Manufacturing Guidelines of Production Design

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

Lecture-23
Design Guidelines for Sheet Metal Working

Namaskar friends, welcome to session 23 of our course on manufacturing guidelines for product design. And we are currently discussing the design guidelines for specific processes and what all processes we have already covered. Let us just have a quick relook on those processes, we have covered the design guidelines for sand casting process as well as for die casting process, we have also covered the design guidelines for some of the plastic moulding processes.

The common moulding processes being the compression moulding, extrusion and injection moulding. In session number 22 we have covered the design guidelines for injection moulding and extrusion and if you remember for moulding the standard guidelines were in context of or the design attributes or design requirements were based on the wall thickness, based on the rib thickness, the height of the ribs as well as we have seen the spacing between the ribs.

The number of design attributes were there and once we are designing a plastic part all these important aspects help us to design our part in a more manufacturable manner. So, when once we are designing a part a lot of imagination a lot of creativity, a lot innovative ideas may come to us. But once we put them on a piece of paper we must have a review of those ideas based on the manufacturability of the product.

Many times the ideas may look very fascinating but from the manufacturing point of view they maybe complete disasters maybe the idea is good, the shape is good, but we cannot manufacture it why because of the limitations of the various manufacturing processes. So, therefore it is very important to look at the established design guidelines for the various processes, so that the product is designed accordingly and then it is manufactured as per the capability or the process capability of the process.

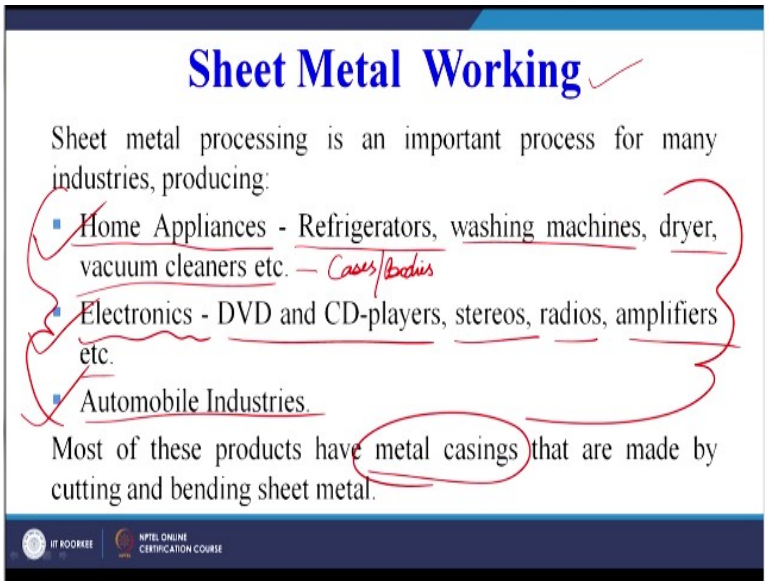
So, for most of the plastic processes we have seen the guidelines were related to the uniform wall thickness, the guidelines were related to the design of the ribs. The guidelines were related to the various other product design attributes. Now we come to a completely different process that is a sheet metal working. Now if you see we see so many sheet metal parts around us the bodies of the automobiles in most of the cases are made by sheet metal.

We have utensils at our homes which are made by sheet metal, then maybe the covering bodies of the microwave or the cases of the microwave or the refrigerators are usually made by metals or the metal sheets. So therefore once we design a product which is going to be made using a metallic sheet what are the guidelines that we must keep in mind. I must again emphasize that we are just discussing the broader guidelines.

Because there will be very specific guidelines related to a specific process. For example the sheet metal working is a big topic. In sheet metal working we can have different types of processes, we can have shearing, we can have punching, we can have blanking, we can have even nibbling. So there are number of processes and for each process there may be specific guidelines that a product designer must keep in mind when he is designing a product which has to be made by sheet metal.

But here in a short time of 25 to 30 minutes we are trying to summarize that what are the types of guidelines that are available that we must keep in mind when we are designing a product which is going to be made by sheet metal. So let us have a quick relook at the process of sheet metal working.

(Refer Slide Time: 04:29)

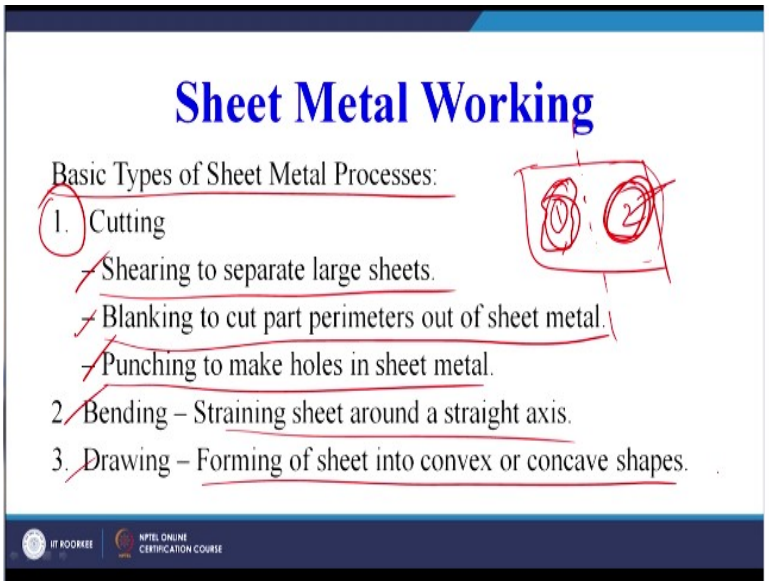


So, the sheet metal processing is an important process for many industries, now what are the industries producing. So, maybe in home appliances already I have taken the example of refrigerators, washing machines, dryers, vacuum cleaners. So, the cases or the bodies of these products may be made by sheet metal, then the second industry is the electronics industry, the DVD and the CD-players, the stereos, the radios, the amplifiers etc., do make use of sheet metal parts.

Then the automobile industries already I have told make use of large types of processes or wide variety of the processes related to the sheet metal fabrication process. So we can see that lot of applications are there and we must take care of the design guidelines for the products which are to be made by the sheet metal parts. So most of these products have metal casings, all the products which are mentioned here that are made by cutting and bending the sheet metal.

So sheet metal is cut it is bend, holes are produced, slots are cut in the sheet metal and lot of punching, blanking operations are done to get the desired product out of the sheet metal piece or the sheet piece. So we will see that what are the standard guidelines that we must keep in mind while we are designing products or designing shapes that are going to be made by the sheet metal.

(Refer Slide Time: 06:10)



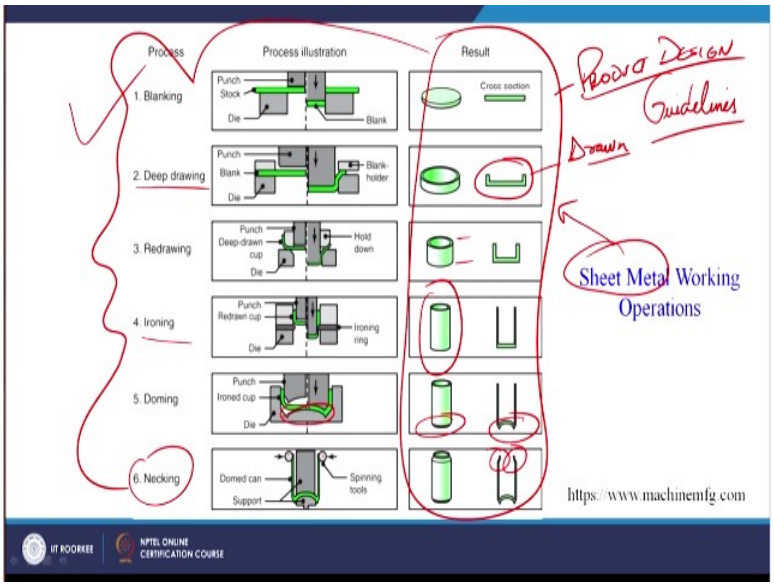
Now let us see the there are different types of sheet metal processes, what are these processes, the cutting of the sheet metal shearing to separate the large sheets. So, you have a large sheet and you want to cut it into 2 pieces maybe the cutting operation of sheet metal, blanking to cut parameters out of the sheet. Now out of this sheet if I want to cut something then maybe this part, part number 1 is called a blank.

So blanking cut part parameters out of the sheet metals for example the pad lock keys are made by the process of blanking. Punching of holes to make in the sheet metal of this is a hole that second number part, if this hole is made in the sheet metal it is called the punching operation. So, blanking, shearing, punching, bending, straining the sheet around a straight axis, then drawing forming of sheet into convex or concave shapes.

Sometimes we make use of the steel plates for having foods, so or that maybe falling under the drawing operation for the sheet metal. So, we can see we can cut the sheet, we can blank the sheet, we can do punching in the sheet, we can do the bending of the sheet, we can do the drawing of the sheet in order to give it a desired shape. So that it can be used as a useful engineering product, so all these operations are done on sheet metal.

And then we need to take into account certain guidelines for the products that we producing using the sheet metal.

(Refer Slide Time: 07:43)

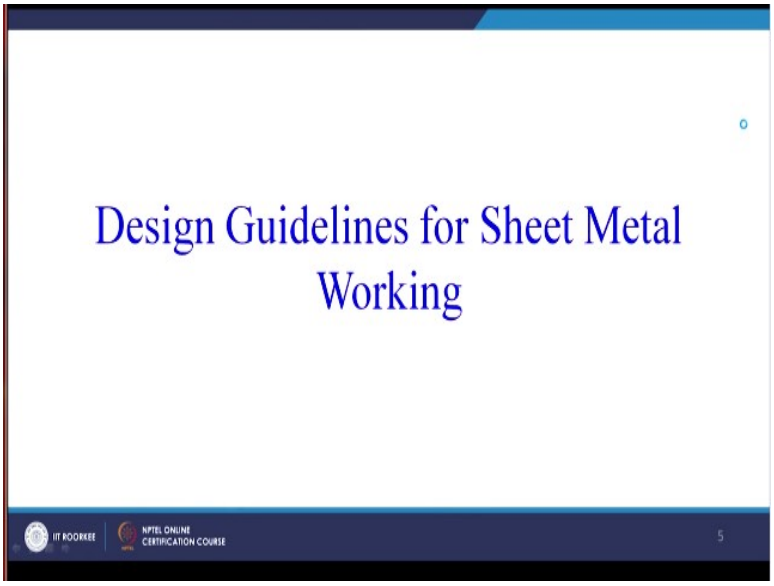


Now let us see just a summary of the processes, this is the first one is the blanking operation, deep drawing, we try to draw the metal sorry sheet metal into a product. Here we can see, this is a drawn part of sheet metal, then the redrawing you can see more height is there for the product ironing. Another process you can see the height then the doming it is done at the bottom, this is the dome that is created if you shown here.

Then necking you can create a neck also here, so this is a necking operation with the help of the spinning tools or the rollers here. So these are the products which are being made from the sheet metal working operations, so for these products what are the guidelines that must be kept in mind. So that these products can easily be made by all the combination of all these processes, so we have seen that we can cut the sheet, we can draw the sheet, we can bend the sheet, we can punch the sheet, we can take out a blank or a product from the sheet.

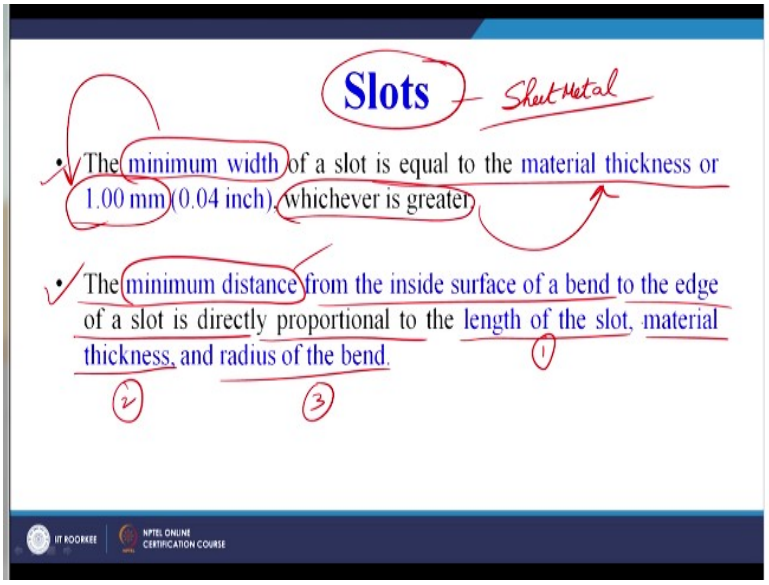
So there are different types of operations which can be done on sheet metal, now what are the guidelines to be taken care of when we are working with sheet metal. And what are the product our main target is to establish the product design guidelines for the products to be made by sheet metal.

(Refer Slide Time: 09:19)



Now let us see that what are the design guidelines for sheet metal working, so one by one maybe we will try to establish the guidelines based on the various product attributes that will help us to make a product in a very efficient and effective manner.

(Refer Slide Time: 09:40)

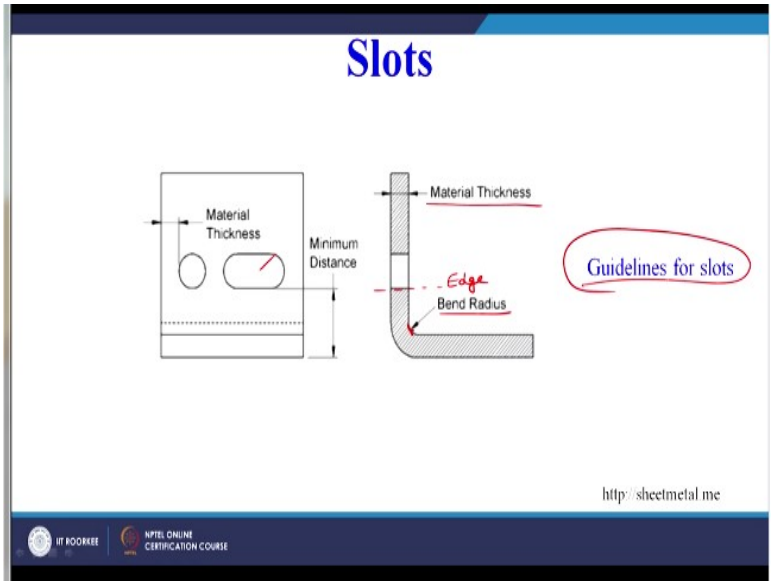


So, first one is that once we have to cut the slots in the sheet metal, so when we have to cut the slots in the sheet metal what are the guidelines that we must keep in mind. The guidelines are listed on your screen, the minimum width of the slot must be equal to the material thickness or 1 millimeter. So, the minimum width of the slot is specified must be 1 millimeter or whichever is greater, so it is related to the material thickness.

So, the minimum slot width must be either equal to the material thickness but if the material thickness is even less than 1 millimeter. Then we must at least have a width which is equal to 1 millimeter, so the slot must at least equal to 1 millimeter. The second guideline is the minimum distance it is the minimum distance from the inside of a bend, so if the sheet metal has been bend from the inside of the bend to the edge of the slot is directly proportional to.

Now suppose there is a bend and we have cut a slot, now the distance between the inside of the bend and the slot or the edge of the slot is proportional to the length of the slot number 1, the material thickness number 2 and the radius of the bend that is number 3.

(Refer Slide Time: 11:15)



So, we will try to understand this with a help of a diagram, we can see here this is the internal bend and this is maybe the edge of the slot. In this case edge, so we can see that the distance between the internal of the inside of the bend and the edge of the slot that we have cut. So, that we have to see, we have to understand it is proportional to the length of the slot what is the length of the slot, what is the thickness of the material and what is the radius of the bend which is shown here.

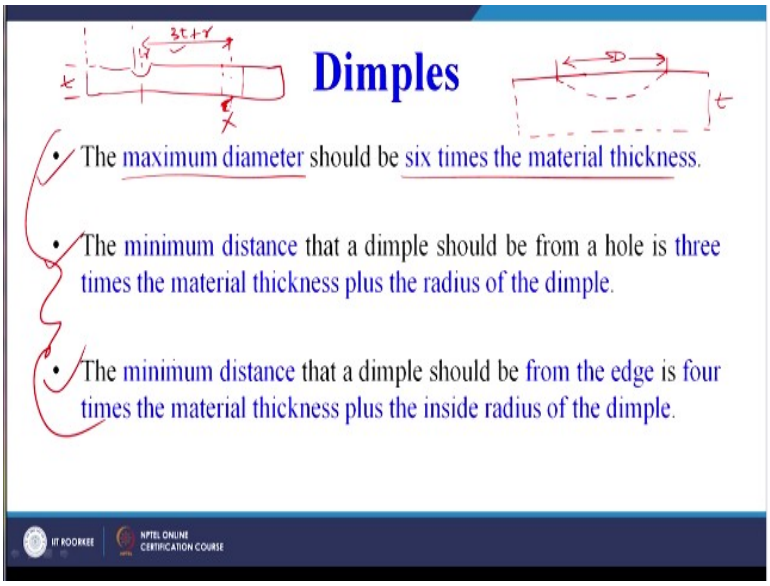
So, the bend radius is given, the material thickness is also given, the bend radius is also given and the third thing we have seen on the screen is the length of the slot. So the length of the slot is also this is a slot which is cut, so the length of the slot is also mentioned, so we have maybe this

minimum distance will depend upon we can again go back to the previous slide, the minimum distance from the inside surface of a bend to the edge of the slot is proportional to the length of the slot, the material thickness and the radius of the bend.

So, these are the guidelines for the slot, suppose we do not follow these guidelines definitely the product will have certain defects. There can be a spring back of the sheet, there can be rupture or failure of the slot, so if we do not follow the guidelines the product we can be made. But in service the product may not deliver as per the required specification or the in service performance requirements of the product.

So, we must follow these guidelines during the designing of the parts, so that we manufacture the part effectively as well as efficiently and the part delivers the performance for which it has been designed. So, these guidelines do help us in designing our parts for better performance, now we talk about the dimples.

(Refer Slide Time: 13:20)



The maximum diameter of the dimple, so maybe suppose we have this is our surface and we want to produce a dimple in the surface. So, the maximum diameter of the dimple, so for example this diameter we say this is a maximum diameter. So, the maximum diameter of the dimple should be 6 times the material thickness, now suppose I am assuming this is my material thickness T.

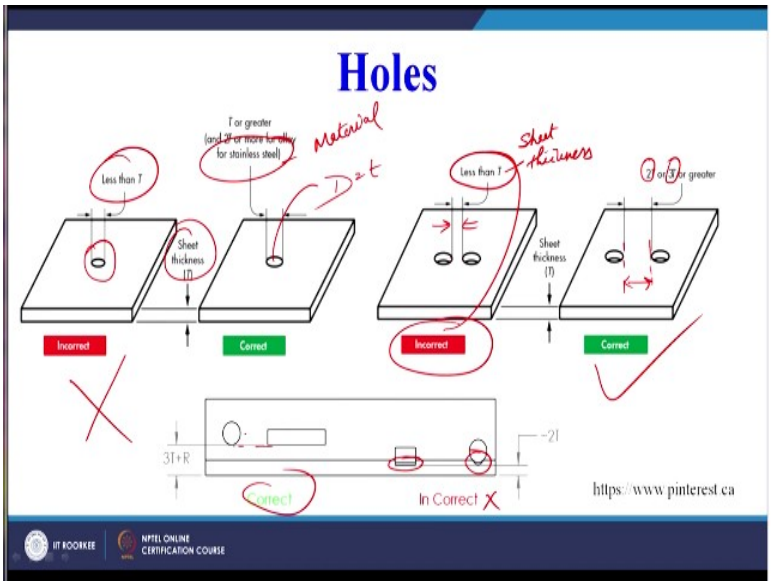
So, our D must be the maximum diameter should be 6 times the material thickness, so this has been made on an exaggerated scale. So, when we want to make a dimple the diameter, maximum diameter should be 6 times the material thickness. So, here in this case D must be equal to 6 times of T, the minimum distance that a dimple should be from a hole. Now for example if there is dimple here suppose this is the dimple, this is a surface, this is thickness and here we have a hole.

So, the minimum distance, so we have to see the minimum distance between the dimple and the center of the hole. The minimum distance that dimple should be from the hole is 3 times the material thickness plus the radius of the dimple. Now suppose this radius is small r and the thickness is given here material thickness 3 times the material thickness T. So, this distance must be 3 times the material thickness plus the radius of the dimple sorry .

The minimum distance that dimple should be from the hole is 3 times the material thickness, 3T is ok plus the radius of the dimple sorry not the radius of the hole, the radius of the dimple, this is r. So, the radius of the dimple, so this is the r here, this is not the r, so these guidelines must be ensured when we are having dimples in our sheet metal. The minimum distance that are dimple should be from the edge maybe this is suppose the edge, from the edge is 4 times the material thickness plus the inside radius of the dimple.

So, this distance is also is specified that how much away the dimple must be from the edge, how much away, how much distance away the dimple must be from the hole. And what is the maximum diameter of the dimple in context of the thickness of the sheet. So, if we follow these type of guidelines we will be able to design our product in a much better manner.

(Refer Slide Time: 16:00)



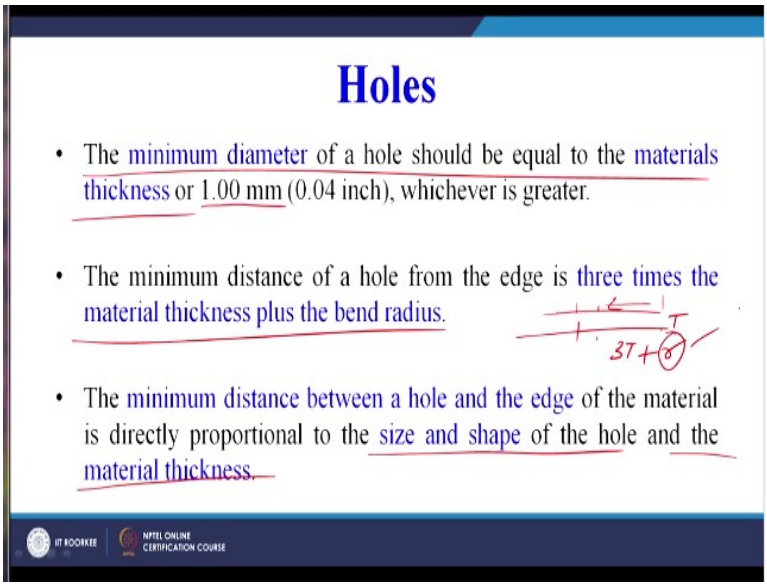
Now coming onto the holes when we have holes in the sheet metal we can see here the diameter of the hole is shown. It is less than T, now what is T, T is the sheet thickness, so this is not correct, so which means that the diameter of the hole which we can call as capital D must be T it must be equal to either T or greater than 2T or more alloys or for stainless steel. So, the diameter of the hole in the sheet metal must be greater than the thickness of the sheet or the sheet thickness maybe 2 times or 3 times more than the sheet thickness depending upon the material.

So, we have already seen that there are process material charts which help us to understand that which process is applicable to which material. And these guidelines also are sometime specific to the specific materials, so as the alloy will change these guidelines may also change. Now when we have to understand that what must be the thickness between the centers of the 2 holes or the edges of the 2 holes.

We can see here this is incorrect why it is incorrect because the distance between the edges here is less than the thickness or the less than the sheet thickness. So, we must provide a proper distance between the 2 holes, so when we have to provide the proper distance it must be 2 times the thickness or 3 times the thickness of the sheet or even greater. So, that will be termed as correct, so this is maybe these are the 2 guidelines related to the holes in the sheet metal parts.

Also we can see there is another guideline related to the edges here, this is correct and this incorrect because here we can see this is and this is not correct why not correct because they are not in a proper alignment which is incorrect. So, this is another guideline which we will see this is 3 times the thickness plus the radius.

(Refer Slide Time: 18:14)



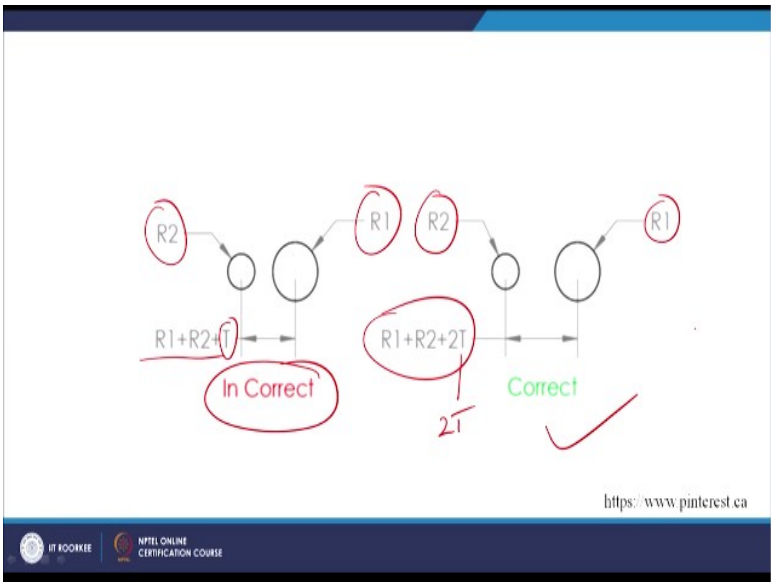
So, let us now come to the textual part of what we have already understood, the minimum diameter of a hole should be equal to the materials thickness which we have already seen with the help of a diagram or if that is not possible the sheet is very very thin. The minimum diameter of the hole must be 1 millimeter, the minimum distance of a hole from the edge is 3 times the material thickness plus the bend radius that also we have seen.

This we have not seen that if we have a hole then from the edge what must be the distance of the hole that is also mentioned here. Here the minimum distance of a hole from the edge is 3 times the material thickness, suppose this is T, so this distance must be 3T+the band radius suppose it is r. The minimum distance between the hole and the edge of the material is directly proportional to the size and shape of the hole and the material thickness.

So, we can see size and shape of the hole will be given by the value r or the band radius in case we have a hole which is away from the maybe this is 1 side of the sheet metal another side, so there is a band here, so what must be the distance of the hole from the bend that is given or from

the edge of the band that is given here. So, we can have similar such guidelines related to the different shapes in sheet metal, so this is given here.

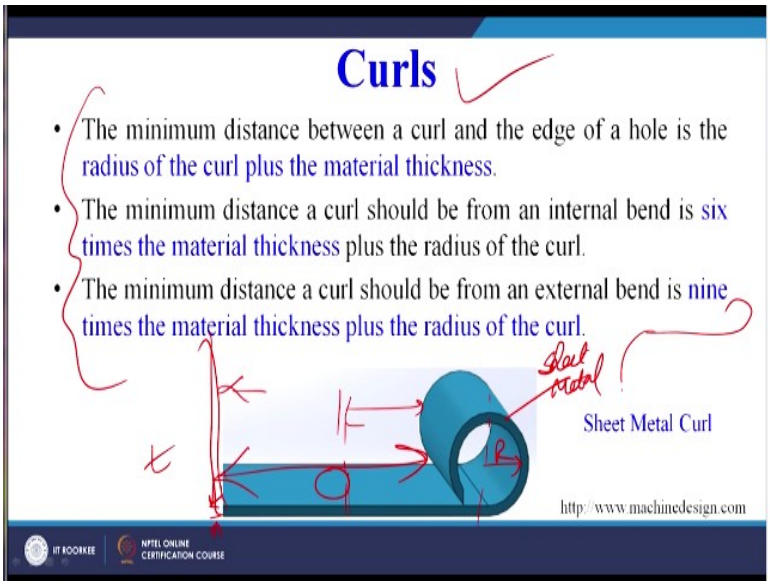
(Refer Slide Time: 19:41)



These are 2 holes R this is having radius R2, this is having radius R1, so center to center distance R1+R2+T, T is the sheet thickness, so this is incorrect. Whereas this is correct center to center distance between the 2 holes the radius is R1, radius is R2 here suppose R1+R2+2 times the sheet thickness. Here it was single T only, so it is 2T which is correct, so many such guidelines are there, so it is giving us that if 2 holes of different diameters or different radii are going to be produced in a sheet metal, what must be the center to center distance.

So that if we are able manage that distance or if we are able to design that distance between the 2 holes of different radii our product will perform better and we will be able to manufacture it in a very productive manner.

(Refer Slide Time: 20:41)



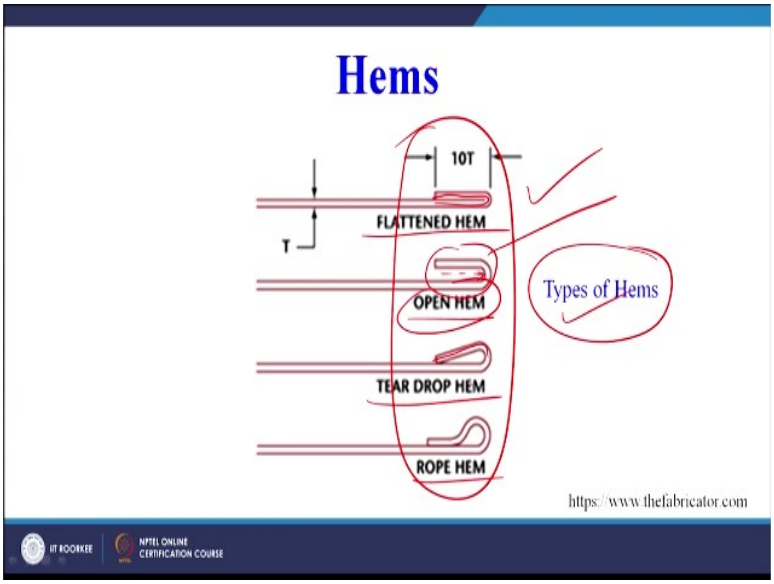
Now coming onto the curls let us see now the minimum distance between a curl and the edge of the hole is radius of the curl plus the material thickness. Now suppose we want to make a hole here, so what must be the distance of this hole from the curl that is specified. So, the minimum distance between a curl and the edge of the hole will come here is the radius of the curl plus the material thickness.

So, radius of the curl means suppose we can take suppose this is the radius and suppose this is the thickness of the sheet. So, very easily we can calculate that what must be the distance of the edge of the hole from the curl. The minimum distance a curl should be from an internal bend is 6 times the material thickness, now suppose we have a internal bend here like this what must be the distance of this internal bend or sorry the edge the minimum distance of a curl should be from an internal bend is 6 times from an internal bend here what must the difference distance sorry not the difference.

The distance between the curl and the internal bend is 6 times the material thickness that we have specified as T, the minimum distance a curl should be from an external bend. So, this is an internal bend, so if we have an external bend like this and here we have a curl like this. The minimum distance a curl should be from an external bend is 9 times the material thickness plus radius of the curl, so radius of the curl can be calculated.

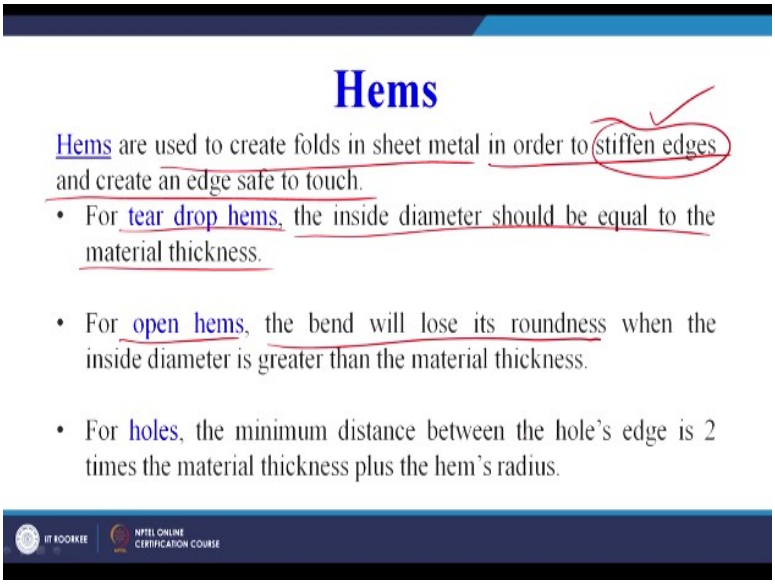
So, internal bend, external bend all those type of guidelines are given for the curls which can be made in sheet metal. So, these guidelines will help us to design the products where we want to curl the sheet metal.

(Refer Slide Time: 22:33)



So, coming on to the hems many times we give the hems in the sheet metals, what are these hems we can see we have flattened hem, open hem, tear drop hem and a rope hem. Now what are these hems and why are the used let us see.

(Refer Slide Time: 22:48)



So, hems are used to create folds in sheet metal in order to stiffen the edges and create an edge safe to touch. So, safety is another parameter now if we have a sheet which have a clear cut edge,

so the edges are sharp, so if you hold them or sometime accidentally your finger or maybe hand may have a bruise or a cut because of the sheet sharp and of the sheet. So, in order to avoid that for we curl or we hem the sheet metal, so that is one thing and hems are use to create folds in the sheet metal in order to stiffen the edges.

One thing is we want make the edges of the sheet stiff and another hand we want to be more on the safety side more on the safer sides. So, that the sheet end does not cut, so it is easier to hold by the worker**,** so we can see that what is the role of the hems. Now what are the different types of hems we have already seen, we can have a flattened hem, completely flattened the sheet metal like this, we can have open hem where there is an internal radius here.

Tear drop hem like this and the rope hem, so depending upon the requirement we can use different types of hems. Now what are the design guidelines for the hems, let us see. So, for tear drop hems the inside diameter should be equal to the material thickness. So, once we are bending it the inner diameter must be equal to the material thickness in tear drop type hems.

In open hems the bend will lose it is roundness, so in open hem let us once again see the open hem, this is the open hem which is written here open hem. So, what are the design guidelines for open hem, in open hems the bend will lose it is roundness. So, it is round and it is open, it will lose it is roundness when the inside diameter is greater than the material thickness, so inside diameter is greater than the material thickness it may lose it is roundness.

So for holes the minimum distance between the holes edge is 2 times the material thickness plus the hems radius. So, this is also an important guideline for holes the minimum distance between the holes edge is 2 times the material thickness plus the hems radius. So, with this we conclude the today's session on sheet metal working and what are the important design guidelines for sheet metal working.

So we have try to understand that the sheet metal working is an important process, we can make different types of products the cases of refrigerator or the bodies of the automobiles or the cases of the microwave oven are made by sheet metal operation. We have also try to see that what are

the various geometrical features that we can create in the sheet metals, we can have dimples, we can have holes as well as we can have hems.

And once all these features have to be included in our product design, what are the important design guidelines that we must keep in mind when we are working with sheet metal products. In our next session we will focus on different types of other processes and the design guidelines related to those processes, thank you.