length, while the inside portion undertake compression and shortens. The neutral axis is the boundary line inside the sheet metal; along neutral axis no tension or compression forces are present. For this reason, the length of this axis remains constant. The variations in length to the outside and inside surfaces can be related to the original length by two parameters, the bend allowance and bend deduction, which are defined as follow.



Neutral Axis

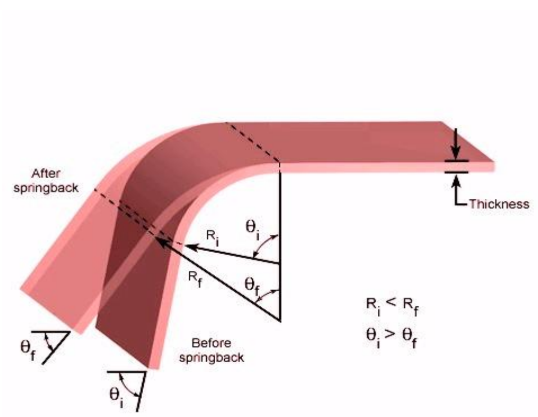
a) Neutral axis - The location in the sheet that is neither incremented nor decremented , and therefore remains at a constant length.

b) K-factor - The location of the neutral axis in the material, computed as the ratio of the distance of the neutral axis (measured from the inside bend surface) to the material thickness. The K-factor is dependent upon several factors (material, bending operation, bend angle, etc.) and is typically higher than 0.25, but can't incremented by 0.50.

c) Bend allowance - The length of the neutral axis between the bend lines, or in other words, the arc length of the bend. The bend subsidy added to the flange lengths is equal to the total flat length.

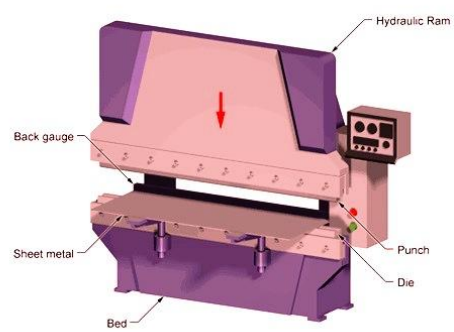
d) Bend deduction - Also called the bend compensation, the amount a piece of material has been screeched by bending. The value equals the difference between the mold line lengths and the total flat length.

When bending a piece of sheet metal, the residual stresses in the material will affect the sheet to spring back feeble after the bending operation. Due to this elastic recovery, it is necessary to over-bend the sheet a precise amount to achieve the required bend radius and bend angle. The final bend radius will be higher than initially formed and the final bend angle will be smaller. The ratio of the required bend angle to the initial bend angle is defined as the spring back factor, K_S. The amount of spring back turn upon several factors, including the material, bending operation, and the initial bend angle and bend radius.

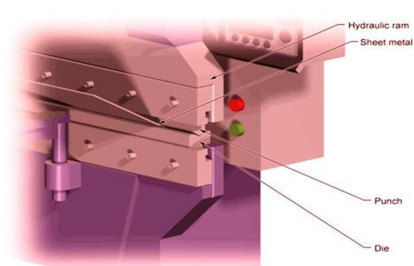


Spring back

Bending is normally performed on a machine called a press brake, which can be manually or automatically operated. Because of this the bending process is also called as press brake forming. Press brakes are available in a range of sizes in order to best suit the given application. A press brake contains an upper device called the punch and a lower device called the die, between which the sheet metal is located. The sheet is carefully located over the die and held in place by the back gauge while the punch lowers and forces the sheet to bend. In an automatic machine, with the help of hydraulic ram the punch is forced into the sheet. The bend angle achieved is calculated by the depth to which the punch forces the sheet into the die. This depth is precisely forced to achieve the desired bend. Standard tooling is generally used for the punch and die, allowing a low initial cost and suitability for low volume production. As per requirement tooling can be used for specialized bending operations but will add to the cost. The tooling material is selected based upon the production quantity, sheet metal material, and degree of bending. Logically, a stronger tool is required to endure larger quantities, harder sheet metal, and severe bending operations. In order of increasing strength, some common tooling materials add hardwood, low carbon steel, tool steel, and carbide steel.

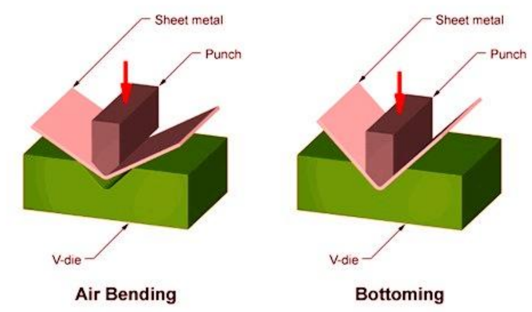


Press Brake (Open)



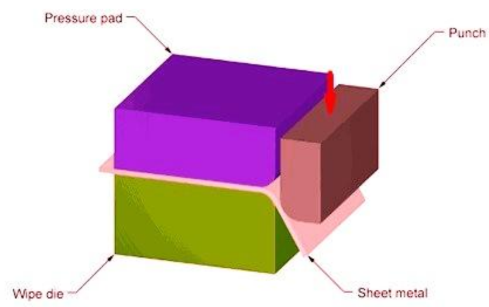
Press Brake (Closed)

When using a press brake machine and standard die sets, there are still some techniques that can be used to bend the sheet. The most common method is known as V-bending, in which the punch and die in "V" shaped. The punch forces the sheet into the "V" shaped groove in the V-die, and bending occurs. If the punch does not force the sheet into the die cavity, leaving space or below air, it is called "air bending". As a result, the V-groove must have a sharper angle than the angle being formed in the sheet. If the punch forces the sheet to the bottom of the die cavity, it is called "bottoming". This technique permit for more organizes over the angle because there is less spring back. However, a higher tonnage press is necessary. In both techniques, the width of the "V" shaped groove, or die opening, is typically 7 to 19 times the sheet thickness. This value is referred to as the die ratio and is equal to the die opening divided by the sheet thickness.



V Bending

Another common bending method with V-bending is wipe bending, sometimes called edge bending. Wipe bending wants the sheet to be held against the wipe die by a pressure pad. The punch then forces against the edge of the sheet that extends beyond the die and pad. The sheet will bend against the radius of the edge of the wipe die.



Wipe Bending

Design rules

a) Bend location - A bend should be located where enough material is present, and preferably with straight edges, for the sheet to be secured without slipping. The width of this flange must be equal to at least four times the sheet thickness plus the bend radius.

b) Bend radius

i. Use a single bend radius for all bends to eliminate additional tooling or setups

ii. Inside bend radius should equal at least the sheet thickness

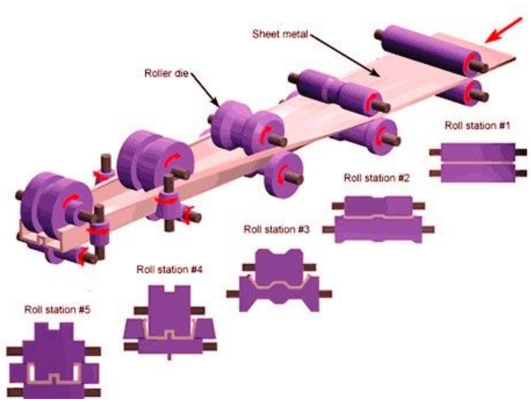
c) Bend direction - Bending hard metals parallel to the rolling direction of the sheet may guide to fracture. Bending perpendicular to the rolling direction is recommended.

d) Any features, such as holes or slots, located very close to a bend may be distorted. The distance of such features from the bend should be equal to at least three times the sheet thickness plus the bending radius.

e) In the case of manual bending, if the design allows, a slot can be cut along the bend line to reduce the manual force required.

Roll Forming

Roll forming, also known as roll forming, is a metal forming process in which sheet metal is gradually shaped through a series of bending operations. The process is performed on a roll forming line in which the raw sheet metal piece is fed through a series of rollers. a roller die, located on both sides of the sheet. The shape and size of the roller die may be only one of its kind to that station, or several identical roller dies may be used in different positions. The roller dies may be on both sides & above and below the sheet or at an angle, etc. As the sheet is forced through the roller dies in each roller station, it plastically deforms and bends. Each roll station performs one stage in the complete bending of the sheet to form the desired shape. The roller dies are lubricated to reduce friction, wear & tear between the die and the sheet. Also, lubricant can allow for a higher productivity, which will also depend on the material thickness, number of roller stations, and radius of each bend. The roll forming line can also contain other sheet metal fabrication operations before or after the roll forming, such as punching or shearing.

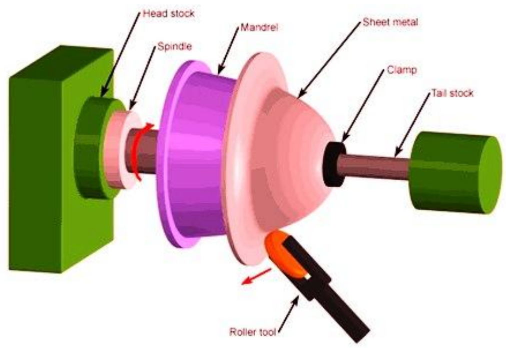


Roll Forming Line

The roll forming process can be used to shape a sheet into a wide variety of cross-section profiles. An open profile is most ordinary, but a closed tube shape can be formed as well. Because the final form is getting through a series of bends, the part does not need a uniform or symmetric cross-section along its length. Roll forming is used to create very long sheet metal parts with usual widths of 1” to 20” and thicknesses of 0.004” to 0.12”. The roll forming process is capable of producing parts with tolerances as tight as ±0.005”. Typical roll formed parts include panels, tracks, shelving, etc. These parts are normally used in industrial and commercial buildings for roofing, lighting, storage units, and Heating Ventilation & Air Conditioning applications.

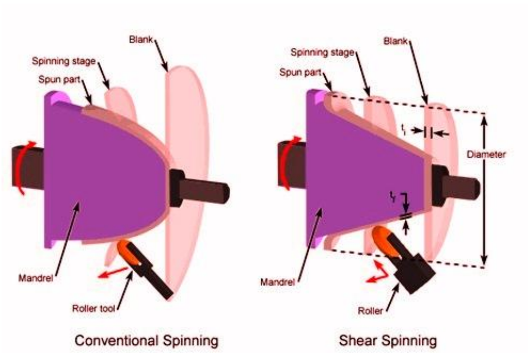
Spinning

Spinning, is also known as spin forming, is a metal forming process used to form cylindrical parts by rotating a sheet metal piece while forces are applied to one side. A sheet metal disc is rotated at very high speeds while rollers press the sheet against a tool, called a mandrel, to form the desired shape. Spin metal parts have a rotationally symmetric, hollow shape, such as a cylinder. e.g. cookware, hubcaps, satellite dishes, rocket nose cones, and musical instruments. Spinning is typically performed manually or CNC lathe and requires a blank, mandrel, and roller tool. The blank is the disc-shaped of sheet metal piece that is first cut from raw sheet and will be formed into the desired part. The mandrel is a solid form of the internal shape of the part, against which the blank will be pressed. For more complicated parts, such as those with re-entrant surfaces, multi-piece mandrels can be used. Because the mandrel does not experience much wear in this process, it can be made from non metal. However, mass production normally utilizes a metal mandrel. The mandrel and blank are clamped together and held between the headstock and tailstock of the lathe to be rotated at high speeds by the spindle. While the blank and mandrel rotate, force is applied to the sheet by a tool, causing the sheet to bend around the mandrel. The tool may make some passes to complete the shaping of the sheet. This tool is typically a roller wheel attached to a lever. Rollers are available in various diameters and thicknesses and are usually made from steel. The rollers are low-cost and experience little wear allowing for low volume production of parts.



Spinning Lathe

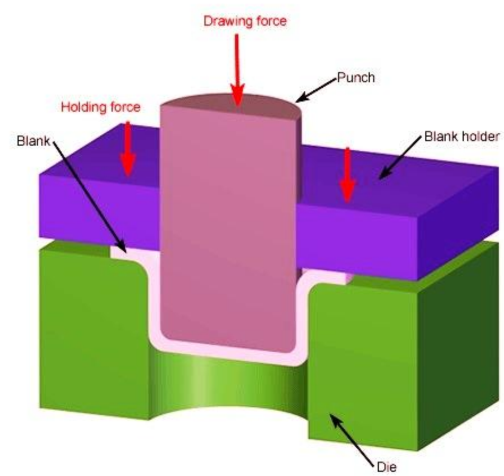
There are two separate spinning methods, referred to as usual spinning and shear spinning. In usual spinning, the roller tool pushes against the blank until it conforms to the contour of the mandrel. The spun part will have a diameter smaller than the blank, but will keep a constant thickness. In shear spinning, the roller not only bends the blank against the mandrel, it also applies a downward pressure while it moves, stretching the material over the mandrel. Due to this, the outer diameter of the spun part will remain equal to the original blank diameter, but the thickness of the part walls will be thinner.



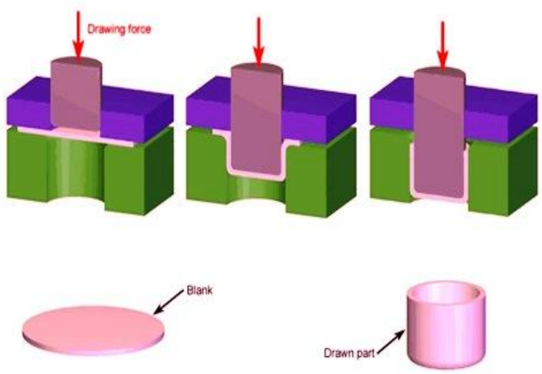
Conventional Spinning vs. Shear Spinning

Deep Drawing

In Deep drawing sheet metal is stretched into the desired shape. A tool forces downward on the sheet metal, forcing it into a die cavity in the desired shape part. The tensile forces applied to the sheet cause it to plastically deform into a required shaped part. These parts can have a variety of cross sections with straight, tapered, or even curved shape, but cylindrical or rectangular parts are most commonly used. Deep drawing is most effective with ductile metals, such as aluminium, brass, copper, and mild steel. E.g. automotive bodies and fuel tanks, cans, cups, kitchen parts, and pans. The deep drawing process requires a blank holder, raw sheet metal piece, punch, and dies. The blank of sheet metal piece, usually a disc or rectangle, which is pre-cut from raw material and will be formed into the part. The blank is clamped on blank holder over the die, which has a cavity in the external shape of the part. A punch moves downward into the blank and draws deformed the material into the die cavity. The movement of the tool punch is usually hydraulically powered to apply required force to the blank. Both the die and punch subjected to wear from the forces applied to the sheet metal and are therefore made from tool steel or carbon steel to reduce wear. The process of drawing the part sometimes occurs in a sequence of operations, called draw reductions. In every step, a punch forces the part into a different die, stretching the part to a greater depth each time. After a part is completely drawn, the tool punch and blank holder can be raised and the part separate from the die. The portion of the sheet metal that was clamped under the blank holder may form a flange around the part that can be cut off.



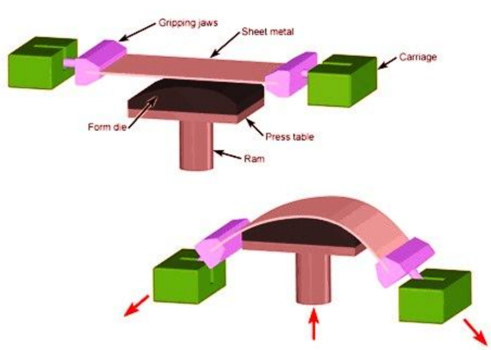
Deep Drawing



Deep Drawing Sequence

Stretch Forming

In Stretch forming sheet metal piece is stretched and bent simultaneously over a die in order to form large contoured parts. Stretch forming is doing on a stretch press, in which a sheet metal piece is securely gripped along its edges by gripping jaws. The gripping jaws are each attached to a carriage that is pulled by pressurised air or liquid force to stretch the sheet. The tool used in this process is called a form die, which is a solid contoured piece against which the sheet metal will be pressed. As the form die is run into the sheet, which is gripped strongly at its edges, the tensile forces increase and the sheet plastically deforms into a required shape. Horizontal stretch presses mount the form die sideways on a stationary press table, while the gripping jaws pull the sheet horizontally around the form die.

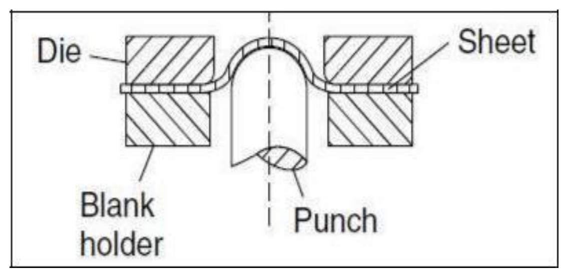


Stretch Forming

Stretch formed parts are usually large and possess large radius bends. The shapes that can be formed vary from a simple curved surface to complicated non-uniform cross sections. Stretch forming is able of shaping parts with very high accuracy and smooth surfaces. Ductile materials are preferable, the most commonly used being aluminium, steel. usual stretch formed parts are large curved panels such as door panels in vehicles or wing panels on aircraft. Other stretch formed parts can be found in window frames and enclosures.

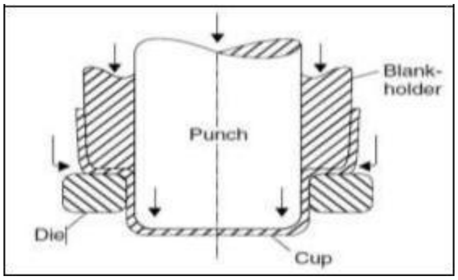
3. FORMING PROCESS AND FORMING LIMIT DIAGRAMS

Metal forming has now became the significant process of modern manufacturing industries, a large variety of products are made by forming processes. The main products which are either used as raw material for other processes such as bars, channels, sheets, draw products such as wires and tubes, forged products such as shafts, gear blanks and gears, automobile parts etc.



Basic Parameters of Forming Process

Sheet metal forming process is the type of forming process which is mostly used in the industries such as auto vehicle, aerospace. Some ordinary sheet metal forming processes used to manufacture wants sheet metal product are cutting off, blanking and punching, bending, deep drawing, hydro-forming, spinning and flow turning



Blanking and punching

Forming Limit Diagram

Sheet metals are used in lots of industries especially in automotive parts hence it is necessary to study the formability of the sheet metals. Forming limit diagram is used to study the formability of the sheet metals. Sheet metal is formed in a great variety of shapes which involve

Complex state of strain paths and total strains. There is no theoretical method to forecast the formability of the sheet metal. The forming limit diagram predicts whether the particular material can withstand certain ratio of strains without failure.

The concept of the forming limit diagram introduced by **KEELER and BACKOFEN** [1] and Goodwin [1] has produced an important how the maximum deformation that a material can withstand during sheet metal forming process can be determined.

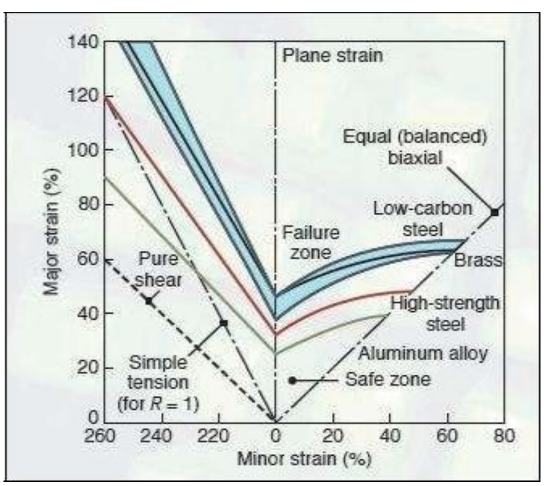


Fig.Forming Limit Diagram[1].

During experimental method, a some diameter of circle in the form grid pattern ate printed on the surface of the sheet metal. The sheet metal gets deformed in stages. The grid pattern is tested during the each stage of deformation. Printing circular grids on the surface of sheet metal gives the advantage that during the deformations the circles will get changed into ellipses with their major or minor axes directed along principal directions of strain.

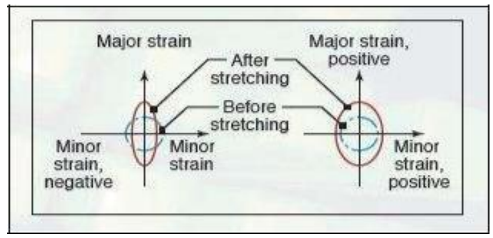


Illustration of the Positive and Negative Minor Strains.

Principal strains and their directions can be calculated by the measurement of the axes and the measurement of diameter of the circles. As the forming progresses, at some region neck may occur. The ratio of strains is calculated at that region. This is a part on finite limit diagram or curve which separates the safe and the unsafe regions. The region above the lines is the failure zone and below is safe. The state of strain in forming must be such that it falls under the curve for a particular material.

The Factors Affecting Forming Limit Diagram:

- Strain hardening and strain rate exponent, Material properties.
- Thickness of sheet, finite limit diagram for thicker sheet is placed higher than for a thinner sheet with small or no change in diagram.
- The forming limit curve of ductile sheet of same alloy and same thickness is positioned higher to that of hard sheet.
- Anisotropy in the sheet.
- Type of covering on the sheet.

- Type of pre-straining prior to testing. The finite limit diagram may be designed by altering strain path. It can be positioned higher by selecting correct strain path.

- The orientation of test sample with respect to rolling direction.

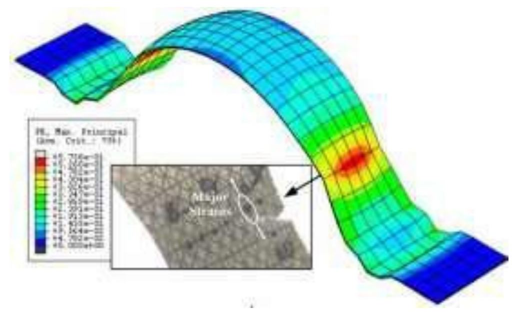
4. METHODOLOGIES

The Different types of methodologies used for sheet metal forming process are explained as follows:

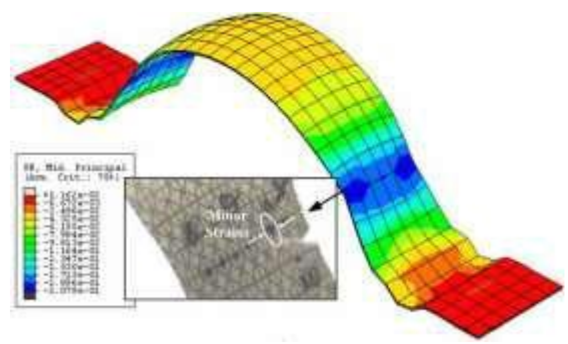
A sheet metal stamping process of mild steel for a wheelhouse used in automobile industry is inspected by means of an explicit nonlinear finite element code and integrating failure analysis and design under uncertainty. Margins of tearing and wrinkling are quantitatively defined on the basis of stress criteria for system level design. A robust design model is manufactured to conduct a probabilistic design, the weighted three-point-based method which approximates the statistical features of the responses of interest, and deliver a systematic method in designing a sheet metal forming process under the framework of design uncertainty.

The result shows the study for strong design in the sheet metal stamping is used to constantly quantify the margin of safety or failure and to powerfully take uncertainties into account to generate a system level strong design model. The strong design for a wheelhouse stamping process is accompanied to maximize the total mean value of margins and to reduce the total alteration of margins. The possibility of the implementation of design and maximization under uncertainty in sheet metal forming processes forced with failure analysis is presented in the paper by Thaweepat Buranathiti, Jian Cao, Z. Cedric Xia and Wei Chen [3].

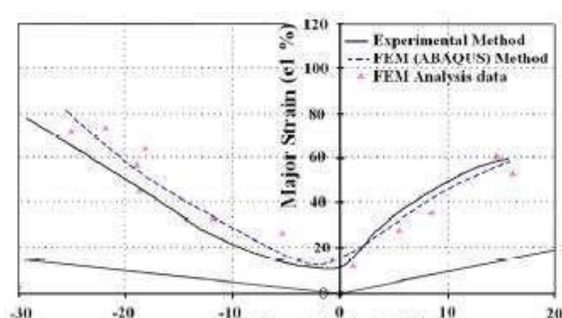
A theoretical failure model is represented for the numerical forecasts of the developing limit strains of automotive sheets which are used in the automobile industries. For the computational forecast of forming limit diagrams under plane stress distortion conditions, a numerical model is established. The numerical model measures the Hill's orthotropic yield condition which is the explanation of the directional variation of yield stress and Swift's diffusive necking and Hill's contained failure criteria are engaged in the distortion induced failure forecast of the sheet metal. Using an improver back stress form of the nonlinear kinematic toughening rule The Bauschinger effect is include correctly in the distortion modelling. A set of algebraic equations are made by converting the failure conditions and plasticity model. These algebraic equations that might be determined incrementally for both proportional and non-proportional deformation tracks.



(a) Major strain of the SM of 30 mm width



(b) Minor strain of the SM of 30 mm width[4]



Comparison of the FLD curves obtained by FEM and experimental method[4].

It is forecast that finite limit curves for steels are correct and conservative in each sides of finite limit diagrams which is offered in the paper by Mehmet Firat [4].

The formability of the sheet metals which is a perilous to the achievement of sheet metal forming processes. The finite limit diagrams have been resolute in this paper experimentally for some grades by accompanying punch stretching, a correctly designed and fabricated tools. The Formability noticed from finite limit diagrams has been connected with mechanical properties and formability parameters i.e. type of punch used, diameter of punch, opposition takes place between punch and sheet, work hardening exponent and plane anisotropy of the sheets. Influence of sheet metals thickness, effect of lubricant such as mineral oil, effect of work hardening exponent and plane anisotropy values on the FLD are also discussed in the paper.

The results shows that with the increasing the thickness of the sheet metals, the forming limit strain is incremented. Dry test is very much sensitive to the small changes in the mechanical properties than that of the lubricated tests. The property of material such as “r” and “n” values affect the finite limit diagrams. The increasing the “n” and “r” value for the sheets which is accessible in the paper by S. A. Jenabali Jahromi, A. Nazarboland, E. Mansouri, S.Abbasi [5].

The numerical finite element method is used for calculating forming limit stress diagrams of the sheet metal piece. The conventional forming limit diagrams is well accepted tool for predicting the formability and protection limit of the material in sheet metal forming processes. The finite limit diagram is a strain based criterion by which principal strains at failure are evaluated. The forming limit diagram is dependent on forming and strain path, but stress based criterion does not confirm this dependency. This criterion is more robust against any changes in strain path happening in the forming process. A determination method of the stress based failure criterion for sheet metal forming was introduced in the paper by Sansot Panich, Vitoon Uthaisangsuk, Jittichai Juntaratin and Surasak Suranuntchai [6].

5. CONCLUSION

Nowadays, most of the finding work has been carried out on sheet metal determining the finite limit diagram by finite element method, experimental method, numerical analysis, material proper use on sheet metals. An attempt is to be carried out on forming examination of an automobile air filter mounting bracket based on above criter.

REFERENCES

[1] Keeler, S. P. & Backofen, W.A. (1964), “plastic instability and fracture in sheets stretched over rigid punches”, ASM trans. Quart. 56,25.

[2] Goodwin, G.M. (1968), “application of strain analysis to sheet metal forming problems in the press shop”, SAE paper no. 680093

[3] Thaweepat Buranathiti, Jian Cao, Z. Cedric Xia, Wei Chen, “Probabilistic Design In A Sheet Metal Stamping Process Under Failure Analysis”, CP778 Volume A. Numi sheet 2005.

[4] Mehmet Firat, “A numerical analysis of sheet metal formability for automotive stamping applications”, science direct, computational material science 43(2008) 802-811.

[5] S. A. JENABALI JAHROMI A. NAZARBOLAND, E. MANSOUL S. ABBASI, “Investigation of Formability of low carben steel sheets By Forming Limit Diagram,” Iranian of science and Technology, transaction B, 2005, Engineering Vol.30,No.B3

[6] Sansot Panich, Vitoon Uthaisangsuk Jittichai Juntaratin and Surasak Suranuntchai, “Determination of forming limit stress diagram for formability prediction of SPCE 270 steel sheet”, Joirnal of Metals, Materials and Minerals, 2011, Vol.21 No.1 19-27.