Manufacturing Processes - I

Prof. Inderdeep Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology, Roorkee

Module - 1
Lecture - 8
Sheet Metal Operations – 2

A warm welcome to all of you in this session on Sheet Metal Operations second. Before we start our discussion on Sheet Metal Operations – 2, we would just like to review what we have discussed in the series of lectures on manufacturing processes. We started our discussion with a very important aspect of manufacturing technology that is Powder Metallurgy. We had three lectures on powdered metallurgy in which the fundamentals as well as the basic mechanisms that are prevalent in the process of powder metallurgy were discussed. After that we started our discussion on Metal Forming.

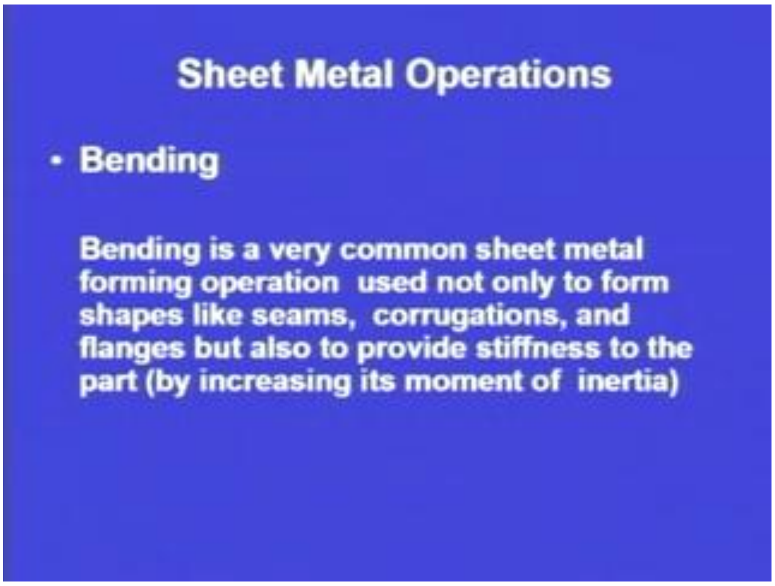
In metal forming, we had first lecture on metal forming fundamentals. In metal forming fundamentals, we discussed what is plastic deformation; how does plastic deformation takes place in metals; then we discussed the basic principles of hot, cold and warm working. Thereafter, we went into discuss the different types of metal forming processes. We discussed forging, different type of forging operations. Then we went on to discuss swaging, we discussed wire drawing. And then finally we discussed some other related operations.

Now, in this session on sheet metal operations 2, we will discuss some of the important aspects of sheet bending. Then we will discuss some of the important aspects of drawing. We have already discussed wire drawing. Today we will discuss some of the aspects of deep drawing or the drawing process used for making shallow and deep utensils. So, bending is also the one of the important aspects of sheet metal forming operations. Why because we see that most of the components all around us are made up of sheet metal.

Now, we have seen in the last of lecture or last session of sheet metal operations one, that what are the various sheet metal operations. We discussed sharing; we discussed nibbling, notching; then we discussed some of the operations like piercing. So, we have a sheet metal and then we change the shape of the sheet metal according to our requirement.

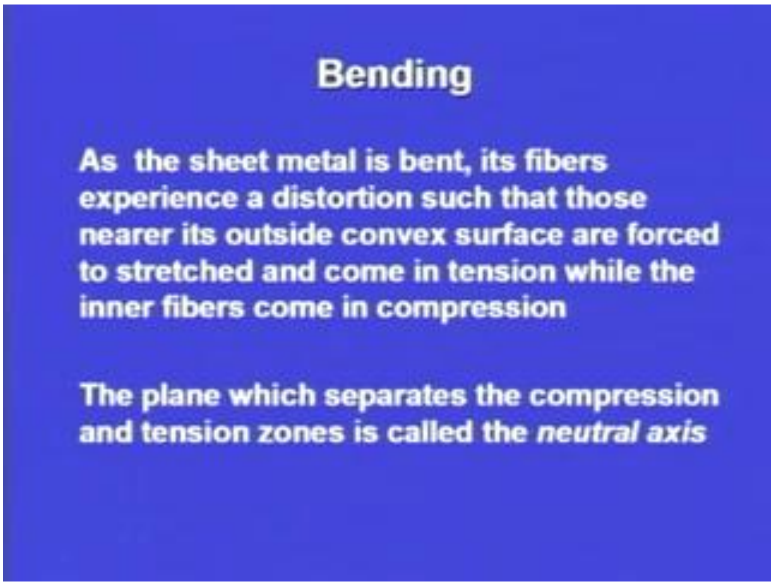
Now, in some of the applications like last lecture we discussed what are the various applications of sheet metal, though the sheet metal applications were like these can be used in automotive car bodies; then they can be used in aerospace industry; they can be used in making beverage cans; so, depending upon the application areas of sheet metal, we have to decide which process we are going to choose. And most of the times we have to bend the sheet metal in order to make it into a usable product. So, bending is an important aspect; that we will address in today's lecture.

(Refer Slide Time: 03:33)



What is bending? So, in sheet metal operations 2, we will start our discussion with bending. So, bending is a very common sheet metal forming operation used not only to form shapes like seams, corrugations and flanges, but also to provide stiffness to the part. So, we can make different type of shapes or different types of geometrical features using the process of bending. These are seams, corrugations or flanges. We can also improve the stiffness of the part. So, how the stiffness will be improved? This stiffness will be improved by increasing the moment of inertia. So, bending has certain advantages. These advantages lie in giving a particular shape to the raw material in order to convert it into a usable product.

(Refer Slide Time: 04:24)

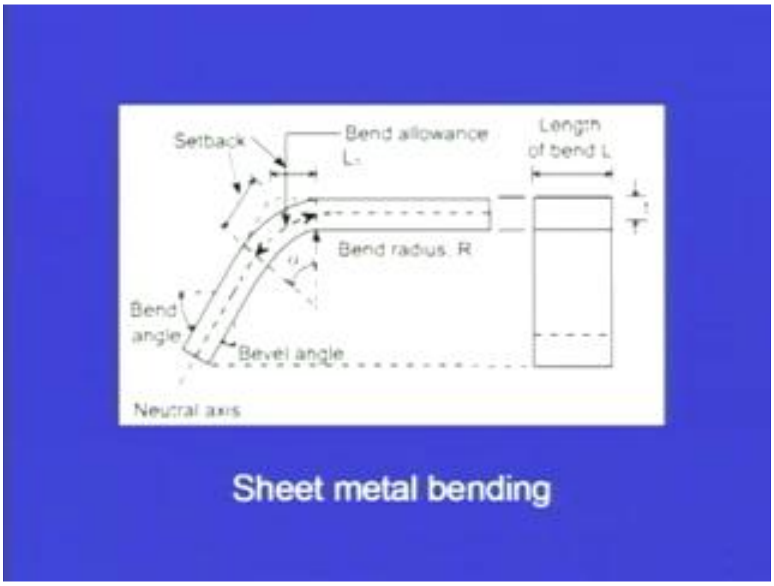


Now, as the sheet metal is bend, for example, we can take an example of this mouse pad. Now, this is the mouse pad; we can bend it; the sheet will also look like this; only there will be a blank of the sheet which can be bend. So, bending is a very common principle. As the sheet metal is bend, the fibers experience a distortion such that those nearer to the outer convex surface are forced to stretch. So, those are near the outer convex surface; they will stretch the fibers and come in tension. So, there will be a graduation from tension to compression. So, outside surface we will have a tension while the inner fibers will come in compression.

Suppose we take the example. If we want to bend it, if we want to bend it like this, this is the sheet metal. Suppose, surface of the sheet if we bend it like this, so, the outside fibers art the outer side. This will be in tension and the inner fibers at this particular surface will be in compression.

So, just to have the review, as the sheet metal is bend, its fiber experiences a distortion such that those nearer its outside convex surface are forced to stretch and come in tension. So, outside surface stretched and come in tension, while the inner fibers come in compression. So, the there are there is a graduation from compression to tension. So, the plane which separates the compression and tension zones is called the neutral axis. So, whenever a sheet metal is bend, there will be a neutral axis and there will be a zone where the fibers are in compression and there will be a zone where the fibers are in tension. So, in bending, we have different types of stress states that will develope in the sheet metal and those will be separated by the neutral axis.

(Refer Slide Time: 06:26)



Now, this is a very simple diagram that explains the principle of bending. We can see this is the sheet metal that is being bend. Two views have been given. This is the side view here. We can see, this is the thickness of the sheet which has been depicted by D and this is the length of the bend that is depicted by L. So, the length of bend is L which will be, if we enter into the plane this will be perpendicular to this plane in which the figure has been depicted. If we go inside, we will go by a length L and the thickness here is depicted by T. Now, this is the bend allowance.

This is the set back that has been if we take it straight line, draw straight line along this edge and a straight line along this edge, so, this is given. This length will depict the set back and we see that there is a bend radius that has been given. That is depicted by capital R. So, this is the bend radius, this arrow. Starting from this origin, we go to this particular point. This is giving the bend radius and there is a bend angle that is given by, that is being given by alpha. So, bend angle can depicted here also. So, there is another angle that is the bevel angle that has been provided. And as we have already discussed, there is a graduation from compression to tensile or from tensile to compression, and on both side of a neutral axis we will have compression and tensile state of stress.

So, where is the neutral axis? The neutral axis has also been depicted in this sheet metal. So, this dotted line, we can see this is the dotted line; you can see on your screen; this dotted line which is running in between the sheet metal depicts the neutral axis. So, all these various parameters like the bend allowance, the bend radius, the bend angle, all these particular parameters will influence the quality or the quality that we are going to achieve when we bend a sheet metal.

Now, this length of the bend will vary. It will it will tapper like this at the point where we are bending the sheet metal and this zone the length will be more and this L will be more at this particular section and L will be less at this particular section. Suppose we call this section as aa. So, the value of L will be more at the section aa and if we call this section as bb, the length will be comparatively less as compared to this section.

So, we can observe this phenomenon If we have a rubber eraser that we use to erase when we write with a pencil. If we try to bend a rubber eraser, we will see that at the point of bend the length will be more and away from the point of bend the length will be less. This can be attributed to the Poisson's ratio. That is one of the important material aspects. So, this value of L will be varying; this will be more in this section and it will be less at this particular section. So, the attribution, already I have told this is because of the Poisson's ratio. This is the basic diagram of sheet metal bending.

(Refer Slide Time: 09:58)

Now, we may see that if I again take the example of this mouse pad, if I bend it like this and leave it, again it is going to come to its original shape. So, if I bend it like this, again we see it has come to its original shape. So, this particular action, in case of metals when we bend a sheet metal and there is a springback action, that is called a springback or if there is a elastic recovery that is called a springback. So, what basically is springback?

When the load is removed, actually I am applying a load on the sheet or this mouse pad. If I take a metallic sheet and apply some load, I am applying a load on the sheet metal. When the load is removed, the plastic deformation is followed by elastic recovery. Although when I applied a load, there is going to be a certain amount of plastic deformation. How that will take place? Why that will take place? That already we have covered in our first lecture on metal forming fundamentals, in which we saw the slip plane theory, why the slip planes are formed, what are the grain deformation, grain fragmentation; all that was discussed in our first lecture.

So, whenever a load will be applied on a sheet metal, there is bound to be a plastic deformation and after the plastic deformation there is bound to be a elastic recovery. So, sometimes this elastic recovery changes the shape. Whatever shape we desire we will not able to get the accurate shape because of this elastic recovery that takes place. So, when the load is removed, the plastic deformation is followed by the elastic recovery. In bending, when we bend a sheet metal, this is called a springback action or this is called a springback. After springback, the final bend radius is larger and the final bend angle is smaller.

So, whenever we plastically deform a sheet metal and it goes, it has some elastic recovery or some springback takes place, the final bend radius will be larger and the final bend angle, after the springback this is the discussion, that is after the springback the final bend radius will become larger, and the final bend angle will become smaller which is not desirable.

We want to give a particular shape to the sheet metal. We are going to bend it. We will see how sheet metal can be bent in the subsequent slide, but we want to give a desirable shape to the sheet metal, but sometime it is possible that springback may take place and we may not be able to get the desired shape. So, we have to check this action of springback. Springback is not only found in case of flat sheet and plates, but it also found in rods wires or bars.

So, I have given an example of a sheet. If we take a sheet, we bend it; there is going to be a springback action, but it is not a common phenomenon in case of sheet metals or this is not a common phenomenon in case of plates only. This is a common phenomenon if we try to bend a rod. You take a rod of any metal and try to bend it. When you leave, when you leave your hands or when you leave it free, it is going to slightly open. So, that is a springback action. So, the springback is not only confined to sheets and plates, it will take place in case of rods, it can take, it will take place in case of wires as well as in bars. So, springback is basically elastic recovery after the plastic deformation has taken place because each and every metal has a finite modulus of elasticity. So, depending on that, there is bound to be a springback that is going to take place.

So, springback can either positive or it can be in some cases in case of some metals, it can be negative also. So, it depends that under what conditions we are trying to bend. So, depending upon the conditions and the requirements it will be seen that either the plastic recovery sorry the elastic recovery will be positive or the elastic recovery will be negative. Thereby, we can say that springback, either it will be positive or the springback can be even be negative. So, now we have to device some ways in which we can avoid this phenomenon of springback. So, how we can avoid springback, we will see in the subsequent slide.

(Refer Slide Time: 14:18)

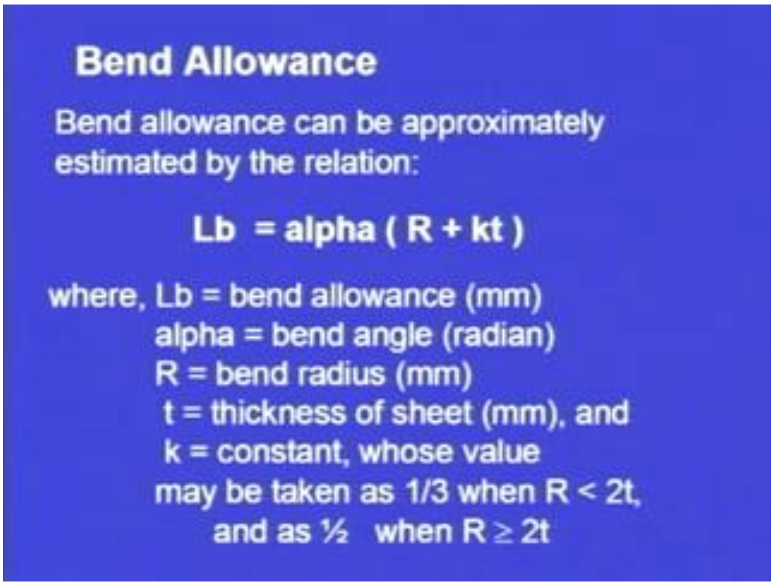
Now, compensation for springback. Springback definitely takes place when we to try to bend a metal sheet or a plate or a rod or a wire or a bar. So, it depends that how we can compensate for the shrinkage. So, first particular remedy for springback is that usually compensated for by over bending the part. Suppose we want to bend it to a particular angle, we will try to bend it to a larger angle so that it will come down to its shape that is desirable. So, over bending is one of the remedies for the springback. Then, stretch bending.

What is stretch bending? In stretch bending, the part that is subjective to tension while being bent also counters the springback. So, another way is the stretch bending in which the sheet metal which we are bending is also under the influence of tension. So, we are having a tension and then we are trying to bend it. So, that will also counter the effect of springback.

Similarly, as we have discussed that we can work either at the room temperature or we can work at the elevated temperature. So, bending may also be carried out at elevated temperatures to reduce the springback. So, whenever we work at a elevated temperature, the effect of springback will be reduced.

Another important point to note here is that the first point where we have addressed that over bending is one of the remedies for springback, but how much to over bend? I say that we want that the sheet to be bend at this angle, one particular angle. So, we say that after it cope after the bending has been done, it will open up. So, we will slightly bend it to a lesser angle. So, that it opens up and we get the desirable angle, but what should be that limit? So, that limit will be found out after a lot of experimentation; that depends up on the type of material the way in which we are going to bend the dye or the punch arrangement that we are going to use. So, all that parameters will influence the over bending, how much over bending we should give so that after the springback, we get the desired quality as as well as the desired angle of the bend.

(Refer Slide Time: 16:43)

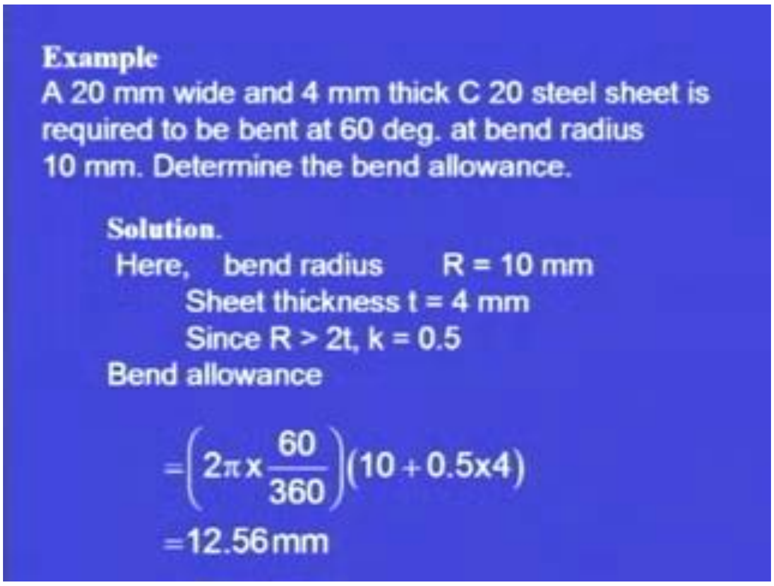


Now, we have seen that there are number of parameters. So, bend allowance is one of the parameters. So, bend allowance can be approximated by the relation. So, a mathematical relation is given for the bend allowance. So, bend allowance basically is given by Lb. It is given depicted by Lb or designated by Lb which is equal to alpha times the summation of R plus kt.

So, what are these different parameters which give us the bend allowance? These are Lb is the bend allowance, already I have told; alpha is the bend angle in radians. So, R is the bend radius, t is the thickness of the sheet in millimeters, and k is the constant whose value may be taken as 1 by 3 when R is less than 2t. When bend radius is less than 2 times the thickness, then we can take the value of k as 1 by 3, and it can be taken as 1 by 2 when R is greater than 2t. So, R and t are two important parameters. When R is less than two times the thickness, we can go for k the value of k as 1.3 and when R is greater than two times the thickness of the sheet metal, we can take the value of k as 1.2 sorry 1 by 2, that is 0.5.

So, depending if we know that what is the bend angle in alpha, that is in radians, if we know the bend angle, we know the bend radius that has to be provided. Then thickness of the sheet obviously will be known to us. And depending upon the value of bend radius and the thickness of the sheet, we can calculate what should be the value of the k that we have to put in our equation. We can very easily calculate the bend allowance.

(Refer Slide Time: 18:30)



So, now we take an example. The example states this is the problem; it is a numerical problem; we can solve it and we can see that how the bend allowance can be calculated. The mathematical formula we have seen in the previous slide. How that formula can be used to calculate the bend allowance that will be seen in this example. So, the problem states that at 20 millimeter wide and 4 millimeter thick C 20 steel sheet.

So, three important parameters has been specified in the first sentence itself. That is a 20 millimeter wide; that is giving the width of the sheet metal. 4 millimeter thick gives the thickness of the sheet metal and C 20 steel sheet and C 20 steel means the material of the sheet has been specified, is required to be bent at 60 degree. So, the angle is also given at bend radius of 10 millimeter. So, the value of R is also given. So, R states here that bend radius is 10 millimeter. So, we have to calculate bend allowance.

So, in this problem, we see that what is given. Bend radius is given that is 10 millimeter; capital R; sheet thickness small t that is given 4 millimeter. Then, since we can here see that R is greater than 2 times the t, so, k will be taken as 0.5.

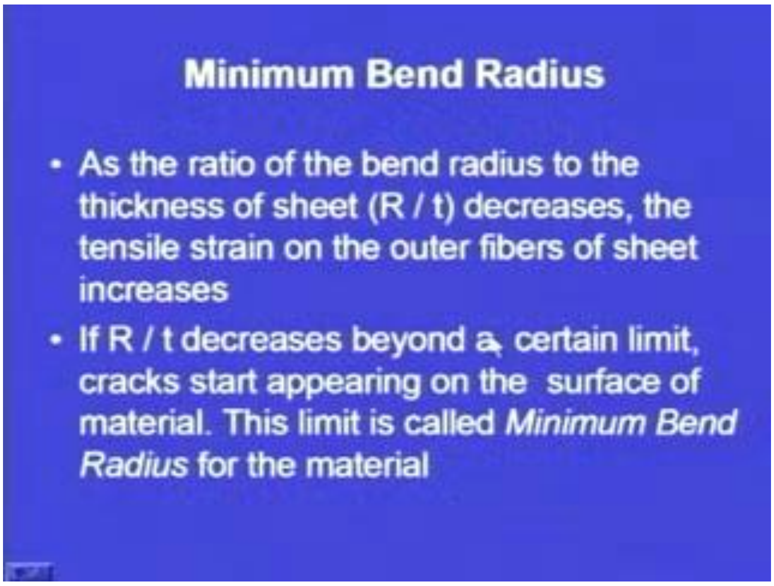
In the previous slide, we have seen that when R is greater than 2 times t, we have to take the value of k as 1 by 2 which has been written here as 0.5. So, knowing the bend radius which has already been provided in the example, knowing the sheet thickness that is already given as 10 sorry 4 millimeter, calculating the value of k from the formula that is given R is greater than 2 by t, k will be 0.5, R is greater than 2 t; k is 0.5. So, bend allowance can be calculated.

This is 2 pi multiplied by 60 by 360. Why 60 by 360? 2 pi 60 by 360 because in the previous slide, we can see that the bend angle should be in this if we want to use this particular mathematical equation. So, the bend angle that is alpha, this should be in radians, but we see that in our problem the angle is given in degrees. So, we have 60 degrees. So, in order to convert the 60 degrees into radians, we are using this particular equation 2 pi 60 by 360. So, 2 pi 60 by 360 will give us the angle in radians into 10 plus 0.5 into 4. So, in our equation, 0.5 is the value of k; 4 is the value of sheet thickness; 10 is the value of bend radius. So, the formula is alpha into summation of R plus kt. Using this equation, we get alpha into the value of R plus k into t we get the 12.56 millimeter.

So, using this mathematical formula, we can very easily calculate that what is going to be the bend allowance. So, this has determined the bend allowance. We have calculated what is going to be the bend allowance.

So, similar type of mathematical formulations can be there in which we can calculate some of the independent, we we can use some of the parameters and we can calculate some of the other dependent parameters. So, independent parameters can be used and we can calculate some of the parameters that influence the final quality of the product that we are making using the sheet metal.

(Refer Slide Time: 22:17)



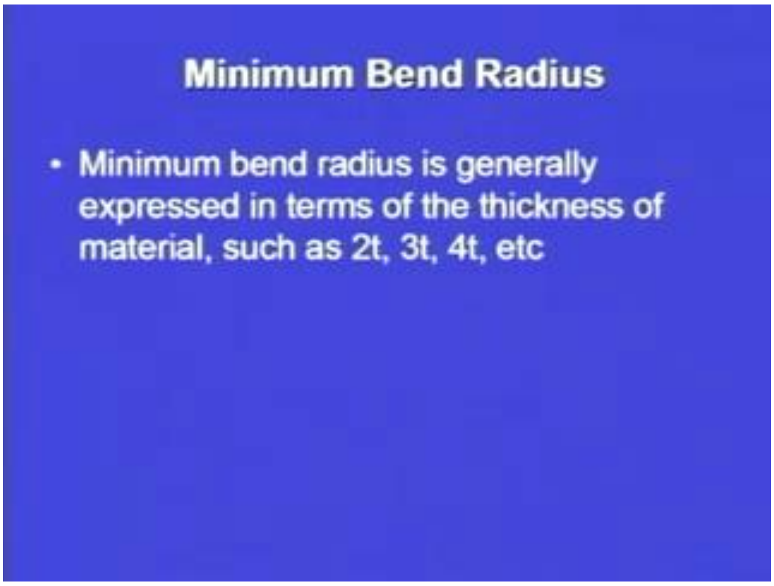
Now, we come on to minimum bend radius. So, what is minimum bend radius as the ratio of the bend radius to the thickness of sheet? So, bend radius is already designated as capital R as the ratio of the bend radius to the thickness of the sheet; the thickness of the sheet has been designated by small t; so, the ratio of bend radius to the sheet thickness that is R by t decreases. So, when this ratio will decrease, the tensile strain on the outer fibers of the sheet will increase. So, whenever this R by t ratio decreases, that is the bend radius to sheet thickness ratio is decreasing, the outer fibers will be having more tensile strain. The tensile strain on the outer fiber of the sheet increases. So, when this ratio R by t, this decreases beyond a certain limit, so, there is a limiting factor.

When this ratio of R by t will decrease beyond our limiting factor or beyond a limiting point, cracks start to appear on the surface of the material. So, whenever we are bending a sheet metal, we do not want any cracks on the surface. If any cracks appear then these small small cracks may further propagate under the surface. So, whenever we are making any product of a sheet metal, it is not that we are directly going to use it in any application where this cracks may not propagate.

So, there are number of instances where whenever under service the small cracks they will propagate and they will meet each other and form a bigger crack, and then this bigger crack may further result into the fracturing of the surface or may even result into a catastrophic failure. So, no particular micro cracks or cracks are desired on the surface.

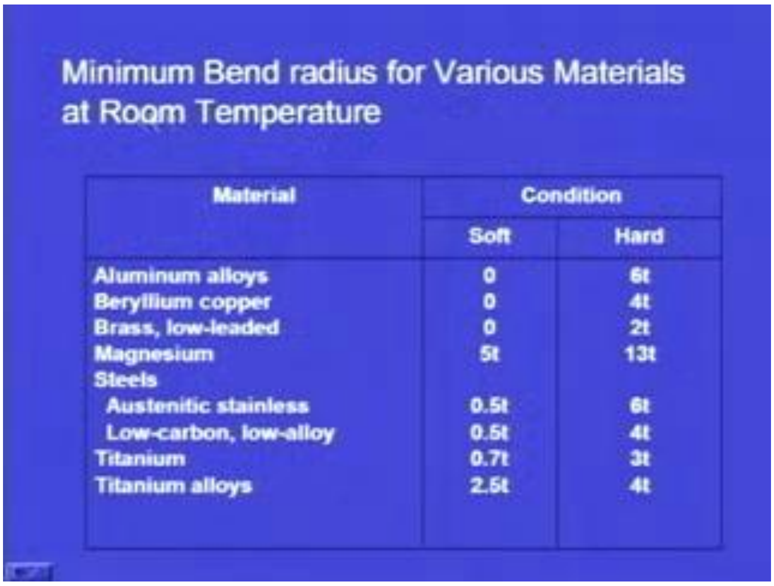
So, whenever this R by t ratio decreases beyond a certain limit, cracks that start appearing on the surface of the material. This limit of the bend radius that is R by t value is called the minimum bend radius of the material. So, at this particular point, wherever the cracks start to appear on the surface is called the minimum bend radius. So, minimum bend radius is generally expressed in terms of the thickness of the material or the thickness of the sheet.

(Refer Slide Time: 24:30)



So, that is bend radius whenever we have to specify, we will specify in terms of thickness of the sheet. So, this thickness of the sheet can be for example, we can say 2t, 3t, 4t 5t. So, depending upon this particular value, we can say that what is going to be the minimum bend radius for that particular material.

(Refer Slide Time: 24:56)


Minimum Bend radius for Various Materials at Room Temperature

Material	Condition	
	Soft	Hard
Aluminum alloys	0	6t
Beryllium copper	0	4t
Brass, low-leaded	0	2t
Magnesium	5t	13t
Steels		
Austenitic stainless	0.5t	6t
Low-carbon, low-alloy	0.5t	4t
Titanium	0.7t	3t
Titanium alloys	2.5t	4t

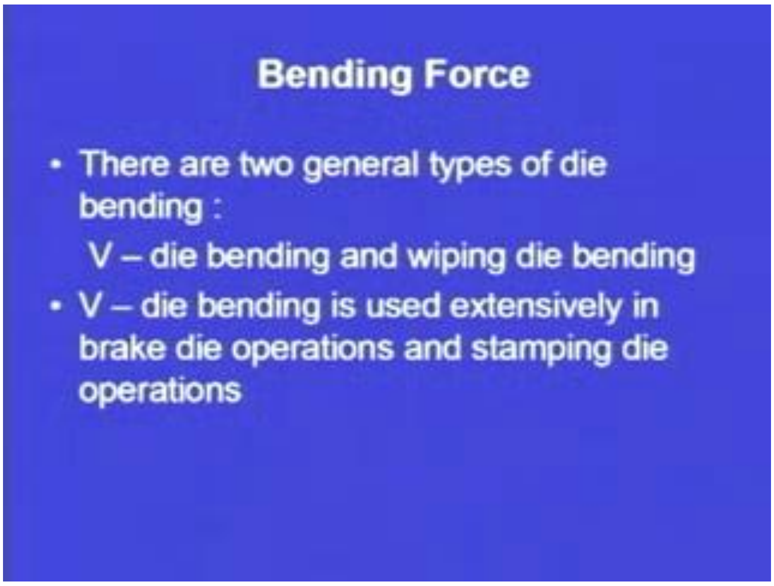
Now, on your screen, you can see a table that is minimum bend radius for various materials at room temperature. So, whenever we work at an elevated temperature, this value may change. So, whenever aluminum alloys we are using, there is for soft condition there is no minimum bend radius; for hard condition it is six times the thickness.

For example Beryllium Copper, for soft condition there is no limit, but hard conditions it is 4 times the value of thickness. If we take examples of Magnesium, for soft it is 5 times t minimum bend radius and in hard it is 13 times t. Similarly, Steels Austenitic steels and low-carbon, low-alloy steels, all depending upon we can see the values are changing for soft also, for hard also. So, we can see that for depending upon the material, the minimum bend radius has been specified.

So, we do not have a generalized bend radius value. It depends upon the type of the materials. So, cracks will start to appear on a sheet metal when it is being bent and the minimum bend radius depends upon the material of the sheet metal. For example, if it is aluminum, the minimum bend radius will be different.

If we are going to you bend a sheet metal of Titanium or Titanium alloys, the minimum bend radius is going to be different. So, bend radius is an important parameter and it depends upon the material out of which we are going to make our final product using the sheet metal forming operation of bending.

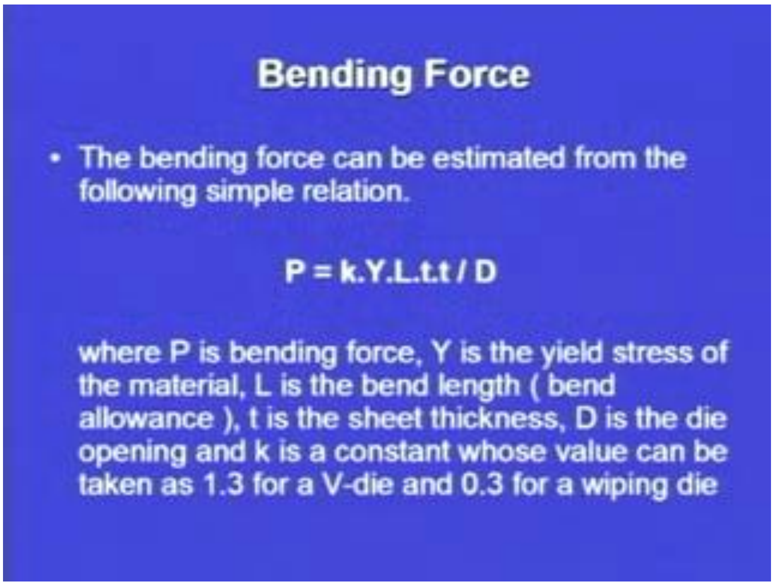
(Refer Slide Time: 26:30)



So, now, we come on to the bending force. We require some force to change the shape of the sheet or in order to bend the sheet according to our desired level or desired specifications. So, there are two types of die bending. So, the force depends upon the die bending. So, V bending and wiping die bending. So, there are two types of die bending processed are possible: first one is the V die bending and second one is the wiping die bending.

So, V die bending is used extensively in brake die operation and stamping die operations. So, wherever we have stamping die operations, we are going to make use of the V die bending. So, we will see in the diagram that what is basically V die bending. Now, the bending force we have seen, already seen that bending force is required. suppose we have a sheet metal, we have to bend it. So, some force is required in order to create that bend.

(Refer Slide Time: 27:22)



So, minimum bend radius already we have seen; it has to be specified and it depends upon the material there for which we are going to form the bend or of which the sheet is basically made of. So, this force required for bending is given by this equation. The bending force can be estimated from the following simple relation.

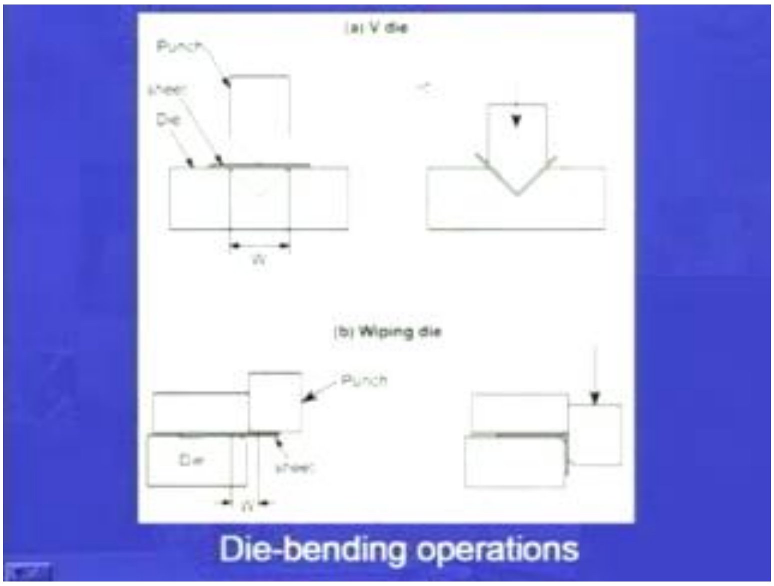
So, this is if P is the bending force, this depends on this can be represented mathematically as P is equal to k times Y times L times t square divided by D. So, again, I will once again go through this mathematical relation; that is P is equal to k multiplied by Y multiplied by L multiplied by t square whole divided by D.

So, what are these individual parameters that finally form a function for the bending force? These are: Y is the yield stress of the material; all of us know what is the yield stress of a material; L is the bend length; already in the first diagram where we have seen what is the bending operation and what are the various parameters in bending. We have seen what is the bend length, bend allowance; t is the sheet thickness, thickness of the sheet which was depicted by small t there; D is the die opening.

So, die opening gives like this. If this is a V type of a die die opening is given by D and then k is the constant whose value can be taken as 1.3 of V die and 0.3 for a wiping die. So, here also we have a constant which is given by k and it will take a value of 1.3 for V die and 0.3 for a wiping die.

So, using this very simple mathematical equation, we are able to calculate the bending force. We can see that the bending force here is a function of a yield stress of the material; that is given by Y. Then, it depends upon the length; that is the bend length or the bend allowance and it also depends upon the thickness of the sheet. So, we can see, it depends if the thick thickness is more or thickness is less. It will have a influence on the bending force that is required to create that bend or to make that bend.

(Refer Slide Time: 29:51)



Now, this is the diagram where we can see a V type of die is there; just to elaborate, what are the labels that have been given, what is the labeling of this diagram. This is the punch; you can see this is the punch; this is the sheet; the solid portion, this is the sheet. And this is the die. This is the die and this is w; that is the die opening in our equation. We have written it as D the die opening, but here in this we have specified it by w; this is the die opening w. So, here the punch will come and it will exert a force on this sheet metal which has been given here and it will form it into a V type of sheet.

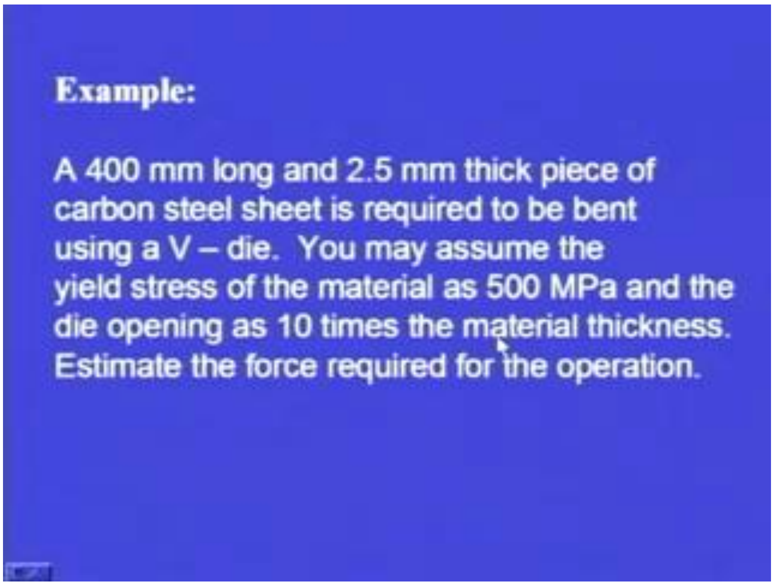
So, this is the V; this is giving the direction in which the punch is going to move. Here, we can see the punch is coming down; the arrow signifies the motion of the punch. The punch is coming down and depending upon the shape of the punch, here it is a V and exact replication is there in the V type of die. The punch is coming down and it is forcing the sheet metal to get into a V shape. This is the final shape that we are going to get when we are going to use a V type of a die.

Then, in this particular case, we are going to take the value of constant k while calculating the bending force. When we calculate the bending force using this mathematical relation, when we are using V type of die, the value of this k that comes into the equation will be taken as 1.3.

Similarly, there is another example. This is the wiping die, the second type of a die. Here we see. This is the die; this is the sheet, the solid sheet which we have to bend; this is the punch and this is the w. Value of w or this we calculate. This we can call as the D in our mathematical equation. The punch will come down and this will force a bend on the die. So, you can see that a radius, small radius is given here on the die. So, in case of a wiping die, we have this is just a spotting block for the sheet metal; this is a die; this is a punch; this arrow depicts the motion of the punch. The motion the punch was initially here. When it comes down, it forces the sheet metal to bend. Initially, we had a straight sheet metal like this the shape was flat. Then we have been able to create a bend using a wiping die. So, whenever we use this equation, whenever we are going to use a wiping die, the value of k will be taken as 0.3.

Now, depending upon the type of the die, either the V die or the wiping die, we will carefully select the value of k for calculating the bending force. So, these are some of the die bending operations. So, we have seen what are the various types of wiping die as well as the V die. We have seen various types of die, that is V die and the wiping die, and the various types of die bending operations.

(Refer Slide Time: 33:12)

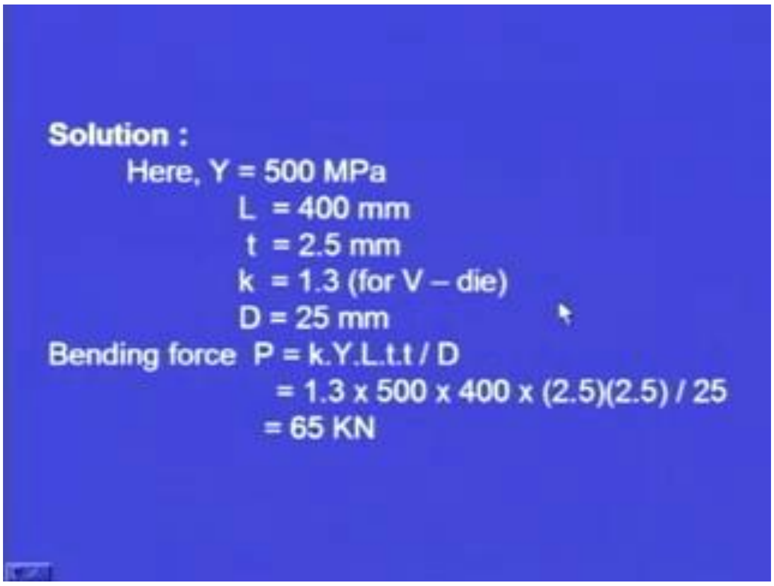


Now, we take a example. In this example, we will calculate the force required for the operation. So, what is a problem statement or what is the example we are going to take? This is a 400 millimeter long. So, the length is specified and a 2.5 millimeter thick piece. So, 2.5 millimeter thick piece means that the thickness of sheet of which we are going to create a bend, in which we are going to create a bend, is 2.5 millimeter of Carbon steel sheets. So, Carbon steel sheet will give us the material property that comes into our mathematical formula to calculate the final bending force.

What was that? That was designated by Y is required to bend using a V die. So, a 400 millimeter long and 2.5 millimeter thick piece of Carbon steel sheet is required to be bent using a V die. Now, the thickness is given, length is given, material is given, and the type of the die is also given. You may assume the yield stress of the material that is designated by Y as 500 mega Pascal and the die opening as how much is going to be die opening, that was shown in the diagram by capital W, as 10 times the material thickness.

So, what is the material thickness? it is 2.5. So, what is going to be the die opening? It is 10 into 2.5. Now, we have to estimate the force required for the operation. Now, the force requirement we have to calculate and the angle may be 90 degree. So, what are the things that have already been given to us? That is the yield stress of the material. What is the yield stress of the material? That is 500 mega Pascal.

(Refer Slide Time: 35:11)

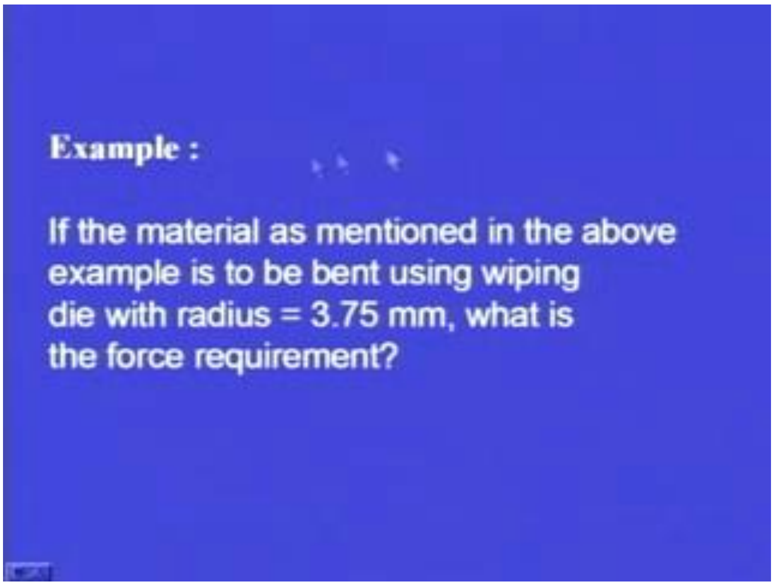


So, yield stress is designated by Y that is 500 mega Pascal. Then the length is given as 400 millimeters. Thickness is given as 2.5 millimeter; that has already been given in the example. Then we have been given that we are going to perform this bending operation using a V type of die. So, the value of k for V type of die is 1.3. Then, we are going to calculate what is going to be the die opening which we have depicted by D or we have designated by D. So, the die opening we can see in the problem is the 10 times the material thickness. So, material thickness is 2.5. So, the value of D comes out to be 2.5 into 10 that is 25 millimeter. And we are making a bend at 90 degree. So, the bending force that is given by the mathematical formula that is P is equal to k multiplied by Y multiplied by L multiplied by t square, and whole divided by D.

So, what is the value of k here? k is the for V die it is 1.3; so, the value of k is 1.3 yield stress; that is the value of yield stress. The yield stress of the material is 500 mega Pascal. So, Y is 500 mega Pascal. So, Y is given 500 mega Pascal; length is given as 400 millimeter that is L; in our problem, you can see a 400 millimeter long sheet; so, the length is given as 400 millimeter; then L is 400 millimeter and then the thickness that is 2.5 millimeter. 2.5 millimeter multiplied by 2.5 millimeter and divided by the die opening. Die opening is 10 times the material thickness that is 2.5 times 10 or 10 times 2.5 that comes out to be, either way it comes out to be 25.

So, we calculate, we get the value that is 65 kilo Newton. So, depending upon the problem where we have 3,4 important parameters, that have already been specified to us. We can put the values in a simple mathematical formula and calculate the bending force. The bending force will also depend upon the angle. Like here we say that it is 90 degree; it is to be bent at 90 degree. So, we are using this formulation. Say if suppose there is some other angle, then the formula may be modified or the same formula may be used with some other formulation.

(Refer Slide Time: 37:54)

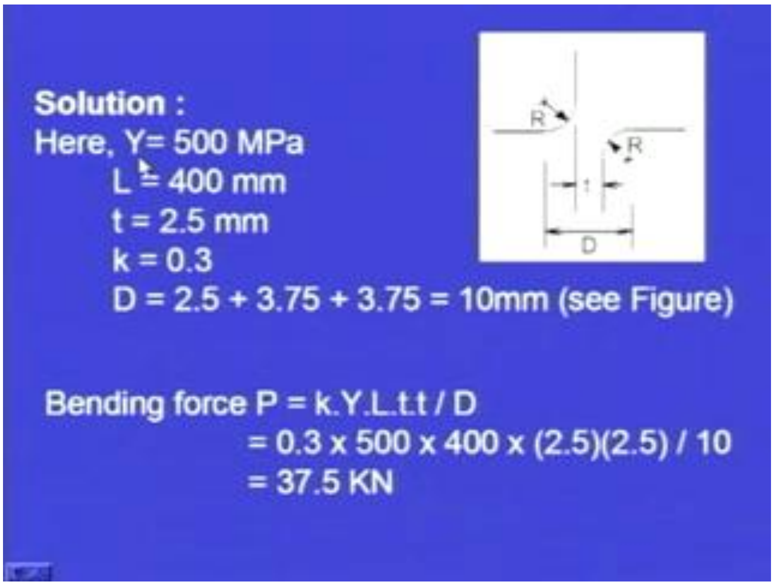


Similarly, we take another example. In the second example, what is stated? If the material as mentioned in the above example, the material here also is same. It means when the material is same the value of Y which was taken as 500 mega Pascal will be same in this particular case also. If the material as mentioned in the above example is to be bent using wiping die and a radius of 3.75 millimeter, what is the force requirement?

So, we have seen in die bending operations, we have two types of dies which are basically used: first one was the V type of a die; second one was a wiping type of a die. So, with first type of die that is V type of die, creating a bend at 90 degree we have seen that what is going to be the force requirement. When the length as well as the thickness, more over the die opening and the material specification that is if the yield stress is given we can very easily calculate the bending force.

Now, using a wiping type of die, if we have to calculate another important parameter that is radius 3.7 millimeter is given. So, in this particular case, how we are going to calculate the particular force requirement for the bending operation, that we will see now.

(Refer Slide Time: 39:19)



Now, here, the value of Y as the material is seen. When the material is seen, the value of Y will be taken as 500 mega Pascal. The length is also 400 millimeter. Same as above, t is 2.5. Now, we have only change that is noted here is k. In case of V type of die, what was the value of k that was chosen? It was chosen as 1.3.

In case of wiping type of die as in this particular problem, we have already been specified that this particular material has to be bent using a wiping die. So, in case of wiping die, the value of k is taken as 0.3. So, now the die opening is calculated. We can see the figure. In previous case, the die opening was given as 10 times the thickness of the material. The thickness of the material was 2.5 and 10 times means we have taken the die opening as 25 millimeter; 10 times, 10 multiplied by 2.5, but here the die opening has not been specified. So, here we see the D is equal to 2.5. What is the what is this 2.5? 2.5 multiplied sorry plus 3.75. What is the value of 3.75? This is R; this is 3.75; this is R 3.75. So, 3.75 has been given as the radius. See this radius has been given as 3.75. This is 3.75, this is 3.75. So, this length, how much is this length? This is R 3.75. This length if you take this as the center, this length is also 3.75; so, 3.75 plus 3.75 plus this t as 2.5. So, initially I was explaining that what is this 2.5; this 2.5 is the value of t.

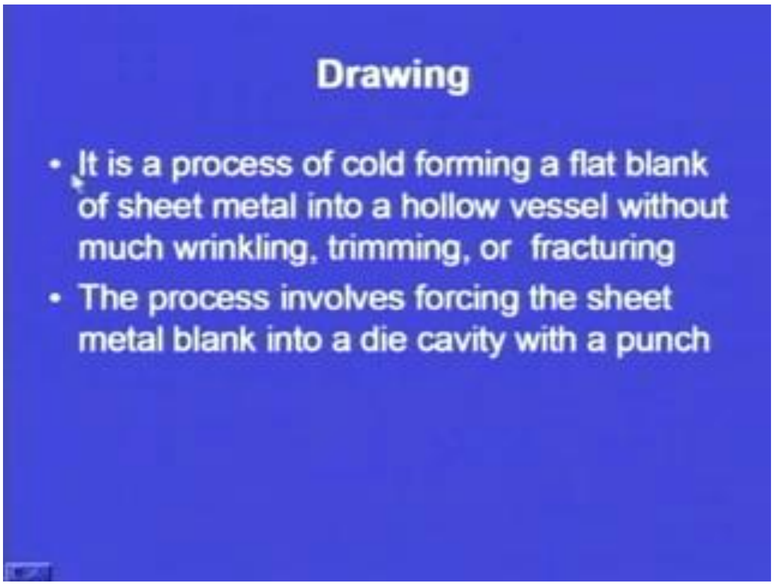
So, once again, just to repeat this is 2.5. In between, these arrows this particular length is 2.5. This particular length from this line to this line is 3.75; that is the radius that has already been provided in the example. Then this also is 3.75. So, 3.75 plus 3.75 plus this value of t this t is 2.5. So, the value of D that is the die opening total comes out to be 10 millimeter. See figure (Refer Slide Time: 42:01). According to this figure, we have been able to calculate the value of D.

So, once we know D, we know k. k for the wiping die is 0.3. We know the thickness that is 2.5. We know the length that is 400 millimeter. We know yield stress or Y as 500 mega Pascal. We are able to calculate the bending force using this formula which is given by P is equal to k times Y Lt square whole divided by D. So, the value of k is taken as 0.3; the value of Y is taken as 500 mega Pascal; the value of L is 400 millimeter; the value of t is 2.5 millimeter and the die opening which has been calculated using this equation using this figure is 10 millimeter. So, whole divided by 10 and we calculate that the bending force is going to be 37.5 kilo Newton.

So, we have seen till now, that you using a very simple mathematical formula and depending upon the die that we are using to change or to convert a one particular sheet metal into a bent sheet metal or when we create a bend in a sheet metal using die operation, we can very easily calculate the bending force. The parameters that are required here are the length of the material, the thickness, more over the type of the die that we are going to use as well as the yield stress of the material; that is given as 500 mega Pascal.

So, now we have seen that what is the basic principle of bending, why bending is required, what are the different types of dies that are used for die bending operations, what is simple formula to calculate the bending force. Then we have seen the springback action, why the springback takes place because of the elastic recovery, and what are the remedies for this springback action. So, that was all that we required to discuss in this particular section on bending operations. Now, we shift our attention to another important aspect of sheet metal forming operations that is the drawing operation. In one of our previous lectures, we already discussed wire drawing which is specifically dedicating to manufacturing of wires or rods or tubes, but here we will focus our attention or drawing of sheet metal.

(Refer Slide Time: 44:28)



So, in drawing, it is a process of cold forming a flat blank of sheet metal. So, here we have a blank of sheet metal. The example can be this. This is the like one blank. This could be made up of sheet metal. So, it is a process of cold forming a flat blank of sheet metal into a hollow vessel. So, this blank will be formed into a hollow vessel without much wrinkling, trimming, or fracturing. So, what are their limiting factors? It is very easy to say that we take a blank of sheet metal, we use the drawing operation, and we make a hollow shape out of that flat blank, but it is not so easy. It is easier said than done.

What are the limiting factors? The three limiting factors have been addressed here. There should be no wrinkling or minimum wrinkling; the wrinkles tend to appear on the edges when we draw a sheet metal into a hollow vessel. Then trimming requirement should be minimum and there should be no fracturing.

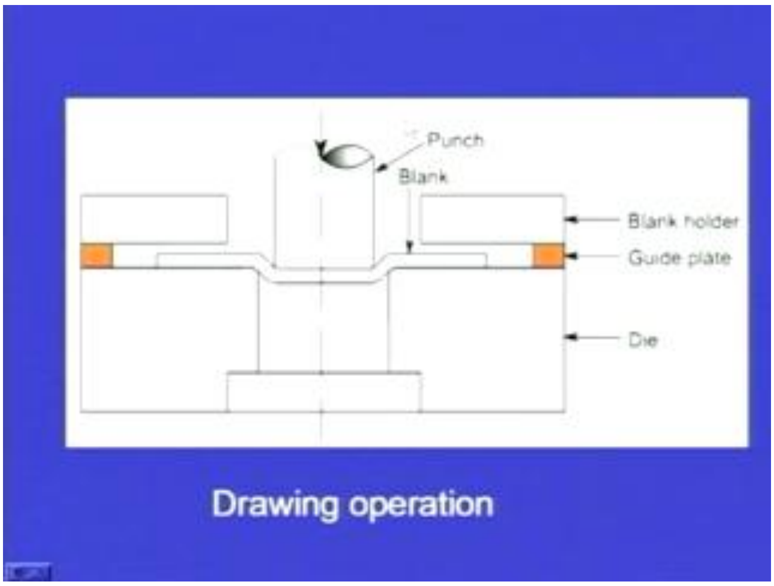
So, when we are able to change the shape of a blank into a vessel that is the drawing operation, but the limitation is that the wrinkling should not take place; more over there should be no trimming, as well as there should be no fracturing. So, the process involves forcing the sheet metal blank into a die cavity with a punch. So, this is also a die punch type of operation in which there is die which will be the exact replication of the final shape that we want to make of the sheet metal, and there will be a punch that will force the sheet metal blank inside that die cavity. So, we have seen this beverage cans that are there for soft drinks. These beverage cans are also made by the process of drawing only.

So, in case of drawing the punch exerts a sufficient force and the metal is drawn over the edge of the die opening and into the die. So, how does the operation of drawing takes place? In drawing operation, the punch, it exerts sufficient force. Sufficient force, it is important to understand that each die will have certain formidability limits.

In our subsequent lectures, we will see what are the formidably limits and what are the different types of tests like Eriksson cupping test is there; that we will discuss in detail, but sufficient force is required. The force should not be too much. If the force exceeds a particular limit, then the sheet metal will crack or the crack will start to appear on the sheet metal. So, the punch exerts a sufficient force and the metal is drawn over the edge of the die. So, the metal will go. It is drawn over the edge of die opening and this the metal will go into the die.

So, this will be explained with the help of a particular diagram and it will become very very clear and it will become crystal clear when the metal will go into the die cavity under the force of the punch. In forming a cup, however the metal goes completely into the die. Suppose we want to make a cup, then the whole blank, when it is being pressed with a punch, will go into a die cavity. So, how this operation will be performed? We will just see.

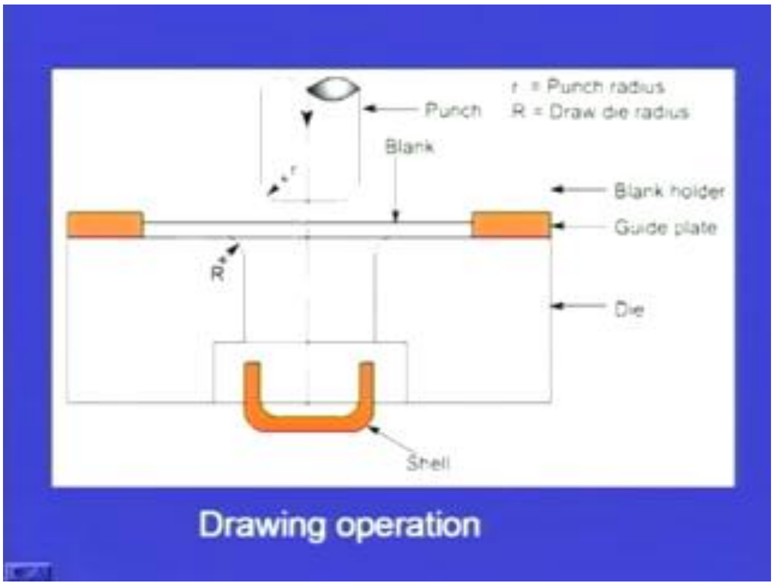
(Refer Slide Time: 48:03)



This is one of the diagrams that explains the principle of drawing. Here we can see that this is the punch; this is the direction of the punch in which the punch is going to move. Suppose we call it z direction. So, the punch will move in z direction. This is the blank; this is the flat blank out of which we want to make the particular shape. So, this is the punch; this is the blank. There is a blank holder. Why blank holder is required? That we will see. And this is the guide plate. This colored portion, this is the guide plate and this is the die.

So, whatever shape that we want to make, whatever requirement is there, the die opening or the die shape will be according to the final requirements or the final shape or the final features that we want to have in the blank that is raw material in this case. So, this is a principle of drawing operation. A punch will exert sufficient force on this blank to force it into the die cavity so that the sheet will take the shape of the die and we will get the final product.

(Refer Slide Time: 49:16)

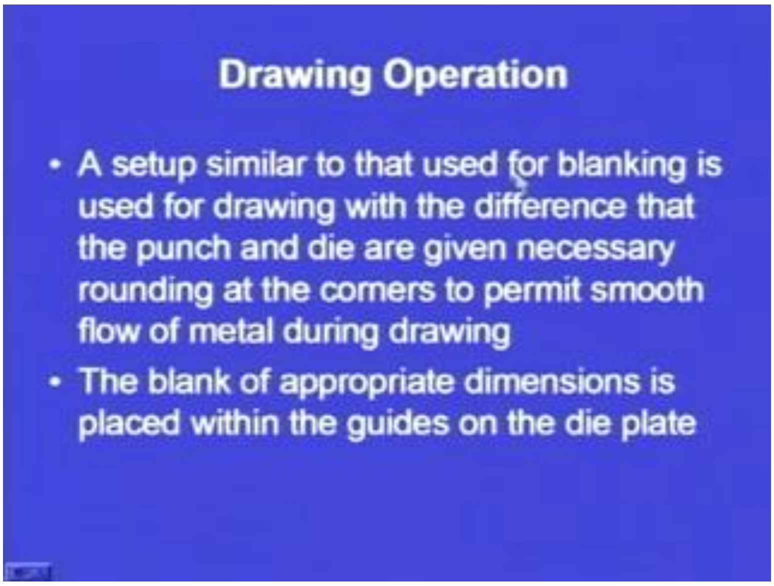


Now, in this particular diagram, we can see that this small r, this is the punch radius. This capital R, that is the draw guide radius; this is the draw die. We can see this is the draw die; the solid portion, this is the draw die. This is the guide plate; colored portion; blank holder. This is the blank and then this is the punch. So, the punch will move in this direction and we will get a shell. This is the final particular portion or the final section that we are attempting to make. So, this is the final product that is required; a cup shape of product has been made.

So, a punch will come. It has some radius here and a radius is also given on the die. So, this is the radius that has been provided on the diagram. Why that radius is provided? We will see. Whatever we are seeing in this diagram, we will just try to summarize it in the form of simple English sentences so that you can memorize or you can remember that what is the basic operations of drawing?

So, here we see there is a counter board portion also. Why the counter board portion is there? This we will see in the subsequent slides. This is the counter board portion here, this portion. As you can see, the mouse is moving; arrow is showing. This is the counter board portion. Why that counter board portion is required? We will see. This is the final product that we have made using the drawing operation.

(Refer Slide Time: 50:55)



So, now, this drawing operation we will try to summarize what we have seen in the form of the diagram. A set up similar to that used for blanking; in blanking also, this type of a set up is used. It is used for drawing with the difference that the punch and the die are given necessary rounding at the corners to permit smooth flow of metal during drawing. So, in blanking, we have not seen this type of radius. In the diagram, we have seen that there is a small r radius that is given on the punch and there is a capital R that is radius that is given on the die. So, these two radius are provided or these two radii are provided on the punch and the die in order to facilitate the smooth flow of metal during the drawing operation.

Here because the metal has go into the die cavity we need to have the smooth flow of metal. If there are edges, the metal or the sheet metal may crack. So, in order to avoid that crack, in order to facilitate the smooth flow, this particular radius is provided and it is provided on both sides. It is provided on the punch side also; it is provided on the die side also. The blank of appropriate dimension is placed within the guides on the die plate. Now, appropriate dimensions here means that we know that what is the final shape of the product that we want to make.

Now, depending upon the final shape and the final volume of metal that will be required to form that shape, we can choose the appropriate dimensions of the blank that is going to serve as the raw material for the drawing operation. So, appropriate dimensions of the blank size will be chosen for making a particular shape using the process of drawing and the operation will make use of a punch and die type of a arrangement. So, here by we come to the end of this session on sheet metal forming operations 2.

We started our discussion in this particular session with a brief review of what we have covered in our previous lectures. Then, we discussed what is the bending; bending as an important manufacturing process or in important sheet metal operation. Then, we saw what are the various parameters in bending; what is the minimum bend radius; what are the bending force requirements. Then we saw that there is a mathematical relation which can be used to calculate the bending force. What are the important parameters in that mathematical relation? That was seen. Then we solved some numerical problems for calculating the bending force.

We also saw that there are different types of die bending operations, different types of dies like V die and wiping type of die that can be used for bending operations. There after we started briefly, the discussion on drawing operations and we are in the middle of the drawing operation. We have just seen what is the basic of the drawing operation. In our next lecture, we will try to address this problem in a little bit of more detail. So, in next lecture, we will start our discussion with the drawing operation.

Thank you.