Department of Mechanical and Production Engineering
Ahsanullah University of Science and Technology (AUST)

IPE 3210: Metal Forming and Sheet metal Working Sessional
Credit Hour: 1.5

General Guidelines:

1. Students shall not be allowed to perform any experiment without apron and shoes.
2. Students must be prepared for the experiment prior to the class.
3. Report of an experiment must be submitted in the next class.
4. The report should include the following:
   - Objectives
   - Apparatus (with specifications if any)
   - Machine used (with specifications)
   - Schematic diagram
   - Data sheet
   - Sample calculation
   - Graphs (if any)
   - Discussion (in passive form)
   - Assignment
   - References

5. Viva & quiz will be taken on the experiments at the end of the semester.
6. Marks distribution:

Total Marks			
Job/project	Report	Attendance and Viva	Quiz
05	30	30	35

Sheet metal:

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

The thickness of sheet metal is commonly specified by a traditional, non-linear measure known as its gauge. The larger the gauge number, the thinner the metal. Commonly used steel sheet metal ranges from 30 gauge to about 8 gauge. Gauge differs between ferrous (iron based) metals and a non-ferrous metal such as aluminum or copper; copper thickness, for example is measured in ounces (and represents the thickness of 1 ounce of copper rolled out to an area of 1 square foot).There are many different metals that can be made into sheet metal, such as aluminum, brass, copper, steel, tin, nickel and titanium. For decorative uses, important sheet metals include silver, gold, and platinum (platinum sheet metal is also utilized as a catalyst).

Sheet metal is available in flat pieces or coiled strips. The coils are formed by running a continuous sheet of metal through a roll slitter.During the rolling process the rollers bow slightly, which results in the sheets being thinner on the edges. So, a tolerance is specified by the manufacturer of sheet metal i.e. deviation from the nominal size.
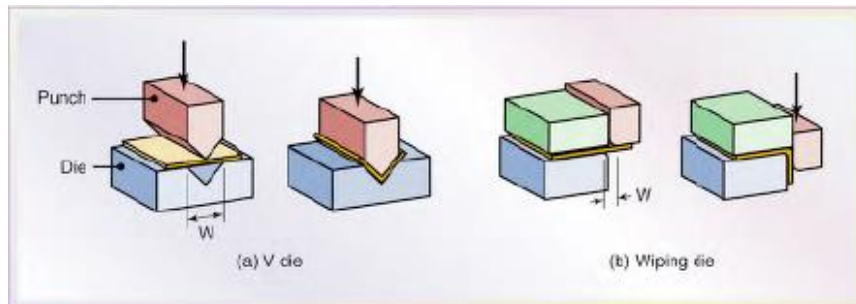
Sheet metal forming processes:

Bending:

Figure 16.21 Common die-bending operations showing the die-opening dimension, *W*, used in calculating bending forces.

Bending is a manufacturing process that produces a V-shape, U-shape, or channel shape along a straight axis in ductile materials, most commonly sheet metal. Commonly used equipment includes box and pan brakes, *brake presses*, and other specialized machine presses. Typical products that are made like this are boxes such as electrical enclosures and rectangular ductwork. Bending forces can be estimated by assuming that the process is one of simple bending of a rectangular beam. The bending force in that case is a function of the strength of the material, the length L of the bend, the thickness T of the sheet, and the size W of the die opening (fig 16.21). Excluding friction, the maximum bending force P is,

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

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.

For a V-die, this equation can often be approximated as,

$$P = \frac{(UTS)LT^2}{W}$$

Where UTS is the ultimate tensile strength of the material. This equation applies well to situations in which the punch radius and the sheet thickness are small compared to the die opening W.

Press brake forming:
In press brake forming, a work piece is positioned over the *die block* and the die block presses the sheet to form a shape. Usually bending has to overcome both tensile stresses and compressive stresses. When bending is done, the *residual stresses* cause the material to *spring back* towards its original position, so the sheet must be over-bent to achieve the proper bend angle. The amountof spring back is dependent on the material, and the type of forming. When sheet metal is bent, it stretches in length. The *bend deduction* is the amount the sheet metal will stretch when bent as measured from the outside edges of the bend. The *bend radius* refers to the inside radius. The formed bend radius is dependent upon the dies used, the material properties, and the material thickness.

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

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

press. Because it requires less bend force, air bending tends to use smaller tools than other methods. Some of the newer bottom tools are adjustable, so, by using a single set of top and bottom tools and varying press-stroke depth, different profiles and products can be produced. Different materials and thicknesses can be bent in varying bend angles, adding the advantage of flexibility to air bending. There are also fewer tool changes, thus, higher productivity.A disadvantage of air bending is that, because the sheet does not stay in full contact with the dies, it is not as precise as some other methods, and stroke depth must be kept very accurate. Variations in the thickness of the material and wear on the tools can result in defects in parts produced.

Air bending's angle accuracy is approximately ±0.5 deg. Angle accuracy is ensured by applying a value to the width of the V opening, ranging from 6 T (six times material thickness) for sheets to 3 mm thick to 12 T for sheets more than 10 mm thick. *Springback* depends on material properties, influencing the resulting bend angle.Depending on material properties, the sheet may be overbended to compensate for *springback.*

Air bending does not require the bottom tool to have the same radius as the punch. Bend radius is determined by material elasticity rather than tool shape.

The flexibility and relatively low tonnage required by air bending are helping to make it a popular choice. Quality problems associated with this method are countered by angle-measuring systems, clamps and crowning systems adjustable along the x and y axes, and wear-resistant tools.

The K-Factor approximations given below are more likely to be accurate for air bending than the other types of bending due to the lower forces involved in the forming process.
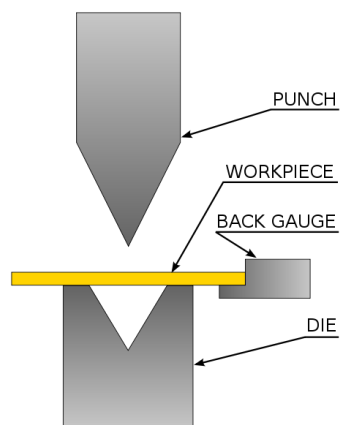

Fig: A schematic of air bending with a *backgauge*.

Coining:
In coining, the top tool forces the material into the bottom die with 5 to 30 times the force of air bending, causing permanent deformation through the sheet. There is little, if any, springback. Coining can produce an inside radius is as low as 0.4 T, with a 5 T width of the V opening. While coining can attain high precision, higher costs mean that it is not often used in metal working.

Three point bending:

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

Roll bending:
A Roll bender is a mechanical jig having three rollers used to form a metal bar into a circular arc. The rollers freely rotate about three parallel axes, which are arranged with uniform horizontal spacing. Two outer rollers, usually immobile, cradle the bottom of the material while the inner roller, whose position is adjustable, presses on the topside of the material. Generally the rolling machine consists of three rollers, one driver and other two driven rollers.
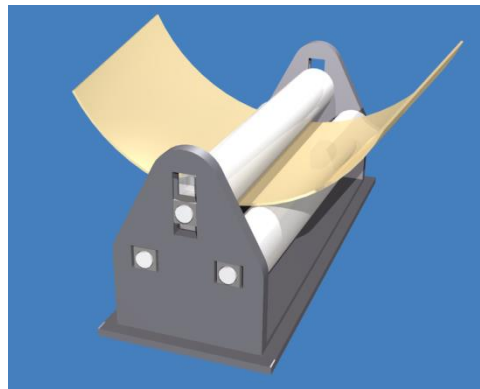
Fig: Roll bending

Calculations:

Legends-

BA=Bending allowance
BD=Bend allowance
R=Inside bend radius
K=K-factor=t/T
T=Material thickness
t=Distance from inside face to the neutral line
A=Bend angle in *degrees*(the angle through which material is bent)

The neutral line (also called the neutral axis) is an imaginary line that can be drawn through the cross-section of the workpiece that represents the lack of any internal forces. Its location in the material is a function of the forces used to form the part and the material yield and tensile strengths. The position of the *neutral axis* depends on the radius and angle of bend. For *ideal case* the neutral axis is the center of the sheet thickness and k=0.5. In the bend region, the material between the neutral line and the inside radius will be under compression during the bend. The material between the neutral line and the

outside radius will be under tension during the bend.Because of the Poisson's ratio, the width of the part (bend length, L) in the outer region is smaller and in the inner region it is larger. (Figure 16.16)
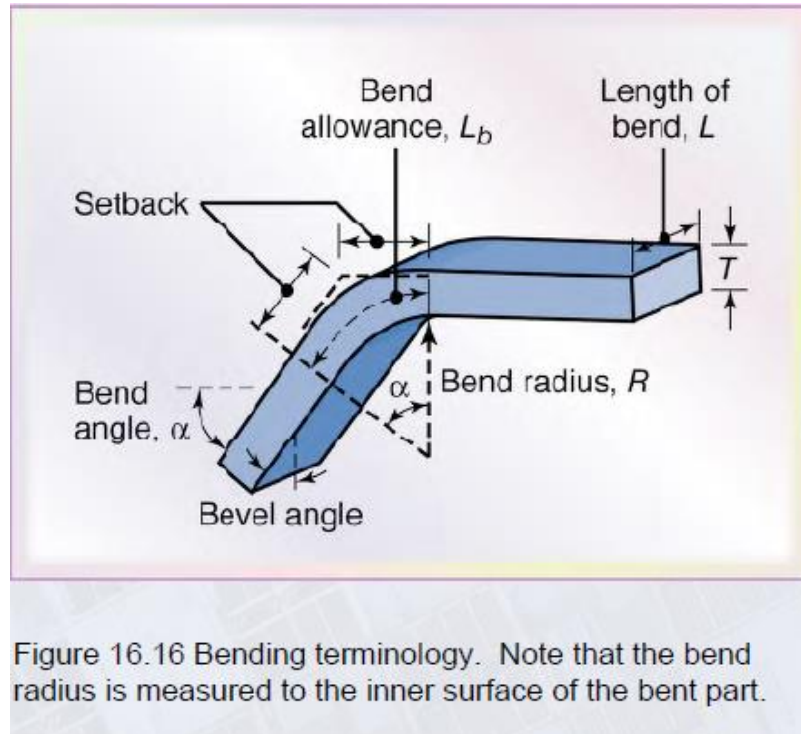

Figure 16.16 Bending terminology. Note that the bend radius is measured to the inner surface of the bent part.

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

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

The BA can be calculated using the following formula:
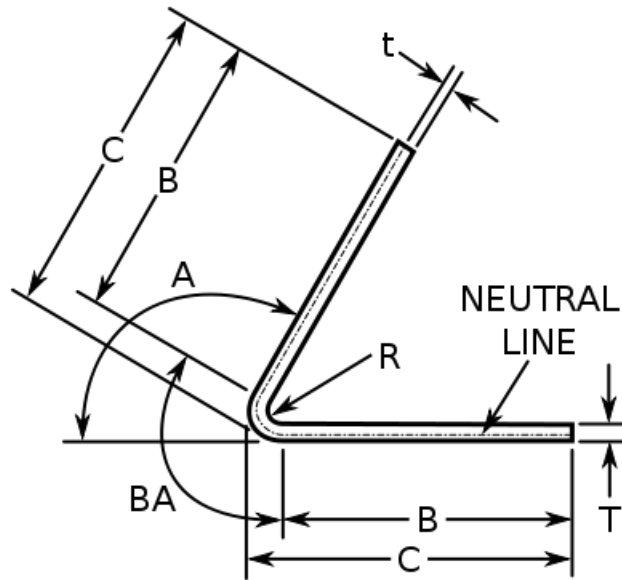
$$BA = A\left(\frac{\pi}{180}\right)(R + K \times T)$$

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

Example:

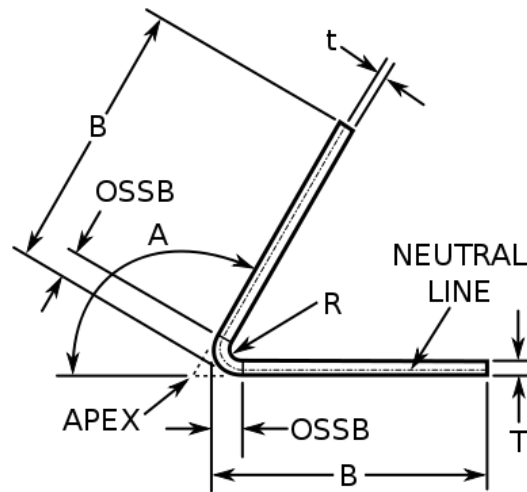| Angle	90^0
pi	3.1416
Radius	1.5
K-factor	0.33
Thickness	6
Bend allowance	5.46708

Bend deduction:

Fig: Diagram showing standard dimensioning scheme when using Bend Deduction formulas.

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

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

The above formula works only for right angles. For bend angles 90 degrees or greater the following formula works, where A is the angle in radians (=degrees*π/180).

$$BD = R\,(A-2) + T\,(kA-2)$$

$$BD = 2 \times OSSB - BA$$

K-factor:
K-factor is a ratio of location of the neutral line to the material thickness as defined by t/T where t = location of the neutral line and T = material thickness. The K-Factor formulation does not take the forming stresses into account but is simply a geometric calculation of the location of the neutral line after the forces are applied and is thus the roll-up of all the unknown (error) factors for a given setup. The K-factor depends on many factors including the material, the type of bending operation (coining, bottoming, air-bending, etc.) the tools, etc. and is typically between 0.33(for R<2T) to 0.5(for R>2T).For *ideal case* k=0.5. The following equation relates the K-factor to the bend allowance:

$$K = \frac{-R + \frac{BA}{\pi A/180}}{T}$$

The following table is a "Rule of Thumb". Actual results may vary remarkably:

Generic K-Factors	Aluminum		Steel
Radius	Soft Materials	Medium Materials	Hard Materials
Air Bending			
0 to Thickness	0.33	0.38	0.40
Thickness to 3 x Thickness	0.40	0.43	0.45
Greater than 3 x Thickness	0.50	0.50	0.50
Bottoming			
0 to Thickness	0.42	0.44	0.46
Thickness to 3 x Thickness	0.46	0.47	0.48
Greater than 3 x Thickness	0.50	0.50	0.50
Coining			
0 to Thickness	0.38	0.41	0.44
Thickness to 3 x Thickness	0.44	0.46	0.47
Greater than 3 x Thickness	0.50	0.50	0.50

The following formula can be used in place of the table as a good approximation of the K-Factor for Air Bending:

$$K=\log(\min(100,\max(20 \times R,T) / T)) / \log(100) / 2$$

__Flat pattern length (L_f):__ Total length of the flat sheet required can be computed by any of the following two formulas:

$$L_f=B+B+BA$$
Or
$$L_f=C+C-BD$$

Where the alphabets represent the conventions in the above figures.
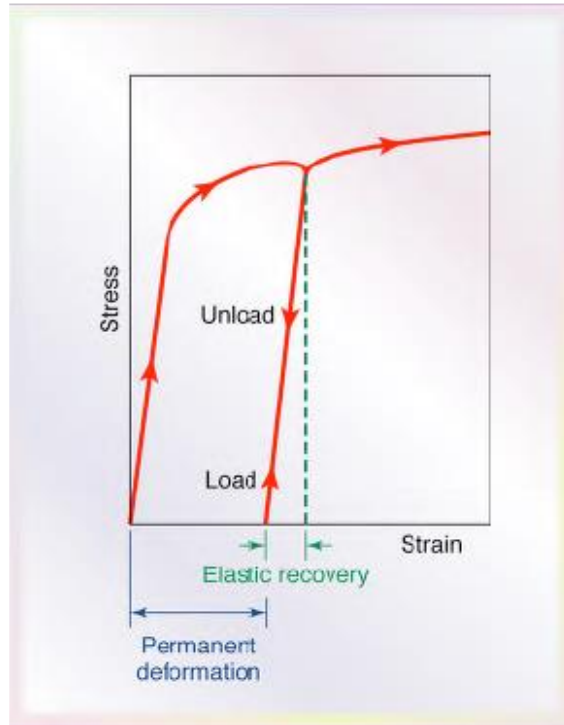
Springback:

Fig 2.3: Schematic illustration of the loading and unloading of tensile-test specimen. Note that during unloading the curve follows a path parallel to the original elastic slope

Because all materials have a finite modulus of elasticity, plastic deformation is followed, when the load is removed, by some elastic recovery (See figure 2.3). In bending, this recovery is called springback; you can easily observe it by bending and then releasing a piece of sheet metal or wire. As noted in the figure the final bend angle after springback smaller and the final bend radius is larger than before bending. Springback occurs not only in flat sheets and plate, but also in rod, wire and bar with any cross section.
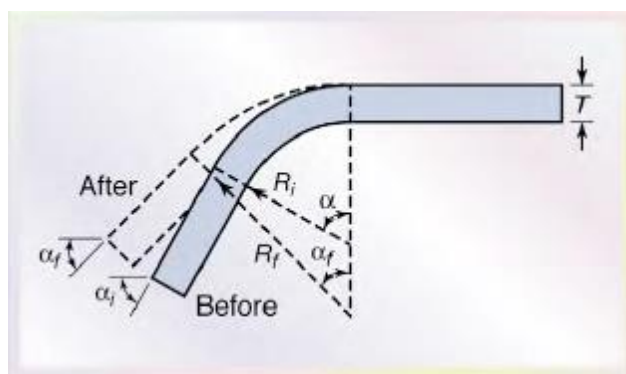

Fig 16.19: Sringback in bending

Springback can be calculated approximately in terms of the radii R_i and R_f (See figure 16.19) as,

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

Note from this formula that springback increases (a) as the R/T ratio and the yield stress Y of the material increase and (b) as the elastic modulus E decreases. In V-die bending it is possible for the material to exhibit *negative,* as well as positive springback. This condition is caused by the nature of the deformation which is occurring just as the punch completes the bending operation at the end of the stroke. *Negative springback* does not occur in air bending (free bending) because of the lack of the constraints that a V-die imposes.

Compensation for springback:

In forming operations, springback is usually compensated for by overbending the part (Figure 16.20 a and b); several trials may be necessary to obtain the desired results. Another method is to use *coin* the bend area by subjecting it to high localized compressive stresses between the technique tip of the punch and the die surface (16.20 c and d); this technique is known as *bottoming* the punch. Another method is *stretch bending* in which the part is subjected to tension while being bent. In order to reduce springback bending may also be carried out at elevated temperatures.
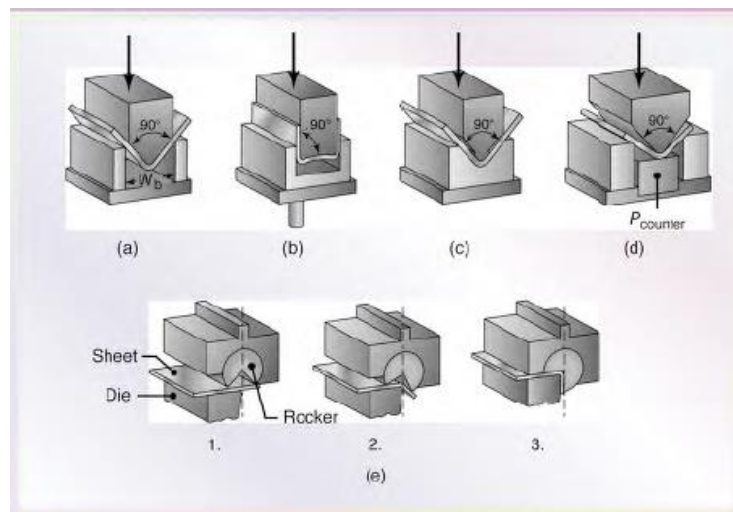

Fig 16.20: Method of reducing or eliminating springback in bending

References:

1. Manufacturing engineering and technology by Serope Kalpakjian.
2. www.wikipedia.org

Bending tube without a mandrel:

Principle of stretching and compressing: When a tube is bent, two things happen to metal (***Figure 1-a***). The outside wall is reduced in thickness due to the stretching of the material and the inside wall becomes thicker due to the compressing of the material (***Figure 1***). The material actually is formed

approximately about the centerline of the tube. The material that forms the outside of the bend has further to travel and therefore is stretched; the inside of the bend has less distance to travel and is compressed.
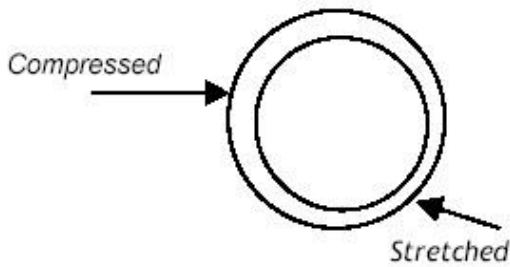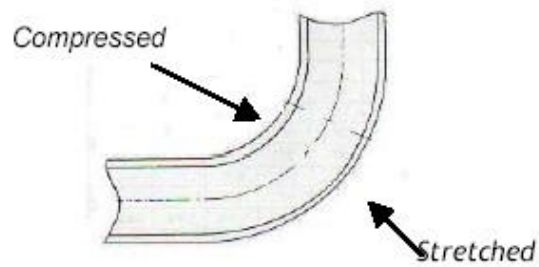

Figure 1-a

Figure 1

Functions of bend die: When the ratio of the tube diameter to wall thickness is small enough, the tube can be bent on a relatively small radius (Centerline radius CLR= 4* tube thickness). Excessive flattening or wrinkling of the bent should not occur. The outside and inside of the bend tend to pull towards the centerline of the tube (flattening). Two factors that help prevent this from happening are a grooved bend die, which supports the tube along the centerline and the natural strength of the tube; round or square (figure 2). **Little or no support is needed within the tube when the tube diameter is small and the wall is thick.** As the size of the tube diameter is increased, the tube becomes weaker. If the wall thickness of the tube is decreased, it also becomes weaker. The forces acting on the tube also becomes greater as the radius of the bend becomes smaller.
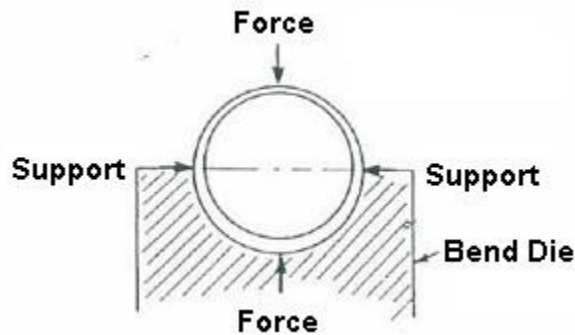

Figure 2

Basic primary tooling: A bend die, clamp die, and pressure die are the minimum essentials for bending tube (*Figure 3*). The bend die helps to prevent the tube from flattening and forms a given radius of bend. The clamp die holds the tube in position while bending. The pressure die forces the tube into the bend die.
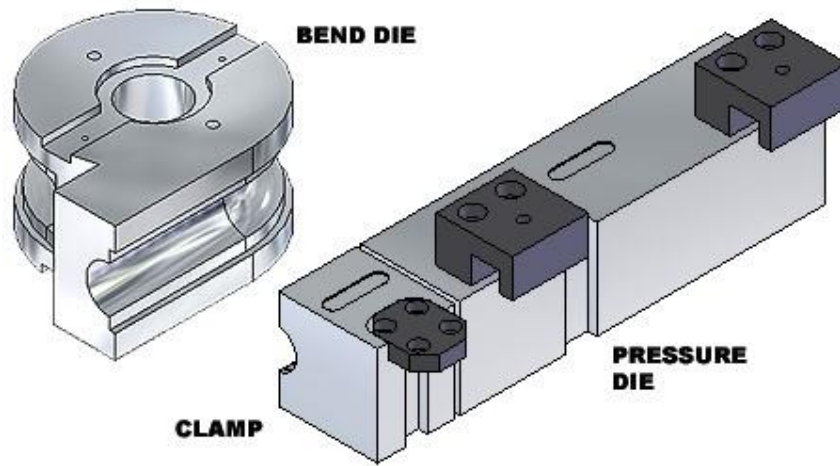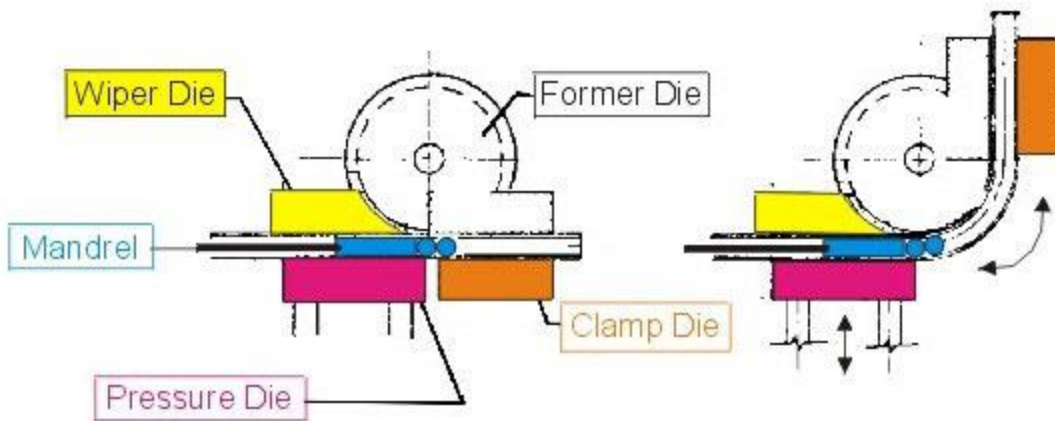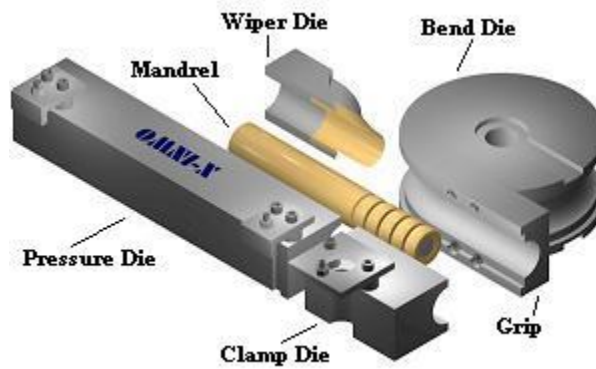
Figure 3

Fig 3 (a)

Figure 3 (b)

Control of springback: Springback is excessive when a mandrel is not used. This should be considered when selecting a bend die. Springback is the term used to describe the tendency of metal that has been formed to return to its original shape. Springback will cause the tube to unbend from

two to ten degrees depending on the radius of bend, and may increase the bend radius of the tube. The smaller the radius of bend the smaller the springback.

Kinked or Buckled bends: The tube may kink or buckle as shown in **Figure 4**.   This may be due to hard material, which will not compress on the inside radius of the bend. The material, not being able to compress, pushes in toward the centerline of the tube. This condition can be corrected (provided the tube is not too hard) by proper set up of the tooling. A plug mandrel (**Figure 5)** is indicated if the tube buckles and is still within the wall factor and the diameter of the bend.
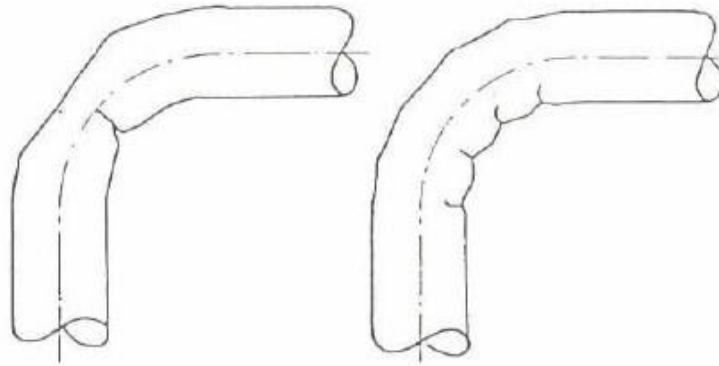

Figure 4

Bending tube with a plug mandrel:

Balanced pressures: **The purpose of a plug mandrel is to prevent the tube from flattening and to bend without wrinkles or kinks.** The mandrel is held in a fixed position while the tube is pulled over it.   The tube stretching process is localized on the outer radius of the bend and the material is work-hardened to retain its shape and not flatten. The material stretching is done on the forward tip of the mandrel **(Figure 5).** This force, acting on the mandrel tip, supports the inner radius of the bend, holding it firmly into the bend die groove.A plug mandrel can be used to produce relatively good quality bends for tubing 10mm diameter and smaller. Exceptions to this are thin wall tubing or a centerline radius that is less than 2 x tube O.D. There also are certain limitations for tubes larger than 10mm diameter.
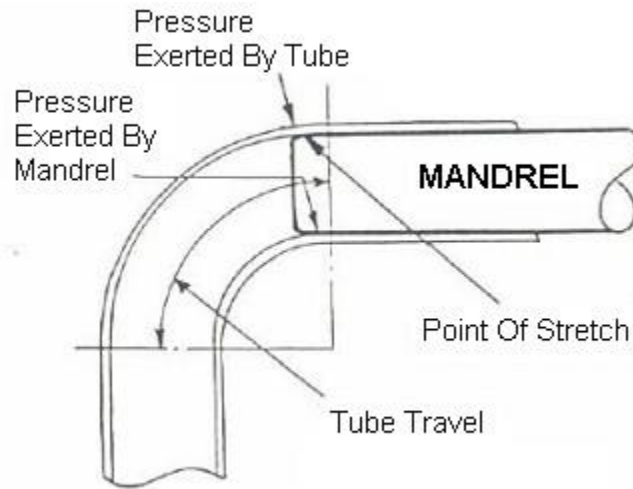
Figure 5

Unbalanced pressures: The pressure die should be adjusted for a light pressure against the tube. **The purpose of the pressure die is to keep the tube against the bend die through the duration of bending.** The pressure die also keeps the mandrel from bending and maintains a straight tube between tangent points of bends (the portion of tubing left on the mandrel after bending). The location of the mandrel affects the amount of springback. The mandrel in a forward position (toward tangent) will stretch the material on the outside of the bend more than is necessary. This increases the length of material on the outside beyond that which is required to make a bend. When the bent tube is removed from the bend die, it will conform to the die and there will be little or no springback. **Figure 6** is an overstated example. The outside of the bend actually is in compression with forces acting at points A and B. Counteracting forces occur at C and D. Forces A and B tend to close the bend while forces C and D act to open the bend.
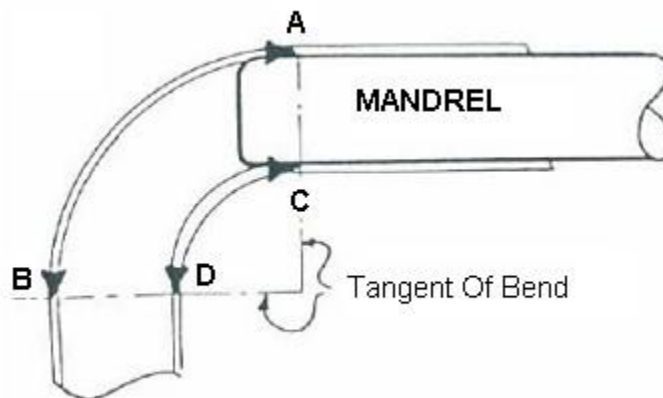

Figure 6

The mandrel in a position away from tangent will not stretch the material on the outside of the bend enough; consequently, there is not enough material to reach from A to B, putting a tension in the material. The forces at A and B are now the reverse of those shown in **Figure 6**, tending to open up the bend. Thus, mandrel location can cause excessive springback, which reduces the angel of bend and

also may increase the radius. The mandrel should be brought forward (toward tangent) when the radius is increased. **There is no given formula for correct mandrel setting.** One thing is clear; when the angle of springback is more than 3 degrees, the mandrel is too retarded and the tube's radius of bend will be larger than the bending die.

When the tube breaks repeatedly, **it may indicate that the material is too hard**. Hard material does not have the ability to stretch sufficiently**.   Working with recently fully annealed material will rule out this likelihood. When the mandrel is set too far forward or the tube slips in the clamp die, breakage may occur also.**
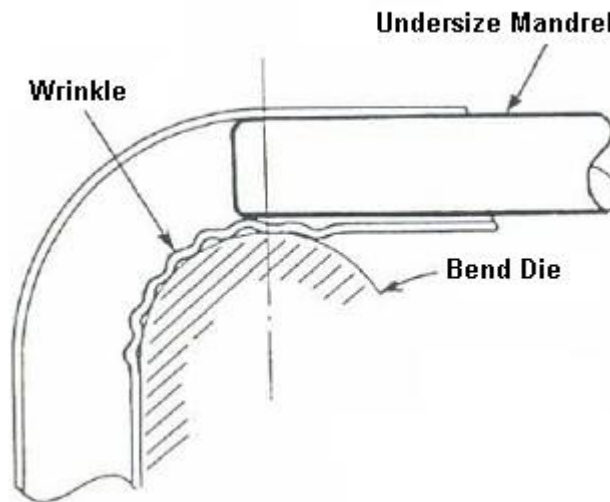

Figure 7

Mandrel too far back: Advancing the mandrel slightly forward, the wrinkles may stop forming in front and begin to form in back of tangent. **The mandrel at this point is still not far enough forward to generate the necessary pressure on the inside of the bend and compress the material.** The bend may start out smooth, but as it progresses past approximately 20 degrees, the material pushes back, forming a ripple or wave at point A (**Figure 8**). This ripple is forming and being flattened continually betweenthe mandrel and the bend die. The ripple, however, does not entirely disappear. When the bent tube is removed from the bend die, there is a large buckle at point A. **It is necessary to continue to advance the mandrel until the material can't squeeze back between the bend die and mandrel. Figure 9** shows what occurs when the mandrel is not full advanced.
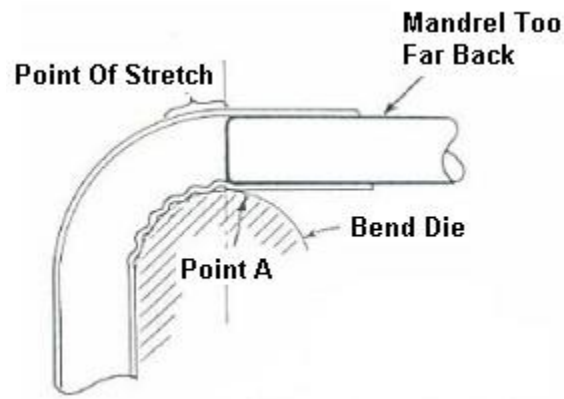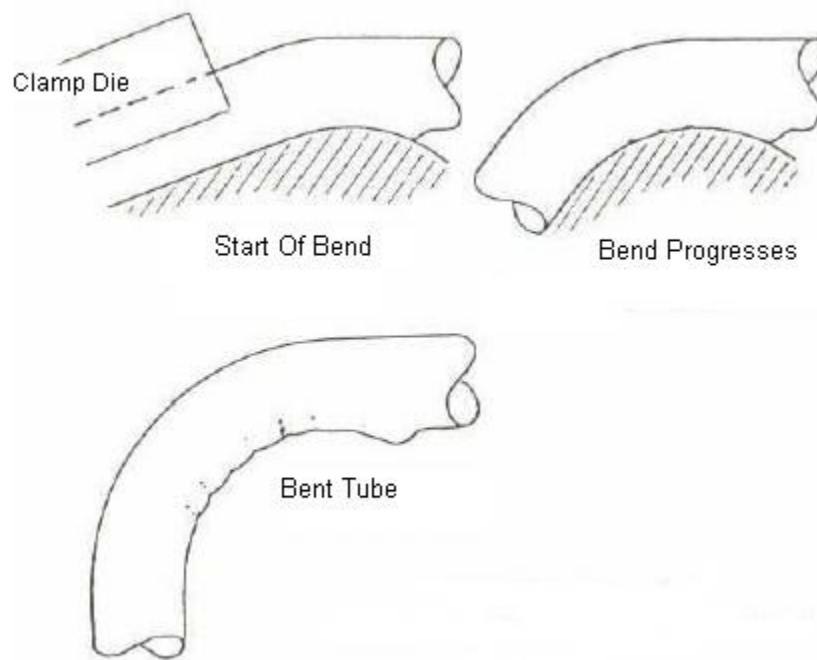
Figure 8

Figure 9

Mandrel too far forward: When the mandrel is too far forward **(Figure 10)** bumps appear on the outside of the bend at the terminal tangent and a step on the inside of the bend at the starting tangent. These are shown on the same tube. They will not always appear at the same time, depending upon shape of the mandrel and bend radius. The bump, obviously, is caused by the mandrel. The end of the mandrel prying the tube away from the bend die forms the step.
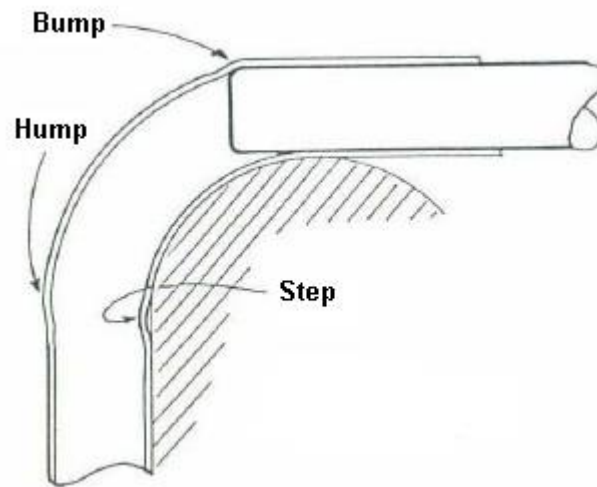
Figure 10

Plug mandrels are inexpensive, easy to maintain and cause little drag.   **Ball-type mandrels, however, should be employed for small radius bends, thin wall tubes or where high quality is desired.** The clamp die should have a minimum length of three times the tube diameter when using a plug mandrel.

Bending with ball mandrel and wiper die: When the radius of the bend is smaller and/or the wall is thinner, it becomes necessary to use a ball mandrel and wiper die. The wiper die is used to prevent wrinkles. The ball mandrel performs like a plug mandrel. The balls are used to keep the tube from collapsing after it leaves the mandrel shank.

Bending issues are enlarged when making tight bends or with thin wall tubing. It becomes more difficult to retain the material during compression. The pressure is so intense the material is squeezed back past tangent and buckles. **This area must be supported so that the material will compress rather than buckle; this is the prime purpose of the wiper die. Note, the wiper die cannot flatten wrinkles after they are formed; it can only prevent them.**
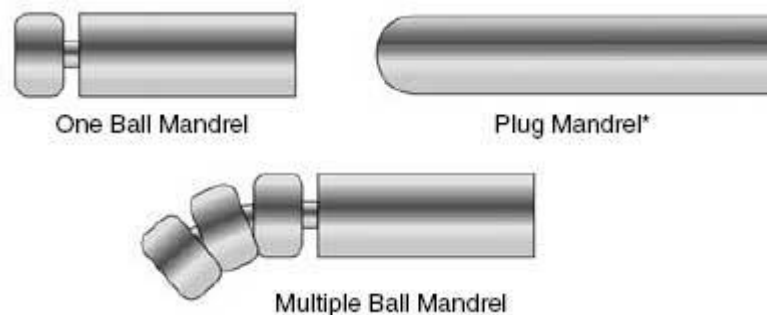

Figure 11: Ball and plug mandrel

Tubes and structural shape bending:

Stretch bending or forming: The work piece is pulled at both ends while being bent over a form block. The method is slow but almost eliminates springback. It is used to make large irregular and noncircular bends without mandrels.

Draw bending: It is dome with the work piece clamped against a form block, which rotates and pulls the material around the bend. The work going into the bend is supported by a pressure bar. A mandrel may be inserted in a tube to restrain flattening. Flexible ball, laminated, cable mandrel etc. are used to provide support around the bend length for delicate work. Draw bending is best for small radii and thin walls and most versatile.

Compression bending or forming: the work piece is clamped to and wrapped around a fixed form block by a wiper shoe. Flat sheet metal is commonly bent in the same way on ungrooved blocks in an operation called wing or tangent bending. Bending radius may be very small. Compression bending can make series of bends with almost no spaces between them. A combination of stretch and compression forming is called radial draw forming and is advantageous for difficult curved parts.

Roll extrusion bending: It is for piper over 130mm O.D. and walls to 16mm thick. A head is rotated inside the pipe with wide thrust rollers on one side and a narrow work roller on the other. The pipe is enclosed by work rings outside of the head. The work roller is cammed in and out as the head rotates to apply pressure to extrude metal in the pipe wall on the side to make it bend. As the material is worked, the pipe is advanced past the head. The method is repeated 10 times faster than others for larger pipes.

Ram or press bending: It is done by pressing the work piece between a moving ram block and two swinging pressure dies as indicated in the figure. A fixed stroke punch press may be used, but an adjustable stroke bending press is better. Equipment cost is little more than for draw bending and angles are limited to about 165^0 but press bending is 3 to 4 times faster than other methods. A different press setup is required for each different bend so the process is suitable only for quantity production, such as furniture factories.

Roll bending: It is done for plates, bars, structural shapes and thick walled tubes using three rolls as shown in the figure. One roll is adjusted between the other two for the desired radius of bend. Continuous coils can be made in this way. Bend radius can be changed easily and the operation is suitable for job work, but angle control is difficult.
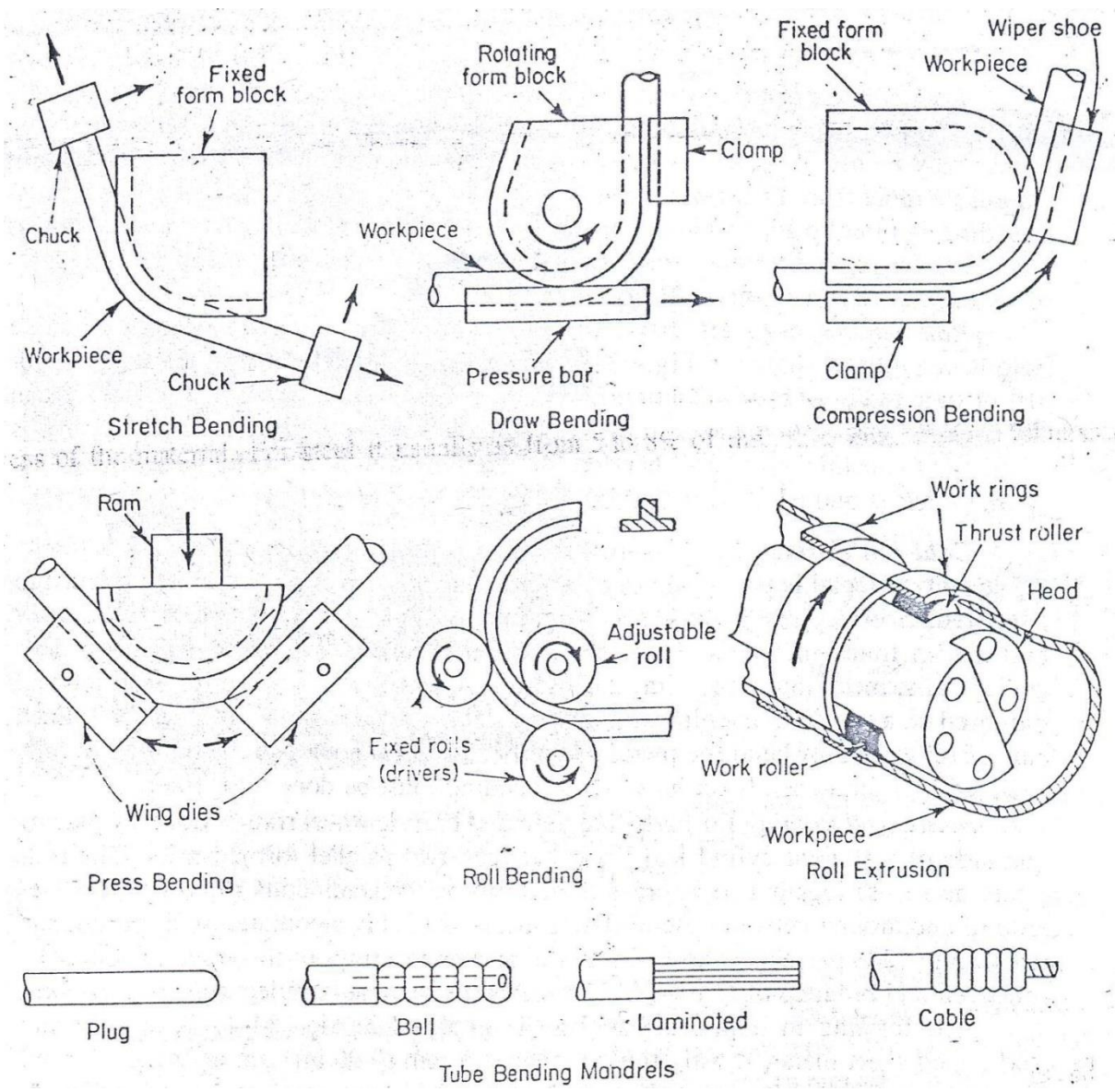
Figure 12: Methods for bending pipes, tubes and structural shape

References:

1. Manufacturing processes and materials for engineers by Doyle
2. R. K. Jain
3. Kalpakgian

Assignment:

1. Write down the causes and suggest ways to eliminate followings:
   (a) Tube breakage (b) Tube wrinkling
2. Explain with neat sketches different tube bending operations.
3. Discuss tube bending mechanism with necessary sketches.
4. Relate tube thickness with different tube bending operations.
5. What are the typical components required for tube bending? Write down their functions.

Experiment 1: Determination of bend allowance and bend deduction for mild steel specimen.

Procedure:

1. Prepare four 4 inch x 1 inch specimen from mild steel sheet.
2. Bend each specimen to 90, 120, 135 and 150 degrees of bend angle separately.
3. Calculate bend radius for each bend angle.
4. Calculate bend allowance, bend deduction and flat pattern length for each bend angle.
5. Plot bend allowance vs. bend angle and bend deduction vs. bend angle in graph paper.

Table:

Bend angle	Bend radius	Bend allowance (BA)	Bend deduction (BD)	Flat pattern length (using BA)	Flat pattern length (using BD)

Formulas:

To calculate bend allowance:

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

To calculate bend deduction for bend angle up-to 90 degree bend angle:

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

For bend angle greater than 90 degrees:

$$BD = R(A - 2) + T(kA - 2)$$

To calculate unbent flat pattern length (L_f):

Using BA: $L_f = B + B +$ Bend allowance

Using BD: $L_f = C + C -$ Bend deduction

Assignment:

1. Calculate the value of bend allowance and bend deduction for the following specimen configuration:

Angle	90^0
pi	3.1416
Radius	1.5 mm
K-factor	Variable for each student
Thickness	6 mm

2. Why do you need to calculate bend allowance and bend deduction? What are the factors that governs the values of those two variables for a certain specimen?
3. Deduce a relationship between bend radiuses and bend deduction, bend radius and bend allowance.

Report writing:

- Objectives
- Apparatus (with specifications if any)
- Machine used (with specifications)
- Schematic diagram
- Data sheet
- Sample calculation
- Graphs (if any)
- Discussion (in passive form)
- Assignment
- References

Experiment 2: Determination of springback and shearing force required for mild steel specimen.

Procedure:

1. Prepare four 4 inch x 1 inch specimen from mild steel sheet.
2. Bend each specimen to 90, 120, 135 and 150 degrees of bend angle separately.
3. Calculate bend radius for each bend angle. This is initial bend radius R_i.
4. Calculate the bend radius again on the next day. This is final bend radius R_f.
5. Calculate the actual and theoretical spring back.
6. Measure the die opening in the bending machine (w) and bend length (L).
7. Calculate bend force required for each specimen.
8. Plot actual springback and theoretical springback vs. bend angle.
9. Plot bending force vs. die opening in graph paper.

Table 1:

Bend angle	Initial radius (R_i)	Final radius (R_f)	Actual springback	Theoretical springback

Table 2:

Bend length (L)	Die opening (w)	Bending force (P)
	$W_1=$	$P_1=$
$L_1=$	$W_2=$	$P_2=$
	$W_3=$	$P_3=$
	$W_4=$	$P_4=$
	$W_1=$	$P_1=$
$L_2=$	$W_2=$	$P_2=$
	$W_3=$	$P_3=$
	$W_4=$	$P_4=$
	$W_1=$	$P_1=$
$L_3=$	$W_2=$	$P_2=$
	$W_3=$	$P_3=$
	$W_4=$	$P_4=$
	$W_1=$	$P_1=$
$L_4=$	$W_2=$	$P_2=$
	$W_3=$	$P_3=$
	$W_4=$	$P_4=$

Formulas:

To calculate springback:

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

To calculate bending force:

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

For 'V' die, this formula can be approximate as:

$$P = \frac{(UTS)LT^2}{W}$$

For mild steel, yield stress, Y= 350 MPa, UTS= 460 MPa.

Assignment:

1. Deduce relationship between bend angle and springback. Also discuss the relationship between material thickness and springback.
2. Deduce relationship between bending force and material strength.
3. Discuss some measures how you can eliminate/reduce springback in bending.
4. Why do you need to calculate springback for a certain specimen?

Report writing:

- Objectives
- Apparatus (with specifications if any)
- Machine used (with specifications)
- Schematic diagram
- Data sheet
- Sample calculation
- Graphs (if any)
- Discussion (in passive form)
- Assignment
- References

Experiment no. 3: To be familiarized with different types of sheet metal forming operations and to make a rectangular/square box with sheet metal specimen.
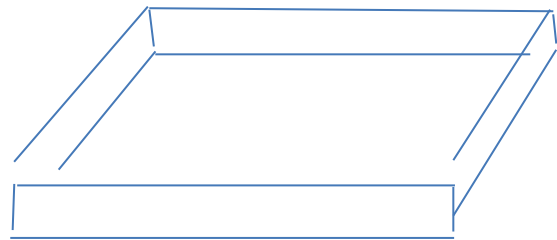
Procedure:

1. Calculate the dimension of required flat sheet to make the box (follow the 'unfolding' technique to do this).
2. Mark on the stocked sheet with scriber.
3. Cut the stocked sheet in the shearing machine along the markings.
4. Mark on the flat sheet on places where to be bent.
5. Connect the diagonals produced on each corner of the flat sheet.
6. Cut the sheet with snip along the diagonals maintaining proper allowances.
7. Bend the flat sheet in the bending machine following 'opposite edge to be the next' technique. Each bend angle should be 90^0.
8. Implement 'cornering operation' on each of the bended corner.
9. Measure the final dimension and compare it with designed one.
10. Discuss reasons behind discrepancy in values, if any.

Figure:
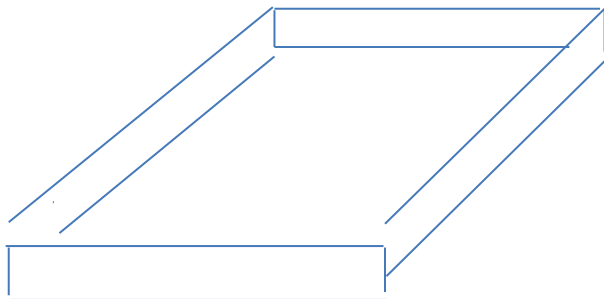
Dimension: 12x8x2 inch (rectangular)   Dimension: 12x12x2 inch (square)
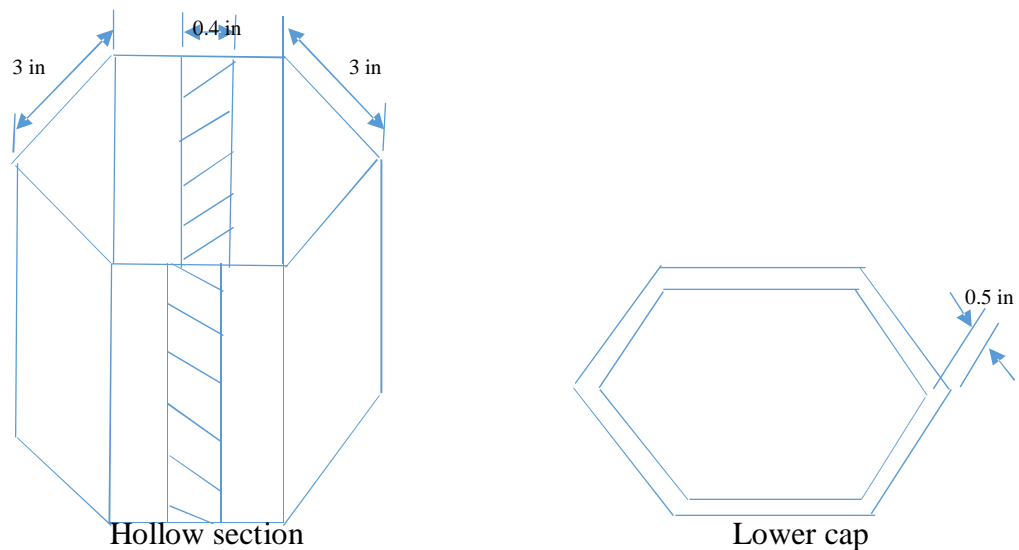
Assignment:

1. What do you mean by 'sheet metal' and 'sheet metal work'?
2. Write down any four characteristics of 'sheet metal'.
3. Discuss different types of sheet metal forming operations.
4. Define the term 'springback'. Which property of the material is responsible for 'springback'?
5. What will be the value of springback for a brittle material like 'glasses'? Justify your answer.

Experiment no. 4: To be familiarized with different sheet metal joining operations and to make a regular hexagonal box with sheet metal specimen.

Procedure:

1. Calculate the flat sheet dimension to make the box (follow the 'unfolding' procedure to do this). Divide the box into two equally dimensioned parts.
2. Mark on the stocked sheet with scriber.
3. Cut the stocked sheet in the shearing machine along the markings.
4. Mark on the flat sheet on places where to be bent.
5. Bend the flat sheet in the bending machine. Firstly bend the extreme edges where seam joint to be produced (bend angle should be nearly 180^0).
6. Produce three bends with bend angle of 60^0 on each parts to make the arms of hexagon. Be careful with the direction of bend, so that opposite parts match together to lock the seam.
7. Lock the seam joint and complete the hollow section.
8. Scribe the outer circumference of the hollow box on a flat sheet.
9. Offset the circumference as required.
10. Join the edges of outer and inner hexagon and cut along the joined line maintaining proper allowance, with the help of snip.
11. Set the hollow box on top of the cap prepared.
12. Implement 'finishing operations' on each component.
13. Measure the final dimension and compare it with designed one.
14. Discuss reasons behind discrepancy in values, if any.

Figure:

3 in   0.4 in   3 in   0.5 in

Hollow section   Lower cap

Assignment:

1. Describe different sheet metal joining operations. Explain with neat sketch: Soldering, Brazing and Resistance welding.
2. Discuss different types of bending operations with neat sketch.
3. Discuss different types of seam joints with illustrations.
4. Write down advantage, limitations and application areas of seam joint.

Report writing:

- Objectives
- Apparatus (with specifications if any)
- Machine used (with specifications)
- Schematic diagram
- Data sheet
- Sample calculation
- Graphs (if any)
- Discussion (in passive form)
- Assignment
- References

Experiment no 5: To be familiarized with sheet metal roll bending process and to make a cylindrical box using sheet metal specimen.

Procedure:

1. Calculate the flat sheet dimension to make the box (follow the 'unfolding' procedure to do this).
2. Mark on the stocked sheet with scriber.
3. Cut the stocked sheet in the shearing machine along the markings.
4. Mark on the flat sheet on places where to be bent.
5. Bend the flat sheet in the bending machine. Firstly bend the extreme edges where seam joint to be produced (bend angle should be nearly 180^0).
6. Bend the flat sheet in sheet rolling machine. Be careful in setting clearance between rollers in the machine. Remember the 'rule of thumb': the closer the clearance, the smaller will be the radius of the cylinder.
7. Lock the seam joint and complete the hollow section.
8. Scribe the outer circumference of the hollow box on a flat sheet.
9. Offset the circumference as required with the help of a divider.
10. Divide the circle into 16 (sixteen) equal parts and mark on the sheet.
11. Cut the sheet along the markings between the outer and inner circle maintaining proper allowance with the help of snip.
12. Bend the split portions with the help of tong.
13. Set the hollow box on top of the cap prepared.
14. Implement 'finishing operations' on each component.
15. Measure the final dimension and compare it with designed one.
16. Discuss reasons behind discrepancy in values, if any.

Figure:

0.5 in

Hollow section

0.5 in

Lower Cap
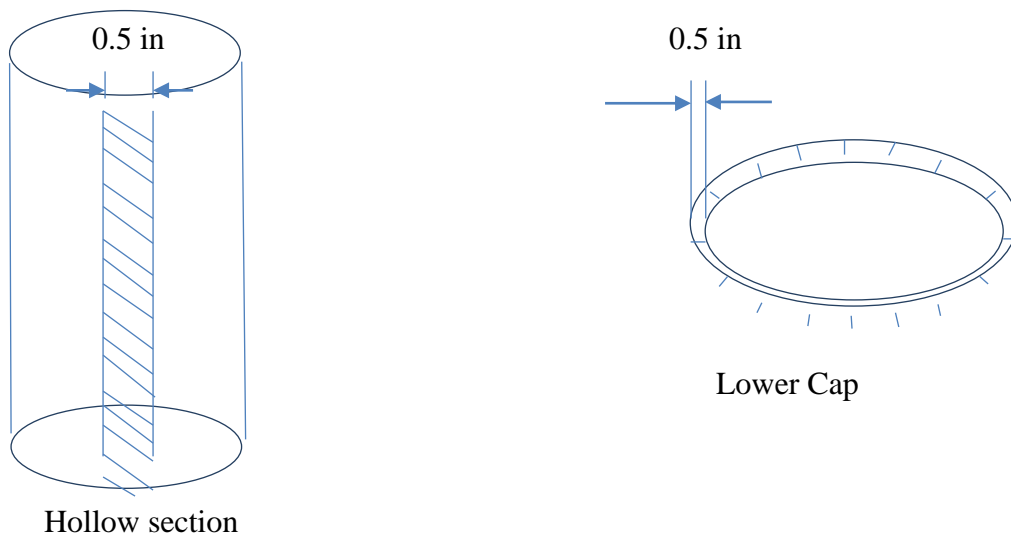
Problem: Find the radius and height of the hollow cylinder having a volume of $64\pi/27\pi/72\pi$ inch^3. Given that, minimum and maximum radius that can be formed in the rolling machine is 2 inch a 10 inch respectively.

Assignment:

1. Elaborate with neat sketch 'Roll bending processes.
2. Can you recommend any alternative method to make a cylinder except 'roll bending'? Discuss.
3. Will the values of 'bend allowance' and 'bend deduction' be larger for 'roll bending' than 'V-die bending'? Justify your answer.
4. How can you find the 'center' of an already drawn circle? What is the value of bend angle for a 'cylinder'?
5. Discuss gas welding process and its advantages, disadvantages.

Report writing:

- Objectives
- Apparatus (with specifications if any)
- Machine used (with specifications)
- Schematic diagram (Rolling machine and job)
- Data sheet
- Sample calculation
- Graphs (if any)
- Discussion (in passive form)
- Assignment
- References

Experiment no 6: To be familiarized with different types of pipe bending operations and various defects in bended pipe.

Procedure:

17. Prepare two pipe specimens from the stocked pipe. Specimen configuration should be: length/Outer diameter/Thickness= (2.5'/1"/1mm) and (2.5'/$\frac{3}{4}$"/1mm). Use hand grinding machine to cut the stocked pipe and finish the cut parts.
18. Set the first specimen on the pipe bending machine. Be sure to pass the pipe in between pressure die & bend die and secure one end of the pipe in the clamp die.
19. Clamp the pipe such that it may flow inside the clam die while pressure is applied to bend the pipe, otherwise the pipe may be distorted if pipe material is not tough enough. Be sure to keep some extra pipe extended further from the clam die in the direction opposite to bend direction; this will facilitate the bending process.
20. Tighten the pressure die with the help of lead screw in the pressure handle. Apply moment in the pressure handle to bend the pipe.
21. Unfasten the pipe from the machine and remove it from the machine. Cut the bended portion in radial direction with the help of hand grinding machine or hacksaw and analyze the bended portion. Make sure if any flattening or wrinkle is present.
22. Repeat from step 2 to 5 for the second specimen.
23. Insert mandrel (sand) in the specimen and repeat from step 2 to 5.
24. Compare observations on the bended pipe without and with mandrel.
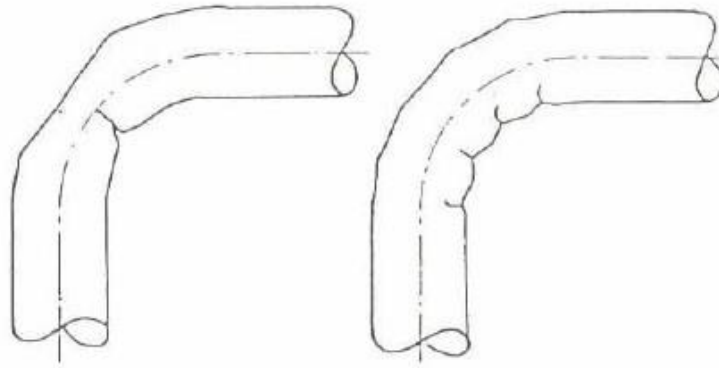
Figure:

Figure: Actual shape of bended pipe

Assignment:

6. Write down the causes and suggest ways to eliminate followings:
   (b) Tube breakage (b) Tube wrinkling
7. Explain with neat sketches different tube bending operations.
8. Discuss tube bending mechanism with necessary sketches.
9. Relate tube thickness with different tube bending operations.
10. What are the typical components required for tube bending? Write down their functions.

Report writing:

- Objectives
- Apparatus (with specifications if any)
- Machine used (with name of the different components)
- Schematic diagram (Experimental setup)
- Data sheet (Pipe configurations)
- Graphs (if any)
- Discussion (in passive form)
- Assignment
- References