2-1 COLD WORKING PROCESSES

2-1 Squeezing processes

Most of the cold working squeezing processes have identical hot working counterparts or are extension of them. The primary reasons for deforming cold rather than hot are to obtain better dimensional accuracy and surface finish.

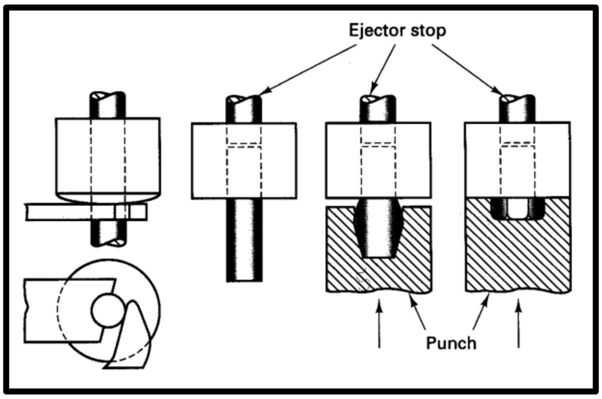
In many cases the equipment is basically the same, except that it must be more powerful.

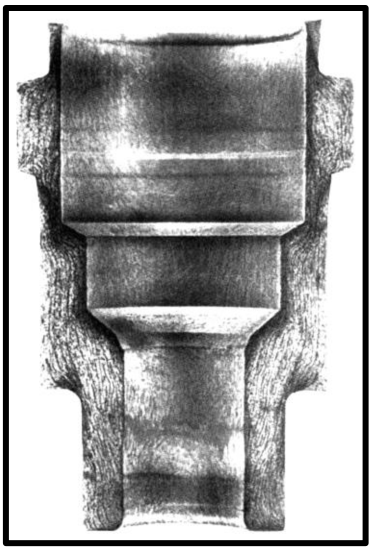
2-1-1 Cold Rolling

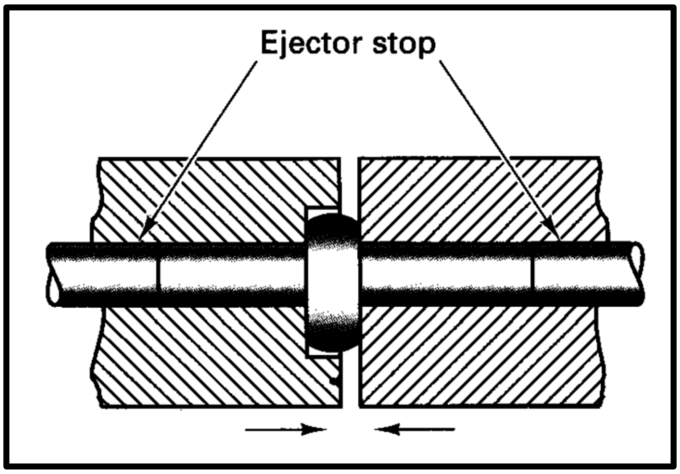
Sheets, strips, bars and rods are cold rolled to obtain products that have smooth surfaces and dimensions.

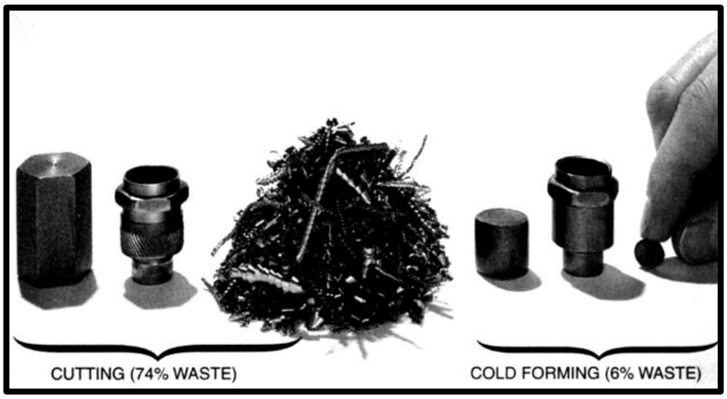
2-1-2 Cold Forging

The metal is squeezed into a die cavity that imparts The desired shape. It is known as cold heading if used for making enlarged sections on the ends of a piece of rod or wire, such as the heads on bolts, nails, rivets, and other fasteners. Upsetting is done in one or more strokes of the heading punches. Enlarged sections at locations other than the ends of rods can also be made.









Parts Made by Cold Forging

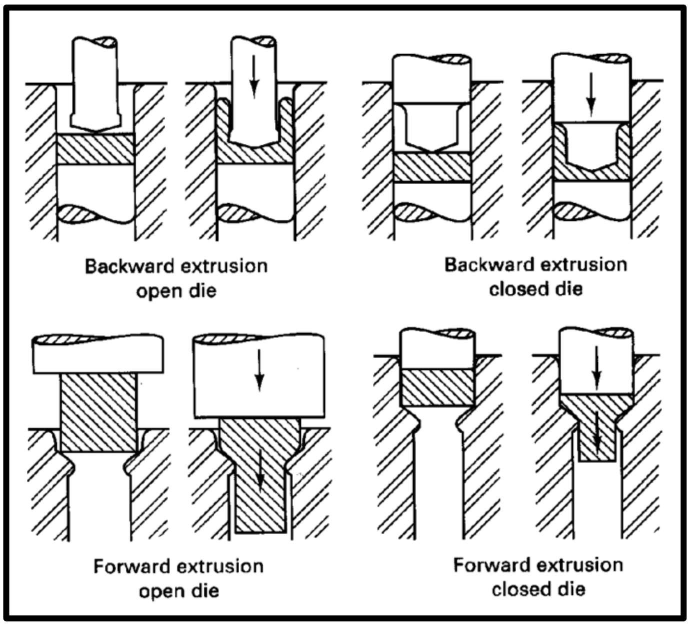
2-1-3 Extrusion

Products like collapsible tubes for toothpaste, medications, and so forth; small cans such as are used for shielding in electronics and electrical apparatus; and larger cans for food and beverages are made by using the process which is often called as impact extrusion. There are:

- **Forward Extrusion**
- **Backward Extrusion**

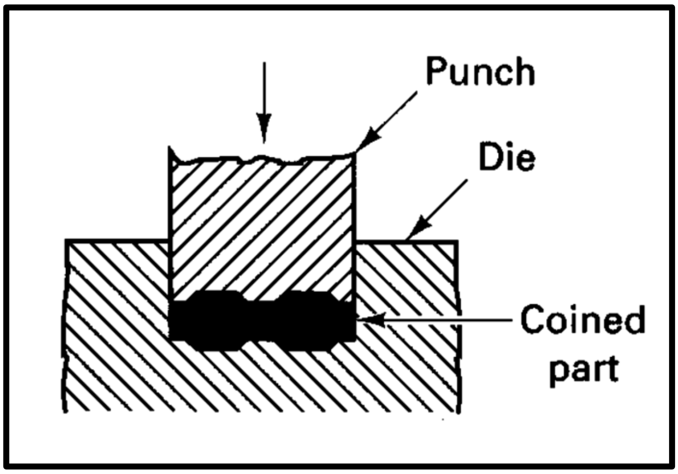
Both may be use open or closed dies

- **Cold - Impact Extrusion**



2-1-4 Coining

It is used to produce coins, medals and other products where exact size and fine detail are required. Metal is confined within a set of dies by means of the positive displacement of the punch, and very high pressure is required.



2-1-5 Surface Improvement (Peening)

Peening involves striking the surface by repeated blows by impelled shot (shot peening) or a round nose tool. The highly localized blows deform and tend

to stretch metal surface. Because the surface deformation is resisted by the metal underneath, the result is a surface layer under residual compressive stresses. This provides resistance against cracking under fatigue conditions.

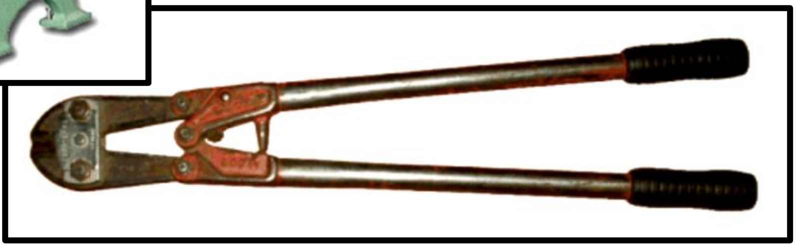
2-2 Sheet metal forming

The raw material for sheet metal manufacturing processes is the output of the rolling process. Typically, sheets of metal are sold as flat, rectangular sheets of standard size. If the sheets are thin and very long, they may be in the form of rolls. Therefore the first step in any sheet metal process is to cut the correct shape and sized 'blank' from larger sheet.

Sheet metal processing is an important process for the largest manufacturers of products such as home appliances (fridge, washer, dryer, vacuum cleaners etc.), electronics (DVD- and CD-players, stereos, radios, amplifiers etc.), toys and PC's. Most of these products have metal parts that are made by cutting and bending sheet metal.

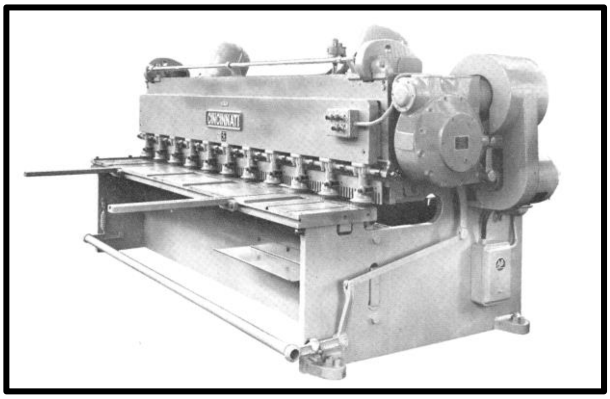
2-2-1 Shearing Processes

Shearing is the mechanical cutting of materials in sheet or plate form without the formation of chips or use of burning or melting. When the two cutting blades are straight, the process is called shearing.



Processes in which the shearing blades are in the form of the curved edges of punches and dies, are called by other names, such as blanking, piercing, notching, shaving, trimming, etc.

Squaring shears which are foot or power operated may be used for shearing sheets of metals along straight lines.

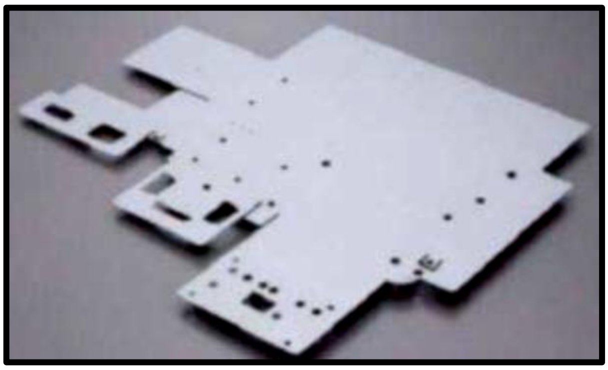


Shearing is similar to the process by which you cut a sheet of paper using scissors. However, the machinery used is a little different. The image below shows two typical machines used to cut sheet metal – the first is a shearing machine which has a long blade to make straight line cuts; it is used to cut long sheets of metal into smaller sheets. This operation is similar to that of a paper-cutting machine: the metal sheet is held on top of a hardened die, and the shearing blade cuts downward, usually driven by an electrical or hydraulic punch.

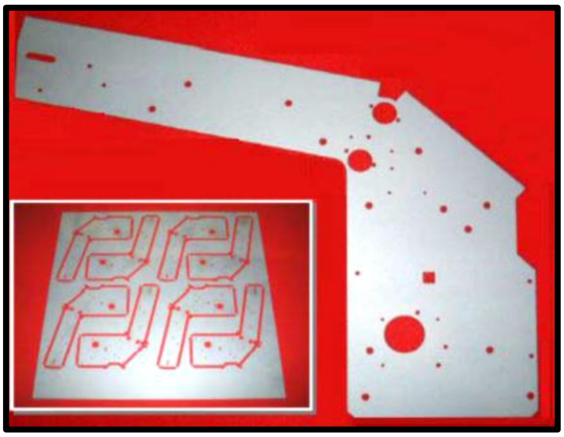
Figure 1. (a) Hydraulic sheet metal shearing machine (b) A CNC turret-type sheet metal punching machine

2-2-2 Punching Processes

The second in above image is a sheet metal punching machine – the cutting tool is a punch, a piece of hard tool steel which is punched down on the sheet to cut a hole. The punch in the image is a turret punch – the turret is a rotating tool holder that can hold tens of different shapes and sizes of punching dies. Typical shapes are rectangular and circular. By punching in a series of steps, a long slot can be cutout. A typical punching operation is similar to that of a paper punch that you may have used to cut holes to hold paper in a 3-ring binder. Main use: to cut large sheet into smaller sizes for making parts.



Typical punched part



Nesting of parts

Figure (2 a): Typical punched part. (b) A square sheet stock used to punch out 8 parts; enlarged view of part

Cutting metal by punching utilizes almost pure from of failure in shear; therefore the cutting forces and the energy required are functions of the shear strength of the material. The figure below shows a schematic of how the metal is sheared between the cutting edges of the punch and the die. Cutting tool is a round/rectangular punch that goes through a hole, or die of same shape.

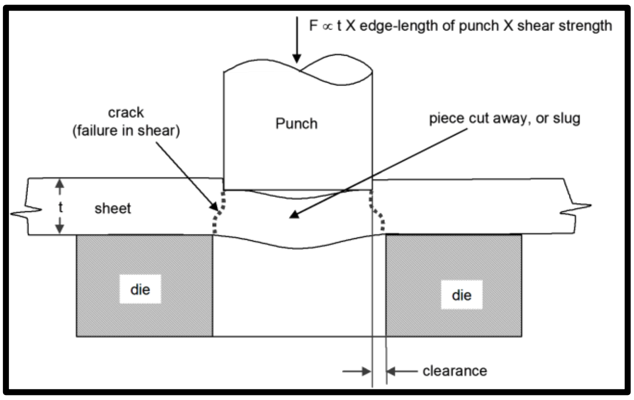
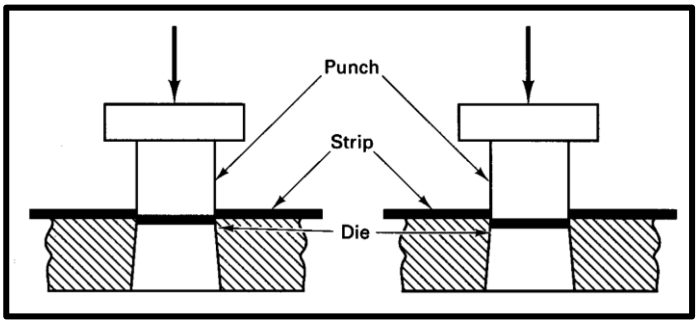


Figure (3):Schematic of the shearing process

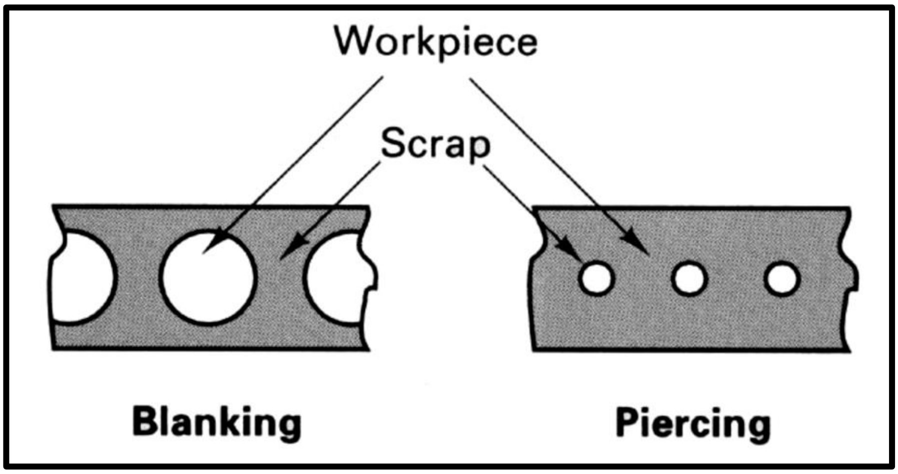
The clearance is typically between 2% and 10% of sheet thickness, t; if the clearance is larger, the part that is sheared away, called the slug, may just flow through the clearance without separating from the sheet.

- **Piercing and Blanking**

The shearing blades take the form of closed, curved lines on the edges of a punch and die. Piercing and blanking are usually done by some type of mechanical press.



In piercing, the piece punched out is the scrap and the remainder of the strip becomes the desired workpiece. In blanking, the piece punched out is the desired workpiece and undesirable features are left on the strip.



Piercing and Blanking Die Sets

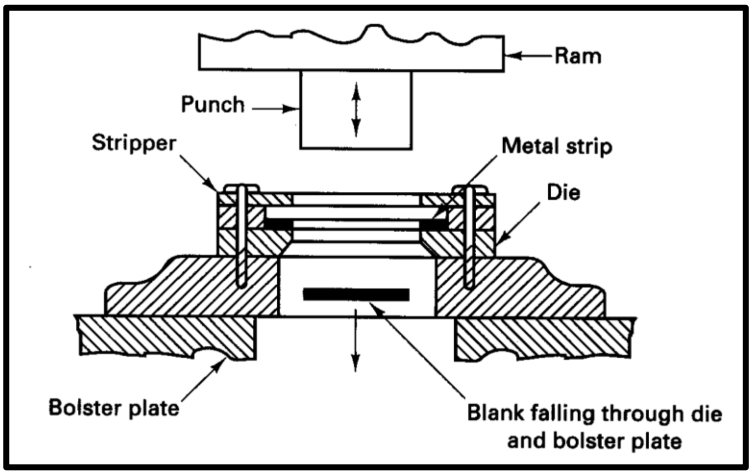
The basic components of piercing an blanking die sets are a punch, a die and a stripper plate. Stripper plate is used to prevent the climbing up of the stock with the punch. The punch and the die have sharp edges. Commonly, the clearance between the punch and the die is about 5 to 7 % of the stock (sheet

metal) thickness.

There are three types of die sets.

1. Simple Die Sets
2. Progressive Die Sets
3. Compound Die Sets

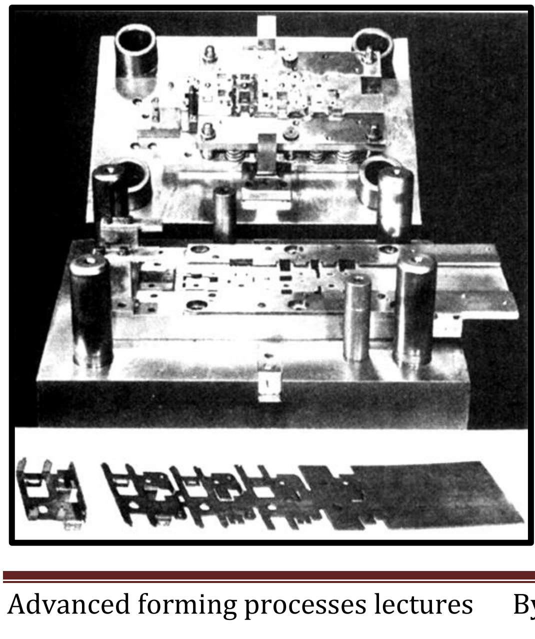
Simple die sets consist of a punch and a die. They can be used for a single process.

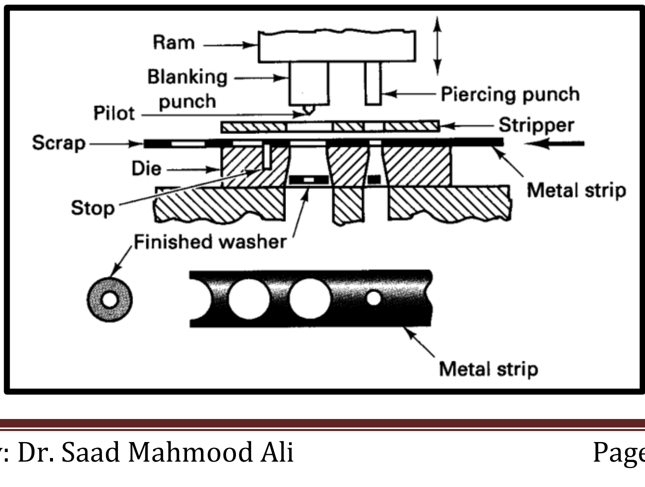


Progressive Die Sets

Consist of two or more sets of punches and dies mounted in tandem.

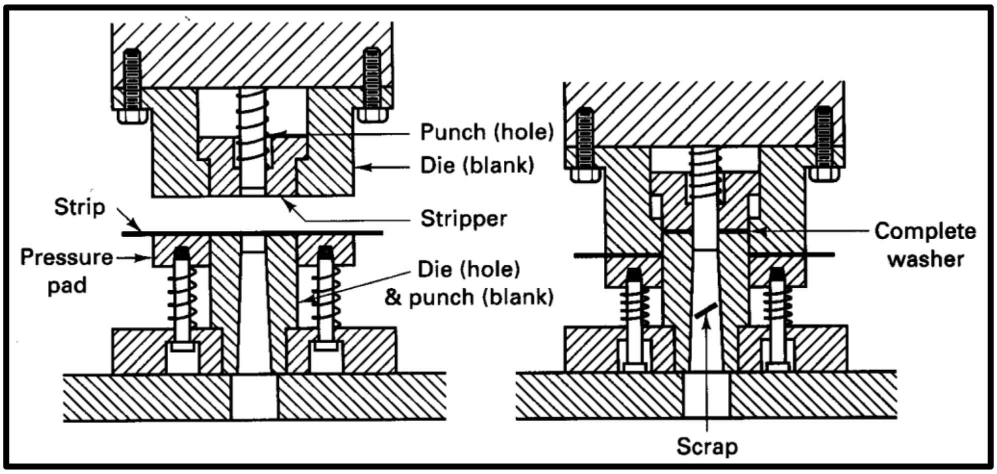
The strip stock is fed into the die and part is completed with successive strokes of the press



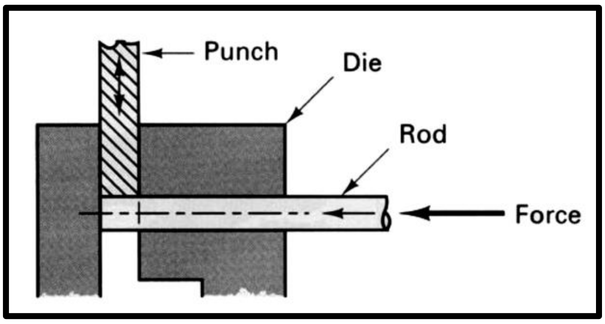


Compound Die Sets

Piercing and blanking, or other combinations occur simultaneously within a single stroke of the ram while the strip of stock remains in one position. Dies of this type are more accurate, but they usually are more expensive to construct and are more subject to breakage and locking.



- **Rod Shearing**



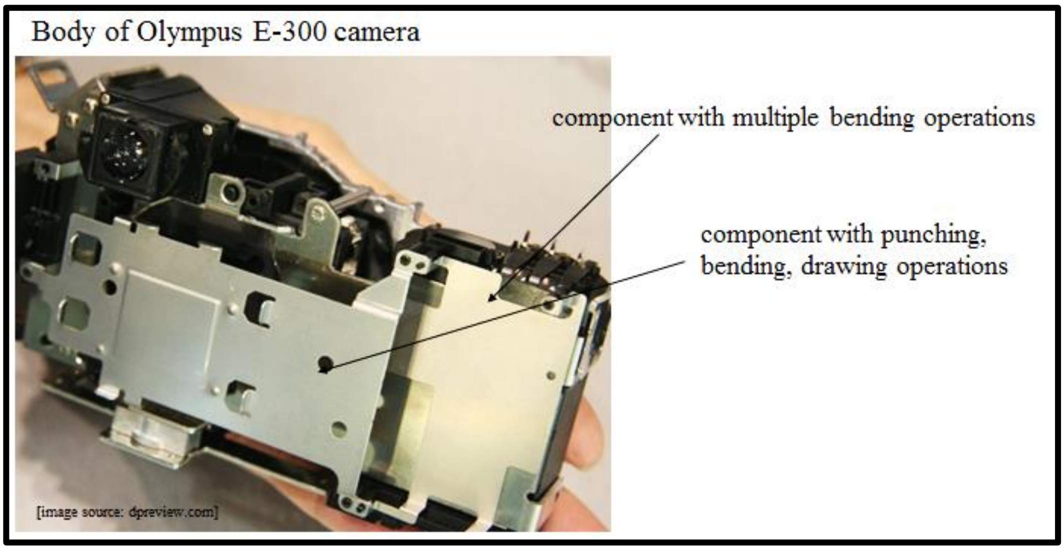
Other methods of cutting sheets

These include flame cutting, sawing, laser cutting, and water-jet machining, some of which we shall cover later.

2-2-3 Bending Processes

A large percentage of sheet metal parts are bent along some lines to get them into the desired shape for use (for example, think of the metal case for a computer). The sheet metal part before it is bent is called a blank. Here, we shall also look at a special case of bending of wires or rods – the reason this is

interesting is that one of the most important mechanical component: springs.



Bending induces plastic deformation metals about a linear axis with little or no change in the area, so the part retains its shape after the bending force is released. However, on studying the stress-strain curves for materials, you will notice that when a material is deformed into the plastic region and then released, some portion (the elastic part) of the strain is released. This phenomenon causes an action called spring-back in the part that we want to bend. Thus, the bending dies must account for the spring-back.

When two or more bends are made simultaneously with the use of a die, the process is sometimes called forming. If the axes about which deformation occurs are not linear or not independent, it is known as drawing, it is not bending. In bending, two axes involved in forming may be at an angle to each other, but each axis must be linear and independent of the other.

Figure (4) shows a simple bend on a rectangular blank. The top profile of the blank undergoes extension – a thin element along the top surface will be longer after the bending than the initial length; likewise, the bottom portion experiences compression. Thus, as we travel from the bottom to the top, there is some layer in the middle which retains its original length – this forms the neutral axis. The location of the neutral axis, and therefore its length, determines the length of the blank we must begin with, in order to get the final part with the correct

geometry.

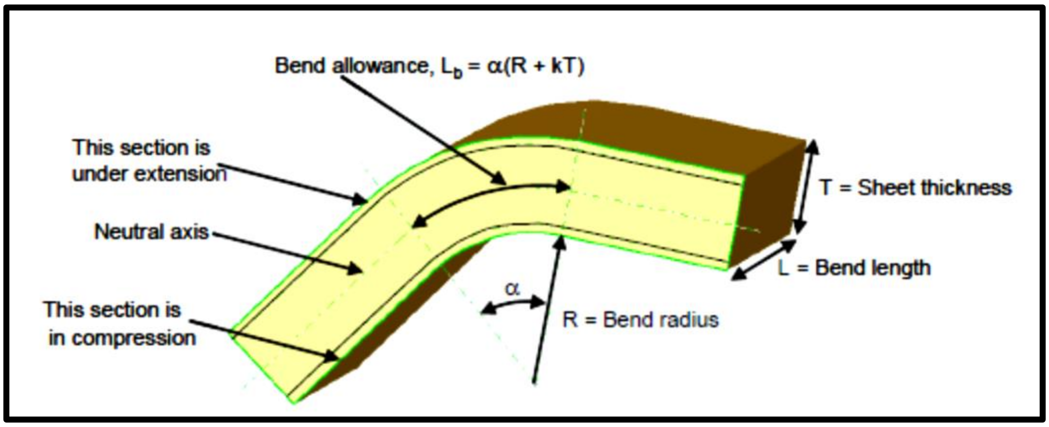


Figure 4. Schematic of bending mechanics,
Ideal case: $k = 0.5$
Real cases: $k = 0.33$ ($R < 2T$) ~~ $k = 0.5$ ($R > 2T$)

Some issues in bending

(i) Minimum bending radius and cracking

When the bending radius is too small, the strain level on the outer layers is too high, and usually the top layer will undergo plastic deformation or cracking. If you keep bending, there will also be failure in buckling at the bottom. The engineering strain during bending is approximated as:

Engineering strain in bending = e = 1/(1 + 2R/T)

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

Thus, as R/T decreases, stress increases and cracking begins. The minimum bend radius is the radius at which cracking begins; it is expressed as a multiple of T; i.e. a minimum bend radius = 3T for a sheet of 1mm means that the bend radius should be larger than 3mm to avoid cracking.

(ii) Anisotropic properties

Since most sheets used as blanks are formed by rolling, they have anisotropic properties (different yield strength along different directions). Thus, the orientation in which you cut the blank from the raw sheet may depend on the bending operations you will perform. Figure 5 demonstrates this.

Bending: cracking, anisotropic effects, Poisson effect

Bending ➔ plastic deformation

Engineering strain in bending = e = 1/(1 + 2R/T)

Bending ➔ disallow failure (cracking) ➔ limits on corner radius: bend radius ≥ 3T

In the figure below, you also see the Poisson effect. When a material is subjected to tensile stress, it elongates; however, due to the law of conservation of mass, the total volume remains the same, and therefore its cross section area decreases. This effect, called the Poisson effect, can be seen in the two ends of the bent metal piece in the figure.

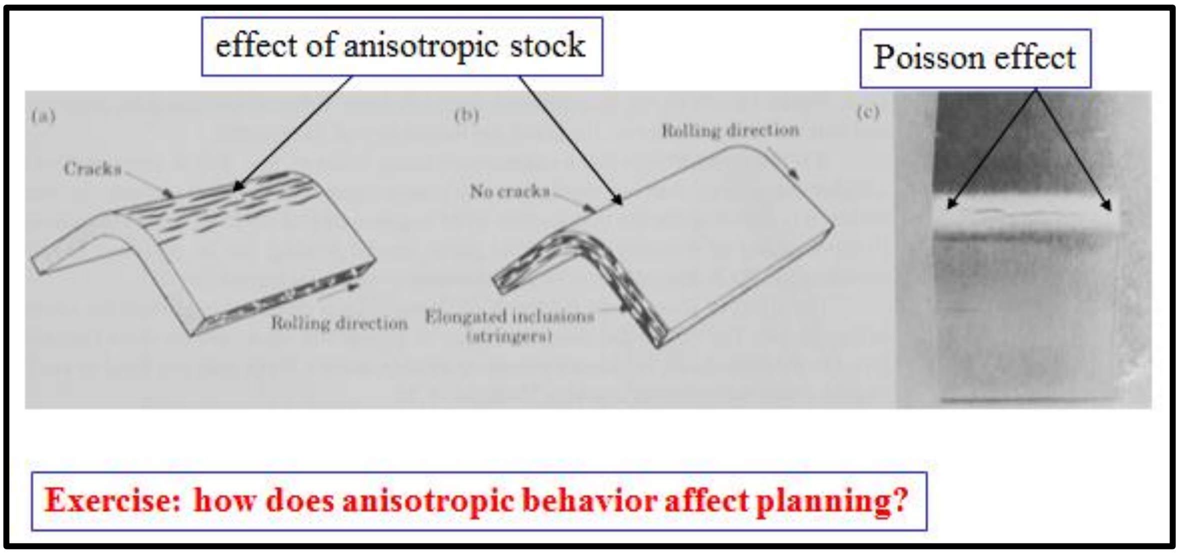


Figure 5. Bending outcome is dependent on anisotropic structure [source: Kalpakjiam & Schmid]

(iii) Springback

Recall from your stress-strain curves that when the stress is released, the all materials experience some elastic recovery. An approximate formula for springback is given terms of the relation between the initial bending radius, R_i, and the final bending radius (after springnback), R_f:

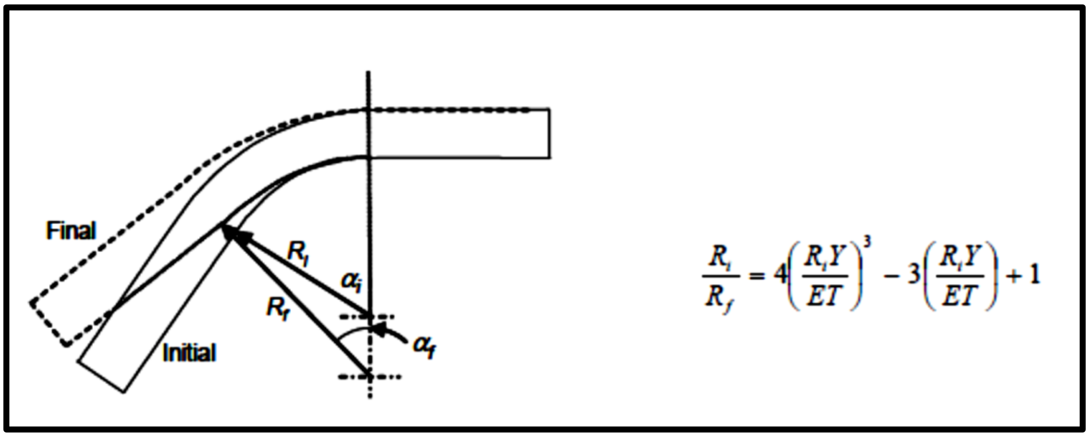
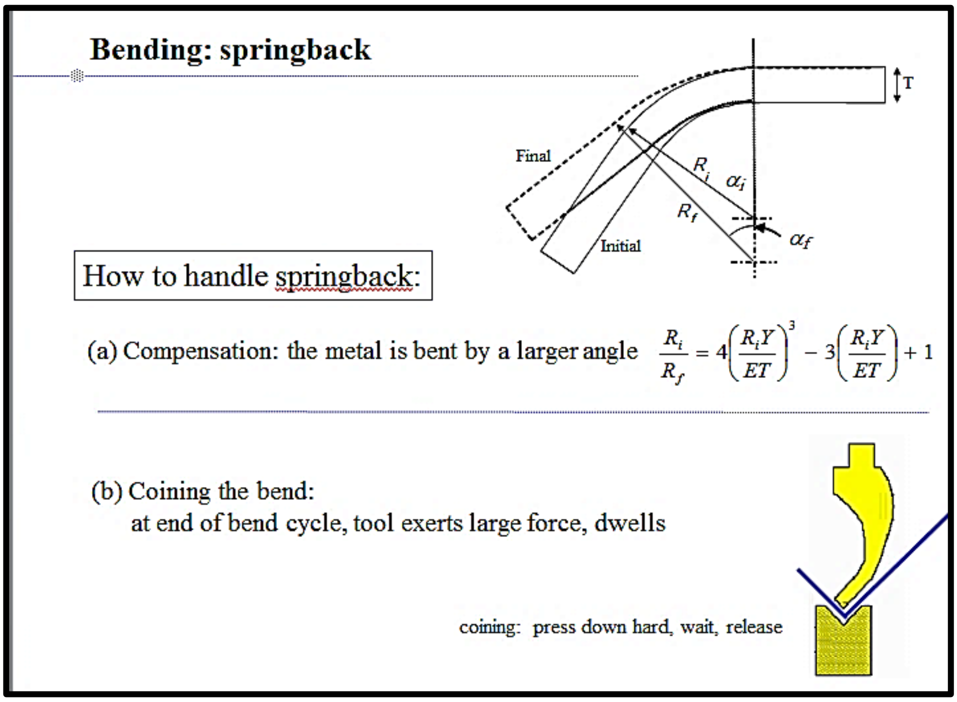


Figure (6): Springback in bending

There are several methods to counter the effect of springback:

(a) Compensation method: here, the metal is bent by a larger angle, such that it springs back to the desired value; namely, the desired angle of bending is set to R_f, and then, using the above formula, R_i is computed; then the blank is bent to R_i. In most companies, some trial and error is required before the exact bending angle that gives the desired result is obtained.

(b) Coining the bend: In this method, the bending is achieved by pushing with a punch, and letting the metal bend into a die; at the end of the cycle, a relatively large squeezing force is exerted, which creates a permanent bend angle.



Typical bending operations and shapes

Figure 7(a) below shows some typical bending operations. Another bending operation is flanging – the flange is usually a 90° bend of uniform width running along a border of a sheet. Take a look at the box covering your computer to see a practical example. Some typical flanges are shown in figure 7(b).

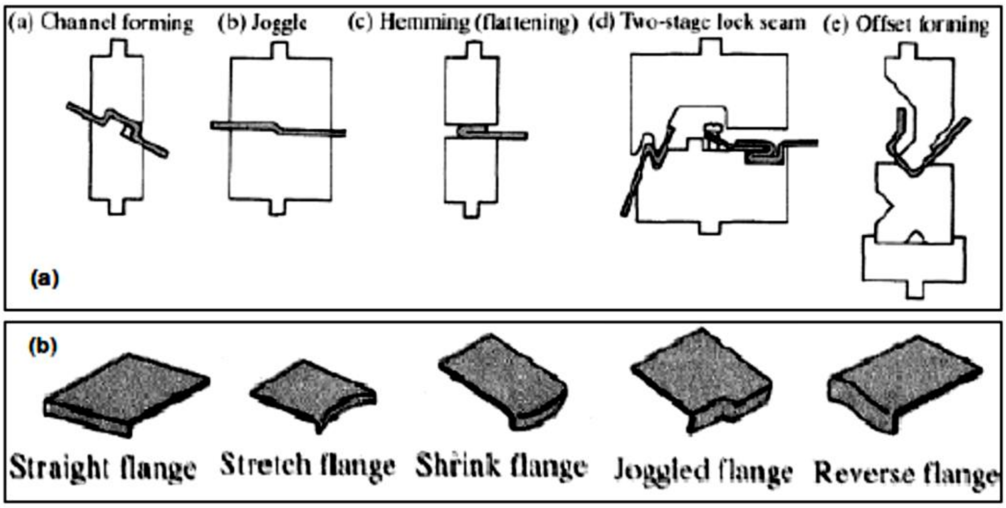
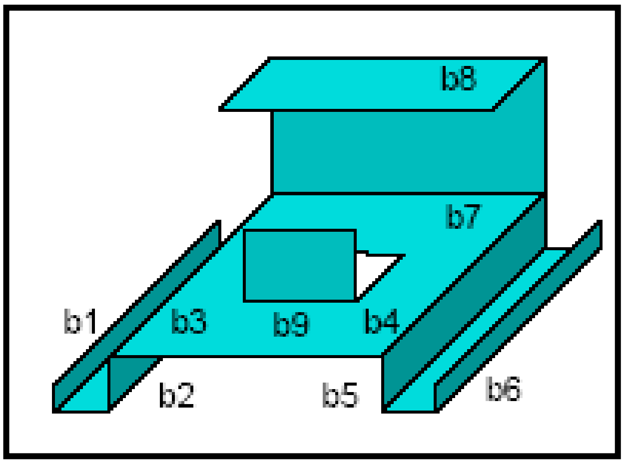


Figure 7.(a) Typical Sheet-metal bending operations (b) Typical flanges [source: Kalpakjiam & Schmid]

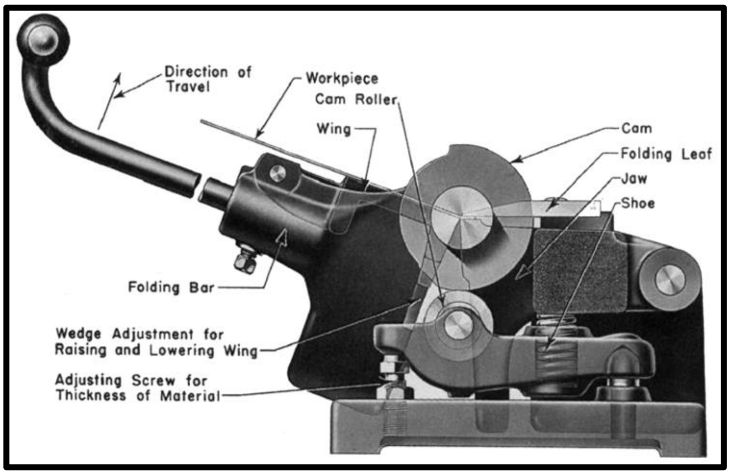
Planning problem: what is the sequence in which we do the bending operations?



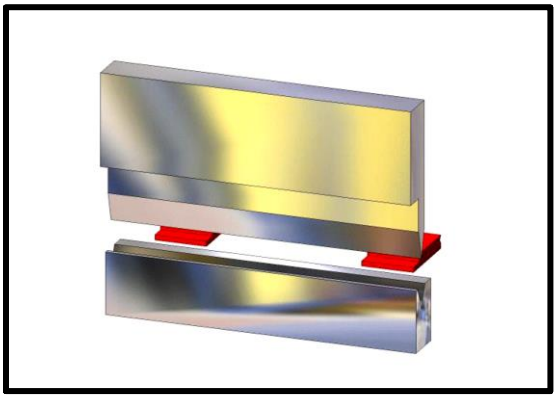
- **Angle Bending**

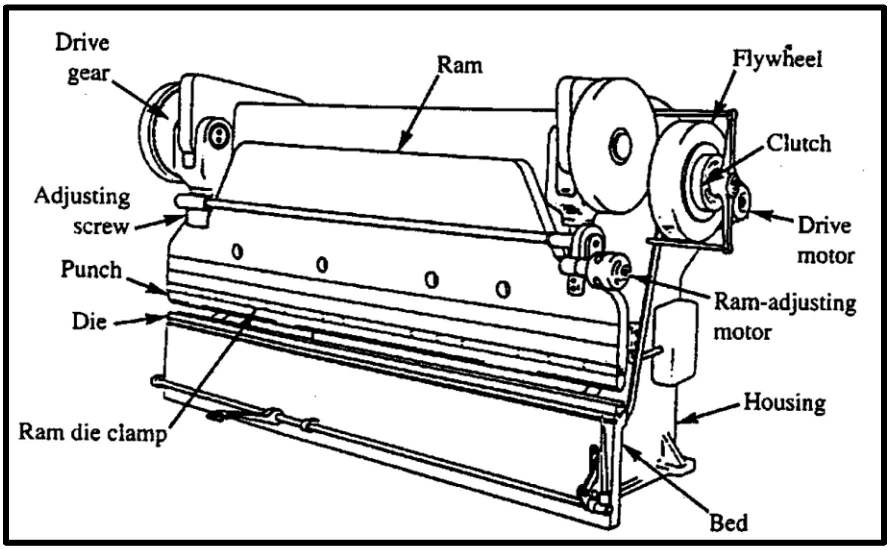
Angle bends up to 150° in sheet metal under about 1.5 mm in thickness

may be made in a bar folder. These machines are manually operated.

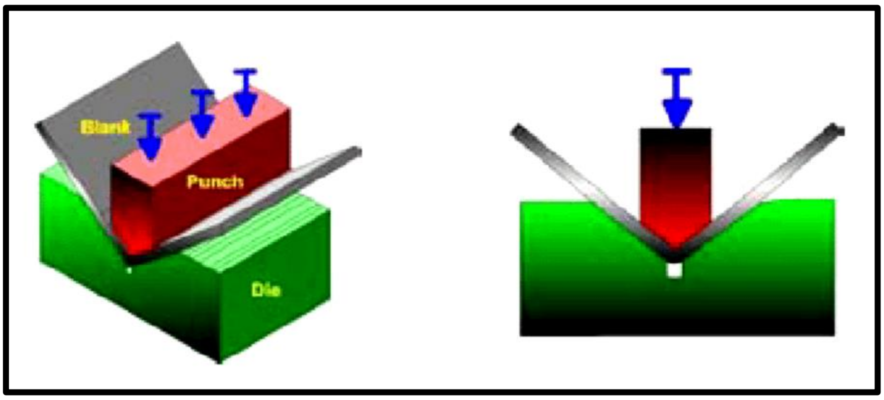


Bends in heavier sheet metal and more complex bends in thinner sheets are made in a press brake which is mechanically or hydraulically driven. The metal is bent between interchangeable dies that are attached to the bed and the ram

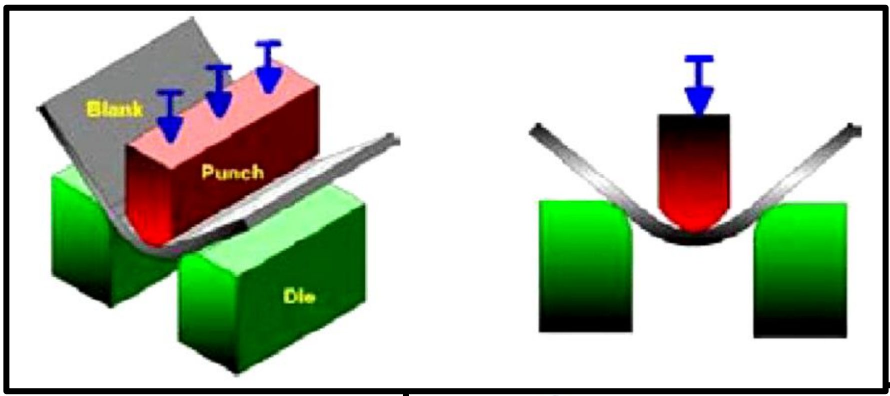


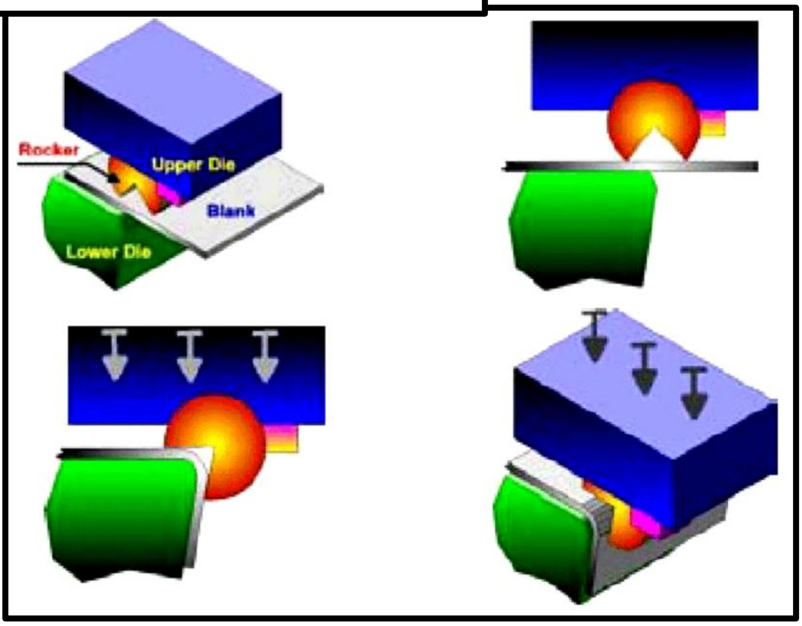


Bar Folder



V Bending

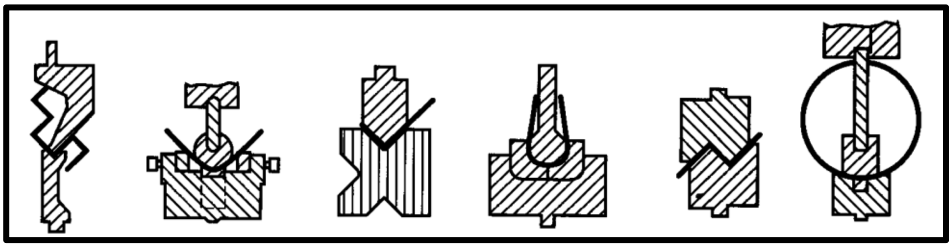


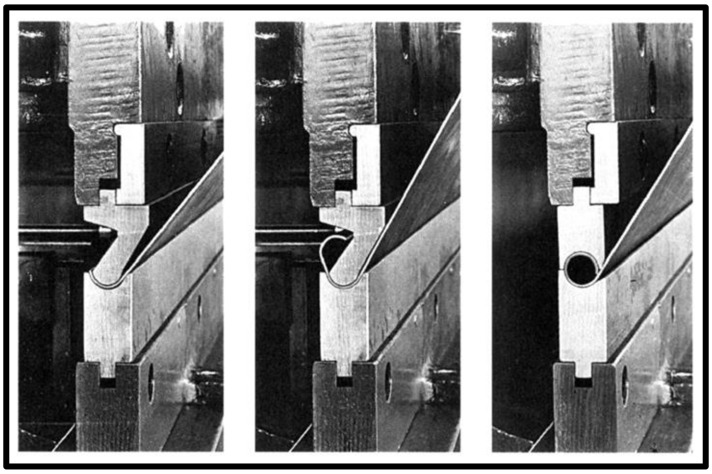


Air Bending

Rotary Bending

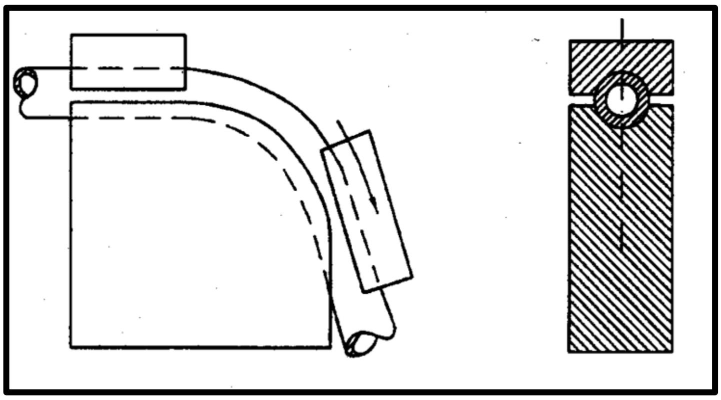
Press Brake Dies





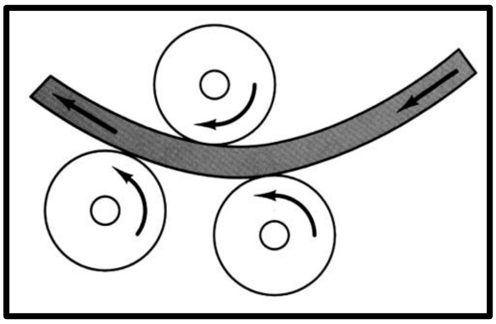
Tube Bending

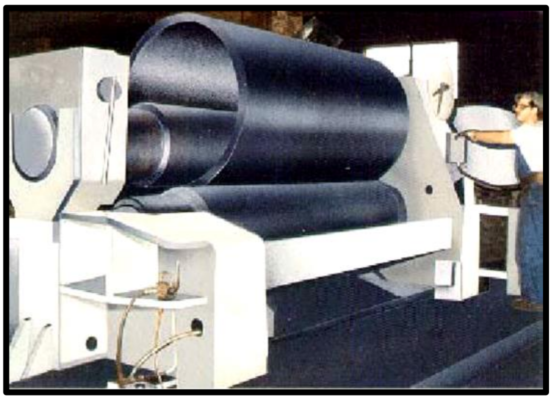
Mostly done by using machines.



Roll Bending

Plates, heavy sheets and rolled shapes can be bent to a desired curvature on forming rolls. These usually have three rolls in the form of a pyramid, with the two lower rolls being driven and the upper roll adjustable to control the degree of curvature.





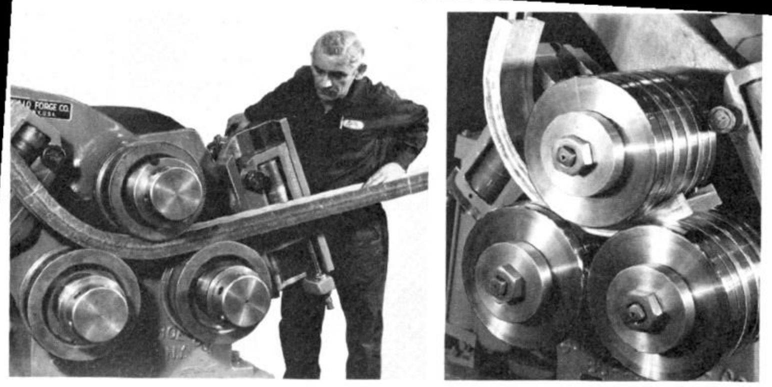
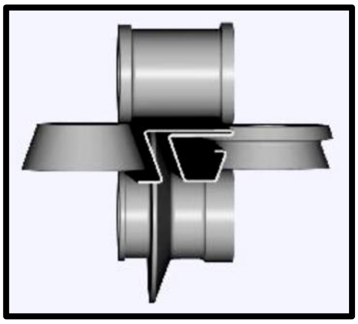
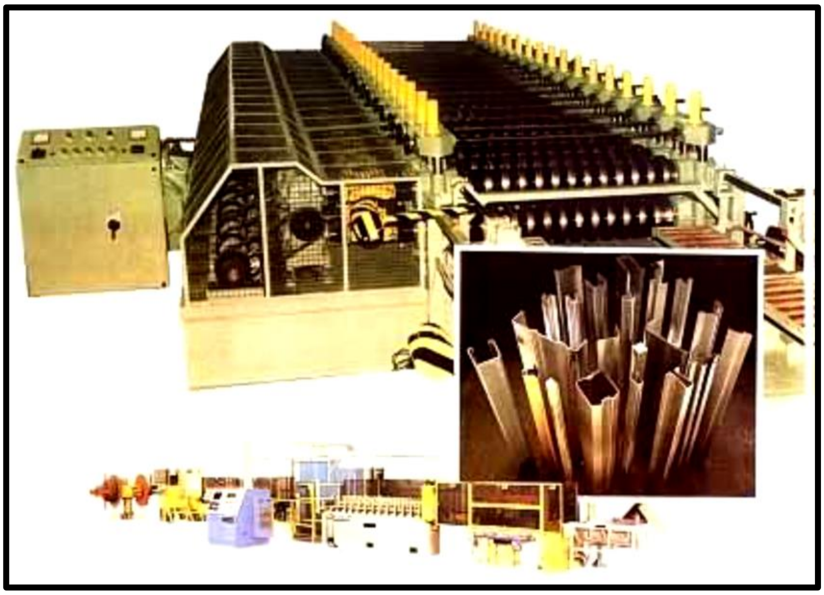


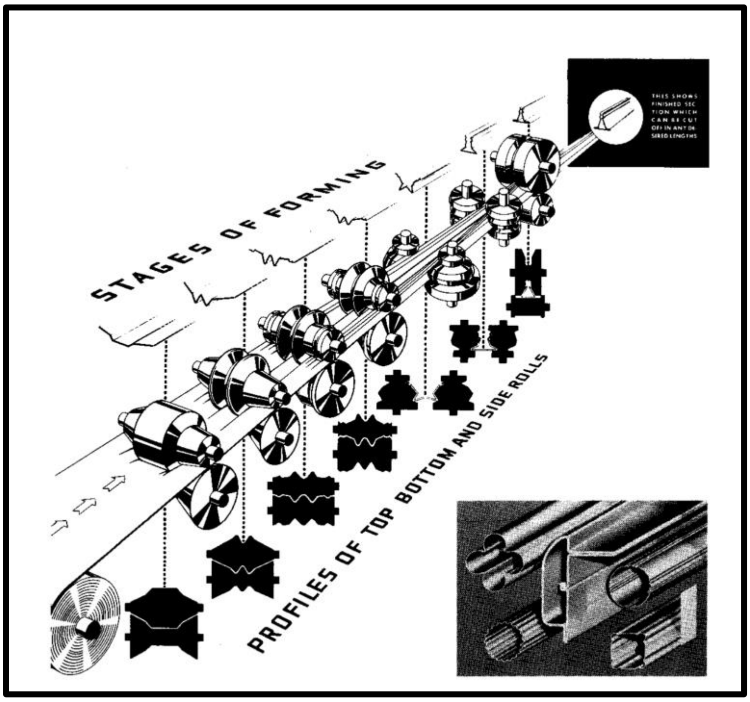
FIGURE 15-28 Cold-roll bending of structural shapes. (*Courtesy Buffalo Forge Company.*)

- **Cold Roll-Forming**

Involves the progressive bending of metal strip as it passes through a series of forming rolls. By changing the rolls, a single machine can be adapted to the production of many different shapes.







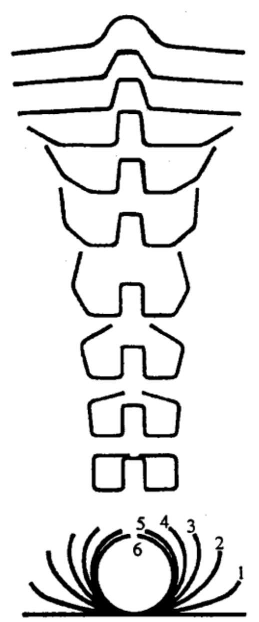
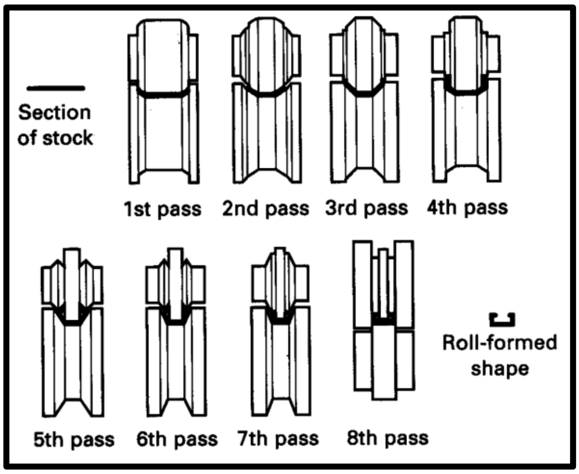


Figure 4.69 Examples of roll forming passes for a) the double channel, b) round tube.



- **Bending Processes – Joining**

Seaming is used to join ends of sheet metal to form containers such as cans, drums etc. by a series of small rollers on seaming machines.

FIGURE 15-32 Various types of seams used on sheet metal.