Module 3

Selection of Manufacturing Processes
Lecture 4

Design for Sheet Metal Forming Processes
Instructional objectives

By the end of this lecture, the student will learn the principles of several sheet metal forming processes and measures to be taken during these process to avoid various defects.

Sheet Metal Forming Processes

Sheet metals are widely used for industrial and consumer parts because of its capacity for being bent and formed into intricate shapes. Sheet metal parts comprise a large fraction of automotive, agricultural machinery, and aircraft components as well as consumer appliances. Successful sheet metal forming operation depends on the selection of a material with adequate formability, appropriate tooling and design of part, the surface condition of the sheet material, proper lubricants, and the process conditions such as the speed of the forming operation, forces to be applied, etc. A numbers of sheet metal forming processes such as shearing, bending, stretch forming, deep drawing, stretch drawing, press forming, hydroforming etc. are available till date. Each process is used for specific purpose and the requisite shape of the final product.

Shearing

Irrespective of the size of the part to be produced, the first step involves cutting the sheet into appropriate shape by the process called shearing. Shearing is a generic term which includes stamping, blanking, punching etc. Figure 3.4.1 shows a schematic diagram of shearing. When a long strip is cut into narrower widths between rotary blades, it is called slitting. Blanking is the process where a contoured part is cut between a punch and die in a press. The same process is also used to remove the unwanted part of a sheet, but then the process is referred to punching. Similarly, nibbling, trimming are a few more examples of cutting process using the same principle of shearing process.

![Figure 3.4.1 Schematic set-up of shearing operation[1](image)]
Bending

Bending is the operation of deforming a flat sheet around a straight axis where the neutral plane lies. It is a very common forming process for changing the sheets and plates into channel, drums, tanks, etc. Two different scheme of bending are shown in the figure 3.4.2. **Spring back** is a major problem during bending of sheets that occurs due to elastic recovery by the material causing a decrease in the bend angle once the pressure is removed. The springback can be minimized by introducing excess amount of bending so that the finished bending angle is the same after the elastic recovery. However, a careful estimate of the elastic recovery based on the mechanical behaviour of the sheet material is necessary to achieve the same.

*Figure 3.4.2* Schematic set-up of (a) air vee bending, and (b) die bending [3]

Stretch Forming

It is a method of producing contours in sheet metal. In a pure stretch forming process, the sheet is completely clamped on its circumference and the shape is developed entirely at the expense of the sheet thickness. *Figure 3.4.3* presents a schematic set-up of stretch forming process. The die design for stretch forming is very crucial to avoid defects such as excessive thinning and tearing of the formed part. The stretch forming process is extensively used for producing complex contours in aircraft and automotive parts.
Deep Drawing

Deep drawing is a sheet metal forming process in which a sheet metal blank is radially drawn into a forming die by the mechanical action of a punch. It is thus a shape transformation process with material retention. The process is considered "deep" drawing when the depth of the drawn part exceeds its diameter. This can be achieved by redrawing the part through a series of dies.

The metal flow during deep drawing is extensive and hence, requires careful administration to avoid tearing or fracture and wrinkle. Following are a few key issues affecting metal flow during deep drawing process and each of them should be considered when designing or troubleshooting sheet metal deep drawing stamping tools.
Type of material used and its thickness
Slightly thicker materials can be gripped better during the deep drawing process. Also, thicker sheets have more volume and hence can be stretched to a greater extent. However, the drawing force will increase with the sheet thickness. The percentage elongation property or ductility of the material is an essential quality for materials to be used for deep drawing.

Tool surface finish and use of Lubricant
Die surface finishes and lubricants are important to reduce the friction between the tool surfaces and metal being drawn, thus allowing materials to flow through tools more easily. Die temperatures can affect the viscosity of the lubricants. Slower deep drawing speed allows better metal flow.

Blank size and shape
Blank that are too large can restrict metal flow. The geometry of parts can also affect the ability of metal to flow during deep drawing process.

Blank Holding Force
Control of the blank holding force (BHF) enables control of friction on the flange during deep drawing process and significantly influences the quality of drawn part. Greater blank holding force may lead to tearing of the flange while inadequate blank holding force may lead to wrinkling of the flanges.

Punching speed
Sufficient punching speed allows time for materials to flow through the tool. Corner cracking will always occur if press speed is too fast in deep drawing process.

Draw radius
Radius on the draw die where the material flows through should be optimum. Too big a die or punch radius can result in wrinkling where as too small a die radius would create cracking at the bottom radius of drawn part.

Draw Bead Height and Shape
This should be selected properly to control metal flow and gripping pressure in deep drawing process. Draw bead height and shape can cause materials to bend and unbend to create restrictive forces going into a tool. Increasing pressure will exerts more force on a material, creating more restraint on material going into the tool.
Defects in Deep Drawing

Wrinkling is a major defect in drawing operation. The movement of the blank into the die cavity induces compressive stress in the flange causing wrinkle. This can be reduced by keeping a blank holder under the effect of a holding force. The magnitude of the force has to be controlled as a function of punch travel to minimize wrinkle. Rapture and excess thinning of the cup wall are also some pronounced defects in deep drawing process. Rapture is caused by incorrect draw ratio and improper die and punch design whereas too little die and punch clearance is responsible for the thinning of the cup wall. Failure generally occurs by thinning of the cup wall under high longitudinal stress.

Hydroforming

Hydroforming, as shown schematically in figure 3.4.5, is a specialized type of die forming that uses pressurized hydraulic fluid to form typical metallic sheets into a desired shape with a die cavity. Hydroforming is a cost-effective way of shaping malleable metals such as aluminium or brass into lightweight, structurally stiff and strong pieces. Hydroforming allows complex shapes with concavities to be formed, which would be difficult or impossible with standard solid die stamping. Hydroformed parts can often be made with a higher stiffness-to-weight ratio and at a lower per unit cost than traditional stamped or stamped and welded parts. Virtually all metals capable of cold forming can be hydroformed, including aluminium, brass, carbon and stainless steel, copper and high strength alloys.

Figure 3.4.5  Schematic set-up of Hydroforming process of sheet metal parts [3]
Design for Sheet Metal Forming

- Design of sheet metal components should be such that it would minimize scrap loss and die cost, and improve efficiency. Figure 3.4.6 illustrates how simple change in part design can improve the utilization of material utilization for blanking operation. Notching a blank along one edge results in an unbalanced force that makes it difficult to control dimension as accurately as with blanking round the entire contour.

![Original layout and improved layout](image)

*Figure 3.4.6* Changes in design to minimize scrap loss in blanking [1]

- When holes are punched in metallic sheet, only part of the metal thickness is sheared cleanly, i.e. a hole with tapered sides is created. If the hole is to be used as a bearing surface, then a subsequent operation will be required to obtain parallel walls inside the hole. Diameter of the punched holes should not be less than the thickness of the sheet or can be a minimum of 0.635 mm. Smaller holes result in excessive punch breakage and
should be drilled. For making threaded hole, the sheet must be at least one-half the thread diameter.

- Bending is the simplest sheet forming operation. The greatest formability in bending is obtained when the bend is made across the metal grain or in the direction of rolling. The largest possible bend radius should be used and it should not usually be lesser than the sheet thickness ‘t’.

- The cost in sheet metal forming operation can be reduced by using thinner sheets if the strength and rigidity are increased by bending and forming into ribs configuration.

- During design, compensation for the spring back has to be incorporated such that the bend part attains the desired shape after spring back as shown in figure 3.4.7. Springback becomes more severe with increase in yield strength of sheet metals (as the same implies greater possibility of elastic deformation and recovery) and section thickness. The final bend radius, $R_f$, and the initial bend radius before spring back, $R_i$, can be estimated by the relation given in equation 1,

$$\frac{R_i}{R_f} = 4 \left(\frac{YR_i}{ET}\right)^3 - 3 \left(\frac{YR_i}{ET}\right) + 1$$

- An important tool in developing sheet metal forming is the Keller-Goodman forming limit diagram (figure 3.4.8). The forming limit diagram typically represents the maximum permissible range of major and minor strains that a typical sheet material can undertake without failure. The forming limit diagram is experimentally measured for each sheet material by placing a grid of circle on the sheet sample and then deforming the same. A comparison of the original and the extensions in the marked...
grids on the sheet sample provide an estimate of the major and minor strain of the sample. The permissible range of the major and minor strains is actually available for a wide range of sheet metals and used considerably for designing of deep drawing operations, in particular.

**Figure 3.4.8** Typical forming limit diagram indicating the safe limits [3]
Exercise

1. During a bending operation, in which case, the amount of spring back will be minimum- when the plane of bending is along the direction of grain or in perpendicular to the direction of grain?
2. The amount of spring back depends on which parameter/parameters?

References