

Basics of Mechanical Engineering-2

Prof. J. Ramkumar

Prof. Amandeep Singh Oberoi

Department of Mechanical Engineering

Indian Institute of Technology, Kanpur

Week 06

Lecture 20

Basics of Forming (Part 5 of 5)

Welcome to the next lecture under the title of Forming. Last week, we covered bulk metal forming processes: rolling, forging, extrusion, and wire drawing. These are the four processes we discussed last week. This week, our primary focus will be on sheet metal forming processes. The moment we say 'sheet,' what comes to mind?

First, the sheets might have holes. The sheet might be sheared along its length. In this lecture, we will focus more on these operations. When we talk about sheet metal, we also consider the sheet metal bending process. Whether it's a car, a bus, a door, or any enclosure, we typically start with a sheet and then bend it.

The moment we start bending it, you may also observe something called springback. Now, we should understand how to reduce springback and achieve the exact shape we desire. So, sheet metal processes are going to be very important.

Contents

- Introduction of sheet metal process
- Sheet metal operation
- Important term
- Other sheet metal operation
- Sheet metal analysis
- Drawing operation
- Analysis
- Defect
- Recapitulate
- References



NPTEL



In this lecture, we will first introduce sheet metal processes. Then, we will explore sheet metal operations. Some of the important terminologies. Other sheet metal operations. A small analysis of the material property in relation to the operation. It is very important because you should know what load I should apply. How thick should my die be.

How thick can I try to take the material. All these things are part of the analysis. Then, the drawing operation. We will look into the analysis of the drawing operation, then the defects. Finally, we will do a small recap and references.

Sheet Metal Working



- It includes cutting and forming operations performed on relatively thin sheets of metal.
- Typical sheet-metal thicknesses are between 0.4 mm to 6 mm.
- Sheet-metal working is usually performed at room temperature (cold working).
- Sheet-metal parts are generally characterized by high strength, good dimensional accuracy, good surface finish, and relatively low cost.
- **Stamping press** and **stamping die(punch-and-die)** are used to perform sheet metalwork.
- The sheet-metal products are also called **stampings**.



<https://makeagif.com/gif/diy-metal-brake-for-bending-sheet-metal-4mrOfu>

This is a sheet metal process in which we place a sheet. And they try to bend the sheet to create an L-type angle. So it includes cutting and forming operations performed on relatively thin sheets of metal. We look at mild steel, copper, titanium, magnesium, and many more materials. Typically, what we discuss is a range from 0.4 millimeters to 6 millimeters.

The moment we talk about 6 millimeters, it is going to be a heavy force which is applied for doing any of these operations. Sheet metal work is predominantly performed at room temperature. The sheet metal parts are generally characterized by their high strength, good dimensional accuracy, good surface finish, and relatively low cost. So, till now, what we were only looking at is bending. Now, let us try to see some new operations like stamping.

Stamping is done in a press with the presence of a die. When I say die, there is a punch and a die. I hope you remember the punch and the die. This is a punch. So, a punch and a die are used to perform the metal forming operations. The sheet metal products are also called stamping products.

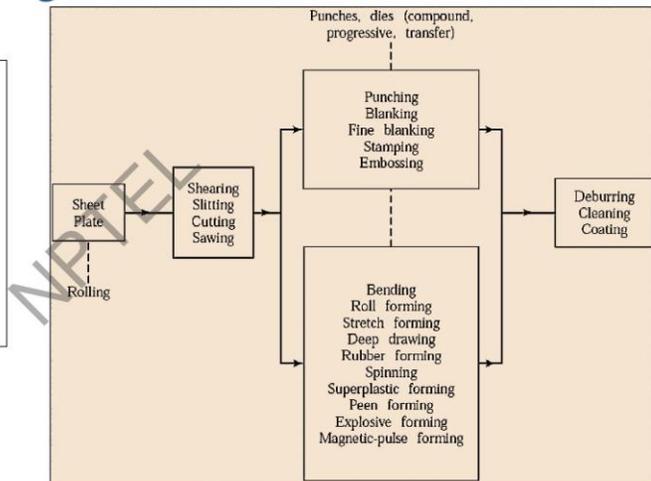
Sheet Metal Working



Categories of sheet-metal processes are

- (1) Cutting
- (2) Bending
- (3) Drawing

- Bending and drawing are used to form sheet-metal parts into required shapes.



M.P. Groover, *Fundamental of modern manufacturing Materials, Processes and system*

So, what are the categories of sheet metal processes? One is cutting, bending, and drawing. So, if you look at the workflow, you will see this is the input which is given as a sheet. Then these are the operations which are done on the sheet.

Shearing and slitting mean cutting. Breaking, right—two pieces. Then cutting—the difference between slitting and cutting is that cutting can be done for a slightly larger width. Slitting is for a small width. Then you can also have sawing.

Slitting, sawing, cutting. Slitting and sawing can predominantly be done on a rotary tool or a circular disc-like thing. But all these are basically operations carried out on sheet metal to produce an output. For these operations, we use machine tools and some other tools. So, these are the operations—these are the processes.

For these operations, we can develop the process. So, punching is one process: blanking, fine blanking, stamping, and embossing. When we look at the other spectrum, we have bending, roll forming, stretch forming, deep drawing, rubber forming, spinning, superplastic deformation, pin forming, explosive forming, and magnetic pulse forming. So here, they bend, right.

They are bent and they try to deform. Here they cut and they remove the waste. So these are operations which are done using a punch die, a compound die, or a progressive die. Once the operation is over, we have to do a post-processing operation, which can be

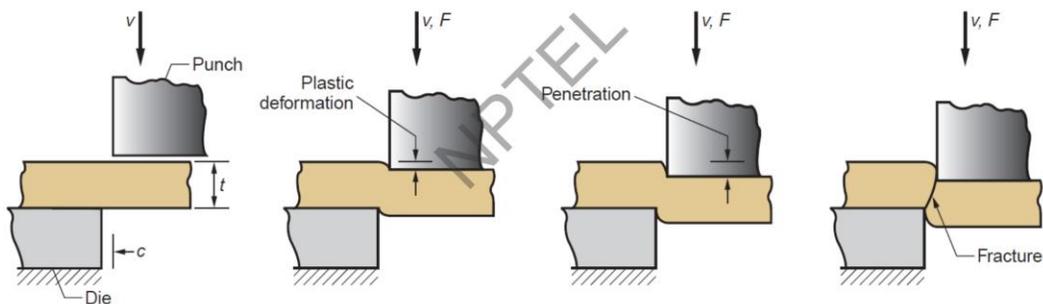
either deburring, cleaning, or coating. So these two are the processes which we will see in detail.

Bending and drawing are used to form sheet metal parts into a required shape. So all these things are forming processes. And here, if you see, these things are cutting processes: cutting, shearing, selecting—all these things come here.

Cutting Operation



- Cutting is used to separate large sheets into smaller pieces, to cut out part perimeters, and to make holes in parts.
- It is accomplished by a shearing action between two sharp cutting edges.



M.P.Groover, *Fundamental of modern manufacturing Materials, Processes and system*

So, cutting operation. Cutting is used to separate a large sheet into small pieces. You cut it into small pieces. You take a vegetable and cut it into small pieces. So, cutting is used to separate a large sheet into small pieces. To cut out parts and to make holes in parts. We try to do this cutting operation.

It is accomplished by a shearing action between two sharp edges. So here is a sharp edge, here is a sharp edge. So here what happens is the shearing action. So this is a punch. The punch comes with a certain velocity.

It can be gradual, it can be impact. So it can be mechanical, it can be hydraulic. The punch moves up and down. There is a die. In this there is a sheet metal which is placed.

So now the punch is moved downwards. So there is a die, sheet metal. So once the punch is moved downwards, there is a plastic deformation which is happening. Because initially they start with the plastic deformation, then further penetration happens. So, that leads to

a fracture. So, the cutting operation is accomplished by a shearing action between two sharp edges.

Important Terms



Rollover

- At the top of the cut surface is a region called rollover.
- This corresponds to the depression made by the punch in the work prior to cutting.

Burnish

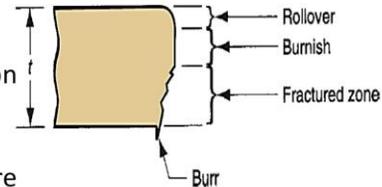
- It is a relatively smooth region, it results from penetration of the punch into the work before fracture began.

Fractured zone

- It is relatively rough surface of the cut edge where continued downward movement of the punch caused fracture of the metal.

Burr

- It is a sharp corner on the edge caused by elongation of the metal during final separation of the two pieces.



So, the important terminology is when you try to shear; you can see these are the terminologies: rollover. Burnish, fractured zone, and burr formation. Rollover at the top of the cut surface is a region called rollover. This corresponds to the depression made by the punch in the work prior to cutting.

So, the initial indentation is nothing but rollover. Burnishing means where there is a smooth region; it results from the penetration of the punch into the work before fracture begins. So, burnishing means you are trying to roll the material along the walls. That is burnishing. Rollover is the initial bend that you create.

Then comes the fracture zone. In the fracture zone, it is a relatively rough surface of the cutting edge where continued downward movement of the punch causes the fracture. So here, rollover; then it is burnishing; then here the fracture starts. After the fracture starts, any excess material which projects out of the surface is called the burr, which has to be removed. If you try to slide or touch and feel just on top of a hole made on a metal sheet, you will always see there are sharp edges.

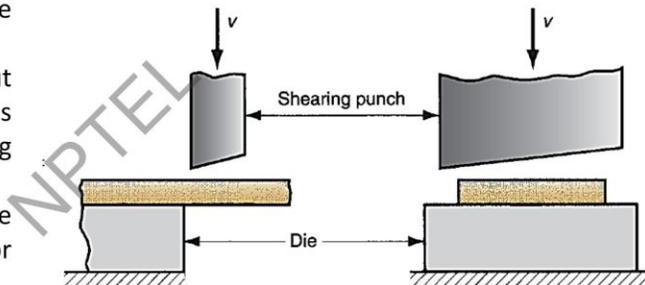
Those sharp edges are called burrs. So when we perform a cutting operation, these are the terminologies that are formed when we take a cross-section and examine it. Rolling, burnishing, fracture zone, and burr formation.

Operation: Sheet Metal



Shearing

- Shearing is a sheet-metal cutting operation along a straight line between two cutting Edges.
- Shearing is typically used to cut large sheets into smaller sections for subsequent pressworking operations.
- It is performed on a machine called a power shears, or squaring shears.



The first operation we saw was cutting; now what we see is shearing. So shearing—this is the die, and here you can see there is a punch.

This is the side view, and this is the front view—the same. So you see here there is a taper. Shearing is a sheet metal cutting operation along a straight line between two cutting edges. Shearing is typically used to cut large sheets into smaller sections. So you can see if you have a large plate or a large sheet, and you want to cut it into several small segments, what we do is perform a shearing operation.

It is performed on a machine called a powered shear or square shear. This is always tapered. Why is it tapered? Because we always start at one edge, which is called the first approach edge. The first approach edge will have a sharp corner, and this sharp corner.

If you see the stress that is being generated, it is force per unit area. The area is very small. So the force will be very—whatever the normal force is, the stress will be very high. Because of the stress, what will happen? The initiation of the crack or the initiation of the shearing can start.

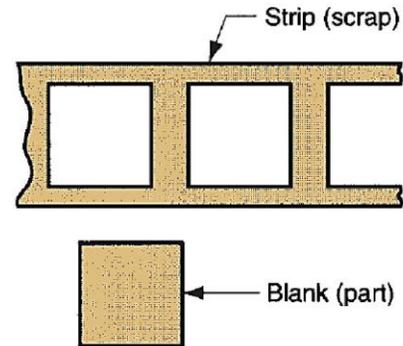
Afterwards, you are trying to go all along the length before it finally shears. So in shearing, we will always have a taper. It is very important. And if you have a very thin punch, then this will lead to bending. To avoid that, what we do is we try to have this tapered edge and a thick plate.

Operation: Sheet Metal



Blanking

- Blanking involves cutting of the sheet metal along a closed outline in a single step to separate the piece from the surrounding stock.
- The part that is cut out is the desired product in the operation and is called the blank.



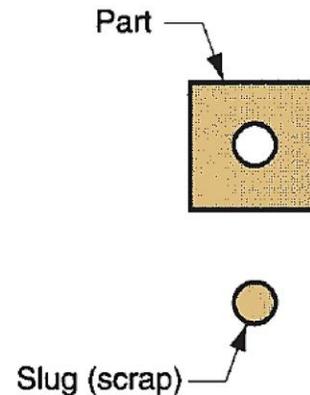
The next operation is blanking. Blanking means you have a strip, you have a punch, and you cut a portion of the sheet, right? Sorry, you cut a portion of the sheet, and what comes out is called a blank. Now, this blank will be the useful part for further operations. So blanking involves cutting the sheet metal along a closed outline in a single step to separate the piece from the surrounding stock.

So this is the stock. So this if you see initially it will be a sheet of thin sheet of metal. So now with the punch you try to blank and get the piece. So now this blank is the starting material or this is the work which can be further used. The part that is cut out of the desired product in the operation is called as the blank Punching.

Operation: Sheet Metal

Punching

- Punching is similar to blanking except that it produces a hole, and the separated piece is scrap, called the slug.
- The remaining stock is the desired part

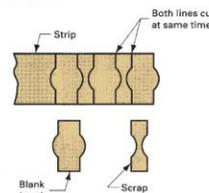


So whatever comes out of the sheet is called as the blank. So now in this blank if you want to make a hole it is called as punching. Punching is a similar operation to blanking except that it produces a hole and the separated piece is a scrap or a slug. In the first one whatever came out is called as a blank which is a useful part for the next operation. In punch, whatever gets punched out is called as a scrap or a slug which is there and you try to create a profile. The remaining stock is the desired part. So, now I am sure you will be able to distinguish punching, blanking, shearing also.

Other Operation: Sheet Metal

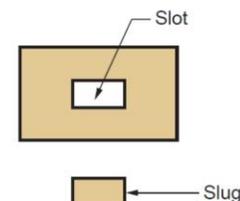
Parting

- It involves cutting a sheet-metal strip by a punch with two cutting edges that match the opposite sides of the blank.



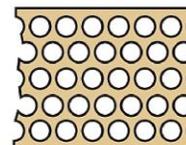
Slotting

- It is a special punching operation that cuts out an elongated or rectangular hole.



Perforating

- It involves the simultaneous punching of a pattern of holes in sheet metal.
- The hole pattern is usually for decorative purposes, or to allow passage of light, gas, or fluid.



So, we have parting operation. Parting, slotting and perforation. So these are hole perforations. Suppose you wanted to sort coins. Then they would have varying diameters, or you wanted to sieve some ore. So they have a sheet sieve with larger diameter holes. The parting operation involves cutting a sheet metal strip by a punch with two cutting heads that match the opposite sides of a blank. So this is called the parting operation. So you have a blank, and this is scrap.

It involves cutting a sheet metal strip by a punch with two cutting edges that match the opposite sides of a blank. What is slotting? A special type of punching operation is called a slotting operation. Here, in punching, generally what we do is try to have a circular hole. In slotting, we have something like a square or a rectangular hole.

Perforating involves simultaneously punching a pattern of holes in sheet metal. The hole pattern is usually used for decorative purposes or to allow light, gas, or fluid to flow through. It can be used like a sieve.

Other Operation: Sheet Metal



Notching

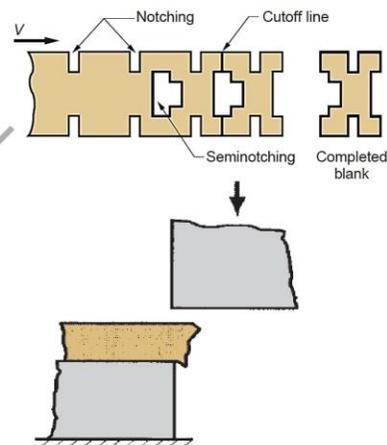
- It involves cutting out a portion of metal from the side of the sheet or strip.

Trimming

- It is a cutting operation performed on a formed part to remove excess metal and establish size.

Shaving

- It is a secondary shearing operation performed with very small clearance to obtain accurate dimensions and cut edges that are smooth and straight.



Notching operation. The notching operation involves cutting out a portion of the metal from the side of the sheet or strip, which is called notching. This is called as notching. Trimming is a cutting operation performed on a formed part to remove excess material and establish size. This is trimming. So notching is on the side if you try to remove

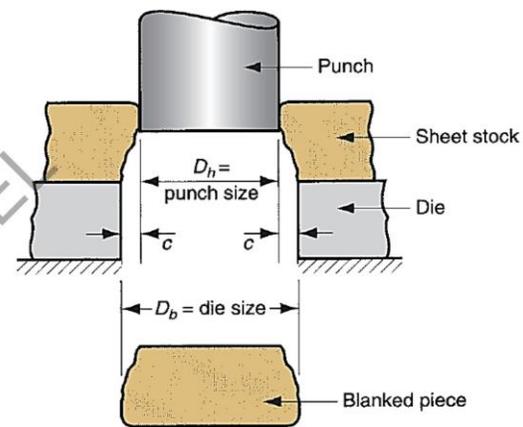
material is called as notching. Trimming it is a cutting operation performed on a formed part.

Please understand this is on a formed part to remove the excess metal of and an established size. Trimming is you take a scissor and cut that like trimming right. The barber does trimming operation cutting. Cutting of the excess metal from the manufactured part is trimming. Shaving operation, this is a shaving operation.

Shaving operation means it is a secondary shearing operation performed with a very small clearance to obtain the accurate dimension. And the cutting edge that of a smooth and a straight one we call shaving. Shaving is removing the excess material, excess material little bit to get it is called as shaving. The secondary shearing operation performed with a very small clearance is shearing operation.

Analysis: Sheet Metal

- Process parameters in sheet-metal cutting are clearance between punch and die, stock thickness, type of metal its strength, and length of the cut.



Now let us do a little bit of analysis in sheet metal. So this is the punch. This is the die. This is the sheet metal that we keep. The punch moves downward. When the punch moves downward, it tries to perform whatever operation we specify.

The four operations—shearing, fracturing—all these things happen. And then it brings out the blank piece. If the blank piece is further used, this becomes a blank. Or if it is scrap, then it becomes a punch. So here what you have is a diameter.

This is the punch diameter; there is always a clearance between the punch and the die. So, that is called c , a clearance. So, if you see, the die size is c . $D_h + 2c$ will be D_b . The die size will be large.

Why? Because as and when you punch, the part has to fall down without any clogging. So, it will push itself and then fall down. So, this is the die size. This is the punch size.

Die size and punch size. Between the die size and punch size, you will always have a clearance. The process parameters in sheet metal cutting are clearance between punch and die. The clearance will have a relationship with respect to the sheet thickness or the stock thickness, type of metal, its strength, and the length of the cut.

Analysis: Sheet Metal



Clearance

The clearance c in a shearing operation is the distance between the punch and die, its usually ranges between 4% and 8% of the sheet-metal thickness t .

$$\text{clearance } (c) = A_c t$$

Here, A_c = clearance allowance; and t = stock thickness(mm)

clearance allowance depends on the type of metal

Punch and Die diameter

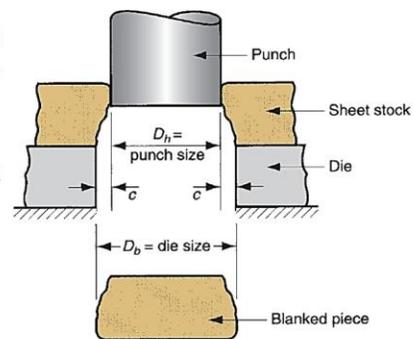
$$\text{Blanking punch diameter} = D_b - 2c$$

$$\text{Blanking die diameter} = D_b$$

Punch and die sizes for a round hole of diameter D_h are determined as:

$$\text{Hole punch diameter} = D_h$$

$$\text{Hole die diameter} = D_h + 2c$$



So, what is clearance? The clearance (C) in a shearing operation is the distance between the punch and the die. It usually ranges from 4% to 8% of the sheet metal thickness. So, the clearance will be 4% to 8%, which is A_c into t . So, A_c is the clearance allowance, and t is the stock thickness.

So clearance c will always be taken. The clearance allowance depends on the type of metal, punch, and diameter. The punch diameter, the blanking punch diameter, will be $D_d - 2c$. The punch and die size for a round hole of diameter D_h are determined as the hole punch diameter is D_h , and the hole clearance will be $D_h + 2c$.

Please make a note for blank: what is it? And for punch: what is it? In the examination, you can be asked a problem based on this. So the blanking punch diameter will be die size – 2c. Whereas in the punch size, the hole diameter will be the diameter of the hole + 2c.

Analysis: Sheet Metal



Cutting Forces (F)

Cutting force determines the size (tonnage) of the press needed.

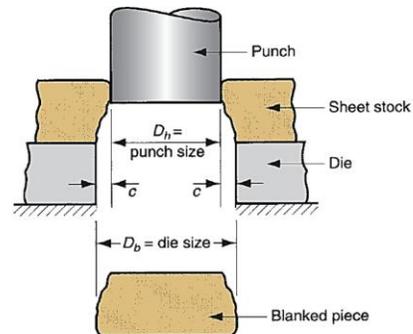
$$F = StL$$

here S = shear strength of the sheet metal (Mpa)

t = stock thickness (mm)

L = length of the cut edge (mm)

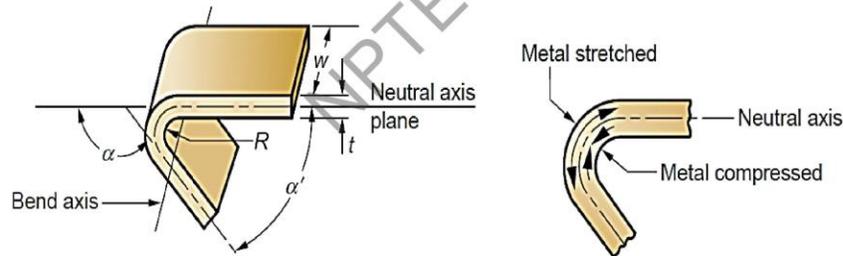
An angled cutting edge on the punch or die reduce the maximum cutting force.



So if we want to find out the force required for shearing, $F = StL$. What is S ? S is the shear strength of the given sheet material. t is the thickness, and L is the length of the cut. And this is the length of the cut, the length of the cut edge. An angled cutting edge on the punch or the die reduces the maximum cutting force.

Bending Operation

- Bending in sheet-metal work is defined as the straining of the metal around a straight axis.
- During the bending operation, the metal on the inside of the neutral plane is compressed, while the metal on the outside of the neutral plane is stretched.
- Bending produces little or no change in the thickness of the sheet metal.



In a bending operation, you can see when you try to buy any vessel, there will always be a bend at the top, right? So that is a bending operation. You can even try to see when you take a milk kettle or a milk-boiling vessel; you can see they will always have a mouth that is rounded off.

All these things are bending operations. When we try to have a rod, we also try to bend the rod. So here we are talking about a sheet. When you can also convert the sheet into a rod, then we call it bending of rods. Bending in sheet metal work is defined as the straining of the metal around a straight axis.

So this is the straight axis, the neutral axis. During the bending operation, the metal on the inside of the neutral plane is compressed. So this is compressed; the inside is compressed, and the outside will be tensile. While the metal on the outside of the neutral plane is stretched or undergoes tensile stress. So when we try to bend, the bottom is compressed, and the top will be tensile.

The bending produces little or no change in the thickness of the sheet. So when we try to bend it, it has to bend in such a way that there is no change in the thickness. This is the thickness; this is the thickness. The T will be uniform all around. So, if you look into it, this is a sheet metal. This is the neutral axis.

It is stretched at the top, compressed at the bottom, and when we bend it, it has a radius. You will not have a sharp bend. So, you will always have a radius of R , and the neutral axis will be bent by an angle called α . This is α' , and this is α .

Bending Operation

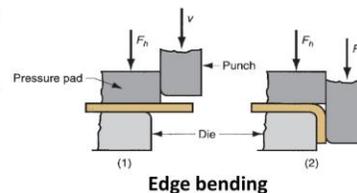
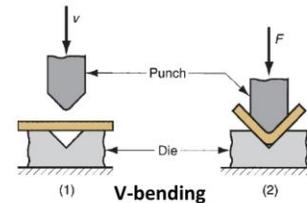


V-bending

- In V-bending, the sheet metal is bent between a V-shaped punch and die.
- It is generally used for low-production operations.

Edge bending

- It involves cantilever loading of the sheet metal.
- Edge bending is generally limited to bends of 90° or less.
- Bend angles greater than 90° can be produced by wiping die.



So, in bending, we can do V-bending or edge bending. V-bending is when the sheet metal is bent between a V-shaped punch and a die. This is a die; this is a punch. So when it presses, it tries to make a V-shaped component. It is generally used for lower production processes. When we go for the edge bending process, like in the glass tumbler edge.

It involves cantilever loading of the sheet metal. The edge is generally bent at 90 degrees, or less. So bending greater than 90 degrees can be produced by a wiping die. So if you want to have more than 90 degrees, it will require a wiping die. There has to be a change in the die.

Bending Operation

Flanging

- It is a bending operation in which the edge of a sheet-metal part is bent at a 90° angle (usually) to form a rim or flange.
- It is often used to strengthen or stiffen sheet metal.



So flanging is a bending operation in which the edge of a sheet metal part is bent at a 90-degree angle to form a rim or a flange. This is straight bending. This is stretch bending. And this is shrink bending. So stretch bending is along like this.

Shrink bending is along in this direction. Straight is this way. And then here you will have a stretch like this. And the other way around is you will have a concave. Then you will have bending. So, this is often used to strengthen or stiffen a sheet metal.

Bending Operation



Hemming

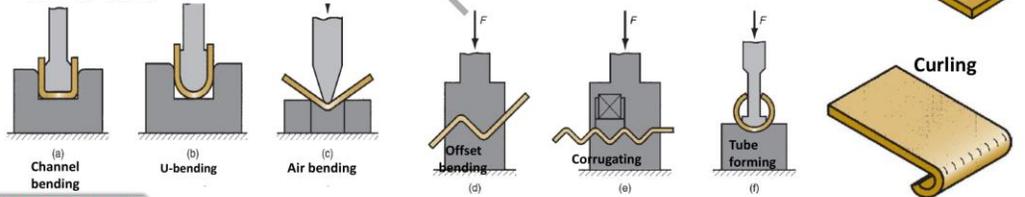
- It involves bending the edge of the sheet over on itself, in more than one bending step.
- This is often done to eliminate the sharp edge on the piece.

Seaming

- It is an operation two sheet-metal edges are assembled.

Curling

- It is also called beading it forms the edges of the part into a roll or curl.



M.P.Groover, Fundamental of modern manufacturing Materials, Processes and system

Hemming is bending again it will try to increase the stiffness of the sheet metal. It is involved in bending the edge of a sheet over on itself in more than one bending step. So, we try to do this because if you want to increase the stiffness we try to do. This is often done to eliminate the sharp edge on the piece.

Seaming is connecting two sheets. It is an operation where two sheet metal edges are assembled. It is almost like this. This is one hemming. This is another hemming.

These two hemming put together is called as seaming. Curling is the other operation. So curling is you try to bend and create a curl. So it is also called beading. It forms the edge of a part into a roll or a curl, this one.

This one is hemming. Then it is seaming, attaching two things. And then it is bending and then you leave it. You also have several of these bending like channel bending. You can bend and create a long channel, channel bending.

And then you can have U channel bending, U bending. Then we have a air bending, so wherein which you have a small bent like this. Offset bending is possible like corrugated sheet you have. Then you have corrugating which is also a bending operation. Then you have a tube forming operation.

Tube forming is you start like this, you press it. Then these two fellows join together and it forms a tube. These are the other bending processes which are generally done on a sheet metal process.

Analysis of Bending

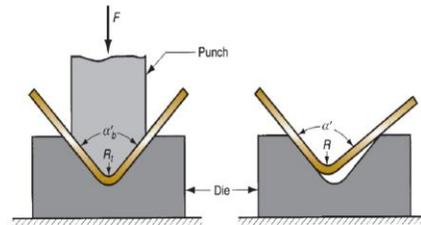


Spring back

- When the bending pressure is removed at the end of the deformation operation, elastic energy remains in the bent part, causing it to recover partially toward its original shape this elastic recovery is called spring back. (it can be also defined as the increase in included angle of the bent part relative to the included angle of the forming tool after the tool is removed)

$$SB = \frac{\alpha' - \alpha'_b}{\alpha'_b}$$

- Here SB= spring back
 α' = included angle of the sheet-metal part, degrees
 α'_b = included angle of the bending tool, degrees



Now let us try to understand the important concept in sheet metal called spring back. Why is this spring back coming into effect? Because when you try to see the stress strain curve, it has elastic. Then maybe it gets into a plastic region. So this is elastic region, this is plastic region. When we try to deform plastically, At certain point what happens is if you are not properly into the plastic region.

There is always a small amount of recovery happens in the plastic zone that leads to the spring back. When the bending pressure is removed at the end of the deformation process. Elastic energy remains in the bend part causing it to recover partially towards its original shape. This elastic recovery is called as spring back. It is also defined as the increase in included angle.

This is the included angle α . Included angle of the bend part relative to the included angle of the formed tool after the tool is removed is called as spring back. Spring back is nothing but

$$SB = \frac{\alpha' - \alpha'_b}{\alpha'_b}$$

So this is the angle whatever it is getting formed. And when the punch is removed you see here this will try to change or increase by this.

So the difference divided by the original will try to give the spring back. The included angle of the sheet metal part in degrees is α . Then the included angle of the bending tool in degrees is α_b . So, the difference between these two is called the springback.

It is a very important phenomenon. Whenever there is a bending operation, there is an elastic energy recovery that will try to bring it back to the original shape. What is the original shape? The original shape was a flat plate. Now you are making it into a V. So now what will happen?

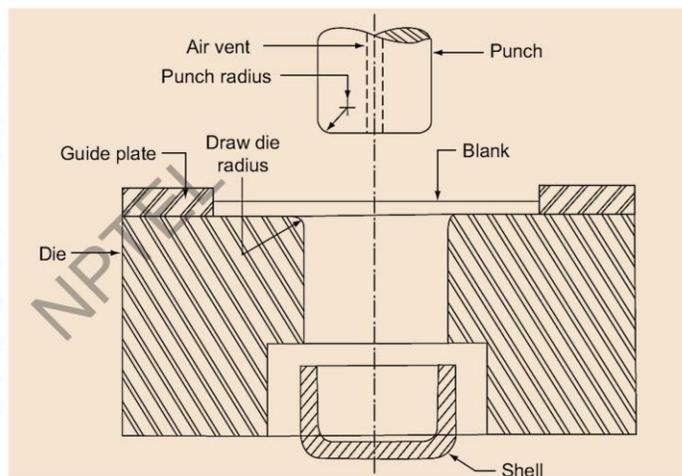
It will become like this. So this is the tool, the punch angle, whatever it is, and this is what it tries to take after it is removed. So bending force, how do you calculate the bending force? The bending force required to perform bending depends on the geometry of the punch and die. The thickness and the length of the sheet metal.

So it will be force is equal to tensile strength into width of the part into thickness of the part divided by die opening dimension. This is a constant which depends on the type of bending. So force = T_s . Generally what we do is $T_s \times 2/d$.

And this is something like a weightage factor D is the die opening dimension die opening dimension. This is d die opening dimension, okay. So width of the part or width of the sheet drawing operation. Till now what we saw was various cutting shearing bending operations right. We also saw a blanking operation, we also saw a punching operation.

Drawing

- It is performed by placing a piece of sheet metal over a die cavity and then pushing the metal into the opening with a punch.
- It is used to make cup-shaped, box-shaped, or other complex-curved and concave parts such as beverage cans, ammunition shells, sinks, cooking pots, and automobile body panels.



Now what we will do is, we will get into a process called as drawing. In drawing, what we do is, we always look for high aspect ratio, high aspect ratio parts. So wherein, we try to talk about diameter of the blank. What is a blank sheet metal you had a punch you pressed it and what fall down was the blank. Now using this blank you are trying to push it against a punch which moves into the die to create high aspect ratio parts.

So here is the diameter to depth. So, here this is what a drawing operation is. In a drawing operation, we always try to make a glass tumbler, a vessel, or deep drawing. It is performed by placing a piece of sheet metal over a die cavity and pushing it against. Now, whenever the punch comes down, the punch is rounded.

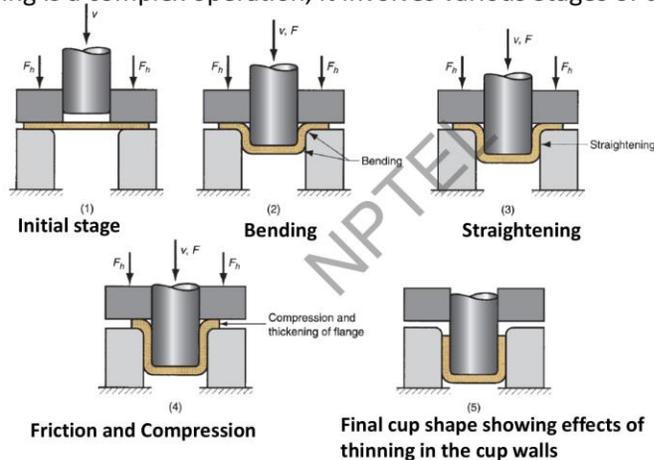
So, in the shearing operation, we had a sharp edge. So, this sharp edge was at an angle. So, it created piercing, right. But here, what we do is we have a radius. The moment you have a radius, what happens?

It will not try to puncture or shear. It will try to push it down and bend the blank as it goes down. So, it is used to make cup-shaped, box-shaped, or other convex or concave-shaped parts. Such as beverage cans are made out of drawing operation; Pepsi cans are made out of drawing. Then ammunition shells, sinks, cooking pots, and automobile body parts are all made out of drawing operation, okay.

So here, there is no cutting of the blank; it will be just bending along it. So here, there is a radius; the die also has a radius. So these radii allow it to smoothly flow. And then it tries to create the cup—okay, this is how it is.

Drawing Steps

Deep drawing is a complex operation, it involves various Stages of deformation of the work



So these are the steps involved in a drawing operation. In a drawing operation, the first drawing is a complex operation involving multiple stages. In the initial stage, what we do is try to have a blank holder, which I did not mention in the previous slide—blank holder. So now, the function of the blank holder is to hold the blank. What is the blank? The yellow part is called the blank.

This is the blank. Why is this blank required? Suppose you don't hold the blank. And when you try to do a bending operation, there can be wrinkles formed on the edges. So, to avoid this and to ensure proper metal flow as the punch moves down, we have a blanking force.

Optimizing the blanking force, the blank holder's force is very important, which is in turn related to the velocity with which you push it down. So the initial stage is that. Then, in the next stage, there is bending that happens to the blank. So the holder continues to have the same force. Now, the punch is moving down.

So it tries to create a bending operation. As it moves down, this bending will try to convert into straightening. And this is a constant-volume process. The material has to flow from the blank area only. So it will have a straight portion.

Then, after it has achieved the straight portion, what it will do is experience friction and compression. You further move it down. In reality, it is a constant blank-holding force.

But if you want to get the best out of it, you can also have a dynamic blank-holding force. That means, depending upon the punch height moving down.

The velocity with which it is moving down. The material which is flowing, the speed with which it is flowing. You can also try to vary the blank holder force. Generally, what we do is we try to have a load, and then we will try to have a spring. Or we will have a nut and then tighten it.

That's how we do it. So you can have the friction, and compression happens when it goes to the final stage. Finally, what happens is all the material below the blank holder is eased out, and then you create a straight wall. So this is called the final cup shape, showing the thinning into a cup wall. So all these things are different steps of the drawing process.

It undergoes all these things. So if you go back to bending, what all did we see in bending? We saw flanging, V-bending, edge bending, flanging, hemming, seaming, and curling. All these things are bending. Here also, it is bending. So bending means there is no cut. It is only trying to change its shape from flat to a cup.

Drawing Analysis



In case of drawing of a cup shaped part

c = Clearance

D_b = Blank diameter

D_p = Punch diameter

R_d = Die corner radius

R_p = Punch corner radius

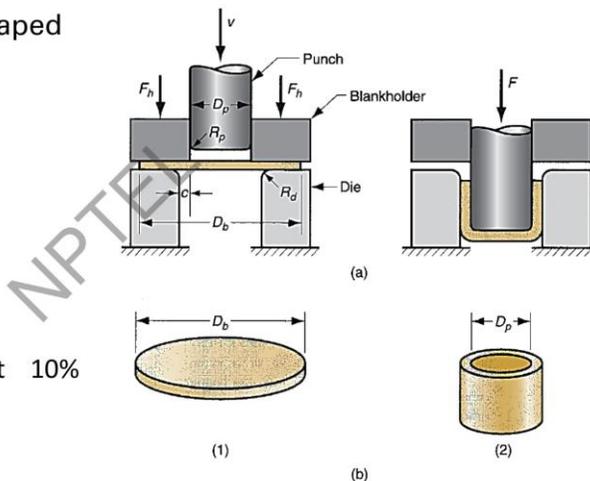
F = Drawing force

F_h = Holding force

Clearance

Clearance in drawing is about 10% greater than the stock thickness.

$$c = 1.1t$$



So here we will try to do a drawing analysis in case of drawing a cup-shaped part. See, you cannot have the punch diameter equal to the die diameter. If these two are equal, then when it is bending, you will not have enough space for the blank to bend.

So there is always a clearance which is given. So c is the clearance which is given, and the blank diameter is this: the blank diameter which is D_b . The punch diameter is called D_p . The die corner radius, this is called the die corner radius, is called R_d . The punch corner radius is called R_p .

The force which is required to push it down is F , and the holding, blank holding, is this. If the holding is very tight, when the punch moves very fast, it can shear. If the holding is very weak, when the punch moves down, you will try to have defects in drawing. So the clearance is always given as 10% greater than the stock thickness. So $c = 1.1t$.

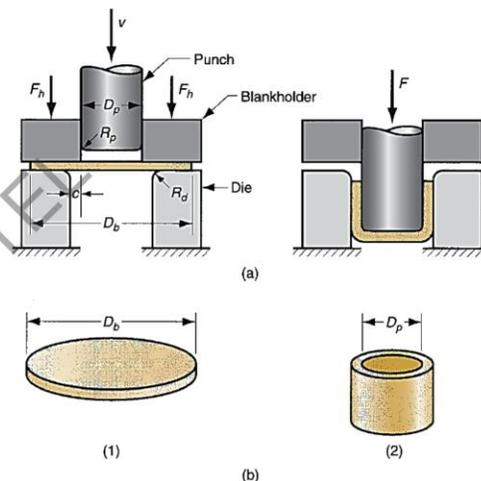
Drawing Analysis

Drawing ratio

Defined as ratio of blank diameter (D_b), to punch diameter (D_p), greater the ratio, the more severe the drawing operation.

$$DR = \frac{D_b}{D_p}$$

The limiting value of $DR (\leq 2)$ depends on punch and die corner radii, friction conditions, draw depth, and quality of the sheet metal like ductility, degree of directionality of strength properties in the metal.



The drawing ratio, which I told you, is the aspect ratio. It is defined as the ratio of blank diameter, which can be written like this: D_b/D_p . The greater the ratio, the more severe the drawing operation is. So we try to do this. Today, what is happening?

So we can also have a lot of water, which is getting filled here. And you try to press. So there is a hydrostatic pressure that is getting applied to the blank. So because of that, you get better stretching. So better stretching means a higher drawing ratio.

The limits of Dr are always equal to or less than 2. Depending on the punch and the die corner radius, friction condition, and draw depth. The quality of the sheet metal, like ductility, degree of directionality, and the strength properties in a metal. We try to have the drawing ratio. So there is something called reduction.

Reduction in the drawing can be written as blank diameter minus the punch diameter. Divided by the blank diameter, which is always less than or equal to 0.5. The thickness-to-diameter ratio is t/d . t is the thickness. The thickness-to-diameter ratio of the blank is t/D_b , which should always be greater than 1%. If it is less, the tendency of wrinkling increases. Wrinkling is a defect.

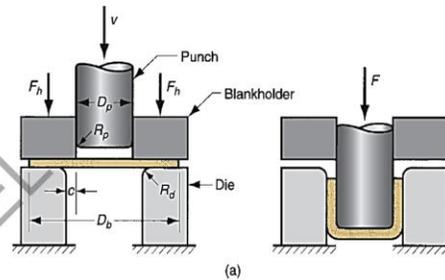
Drawing Analysis



The maximum drawing force, F , can be estimated approximately by the following equation .

$$F = \pi D_p t \sigma_{UTS} \left(\frac{D_b}{D_p} - 0.7 \right)$$

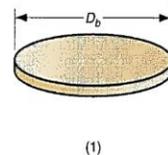
Correction factor for friction



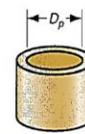
The holding force, F_h , is given by,

$$F_h = 0.015 \sigma_{ys} \pi \left\{ D_b^2 - (D_p + 2.2t + 2R_d)^2 \right\}$$

$$F_h = \frac{F}{3} \quad (\text{approx. holding force is one-third of drawing force})$$



(1)



(2)



M.P.Groover, *Fundamental of modern manufacturing Materials, Processes and system*

We will see wrinkling as we move forward. So the drawing analysis is F is

$$F = \pi D_p t \sigma_{UTS} \left(\frac{D_b}{D_p} - 0.7 \right)$$

So through this, we get the force. The holding force is the force with which the punch is moving. The holding force is

$$F_h = 0.015 \sigma_{ys} \pi \left\{ D_b^2 - (D_p + 2.2t + 2R_d)^2 \right\}$$

Approximately,

$$F_h = \frac{F}{3}$$

Operation Drawing

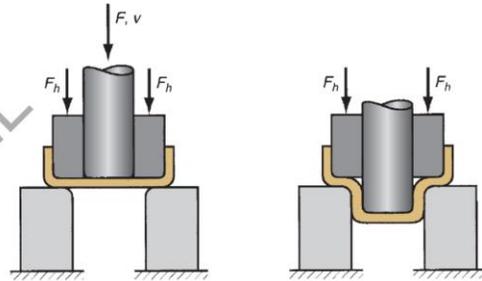


Redrawing

- In some cases (at higher drawing ratio) final shape of part requires more than one drawing step, The second drawing step, or any further drawing steps if needed, are referred to as redrawing.

For redrawing to be successful

- First draw: Maximum reduction of the starting blank - 40% to 45%
- Second draw: 30%
- Third draw : 16%



You can also have redrawing. Redrawing means you try to generate a cup. And then next, you put one more step and try to create a drawing.

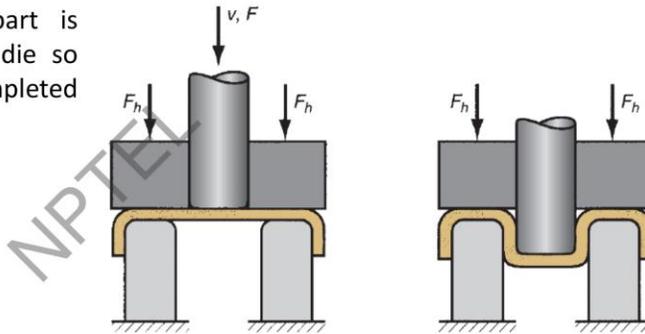
So in some cases, the final shape of the part requires more than one drawing step. First, you create this; then you create this. The second drawing step or any further drawing step, if needed, is referred to as redrawing. The maximum you can draw in one step is 40 to 45% only. In the second drawing, you can go 30%; in the third drawing, you can go 16%.

Why does the percentage of drawing go down? Because more and more strain hardening happens. Because of the strain hardening, it can go down. The percentage goes down; the maximum reduction percentage goes down.

Operation Drawing

Reverse drawing

- In this process a drawn part is positioned face down on the die so that the second drawing is completed in the direction of initial bend.



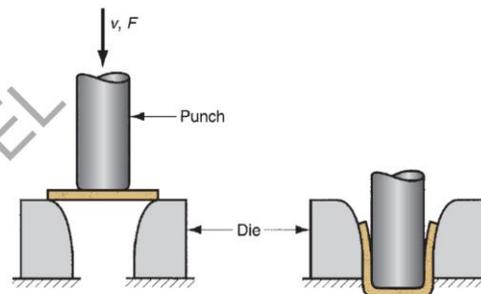
Reverse drawing is also possible. Redrawing and reverse drawing are different. Redrawing means on the same side; if you move the punch down, it is called redrawing. In reverse drawing, what we do is draw it in one direction. Then we flip it, place it on the other side, and try to push it down. So in this process, a drawn part—this is a drawn part—is positioned face down on the die.

So that the second drawing is completed in the direction of the initial bending. So initial bending is done. So redrawing and reverse drawing are different.

Operation Drawing

Drawing Without a Blank holder

- Blank holder prevent wrinkling of the flange while the cup is being drawn.
- The tendency for wrinkling is reduced as the thickness-to-diameter ratio of the blank increases.
- If the t/D_b ratio is large enough, drawing can be accomplished without a blank holder.
- The limiting condition for drawing without a blank holder-



$$D_b - D_p < 5t$$

So now let us see drawing without a blank holder. If there is no blank holder, the aspect ratio will be reduced, and thickness variation along the drawing can occur.

So that is the problem when we do it without a blank holder. Drawing without a blank holder prevents wrinkling of the flange while the cup is being drawn. See why is that wrinkling coming? We will see wrinkling later. Once there is a restriction here.

So at the moment, there is a restriction here. There is a restriction for flowing, but the material is trying to flow. There can be tearing, breaking, or there can be a loose edge forming on the top surface. So the tendency for wrinkling is reduced as the thickness-to-diameter ratio of the blank increases. The T -by- d_B ratio is large enough.

The drawing can be accomplished without a blank holder. So the limiting condition for a blank without a blank holder. Will be d_B minus d_p (diameter of the die minus diameter of the punch), which is less than 5 times the thickness.

Operation Drawing

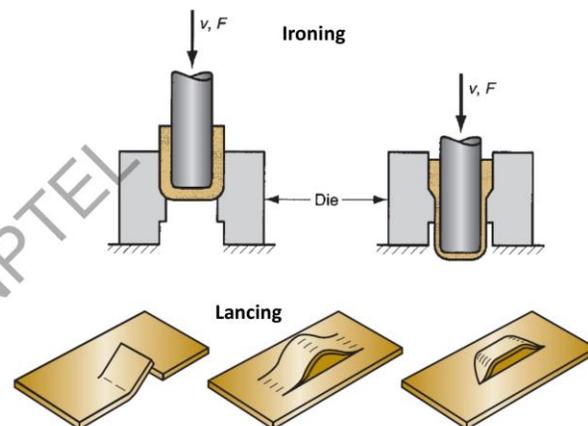


Ironing

- It makes the cylindrical cup more uniform in wall thickness.

Lancing

- Lancing is a combined cutting and bending or cutting and forming operation performed in one step to partially separate the metal from the sheet.



Ironing can also occur. Ironing means the reduction in thickness can happen. It makes the cylindrical cup more uniform in wall thickness. So when you move down in ironing, you will get a uniform thickness. Lancing is an operation that is also performed. Lancing is a combination of cutting and bending or cutting and forming operations performed in a single step. To partially separate the metal from the sheet.

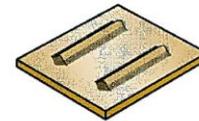
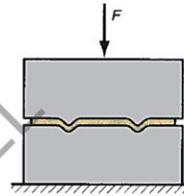
So what you do is you try to cut it, and then you try to bend it. So that is lancing. A combination of cutting and bending or cutting and forming operation in one step. To partially separate the metal from the sheet is done in lancing.

Operation Drawing



Embossing

It is a forming operation used to create indentations in the sheet, such as raised (or indented) lettering or strengthening ribs.



Embossing is basically to increase the surface area. So this was a flat plate. Now you have bumps on it. The surface area is increased. It is a forming operation used to create indentations in the sheet just by raising, lettering, or stretching ribs. So it is basically to increase the surface area.

So if it is a smooth sheet, you might slip. So if you have this embossing done, it increases the friction, and you will not fall down. So this is what I was trying to tell you about wrinkling. Wrinkling means something like making a frill around the flange. This is called as a flange surface, flange, where you have a blank when it is bending, this becomes a flange.

Drawing Defect



Wrinkling in the flange

- Drawn part consists of a series of ridges that form radially in the undrawn flange of the work part due to compressive buckling.

Tearing

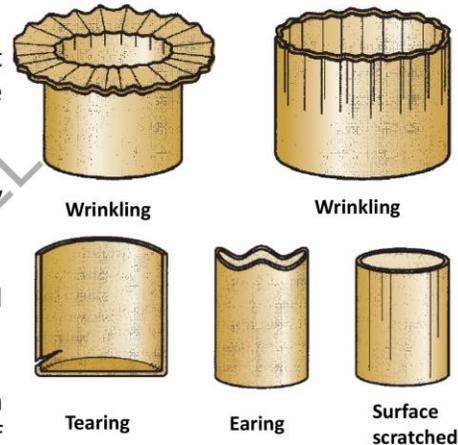
- It is an open crack in the vertical wall, usually near the base of the drawn cup.

Earing

- This is the formation of irregularities (called ears) in the upper edge of a deep drawn cup,

Surface scratche

- Surface scratches can occur on the drawn part if the punch and die are not smooth or if lubrication is insufficient.



Wrinkling in the flange, drawn part consists of a series of ridges that form rapidly in the undrawn flange of the work due to compressive buckling. So because of this wrinkling happen the material flow will not happen uniform. Tearing means when you are trying to push the punch to a far extent. And you have hit the limits or there is not much of material to flow.

So there comes the tearing. It is an opening crack in the vertical wall usually near the base of the cup is called as tearing. Yearing means here you do not have a straight on top, you will have something like a year which is created. It is a formation of irregularities in the upper edge of the blank or the flange, drip drawn cup. Then surface scratches, surface scratches are there are friction which comes into existence.

And if there is improper finish of the die. And the punch whatever it is, it tries to create scratches which increases the friction. This is the first step wrinkling. This is the second step. So you have compressive buckling happens. If you further push it down on the edge, the top edge of the cup, you will see undulations.

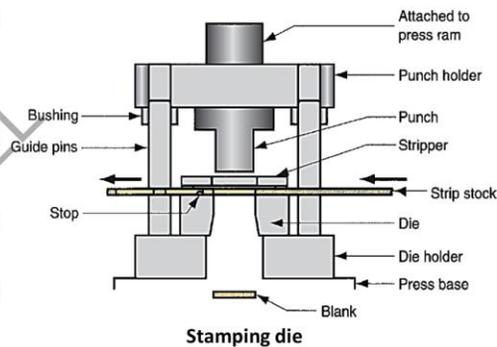
Sheet Metal Dies



- Stamping die is used for high production in sheet metal working, these stamping dies are generally made of tool steel.

Types of Stamping Dies

- **Simple die**
- **Compound dies**-Two operation at one time at one station
- **Combination dies** - Two operations at two different stations
- **Progressive dies**-Two or more operations on a sheet-metal coil at two or more stations



M.P.Groover, *Fundamental of modern manufacturing Materials, Processes and system*

Sheet metal dies, stamping dies are used for stamping. Stamping is a process; stamping means like having a stamp to create an emboss. Or having a stamp to create a hole, a blank, or a punch, right? Stamping dies are used for higher production.

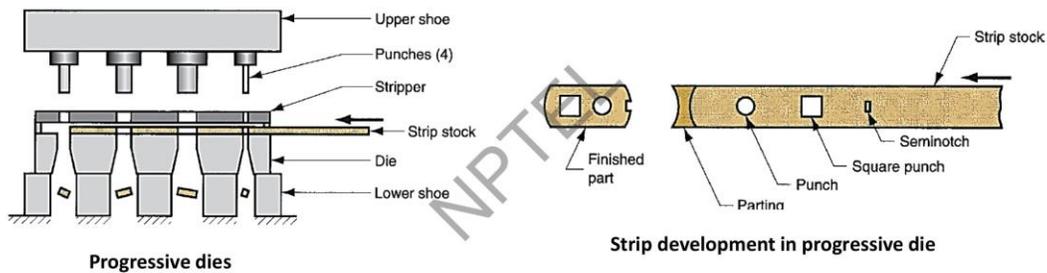
What happens is, you punch it once, then take it up, and it advances. Then you punch it, and it advances, something like that. Stamping dies are used for higher production in sheet metal working. These stamping dies are generally made out of tool steel. The stamping die has several varieties.

They are simple, compound, combination, and progressive dies. Progressive dies progress one after the other. Two or more operations on a single metal coil at two or more stations are called progressive. A simple die is simple: the punch comes down, the blank or the punch, the slug falls down—simple. If you perform two operations at one time, it is called a compound operation.

Two operations at two different stations—it is called a combined die. Compound die, combined die—two different things. Please understand the distinction between these two—very important. A combined die is two operations in two different stations: first station, second station. In the first station, you try to punch; in the next station, you try to bend. Or, in the first station, you try to cut; in the next station, you try to bend, right?

So, those things are called combination dies. Compound dies are in the same—that means to say you have a redrawing, right? So, you have a draw—first draw made. And then you have a second drawing—that is a compound die. So, these are the different types of stamping dies which are available. And they are all used for increasing the production rate.

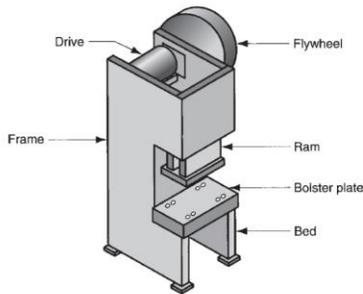
Sheet Metal Dies



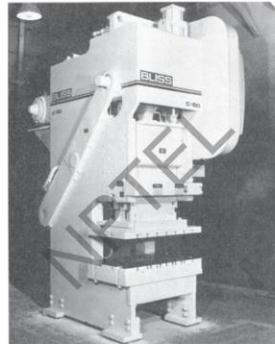
So, this is a progressive die. In a progressive die, you have an upper shoe and a lower shoe, right? And then you have a punch strip. What is a progressive die?

Two or more operations on a sheet metal coil at two or more different stations are called a progressive die. So the strip developed progressively; this is the finished part, and this is the starting part. So first you do parting, then punching, then square punch, then semi-notch. Then you will also try to make a strip notch. So these are all progressive, one after the other.

Sheet Metal Press



Stamping press components



Gap frame press



Press brake.

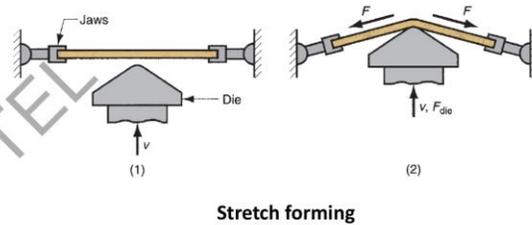


So these are the different types of stamping press component machines. There is a flywheel which stores the energy, and then there is an impact load that comes. This is a gap frame press. This is a press brake which is hydraulically operated. The other metal forming processes are stretching.

So in stretching, what we do is we try to have a blank. We try to hold the blank on both sides. Then we try to have a die which has something like a conical shape. So now when this is pushed, it tries to stretch the sheet metal. Stretching is important because you are trying to strain-harden the material.

Other Metal Forming

- Stretch forming is a sheet-metal deformation process in which the sheet metal is intentionally stretched and simultaneously bent in order to achieve shape change.
- The work part is gripped by one or more jaws on each end and then stretched and bent over a positive die containing the desired form.



Stretch forming is a sheet metal deformation process in which the sheet metal is intentionally stretched. And simultaneously bent in order to achieve the shape. So, basically, what they do is they try to stretch and bend. This is bending. So, the workpiece is gripped by one or more jaws on each end and then stretched. Then the stretching happens, then the bending happens. So, this is the stretch forming process.

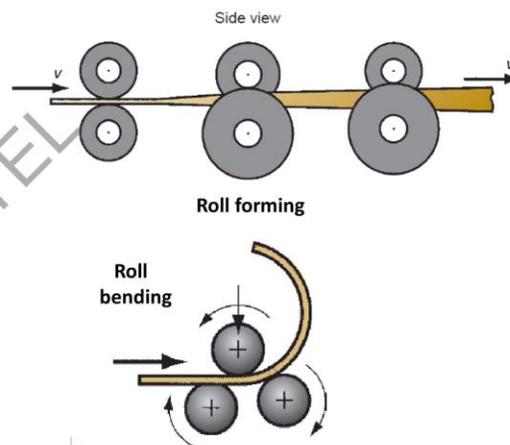
Other Metal Forming

Roll forming

- Roll forming (or contour roll forming) is a continuous bending process in which opposing rolls are used to produce long sections of formed shapes from coil or strip stock.

Roll bending

- It is an operation in which (usually) large sheet-metal or plate-metal parts are formed into curved sections by means of rolls.



The roll forming process is also there; it is a continuous bending process. In which opposing rolls are used to produce long sections of formed shapes from coil or strip. So, it starts from here, then it forms a long strip, right?

These are side views, and then you have the V, which is the form. Rolls—these are the rolls—and here, what we do is we try to give a coil or a strip. Roll bending: suppose you have a flat plate, then you have to make a shell. They use it as roll bending. For example, if you see all the HPCL, BPCL, IOCL, huge oil tanks where they store petrol, diesel, and all, they are all large diameter ones.

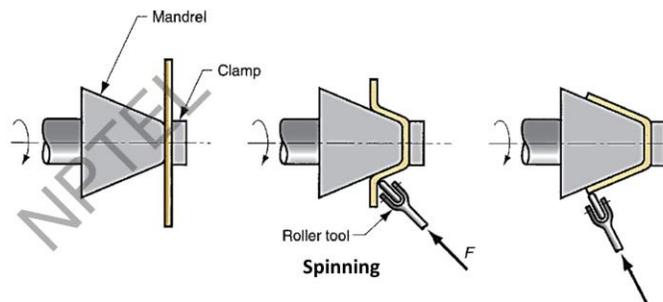
So these large diameter structures are made by roll bending. So what they do is push a sheet, and then they have two rollers, with one roller in the opposite direction. It goes back and forth multiple times, and finally, it forms a ring. A ring or a circular radius thing is formed. So, it is an operation in which large sheet metals or plate metal parts are formed into curved shapes to meet the requirements. So roll forming and roll bending are other operations used there.

Other Metal Forming



Spinning

It is a metal-forming process in which an axially symmetric part is gradually shaped over a mandrel or form by means of a rounded tool or roller. The tool or roller applies a very localized pressure (almost a point contact) to deform the work by axial and radial motions over the surface of the part.



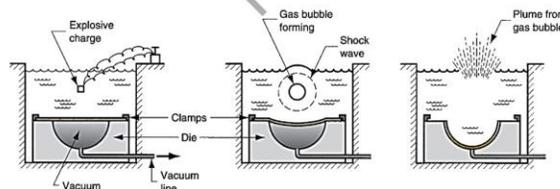
Spinning—almost all the vessels and artifacts are made using the spinning process. In the spinning process, you place a blank and then clamp it. You use a tool that pushes the blank against the mandrel. So it takes the shape of it.

It is almost like deep drawing. But here, there is no blank, and it is spun on top of a mandrel. So it is a metal forming process in which an axially symmetric part is gradually shaped over a mandrel using a rounded tool or roller. So this is a rounded tool or roller—no cutting or anything. The tool or the roll applies very localized pressure, deforming the material to meet the requirements.

Other Metal Forming

Explosive Forming

- Explosive forming involves the use of an explosive charge to form sheet (or plate) metal into a die cavity.
- The work part is clamped and sealed over the die, and a vacuum is created in the cavity beneath.
- The apparatus is then placed in a large vessel of water.
- An explosive charge is placed in the water at a certain distance above the work.
- Detonation of the charge results in a shock wave whose energy is transmitted by the water to cause rapid forming of the part into the cavity.



In explosive forming, it is very simple. We have a sheet which is there and then placed against a die. Then we have an explosive material. We try to explode it. When the explosive material is exploded, it creates a lot of shock waves.

These shock waves are used to form it against the die. Many of the dissimilar joinings of sheet metal. If you want to have bimetallic metals, we do it by explosive forming. So explosive forming involves the use of an explosive charge to form a sheet metal into a die cavity. The work part is clamped and sealed over the die, and a vacuum is created beneath.

So that when the explosive die form is there, you try to remove the vacuum, you try to get the shape. The apparatus is placed in a large vessel of water. An explosive charge is placed in the water at a certain distance from the workpiece. The detonation of the charge results in a shock wave whose energy is transmitted through water to cause rapid forming. Any complex shapes it can do, and any dissimilar material joining it can do.

So, before we do recap, let us try to do some experiments. You try to take a rod of steel and bend rod of diameter 4 mm and bend, bend it to any shape, bend. Take a wire, aluminium wire and try to bend in the same shape as 1. Whatever it is you did for rod right. Third, try to take a steel sheet, try to take a steel sheet.

And may be here wire when I said it is 1 millimeter, sheet you try to take may be again 1 millimeter. And bend as the same shape as above. So now you will see when you bend it, you bend it. And then you draw the profile on A4 sheet and leave it there for 24 hours. You will see that there can be a spring back happening.

A spring back happening. So, you will note down do the experiment for all the three sheet bending, wire bending, rod bending. And then you will try to mark. So, if you find this is too heavy for you, you can try to take 2 millimeter. Bend it and make a shape and then leave it.

So, record the springback for all these three materials. Now, look at the mechanical properties. And see if there is any relationship with respect to the bending you have done. And here, what are we interested in? To find out what is springback.

Next, we would like you to take two sheets of material—0.4 millimeter sheets of any alloy, the same. And try to do a hemming operation. Hemming, and then you also try to do locking. See how easy or difficult it is. So, if you try to do some of these experiments, you will be easily able to understand what we studied and how difficult or easy it is.

To Recapitulate



What is sheet metal process?

- Sheet metal operation
- Important term
- Other sheet metal operation
- Sheet metal analysis
- Drawing operation
- Analysis
- Defect, etc

NPTEL



So, in this lecture, what did we go through? What is the sheet metal process we saw? Various sheet metal processes: blanking, punching, shearing, cutting, drawing. These are some of the operations we saw.

Then we also saw forming. Important terminologies regarding the die were discussed. Other sheet metal operations were studied. And for two of the processes, we analyzed the metal to determine the force required for cutting, shearing, or deformation.

Then, the drawing operation, analysis of the drawing operation, and defects like wrinkling, tearing, and scratching were covered.

References



1. Hosford, W.F. and Caddell, R.M., 2011. Metal forming: mechanics and metallurgy. Cambridge university press.
2. Banabic, D., 2010. Sheet metal forming processes: constitutive modelling and numerical simulation. Springer Science & Business Media.
3. Callister Jr, W.D. and Rethwisch, D.G., 2020. Materials science and engineering: an introduction. John wiley & sons.
4. Schmid, S.R. and Kalpakjian, S., 2006. Manufacturing engineering and technology. Pearson Prentice Hall.
5. Groover, M.P., 2010. Fundamentals of modern manufacturing: materials, processes, and systems. John Wiley & Sons.
6. Rao, P.N., Manufacturing Technology Foundry, Forming and Welding, 2008.

For preparing the slides, we referred to these books.

Thank you so much.