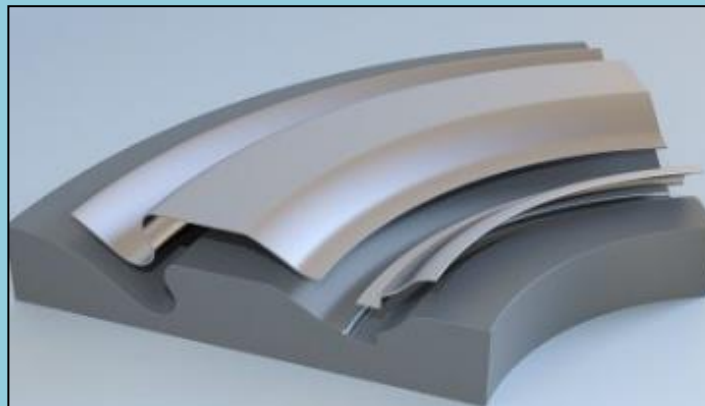
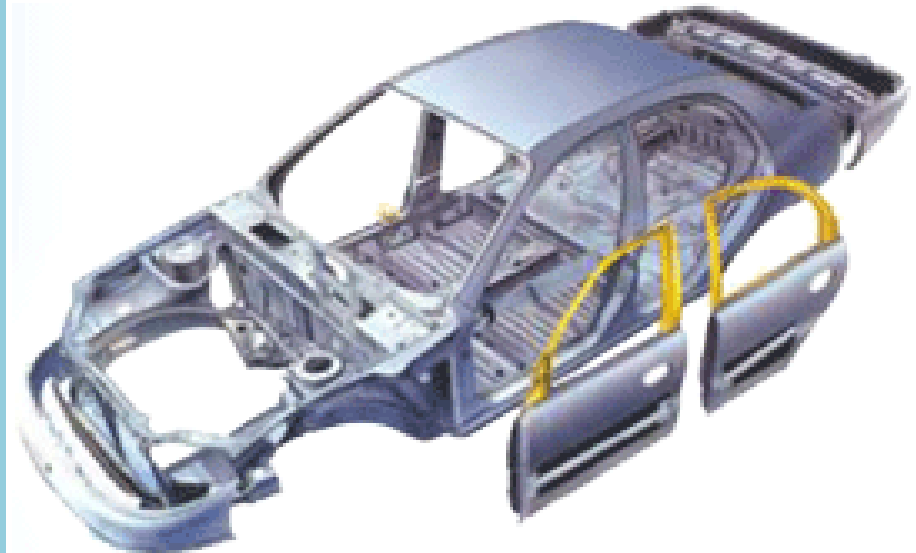


LECTURE-03: SHEET METAL FORMING PROCESSES



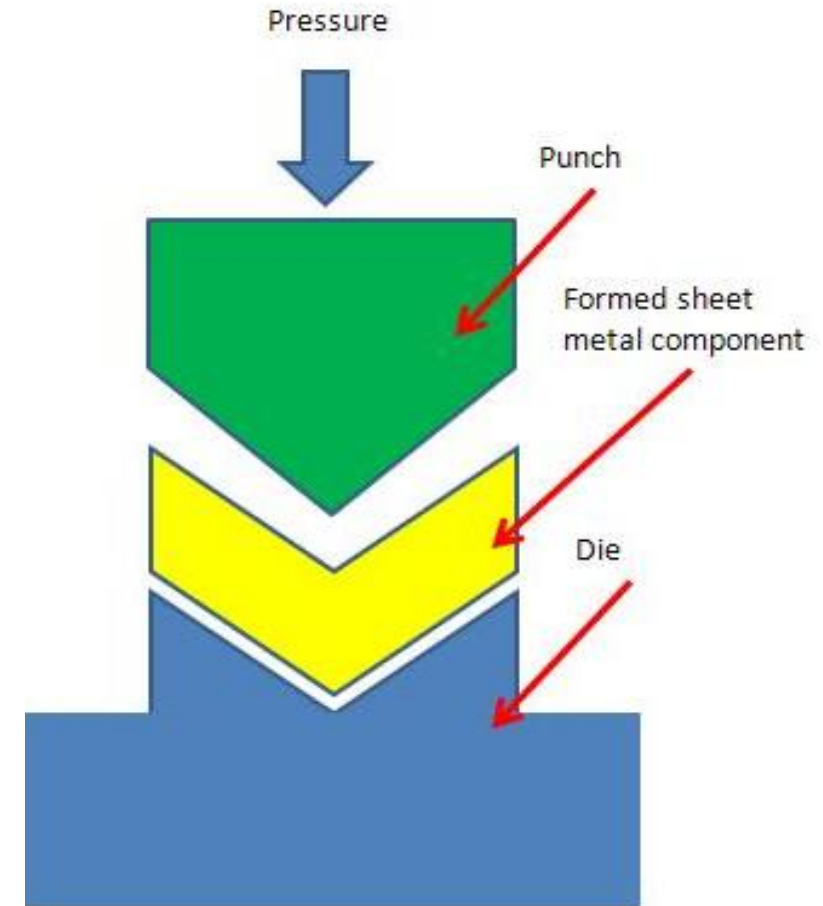
DR. NIKHIL R. DHAR

Professor, IPE Department
BUET

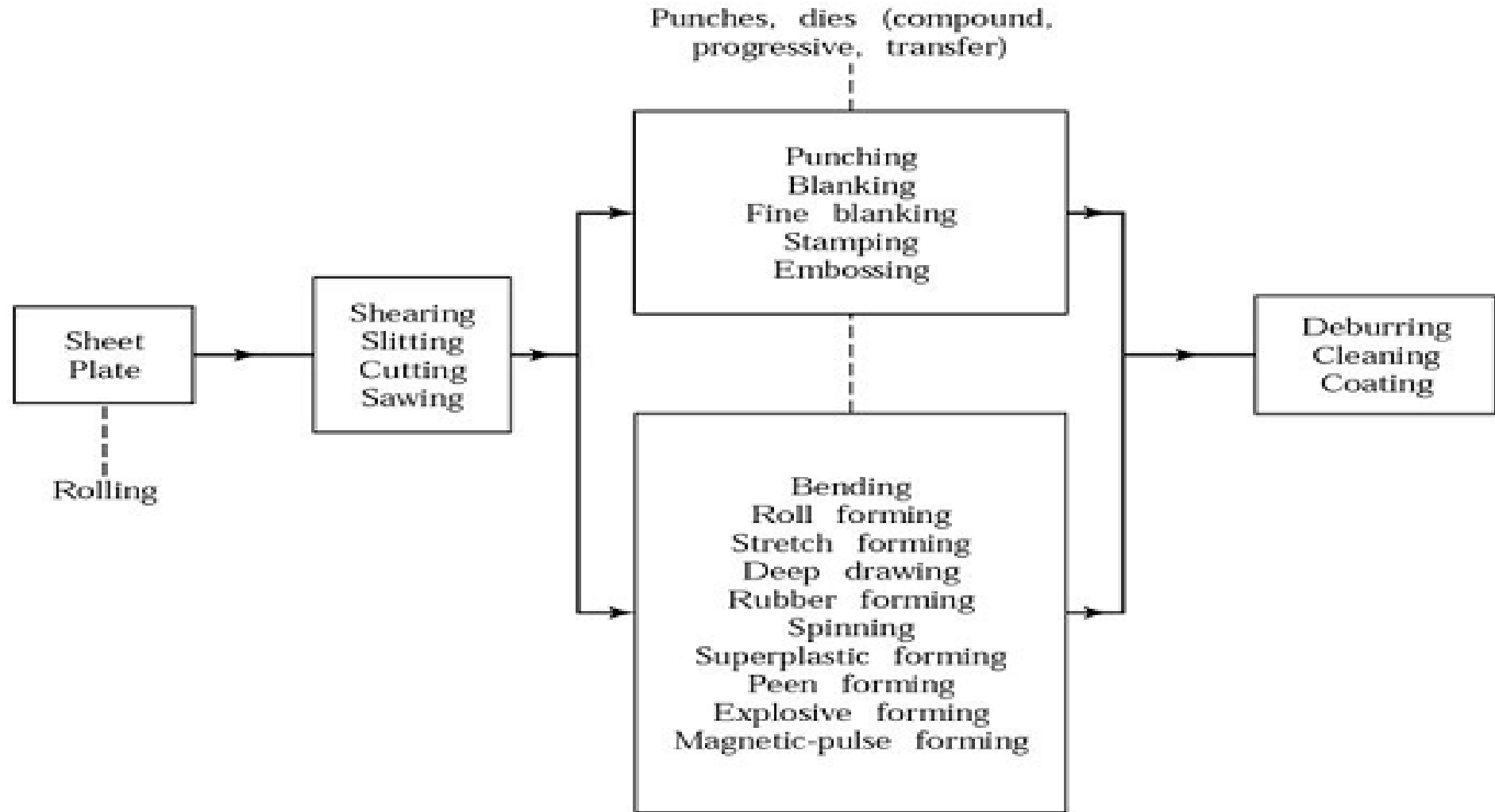
Introduction

- Sheet metal forming is a grouping of many **complementary processes** that are used to form sheet metal parts. One or more of these processes is used to take a flat sheet of **ductile metal**, and mechanically apply deformation forces that alter the shape of the material. Before deciding on the processes, one should determine whether a particular sheet metal can be formed into the **desired shape without failure**. The sheet metal operations done on a press may be grouped into **two categories**,

- **Cutting (Shearing) Operations**
- **Forming Operations.**

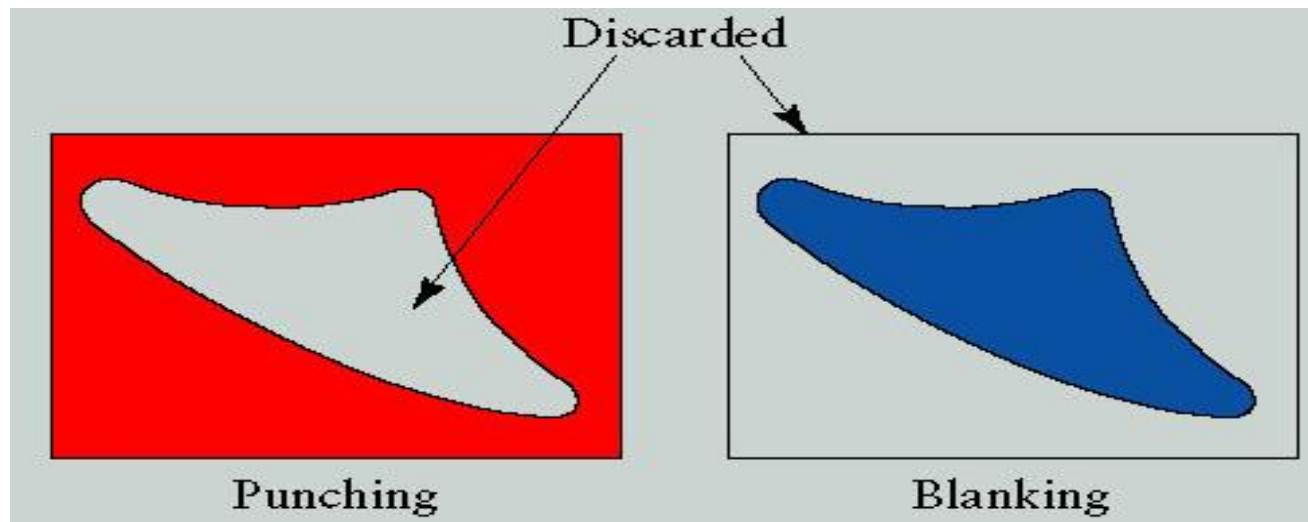


Outline of Sheet Metal Forming Processes



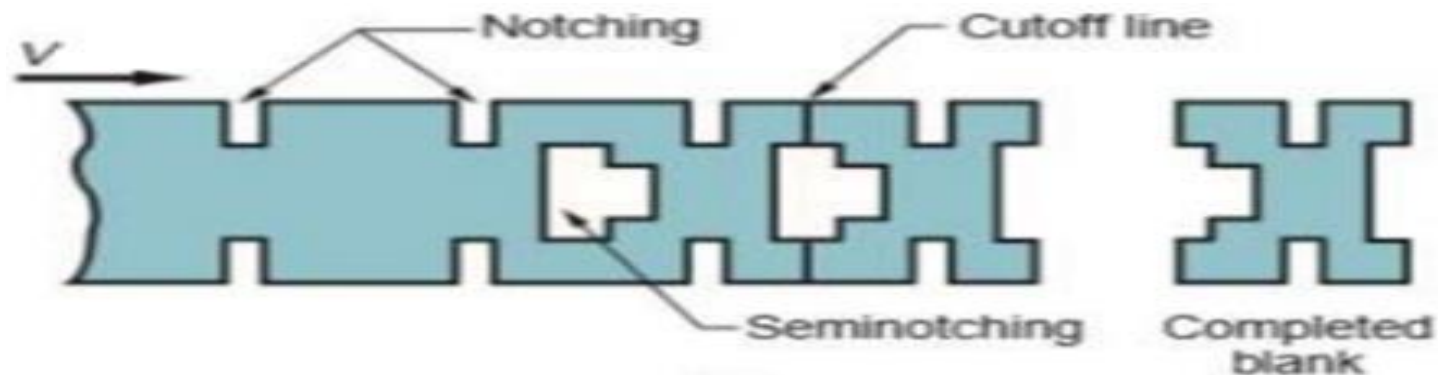
Cutting (Shearing) Operations

- **Punching (Piercing)**: It is a cutting operation by which various shaped holes are made in sheet metal. Punching is similar to blanking except that in punching, the hole is the desired product, the material punched out to form the hole being waste.
- **Blanking**: Blanking is the operation of cutting a flat shape sheet metal. The article punched out is called the blank and is the required product of the operation. The hole and metal left behind is discarded as waste.

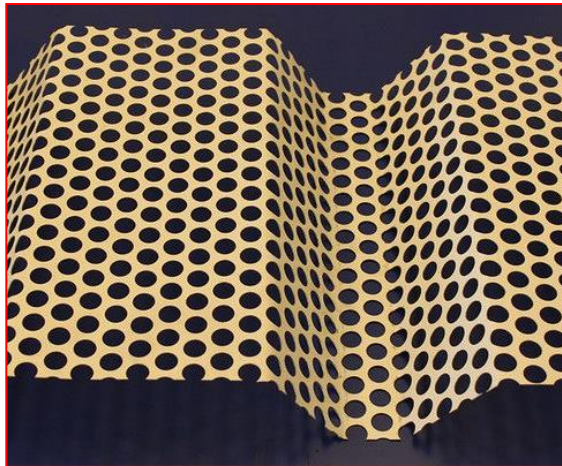


- **Notching**: Notching is a shearing process during which a metal scrap piece is removed from the outside edge of a metal workpiece. Notching is typically a manually operated, low-production process. During a notching operation, the metal workpiece has an outside edge removed by the use of multiple shear blades that are set at right angles to each other. Characteristics of the notching process include:

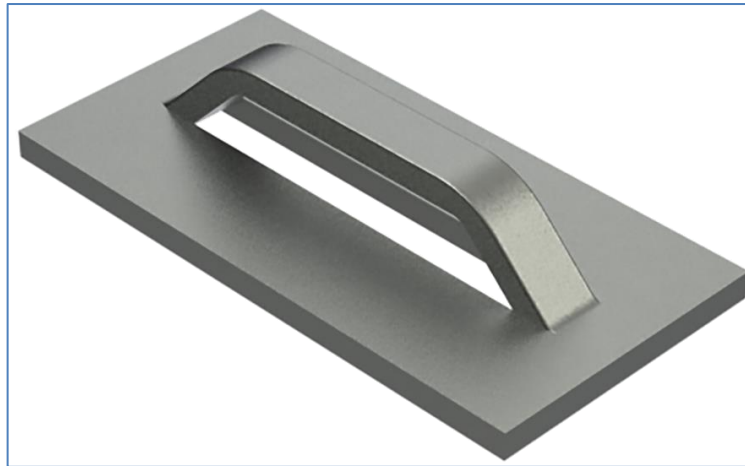
- Only removes metal from the outside of a workpiece
- Capable of making different angle notches
- Produces cuts not possible when using the shearing process



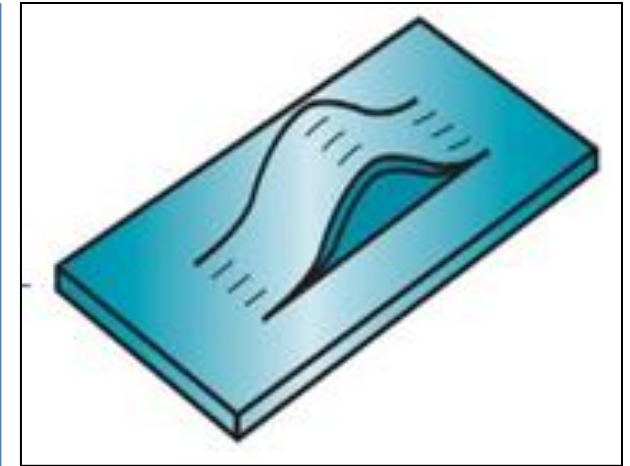
- **Perforating**: This is a process by which multiple holes which are very small and close together are cut in flat work material.
- **Slitting**: It refers to the operation of making incomplete holes in a workpiece
- **Lancing**: is a piercing operation in which the workpiece is sheared and bent with one strike of the die. A key part of this process is that there is not reduction of material, only a modification in its geometry. This operation is used to make **tabs**, **vents**, and **louvers**.



Perforating



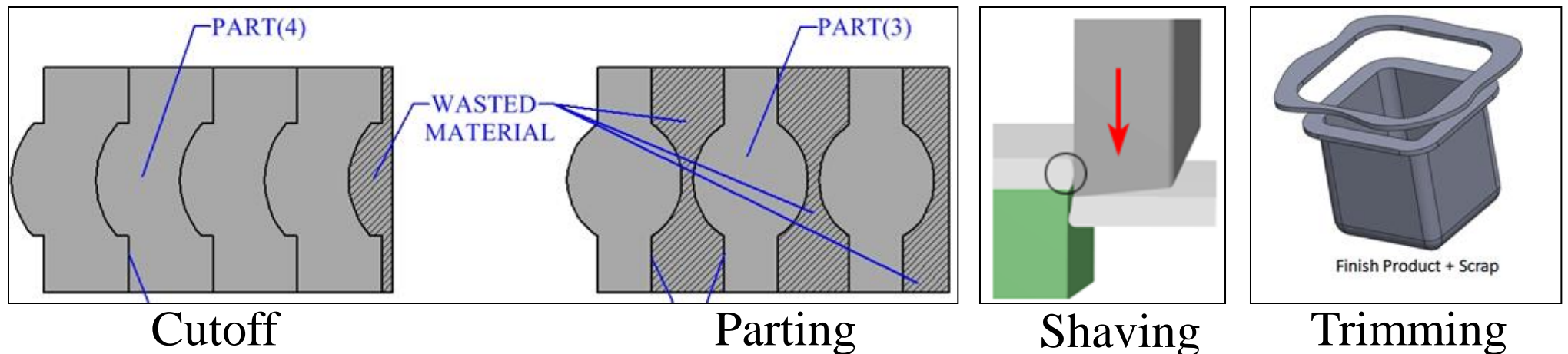
Slitting



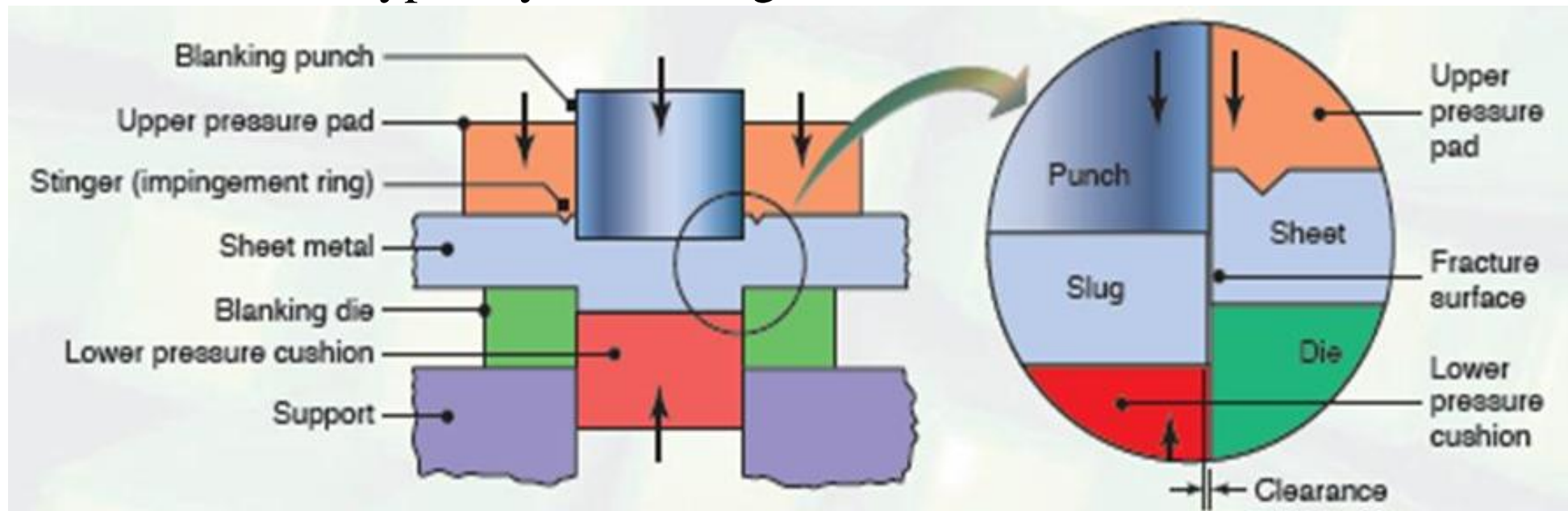
Lancing



- **Parting**: Parting involve cutting a sheet metal strip by a punch with two cutting edges that match the opposite sides of the blank. Parting is less efficient than cut off in the sense that it results in some wasted materials.
- **Shaving**: The edge of blanked parts is generally rough, uneven and unsquare. Accurate dimensions of the part are obtained by removing a thin strip of metal along the edges.
- **Trimming**: This operation consists of cutting unwanted excess material from the periphery of previously formed components.



- **Fine Blanking**. **Very smooth** and **square edges** can be produced by fine blanking. A V-shaped stinger or impingement mechanically locks the sheet tightly in place and prevents the type of distortion of the material. The thickness of the sheet may typically range from 0.5 to 13 mm. A suitable sheet hardness is typically in the range of 50-90 HRB

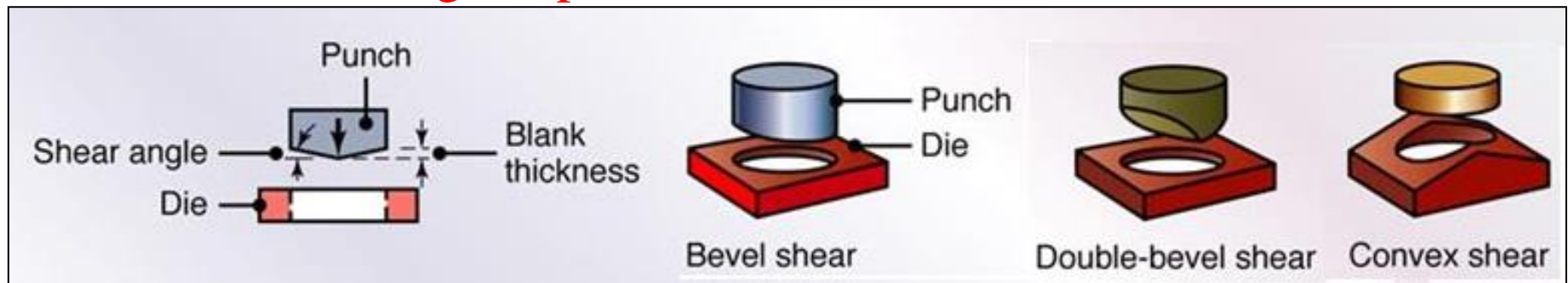


Schematic illustration of one setup for fine blanking



Shearing Dies

- Because the formability of a sheared part can be influenced by the **quality of its sheared edges**, **clearance control** is important. In practice, clearances usually range between 2% and 8% of the sheet's thickness; generally, the thicker the sheet, the larger is the clearance (as much as 10%). However, the smaller the clearance, the better is the quality of the edge.
- As the surfaces of the punch and die are flat; thus, the punch force builds up rapidly during shearing, because the entire thickness of the sheet is sheared at the same time. However, the area being sheared at any moment can be controlled by **beveling the punch and die surfaces**.

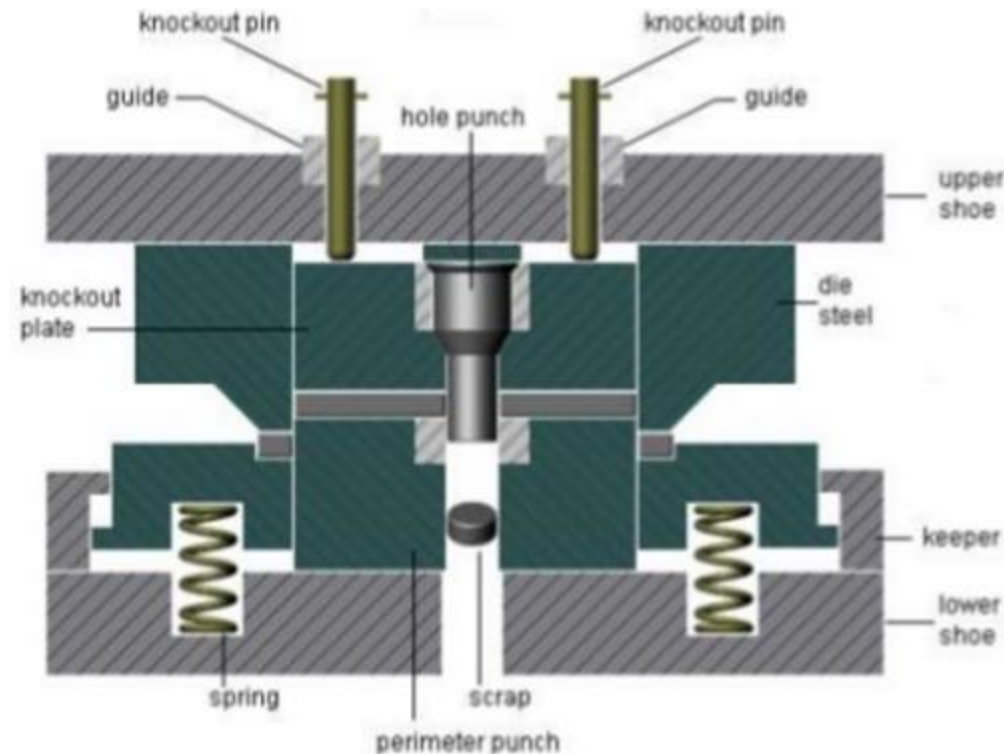


Examples of the use of shear angles on punches and dies.

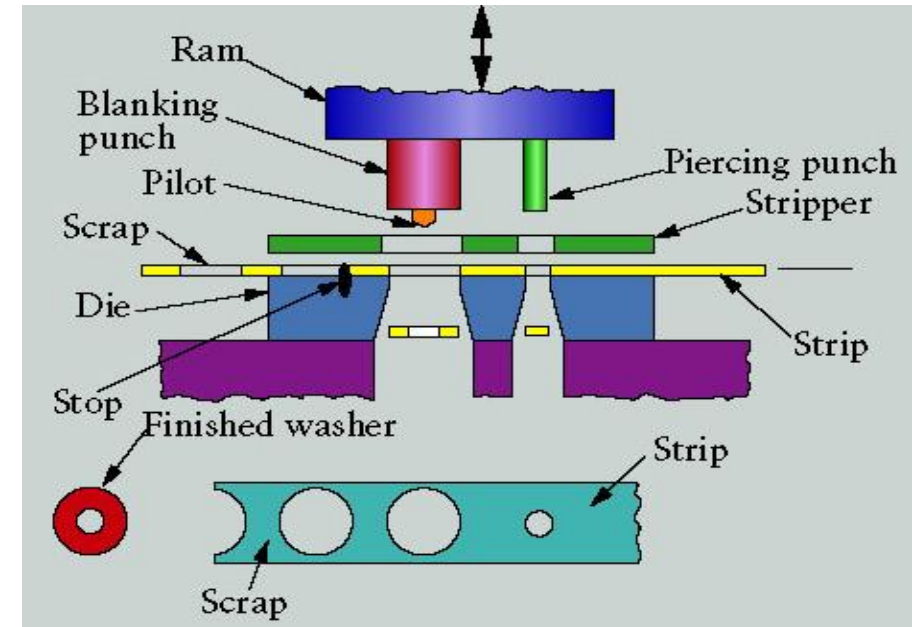


Dies used in Sheet Metal Operations

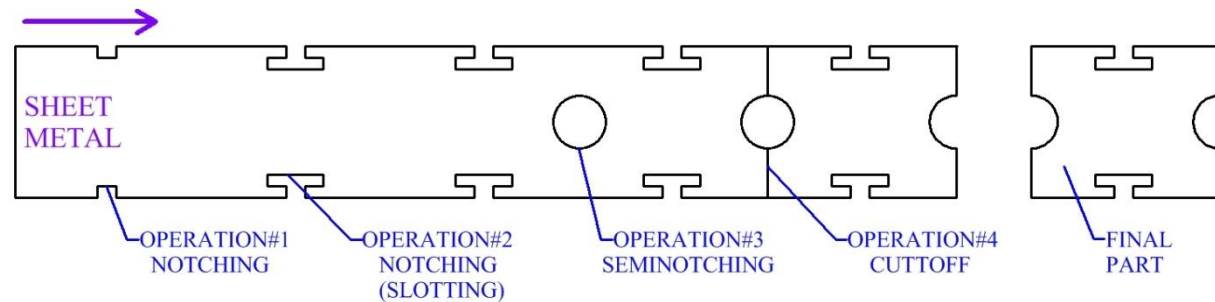
- **Compound Dies**: Several operations on the same strip may be performed in one stroke with a compound die in one station. These operations are usually limited to relatively simple shearing because they are somewhat slow and the dies are more expensive than those for individual shearing operations.



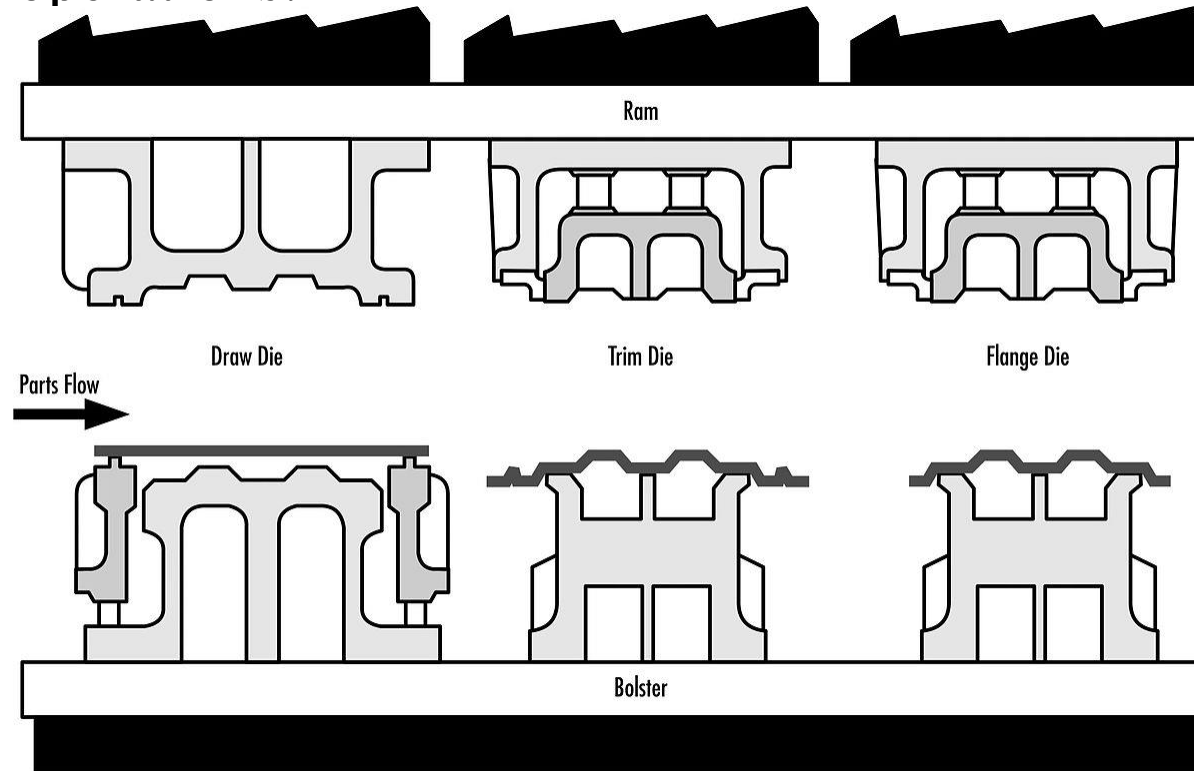
- **Progressive Dies**: Parts requiring multiple operations, such as punching, blanking and notching are made at high production rates in progressive dies. The sheet metal is fed through a coil strip and a different operation is performed at the same station



PROGRESSIVE PROCESSING OF SHEET METAL



- **Transfer Dies**: In a transfer die setup, the sheet metal undergoes different operations at different stations, which are arranged along a straight line or a circular path. After each operation, the part is transfer to the next operation for additional operations.



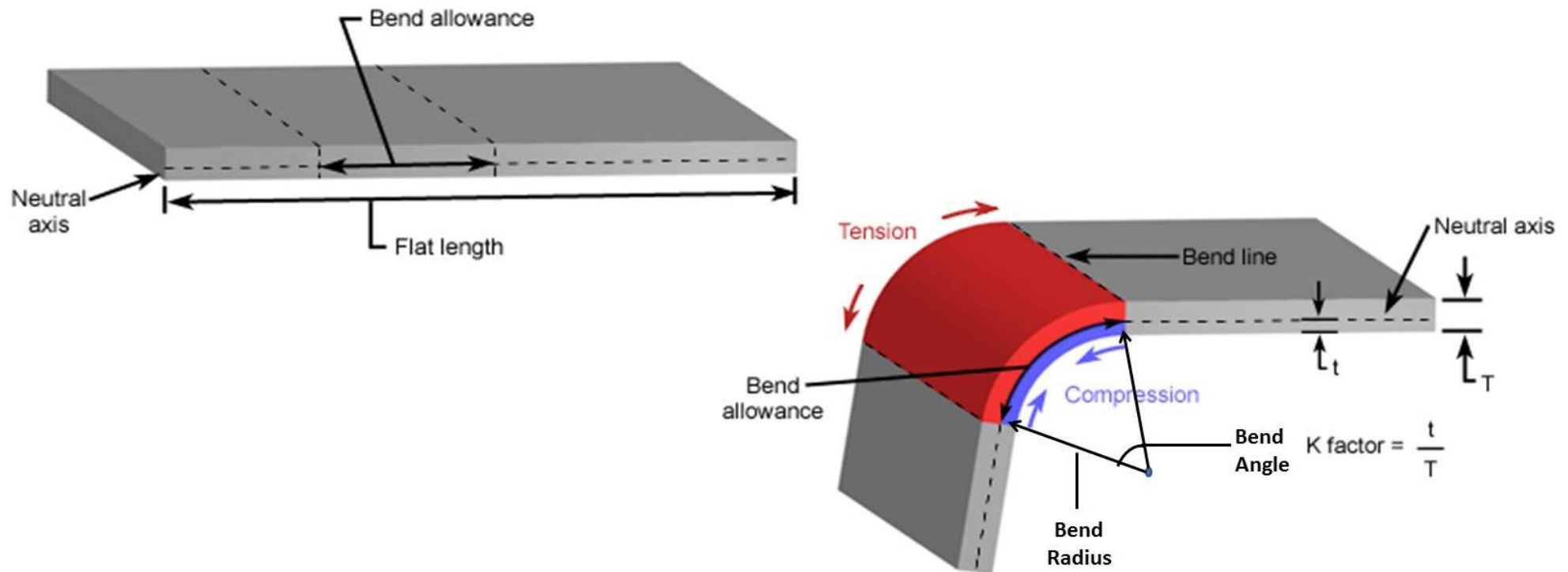
Forming Operations

- In this operation, the stresses are below the ultimate strength of the metal. In this operation, there is no cutting of the metal but only the contour of the sheet is changed to get the desired product. The forming operations include:
 - **Bending**: In this operation, the material in the form of flat sheet, is uniformly strained around a linear axis which lies in the neutral plane and perpendicular to the lengthwise direction of the sheet or metal. The bending operations include: **Flanging, Dimpling, Press break forming, Beading, Roll forming, Tube forming, Bulging and Stretch forming.**
 - **Drawing**: This is a process of a forming a flat sheet into a hollow shape by means of a punch, which causes the blank to flow into die cavity.
 - **Squeezing**: Under this operation, the metal is caused to flow to all portions of a die cavity under the action of compressive forces.



Sheet Metal Bending

- Bending is a most common sheet metal forming operation. Many products like **automobile components** and **its body**, **home appliance**, **paper clip**, **metal doors** etc. are shaped by bending process. Before discussing about bending process and its types, first you should learn about **basic terminologies** used



-
-
- Neutral Axis: It is an imaginary axis which does not undergo any stress during bending.
 - Outer Fibers: Those fibers which are under tension during bending process, known as outer fibers. It is shown by red color in figure.
 - Inner Fibers: Those fibers which are under compression during bending process, known as inner fibers. It is shown by blue color in figure.
 - Bend Allowance: The length of the neutral axis in the bend zone is known as bend allowance.
 - Bend Angle: The angle form by the bend area at the center of bend is known as bend angle.
 - Bend Radius: Distance between bend center and neutral axis is known as bend radius. It is denoted by R.
 - Minimum Bend Radius: The minimum bend radius is the bend radius at which a crack appears on the outer surface of the bend. It is usually expressed in term of **sheet thickness**, such that **2T, 3T, 4T** etc. It is different for different material.



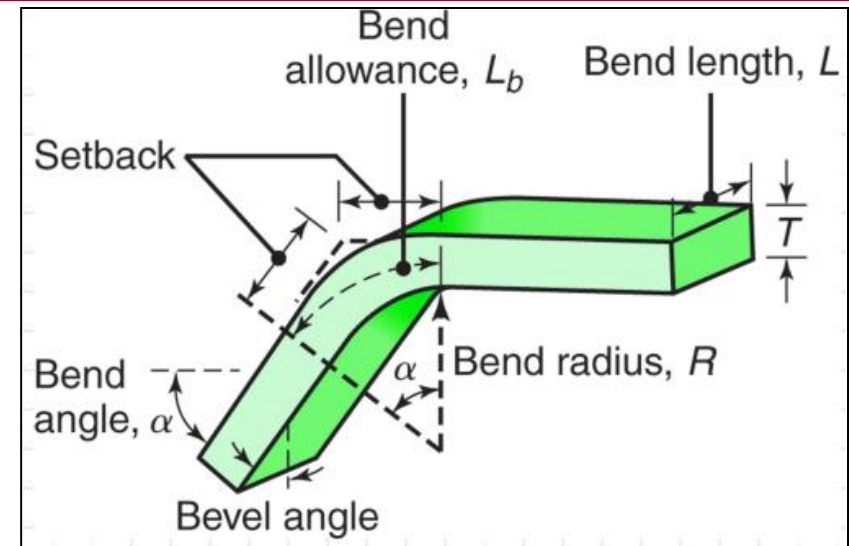
Bend Allowance

- The bend allowance, L_b , is the length of the neutral axis in the bend. It is used to determine the length of the blank for a part to be bend. It depends on the radius and the bend angle. An approximate formula for the bend allowance is,

$$L_b = \alpha (R + kT)$$

- Where α is the bend angle (in radians), T is the sheet thickness, R is the bend radius and k is a constant. In practice, k values typically range from 0.22 ($R < 2T$ to 0.5 ($R > 2T$). In ideal case, $k=0.5$, hence bend allowance is

$$L_b = \alpha (R + 1/2T)$$



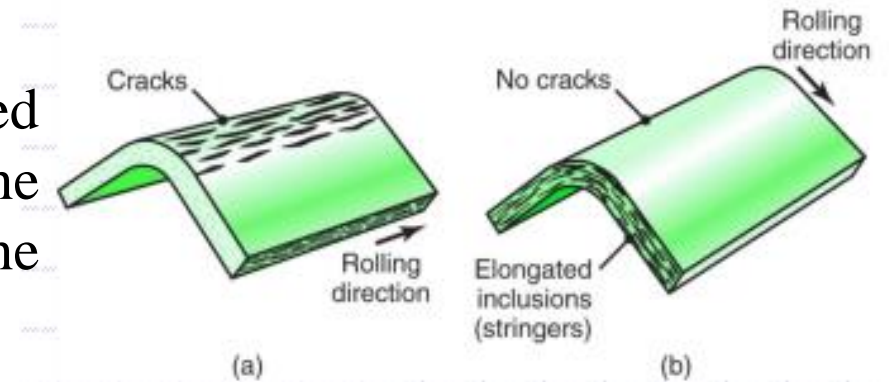
Minimum Bend Radius

- The radius at which a crack first appears at the outer fibers of a sheet being bent is referred to as the minimum bend radius. It can be shown that the **engineering strain (e)** on the outer and inner fibers of a sheet during bending is given by the expression

$$e = \frac{1}{(2R/T) + 1}$$

- Thus, **as R/T decreases** (that is, as the ratio of the bend radius to the thickness becomes smaller), the **tensile strain at the outer fiber increases** and the **material eventually develops cracks**.

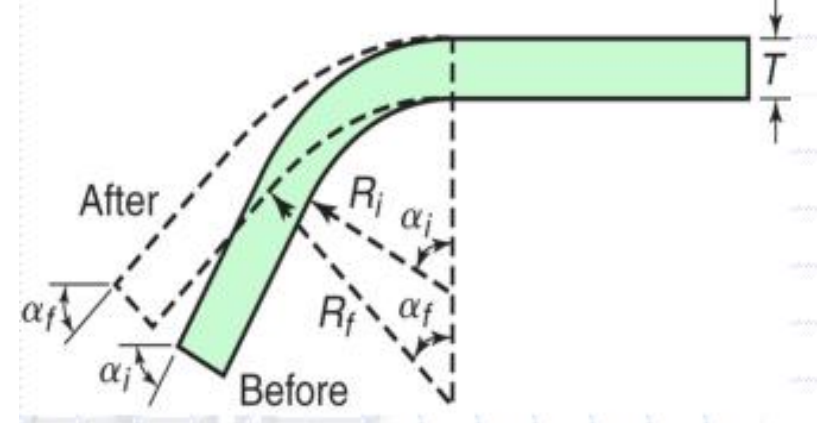
Figure (a) and (b): The effect of elongated inclusions on cracking as a function of the direction of bending with respect to the original rolling direction of the sheet. b



Springback

- Because all materials have a finite modulus of elasticity, plastic deformation always is followed by some elastic recovery when the load is removed. In bending, this recovery is called **springback**, which can be observed easily by bending and then releasing a piece of sheet metal or wire. **Springback occurs not only in flat sheets and plates, but also in solid or hollow bars and tubes of any cross section.**

$$SB = \frac{\alpha_i - \alpha_f}{\alpha_f}$$



Springback in bending. The part tends to recover elastically after bending, and its bend radius becomes larger. Under certain conditions, it is possible for the final bend angle to be smaller than the original angle (**negative springback**).



- The **springback of the bent sheet** can be approximated using the expression:

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

where, R_i and R_f are the initial and final bend radii, Y is the yield strength and E is the elastic modulus of the metal, and T is the thickness of the sheet.

- The **bending force** (P_b) required to bend a length L about a radius R may be estimated from

$$P_b = \frac{YLT^2}{2(R + T/2)} \times \tan \frac{\alpha}{2}$$

The **limiting drawing ratio** (R) measures the normal anisotropy, which denotes high resistance to thinning in the thickness direction.

$$R = \frac{\ln\left(\frac{W_o}{W}\right)}{\ln\left(\frac{T_o}{T}\right)} = \frac{\ln\left(\frac{W_o}{W}\right)}{\ln\left(\frac{WL}{W_o L_o}\right)}$$



- **Problem-01**: A straight bead is being formed on a 1-mm thick aluminum sheet in a 20-mm-diameter die cavity. Considering springback, calculate the outside diameter of the bead after it is formed and unloaded from the die. Assume $Y = 150$ MPa and $E = 70$ GPa.

- **Solution**: $R_i = (20/2) - 1 = 9$ mm

We know,
$$\frac{R_i}{R_f} = 4 \left(\frac{R_i Y}{ET} \right)^3 - 3 \left(\frac{R_i Y}{ET} \right) + 1$$

$$\frac{R_i Y}{ET} = \frac{(9/1000) \times 150}{70000 \times (1/1000)} + 1 = 0.0193$$

$$\frac{R_i}{R_f} = (0.0193)^3 - 3 \times 0.0193 + 1 = 0.942$$

$$R_i = \frac{9}{0.942} = 9.55 \text{ mm}$$

$$OD = 2 \times R_f + 2T = 2 \times 9.55 + 2 \times 1 = 21.1 \text{ mm}$$



- **Problem-02**: A tension test on a special deep-drawing steel showed a 30% elongation in length and a 16% decrease in width. What limiting draw ratio would be expected for the steel?

- **Solution**:

$$\frac{L - L_o}{L_o} = 0.30$$

$$\frac{L}{L_o} = 1.30$$

$$\frac{w - w_o}{w_o} = -0.16$$

$$\frac{w}{w_o} = 1 - 0.16 = 0.84$$

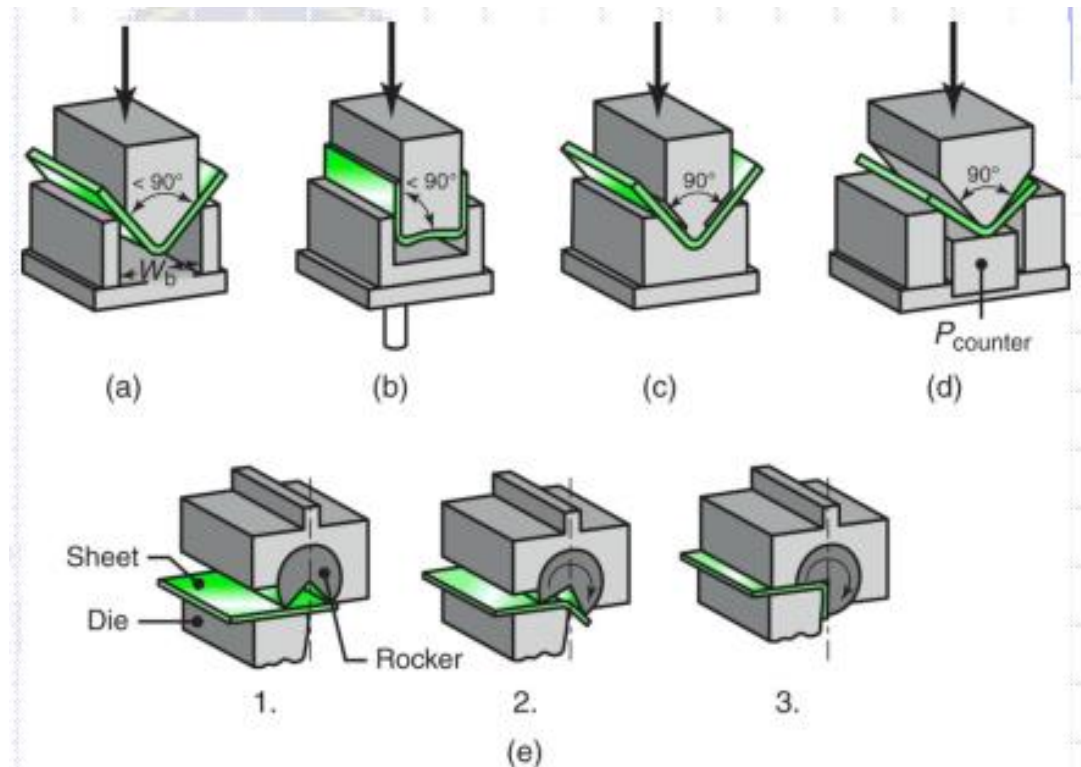
$$R = \frac{\ln(w_o / w)}{\ln((w / w_o)(L / L_o))} = \frac{\ln(1 / 0.84)}{\ln(0.84 \times 1.30)} = \frac{\ln 1.190}{\ln 1.092} = 1.98$$



Compensation for Springback

- Springback in forming operations usually is compensated for by **over-bending** the part. Several trials may be necessary to obtain the desired results. **Another method** is to **coin the bend area by subjecting it to highly localized compressive stresses** between the tip of the punch and the die surface

Methods of reducing or eliminating springback in bending operations.



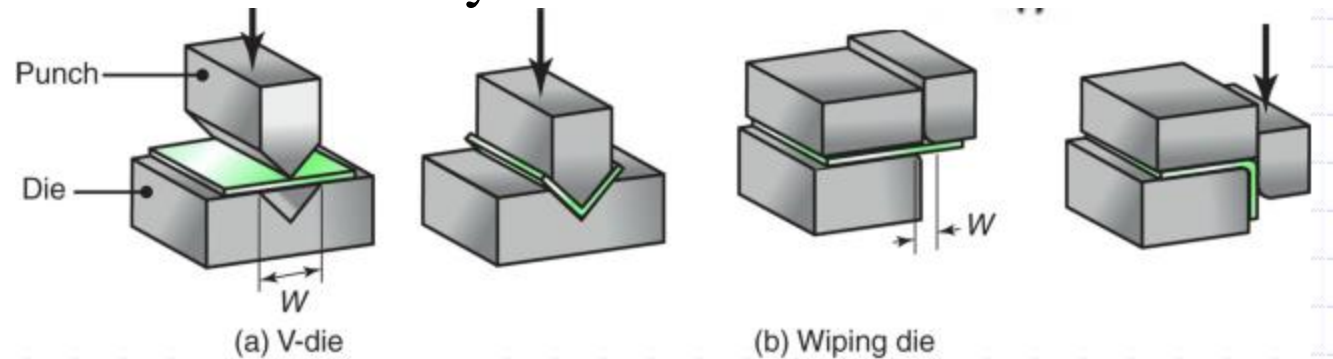
Bending Force

- The bending force for sheets and plates can be estimated by assuming that the process is one of simple bending of a rectangular beam, as described in texts on mechanics of solids. Thus, **the bending force** (P_b) is a function of the strength of the material, the length, L , of the bend, the thickness, T , of the sheet, and the die opening, W

$$P_b = \frac{kYLT^2}{W}$$

where the factor k ranges from about 0.3 for a wiping die, to about 0.7 for a U-die, to about 1.3 for a V-die and Y is the yield stress of the material.

Common die-bending operations showing the **die-opening dimension**, W , used in calculating bending forces.

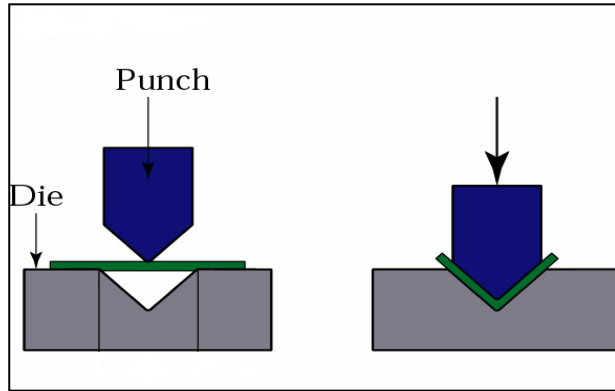


-
-
- **Problem-03**: Estimate the maximum bending force required for a 0.3175 cm-thick and 30.48 cm-wide Ti-5Al-2.5Sn titanium alloy in a V-die with a width of 15.24 cm. For a V-die, an average value of $k = 1.265$. Assume $UTS=860$ Mpa.
 - **Solution**: The problem statement gives us $L = 30.48$ cm, $T = 0.3175$ and $W = 15.24$ cm.

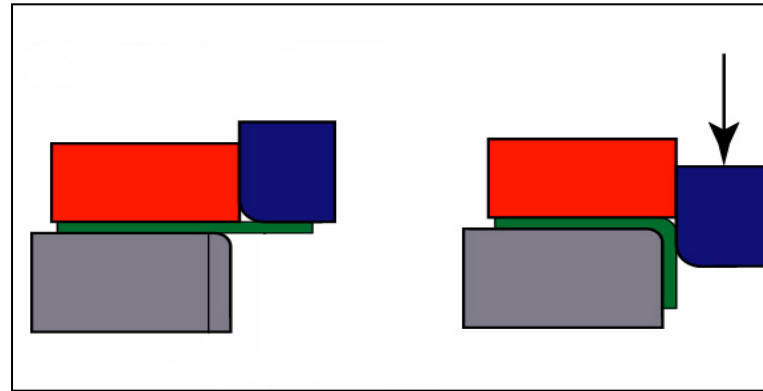
$$P_b = \frac{kYLT^2}{W} = 1.265 \frac{860 \times 30.48 \times 0.3175^2}{15.24} = 2241 \text{ kg}$$



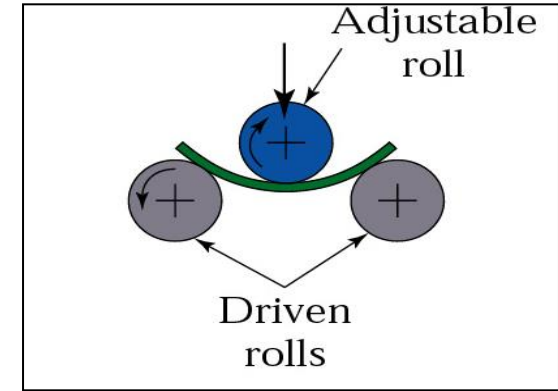
Examples of Various Bending Operations



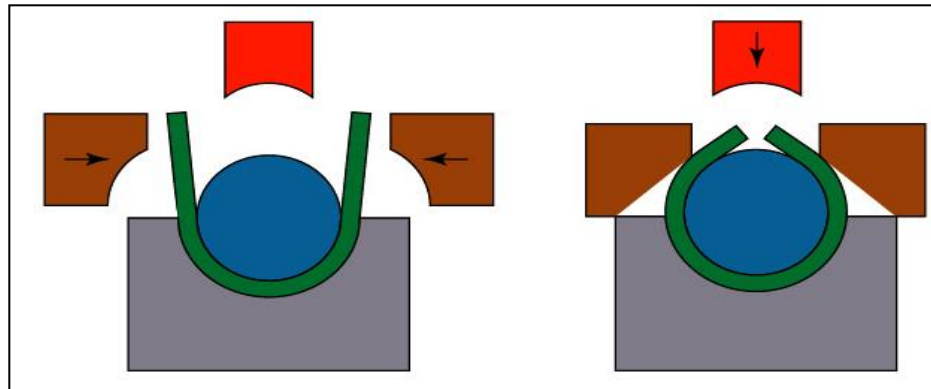
V-bending



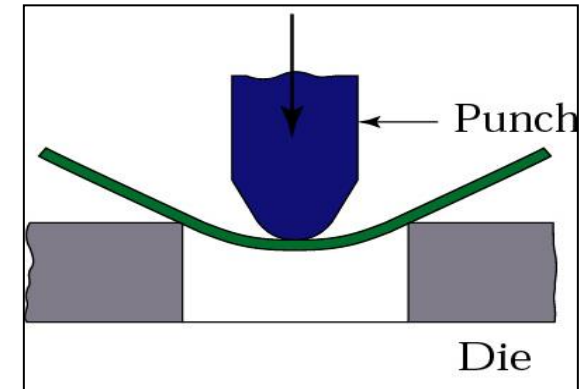
Edge bending



Roll bending



Bending in 4-slide machine

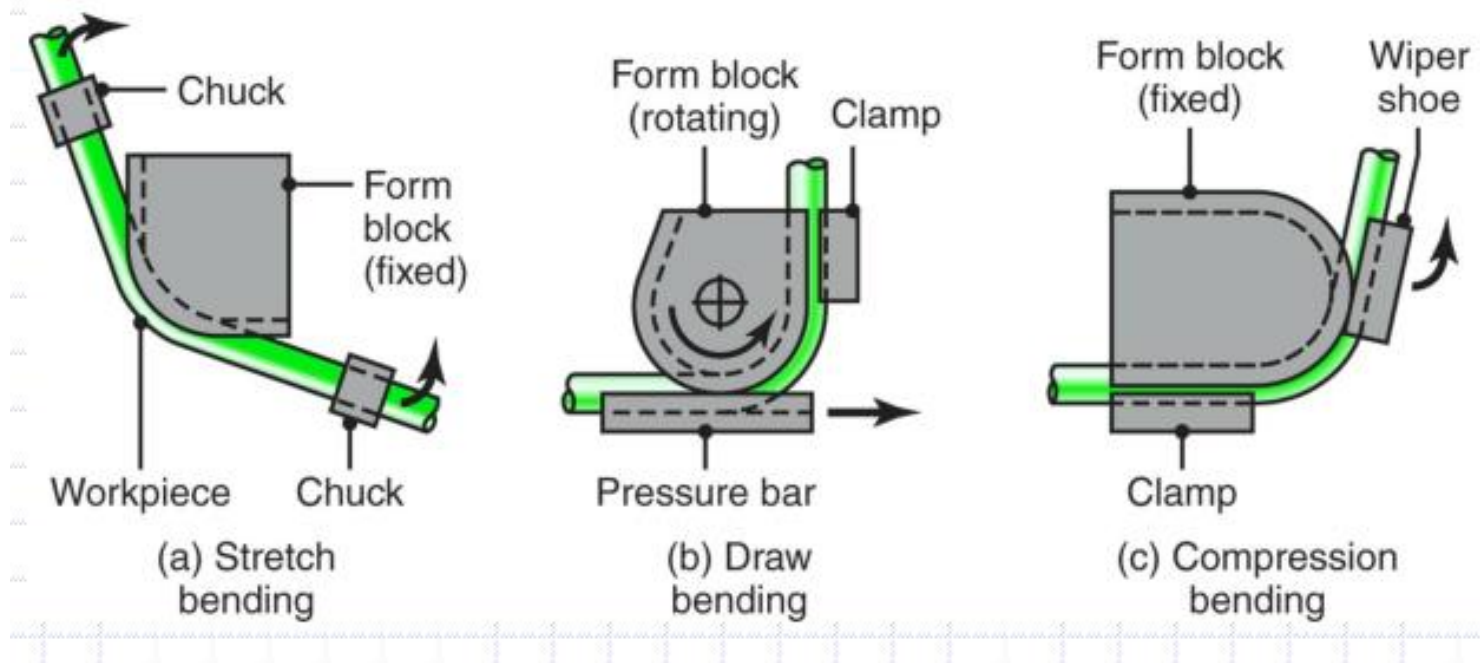


Air bending



Methods of Bending Tubes

- **Internal mandrels** or **filling** of tubes with particulate materials such as **sand** are often necessary to prevent collapse of the tubes during bending.



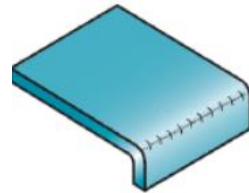
$$\text{Wall factor} = \frac{\text{tube outside diameter}}{\text{tube wall thickness}}$$

$$D \text{ of bend} = \frac{\text{bend centerline radius}}{\text{tube outside diameter}}$$

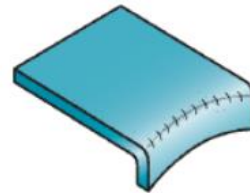


Different Sheet Metal Bending Process

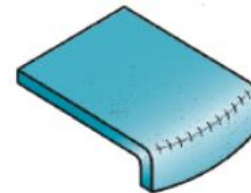
- **Flanging**: Flanging is a bending in which the **edge of a sheet metal** is **bent** at a **90° angle** to form a rim or flange. It is often used to strengthen or stiffen sheet metal. The flange can be straight, or it can involve stretching or shrinkage.



Straight flanging

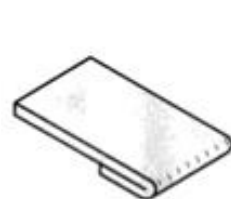


stretch flanging

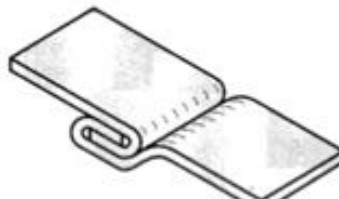


shrink flanging

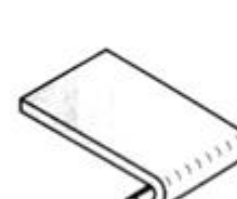
- **Press Brake Forming**: Sheet metal or plate can be bent easily with simple fixtures using a press. **Long and relatively narrow pieces** are usually bent in a press break. This machine utilizes long dies in a **mechanical** or **hydraulic press** and is suitable for **small production runs**.



Hemming



Seaming

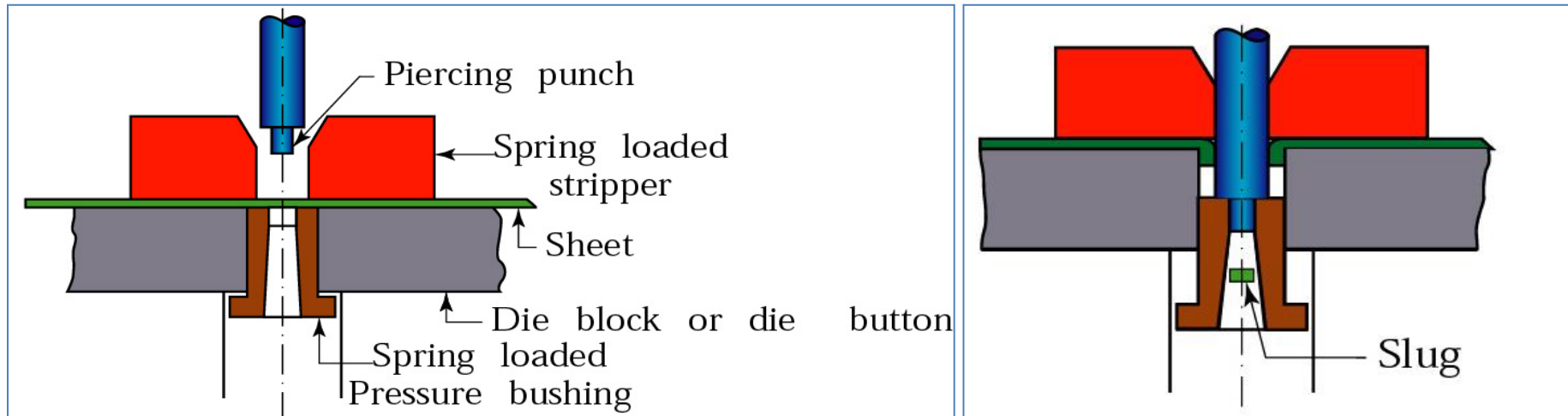


Curling

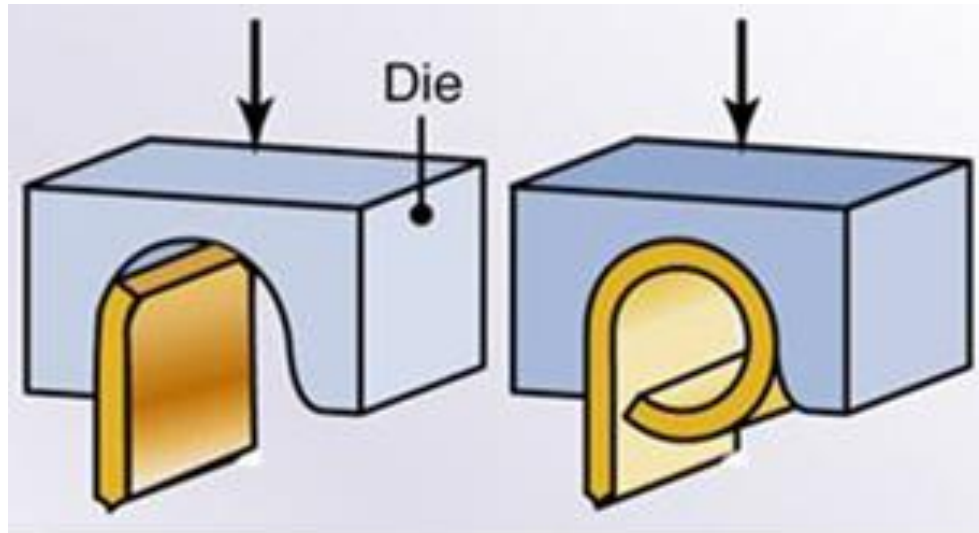


■ Dimpling

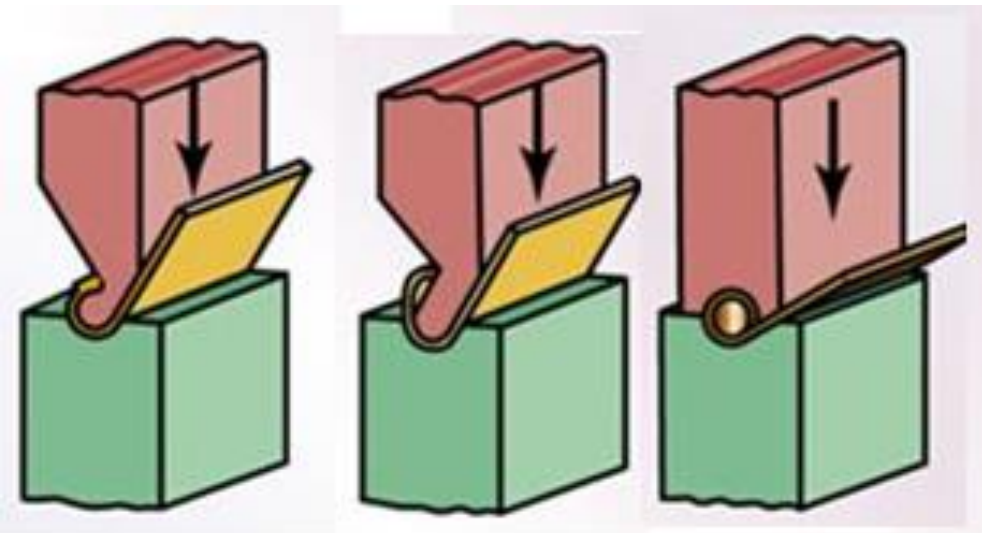
- First hole is punched and expanded into a flange
- Flanges can be produced by piercing with shaped punch
- When bend angle < 90 degrees as in fitting conical ends its called flanging



- **Beading**: In beading the edge of the sheet metal is bent into the cavity of a die. The bead gives stiffness to the part by increasing the moment on inertia of the edges. Also, it improves the appearance of the part and eliminates exposed sharp edges

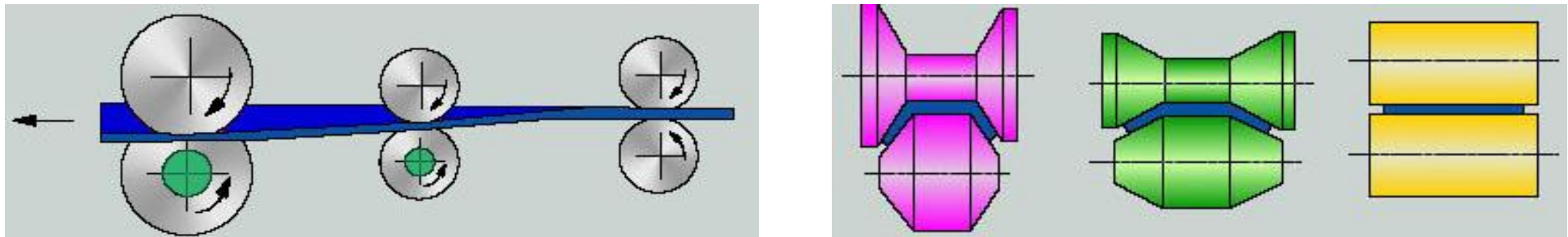


Bead forming with a single die

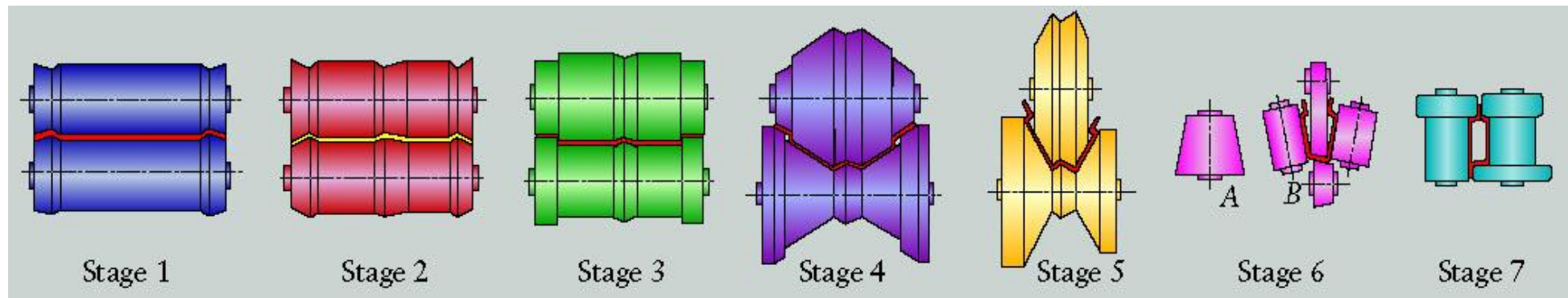


Bead forming with two dies

- **Roll Forming:** For bending continuous lengths of sheet metal and for large production runs, roll forming is used. The metal strip is bent in stages by passing it through a series of rolls.



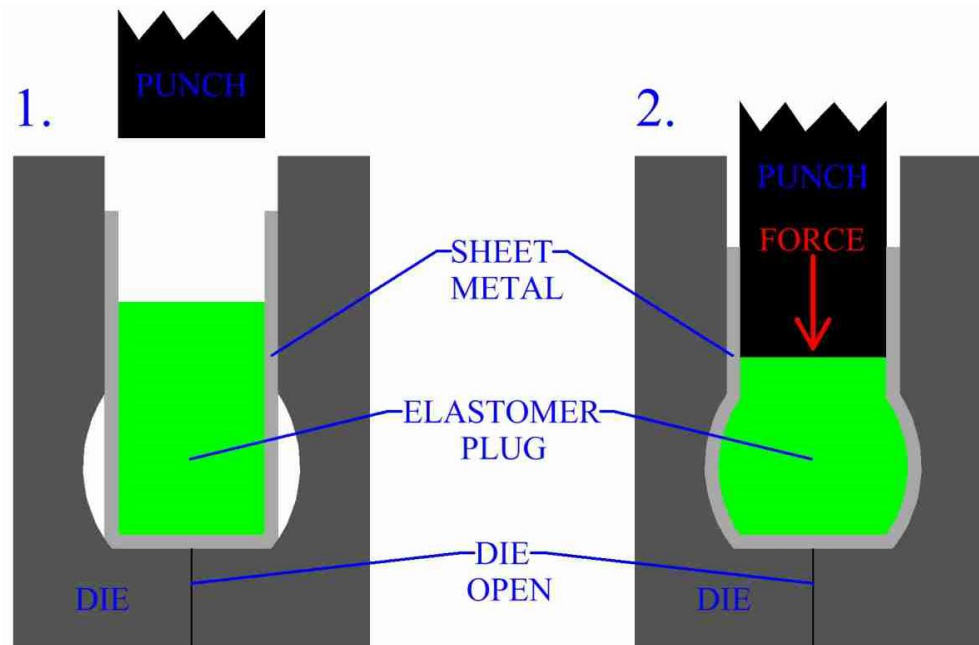
Roll-forming Process



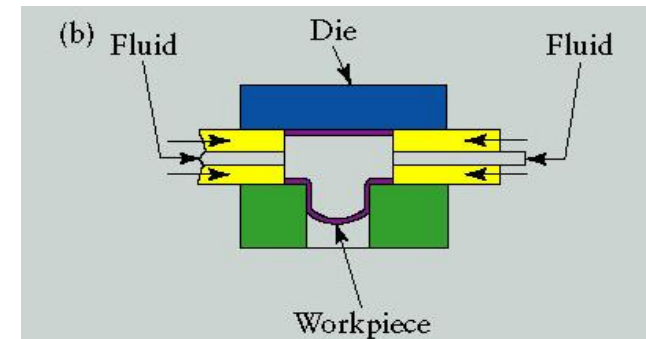
Stages in roll forming of a sheet-metal door frame. In Stage 6, the rolls may be shaped as in A or B.



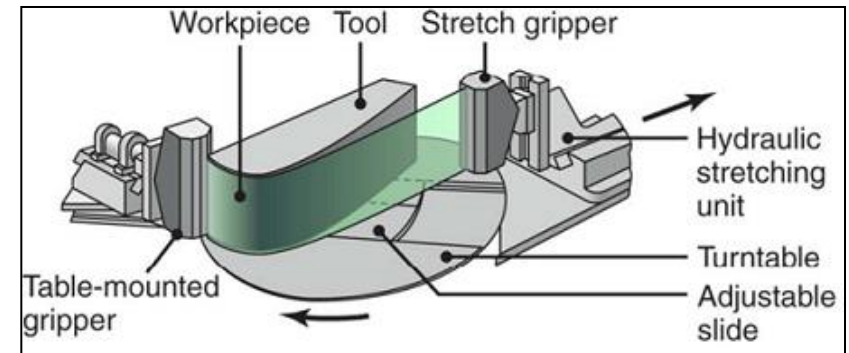
- **Bulging:** The basic forming process of bulging involves placing tabular, conical or curvilinear part into a split-female die and expanding it with, say, a polyurethane plug. The punch is then retracted, the plug returns to its original shape and the part is removed by opening the dies.



Bulging of a tubular part with a flexible plug.
Water pitchers can be made by this method

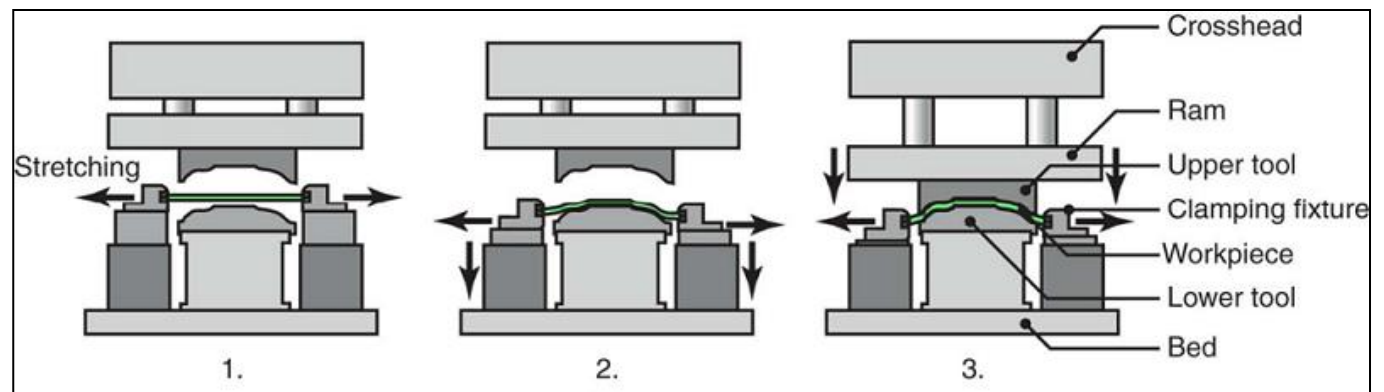


- **Stretch Forming**: In stretch forming, the sheet metal is clamped along its edges and then stretched over a male die. The die moves upward, downward, or sideways, depending on the particular design of the machine

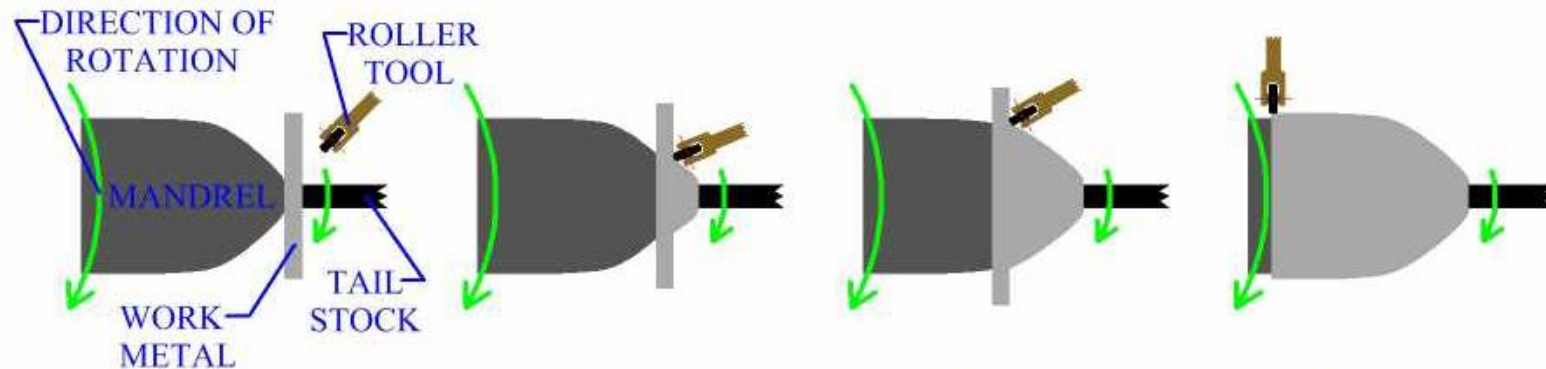


- Stretch forming is used primarily to make aircraft wing-skin panels, fuselages, and boat hulls. Aluminum skins for the Boeing 767 and 757 aircraft, for example, are made by stretch forming-with a tensile force of 9 MN. Although this process generally is used for low volume production, it is versatile and economical, particularly for the aerospace industry

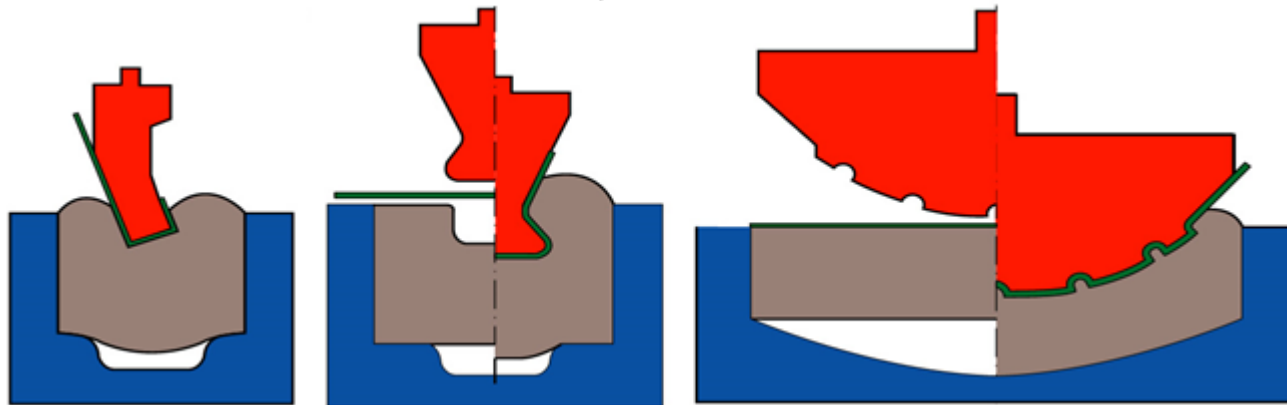
Aluminum skins for aircraft can be made by this method.



- **Metal Spinning**: Metal spinning also called conventional spinning or just spinning, is the forming of an axis-symmetric part over a **rotating mandrel**. Force is delivered by a tool with a **rounded end or a roller**. A flat, or possibly preformed, work piece is held between a mandrel and tail stock. As the apparatus rotates, the tool applies **localized pressure** to the work while it gradually moves up the mandrel. This causes the sheet metal work to be wrapped over the mandrel, thus taking its shape. Spinning is commonly performed cold, but in some cases parts may be subject to warm or hot. Spinning is capable of producing large parts with diameters as high as 20 feet.

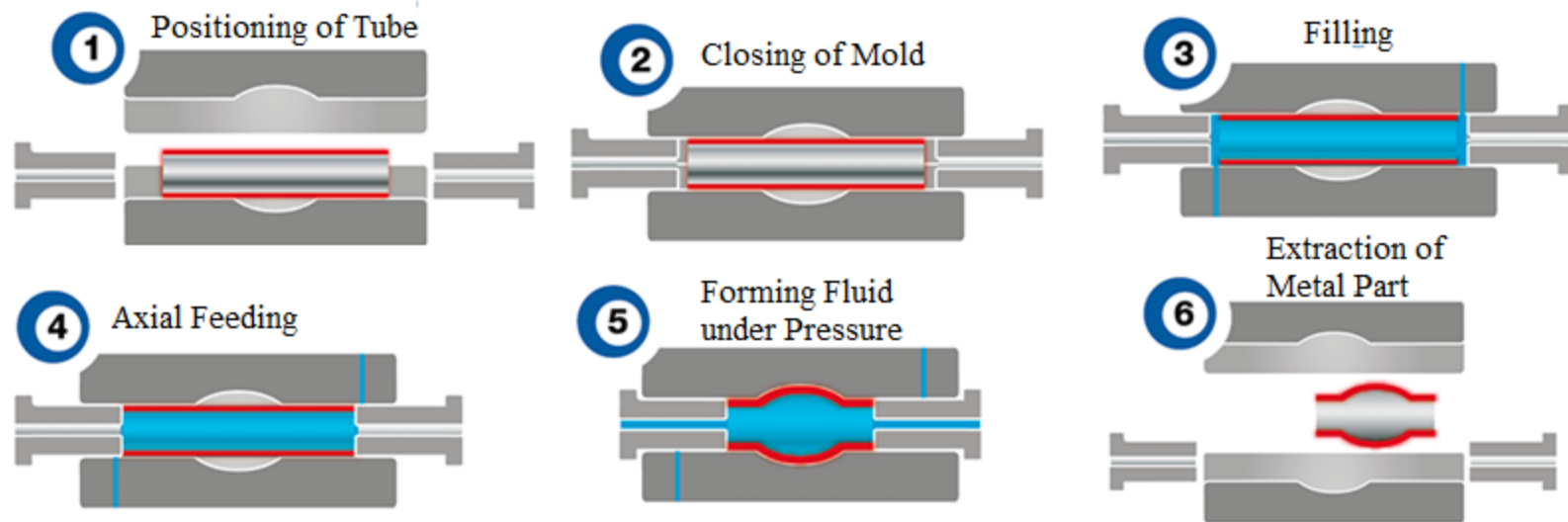


- **Rubber Forming**: In rubber forming, one of the dies in a set is made of **flexible material**, such as a **rubber** or **polyurethane membrane**. Polyurethanes are used widely because of their resistance to abrasion, long fatigue life and resistance to damage by burrs or sharp edges of the sheet blank. In **bending** and **embossing** sheet metal by the rubber forming method, as shown in the following Figure, the female die is replaced with a rubber pad. Parts can also be formed with **laminated sheets** of various **nonmetallic material** or **coatings**.

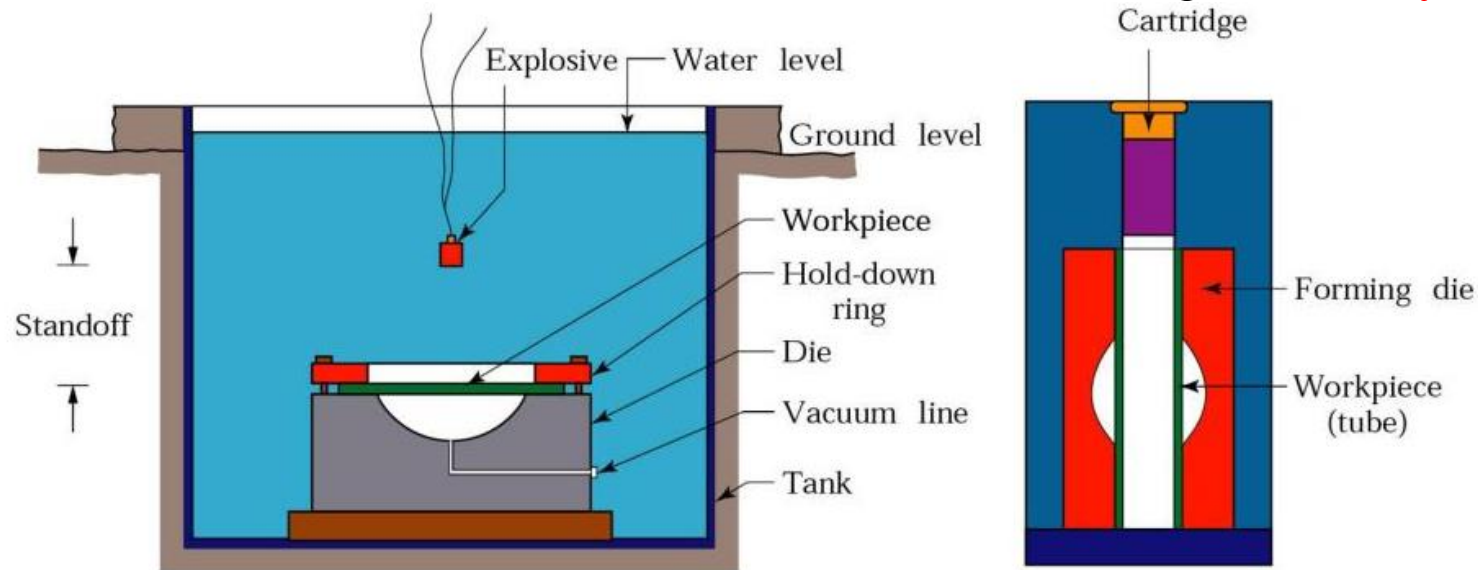


Bending and the embossing of sheet metal with a metal punch and with a flexible pad serving as the female die.

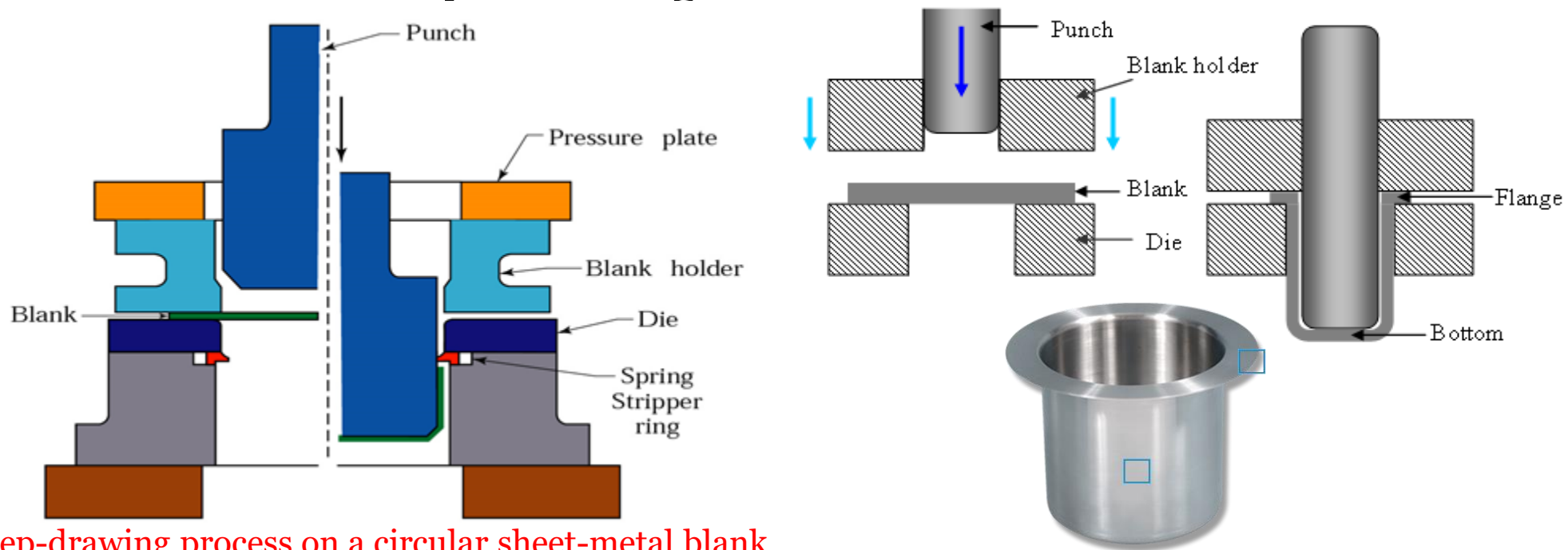
- **Hydroforming**: Hydroforming is an effective sheet metal forming process. Hydroforming can typically obtain deeper draws than conventional deep drawing operations. Hydroforming uses a rigid punch to push a sheet metal work piece into a rubber membrane. Behind the rubber membrane is a chamber of pressurized fluid. When the work is pressed into the chamber, the rubber membrane surrounds it completely and the pressure of the fluid forces the sheet metal to form on the punch.



- **Explosive Forming Process:** The sheet metal work piece blank is clamped over a die and the assembly is lowered into a tank filled with water. The air in the die is pumped out. The explosive charge is placed at some predetermined distance from the work piece. On **detonation** of the **explosive**, a **pressure pulse of very high intensity** is produced. A gas bubble is also produced which expands spherically and then collapses. When the pressure pulse impinges against the work piece, the metal is deformed into the die with as high **velocity as 120 m/s**.



- **Deep Drawing**: Drawing operation is the process of forming a flat piece of material (blank) into a hollow shape by means of a punch, which causes the blank to flow into the die-cavity. Round sheet metal blank is placed over a circular die opening and held in a place with blank holder & punch forces down into the die cavity. Wrinkling occurs at the edges.



Deep-drawing process on a circular sheet-metal blank

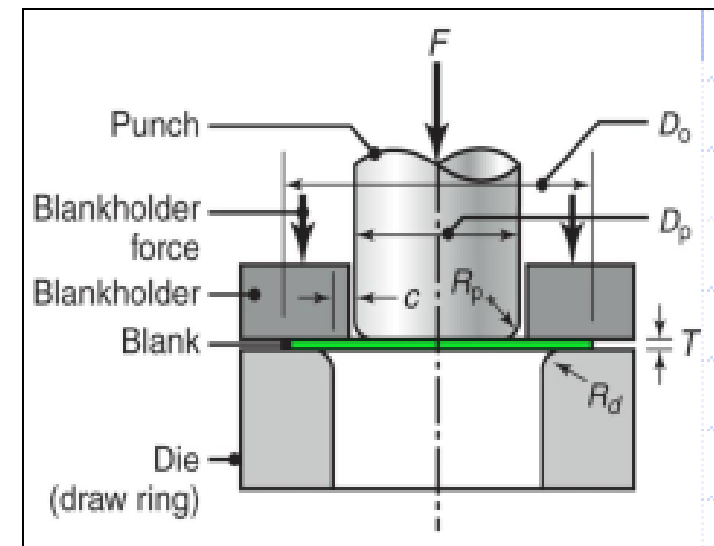


Punch Force during Deep Drawing

- During the drawing operation, the movement of the blank into the die cavity induces compressive circumferential (hoop) stresses in the flange, which tend to cause the flange to **wrinkle** during drawing. Wrinkling can be reduced or eliminated if a blank holder is loaded by a certain force. In order to improve performance, the magnitude of this force can be controlled as a function of punch travel. Maximum punch force (F_{\max}) can be estimated from the formula:

$$F_{\max} = \pi D_P Y \left[\left(\frac{D_o}{D_P} \right) - 0.7 \right]$$

The important variables in deep drawing are the properties of the sheet metal, the ratio of blank diameter (D_o), the punch diameter (D_p) and the clearance (c).



-
-
- **Problem-04**: What is the force required to punch a square hole 60 mm on each side in a 1-mm thick 5052-O aluminum sheet by using flat dies? UTS is 190 MPa for 5052-O aluminum.

Solution: For square or rectangular hole, $F=0.7 T L$ (UTS)

$$F=0.7(0.001)(.06 \times 4)(190 \times 10^6)=31.9 \text{ kN}$$

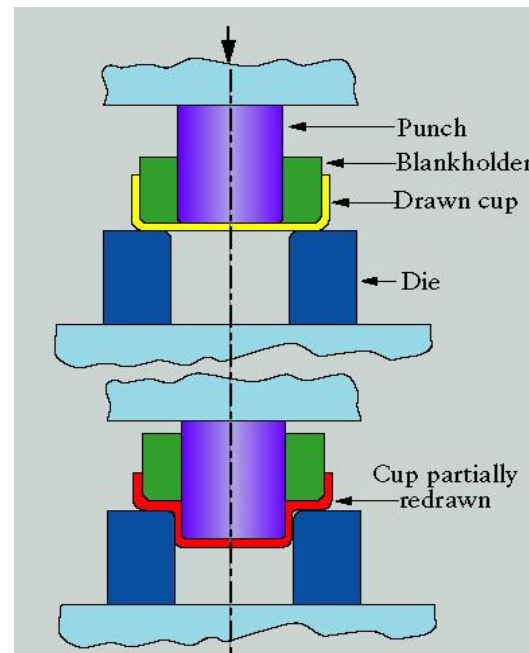
- **Problem-05**: Calculate the press force required in punching 0.5-mm-thick 5052-O aluminum foil in the shape of a square hole 30 mm on each side.

Solution: For square or rectangular hole, $F=0.7 T L$ (UTS)

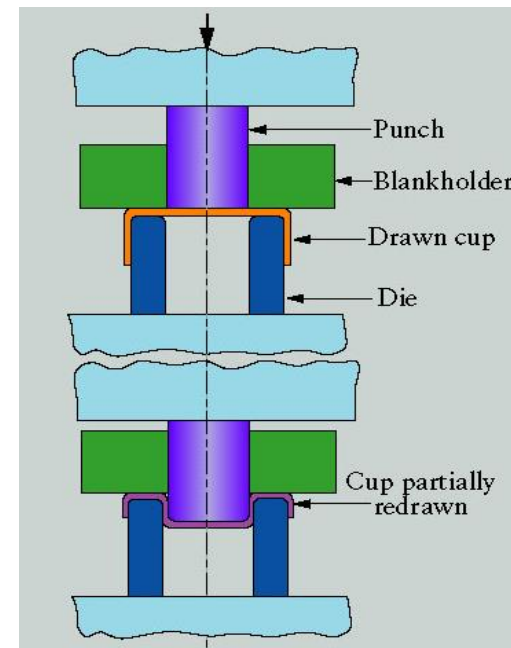
$$F=0.7(0.001)(.05 \times 4)(190)=7980 \text{ N}$$



- **Redrawing Operations**: Containers or shells that are too difficult to draw in one operation are generally redrawn. In reverse redrawing, shown in following Figure, the metal is subjected to bending in the direction opposite to its original bending configuration. This reversal in bending results in strain softening. This operation requires lower forces than direct redrawing and the material behaves in a more ductile manner.



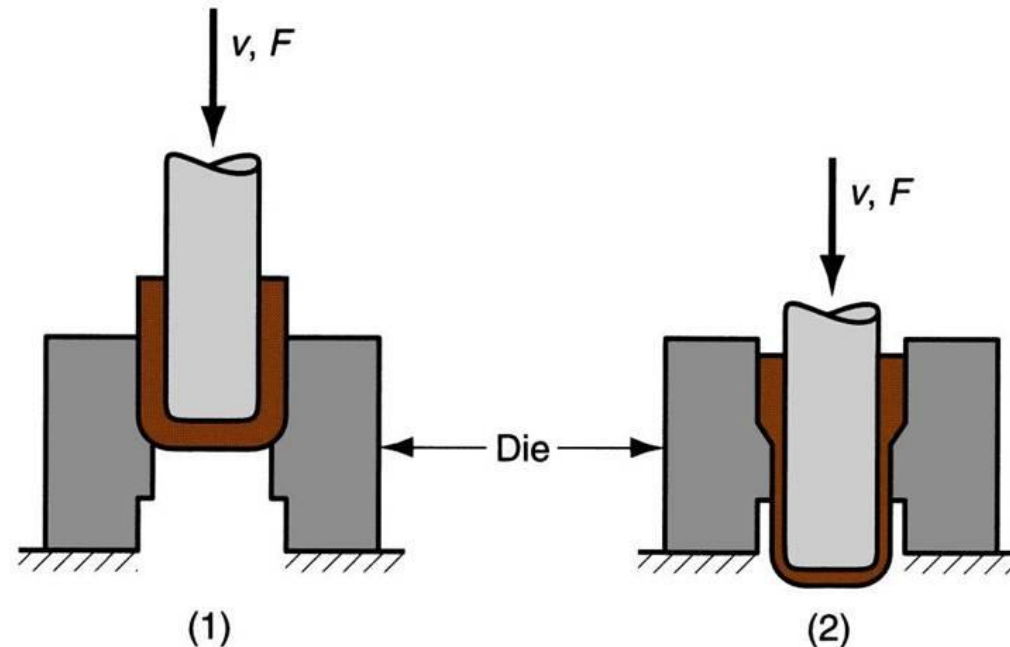
Conventional redrawing



Reverse redrawing.



- **Ironing Process**: If the thickness of the sheet as it enters the die cavity is more than the clearance between the punch and the die, the thickness will have to be reduced; this effect is known as ironing. Ironing produces a cup with constant wall thickness thus, the smaller the clearance, the greater is the amount of ironing.



Ironing to achieve more uniform wall thickness in a drawn cup: (1) start of process (2) during process. Note thinning and elongation of walls



Manufacturing an Aluminum Can

