Design for Quality, Manufacturing And Assembly

Prof. Palaniappan Ramu
Department of Engineering Design
Indian Institute of Technology, Madras

Lecture – 38
Sheet Metal Working

Welcome back under the design for manufacturing topic, injection moulding was discussed. The couple of other things that I will discuss part of the design for manufacturing is Sheet Metal Working. So, as I pointed out in the beginning of the course and also multiple times across the course.

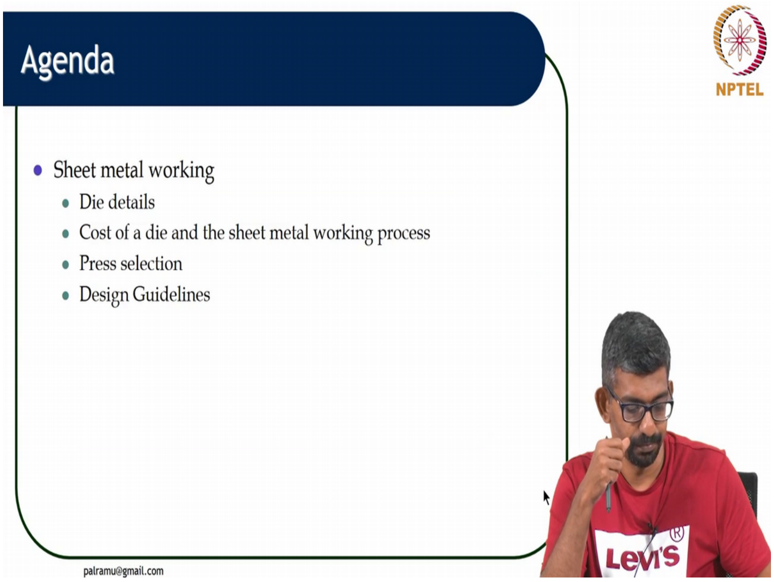
The design for manufacturing just like design for assembly again can be seen as a set of guidelines if you are going to work on a sheet metal. By the way what is sheet metal working? That you are talking about many of the components that use metal for instance they are made out of sheet metals.

For instance, many of the car parts are made out of sheet metals. Sometimes the doors are tailor welded blanks, but they are also made out of sheet metals. Any kind of a washer is a result of sheet metal working.

So, in these cases what are the guidelines, what are the best practices that we need to use? To make sure we say one time, we say one cost. Those guidelines is something that we will talk here more from a design for manufacturing perspective. As a designer what should I know? that I need to take care of those pitfalls in manufacturing in a designed stage. Otherwise you design it is approved, there is a lot of time between the approval and your designing then it goes to manufacturing where they figure out that this design cannot be manufactured.

Then it has to come back, this is something that we have discussed in the very beginning as well as a multiple parts across the course. So, in that context we are going to talk about sheet metal working, what are the best practices and what are the guidelines. Again, as I pointed out throughout the course this is not comprehensive this only opens up this field as an avenue for you.

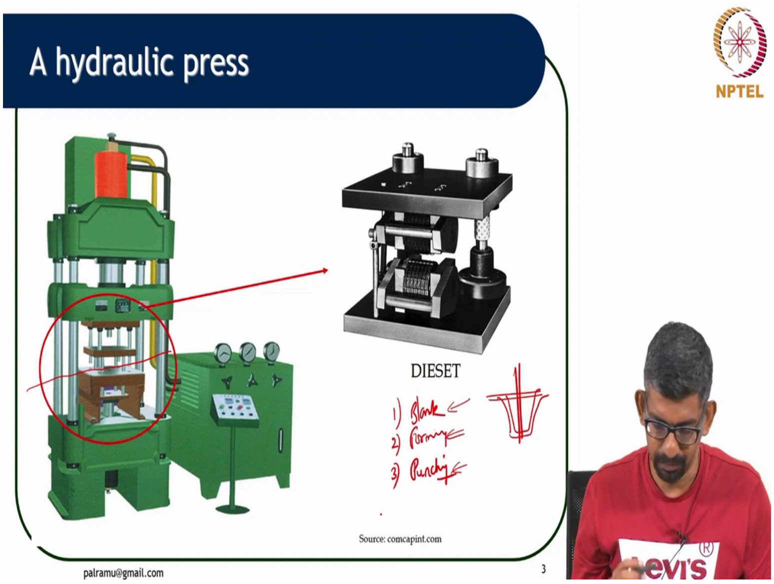
(Refer Slide Time: 02:23)



What is the agenda for the presentation? We are going to look at the die details just give you an idea very quickly and how to go about costing.

The cost of a die and the sheet metal working process, press selection, how do I go about selecting a press to do these sheet metal working and what are the design guidelines.

(Refer Slide Time: 02:49)

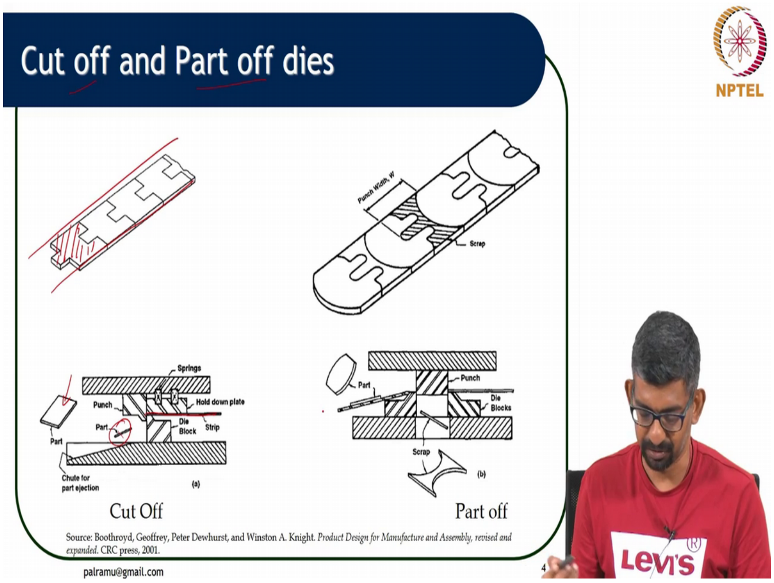


So, this is a hydraulic press, sometimes it is also these are called your die set or your press tools, right. So, usually the press tool has 3 elements at the top and 3 elements at the bottom. There are different process that a particular component goes through before it becomes a component from the sheet metal. So, what usually happens is the sheet metal is supplied in this region and this is the moving part. These 4 bars here are fixed and this die set is going to move up and down to operate or to provide the sheet metal operation.

When the sheet metal is going like this one of the first operation is you want to blank it. Blanking is nothing, but cutting to the shape that is required. Then you want to form the sheet was like this and I want to form it like this so, that is your forming. The third one is your punching after it is formed maybe there is another punch that goes through it, there is a whole something of that sort.

So, basically this is the 3 things that are here ok, there will be a blanking and usually the press tool comprises of tool sets that will do all 3 of them and usually the blank will have a larger a tool the forming will be inside that and the punching tool will be within that. And they will work in a asynchronous manner such that first it will blank then the smaller one will come and form it to the shape that is required and then finally, the punching happens. So, in just one operation the whole thing the component is ready.

(Refer Slide Time: 04:51)



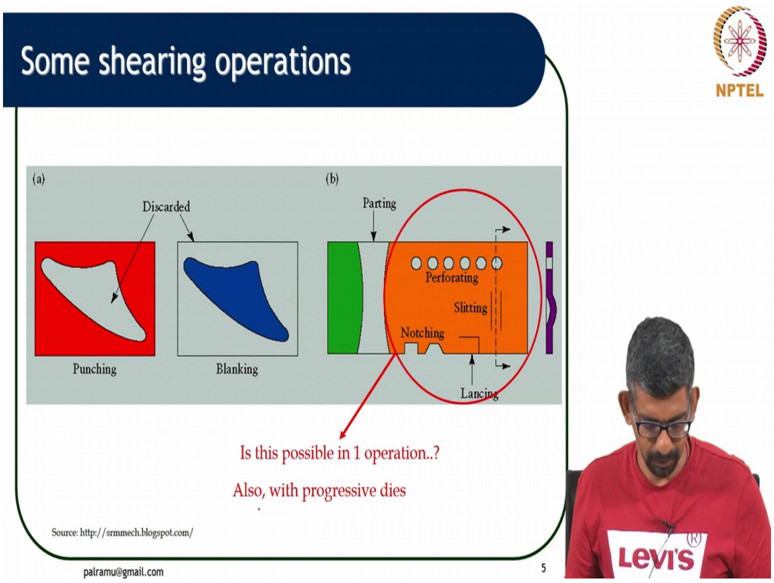
So, there are a couple of things in terms of the dies depending on the finished product that we get or the finished component that we get they can be classified as a cut off part cut off or part of dies. What is the cut off? Take such a design there will be no scrap per say ok.

So, you take a sheet metal like this and I am going to take components like this. So, there is no scrap in this, you just right after that. So, in this case if you look at it there is a punch and there is a die block here. So, we are only showing you the cross section part of it and then your material is coming here.

I have some kind of a hold on plate and when this comes there is a punch that goes and it cuts it, there is a sheet of stuff comes in and there is a cutting stuff that keeps coming up and down. So, it punches the part and then the part ejects that is all this is the part, so, it is simple. So, these are called cut off so, you cut off and then the part comes in right.

The other one is the part off; what happens is after you cut because, this is the component that I want, but there is some kind of a scrap that is also associated with it. So, that scrap will fall in between and then the part is ejected at the other end that is it. So, this is a part of time.

(Refer Slide Time: 06:18)

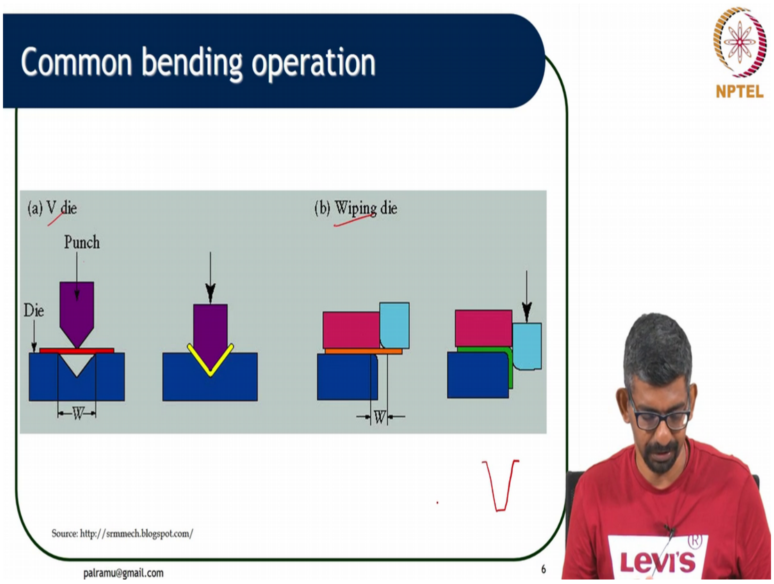


Some shearing operation just to motivate the discussion here, this is a punching operation ok, there is a punch. So, wherever you see grey there is a whole that is what it means within the component. This is a blanking operation, so, you are cutting out this guy, this is what you are going to work upon. You are eliminating the unwanted that is called punching, this is blanking.

So, now, there is a question for you guys. So, there are some perforations here, there is some slitting here, there is some notch here, can we do all this in one pass? So, there is one sheet metal ok, I told you that in one operation you can at that one particular position we can do 3 operations right. But in this they are different, perforations, notching, slitting all of them are punching operations. In this it is it possible to do this in one shot?

There is a possibility to do that using something called progressive dies. So, the die has progressive operations across. So, you can do that.

(Refer Slide Time: 07:24)



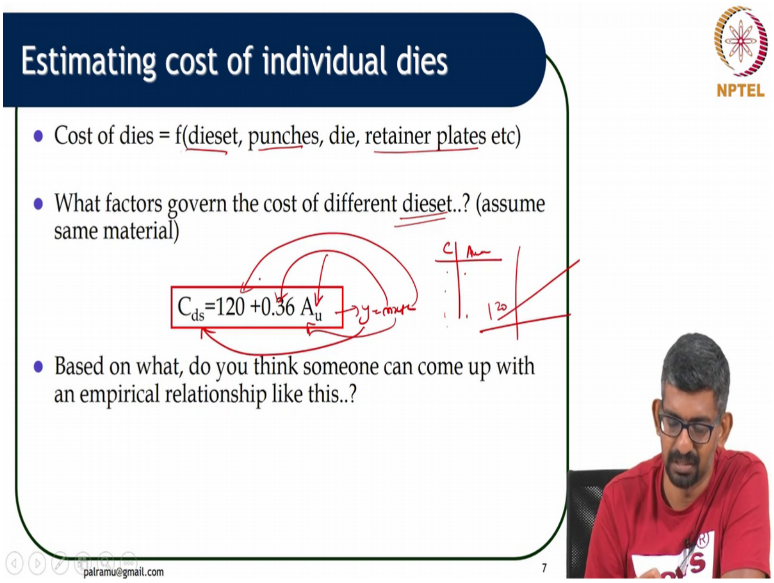
Common bending operations, you have a question?

Student: No.

Common bending operation there are 2 types there are other types also these are the broad classes. One is the V die and the other one is a wiping die. The wiping die is something that we saw as an example in the previous one. So, there will be a holding plate and then the punching part comes in and this is what happens to your, it bends it, forms, both of them are forming only.

In this case it is a punch, we punch it comes it is held on these 2 sides and it will go and deform into a V shape. Many of the vessels that we use in our day to day stuff is formed in this manner from sheet metal. So, the classical our you know what we call the glass or the tumbler is made out of stainless steel is made out of such operations.

(Refer Slide Time: 08:16)



Similar to the other one we are interested in estimating the cost ok. So, in this particular case; obviously, the die cost itself is also included and how do you go about it? So, the cost of the die is the die set, the male and the female part, the punches, the retainer plates. What factors govern the cost of the different die set?

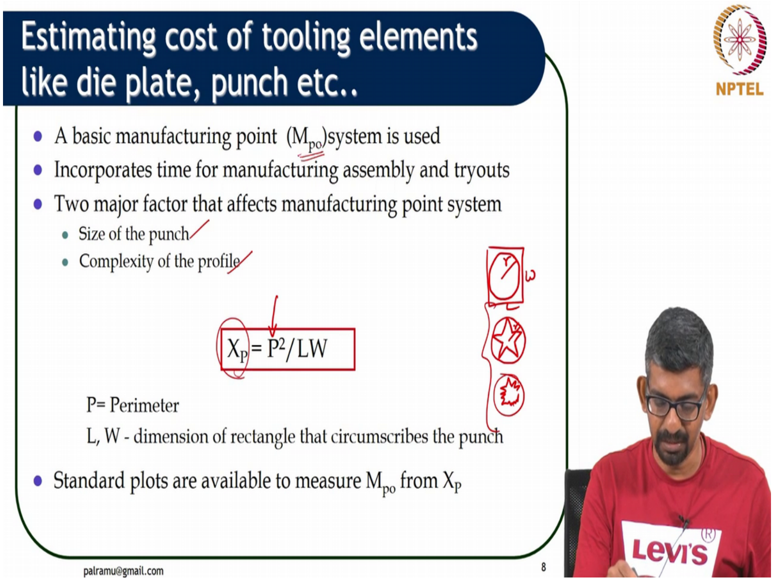
Let us assume the same material, what would be the differentiating cost? If you imagine it is going to be on the part that you are going to make, but does it mean that I am going to use different die sets for different parts? I will use because when I say die set yeah.

So, this is the entire die set right so, I am talking about this not about these individual parts, not about the blanking those kind of stuff.

So, that one thing that will be common is the area ok. So, it is a direct relationship, someone did a study, they went around and then they checked and then they came up with this. So, it is basically it is proportional in one sense 2.3 that is some number 0.4 is a number times A u, A is the area of the die set that you are looking at ok. Based on what do you think someone can come up with an empirical relationship like this? Very simple you go across some kind of a manufacturing valley or something fabrication valley where what I mean by valley is there a lot of shops that can fabricate.

So, what you ask them is what was the cost of the die set? And what is the area of the die set? That is all. So, you do this and you do a simple line fitting that is what has been done here. And this will cut at 120 and this is basically a kind of m x. So, the slope of the line was 0.4 that is all ok. So, you can just see this as a y is equal to m x plus c. Where your y is your C d s, the cost of the die set and your m was this guy, your x is your A u and C, the constant that is added up.

(Refer Slide Time: 10:47)



So, how do you go about estimating the cost of the tooling element? The die set is over, the next one will be your tool like die plate, punch etcetera there are multiple as we pointed out for the punching, for the blanking and for the forming. How do you go about it? Because you know you cannot just have one tool set and do different types. So, you will have to change them right. So, how does it?

So, basically there is one way suggested in this particular book is a basic manufacturing point M p o operating time for manufacturing assembly and tryouts they take this into account. 2 major factors that affect the manufacturing point system is the size of the punch and The complexity of the profile ok.

So, for instance I want to punch something like this. So, you say sir let us go ahead and look at the size of the punch. The size of the punch is the same imagine ok, this is the component that I want to fabricate at the end so, the size of the punch is also the same. Size of the punch is what? It is described by this R.

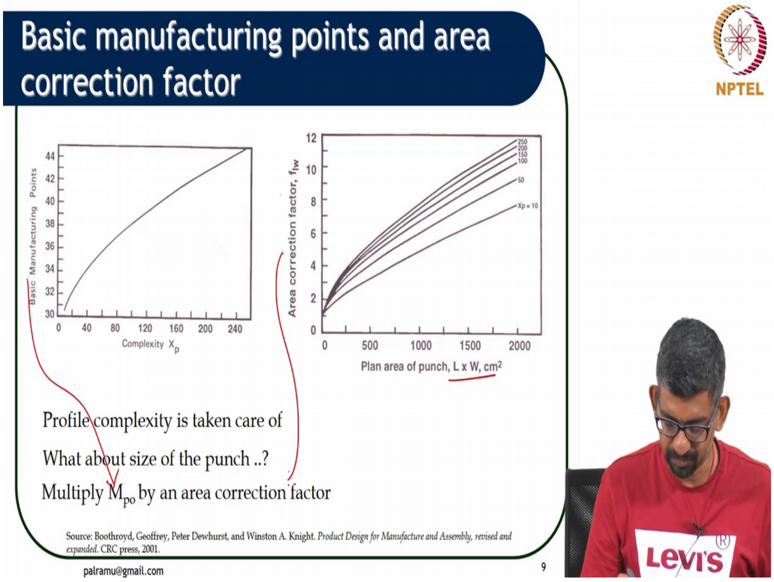
Now, I want to fabricate such a shape so, what is the size of the punch? Supposed to be same let us say that this was also R, but do you think these 2 will cost you the same? No. So, this is only one guy, what is the second one? It says complexity of the profile, how complex is this? This is nothing, it is just a circle here I have some kind of assertion or abbreviation whatever you call it ok.

And; obviously, it will be even more if I do this correct so, this it is even more complex. So, there should be a way in which I can account for these guys. How can you do that? You look at the perimeter, larger the perimeter I am going to penalize U. I am going to do P square it, LW is your dimension imagine that you are putting this guy in this is L and this is W that is all ok.

It is a perimeter LW dimension of the rectangle that succumbs as the punch that is all. This is one simple way of looking at it people might have come up with even more accurate and better ways of doing it, but this one simple way in which you can do that. For instance the perimeter for this is much larger than this hence the cost will be larger which we can imagine directly.

So, there is a standard plot that is available that relates your X P versus the manufacturing point system.

(Refer Slide Time: 13:37)

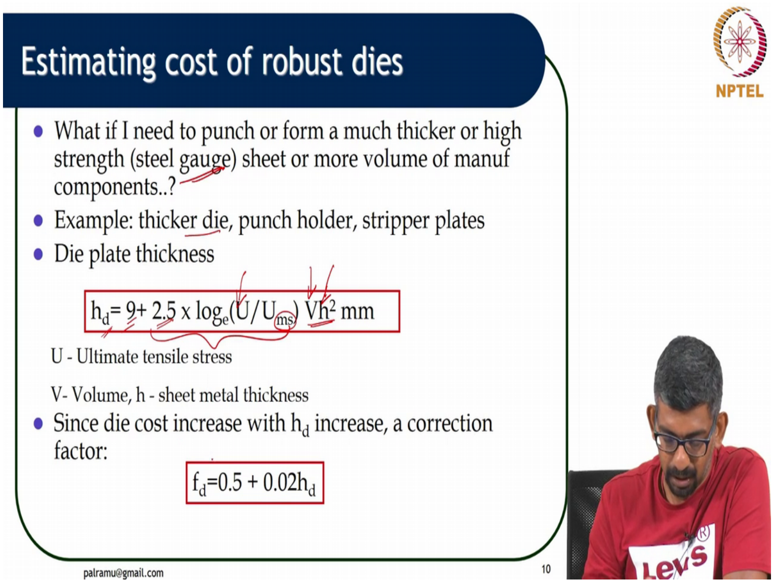


We will see what it means. So, the complexity X P can be estimated from the previous slide and the manufacturing points is given, these are probably plots that is an output of a PhD. This is or research that has been conducted for several years.

The profile complexity is taken care, but what about the size of the punch? So, right now what we have done is we have taken care of the profile complexity, but what about the size of the punch? There is another additional stuff. So, that is from the plan area, basically L times W.

And if your X P was 10 here, X P was 50, 100 that you take from here there are different curves and then you do an area correction factor f, l, w. So, what we do is we multiply that M P o which is a basic manufacturing points here from here by an area correction factor which will come from here that is all.

(Refer Slide Time: 14:36)



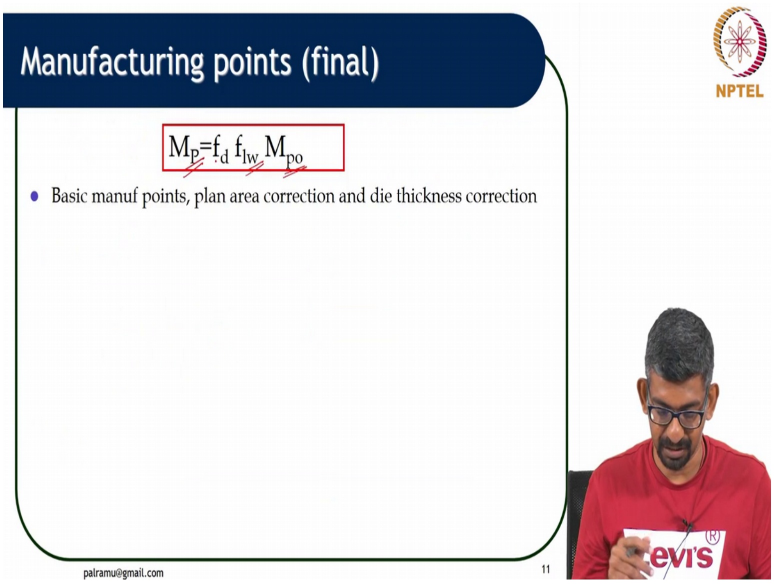
So, how do we go about estimating the cost of such dies? What if I need to punch or form a much thicker or higher strength? Ok so, right now we are only talking about some sheet metal. But, in case if I am increasing the sheet gauge the steel sheet gauge or more volume of the manufacturing components. So, how do you do is there is an example, a thicker die, punch holder and stripper plates.

The die plate thickness see these are you know I am not going into the mechanics and finding out how these equation, I am just you will have to take it from me for granted this is only a kind of I am only scratching the surface of this discussion. So, that you have an exposure.

So, basically what we are trying to talk about is the height of the die plate will be some constant times another constant sorry plus some constant times log of U over U m s, U m s is nothing, but your mild steel in this particular case and U is your ultimate tensile stress. So, whatever material that you are looking at your kind of normalizing it with mild steel.

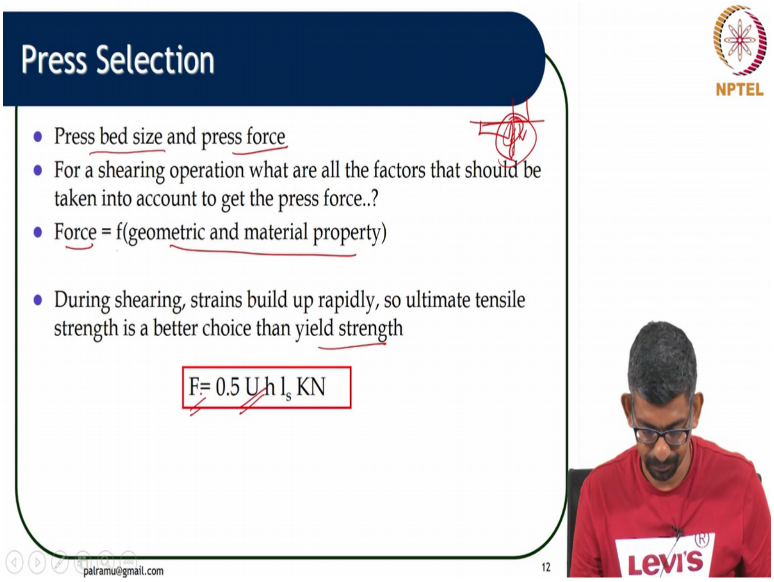
So, if it is the same then this is only one so, if this guy is larger then this particular quantity will become larger that is all the point is and V h squared, what is h? It is a sheet metal thickness. So, larger the thickness I am penalizing U and V is the volume that you are talking about that is all ok. Since, the die cost increases with h d, when h d when h increases by h d then there is a cost die cost also increases or correction factor has to be introduced.

(Refer Slide Time: 16:16)



So, finally, we get to a manufacturing point the final one, it is a function of the basic manufacturing point times the f l w and the f d that we saw in the previous one ok. So, this is f d. So, f l w is the area correction factor that we are talking about ok. So, area correction factor and then there is another correction factor provided the height or the gauge thickness, the thickness of the gauge varies ok.

(Refer Slide Time: 16:49)



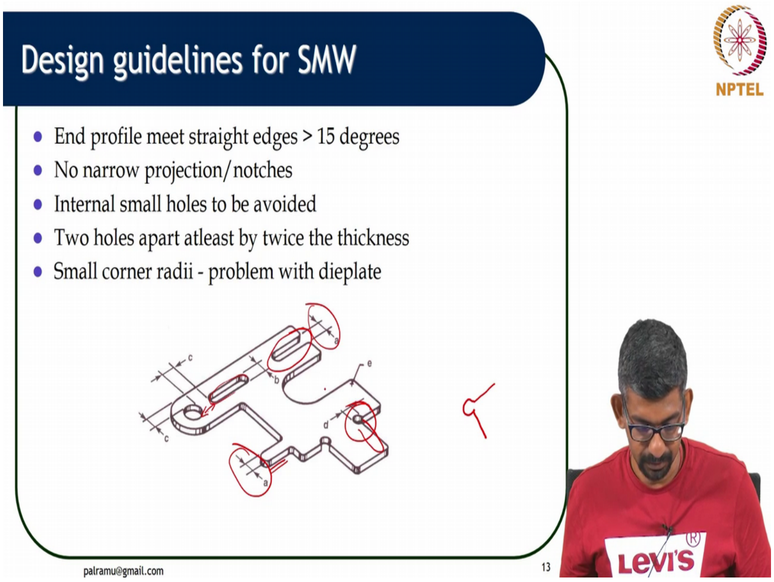
So, how do we go about selecting the press? So, the basically this is a specification, very similar to buying a motor bike or a car. What is it that you look at? You know I look at what is the performance of the bike. For each one the performance of a motorcycle might vary. For someone it will be the power that it can generate, for someone it will be the mileage that it will do. So, you will have to also be able to select a press that will deliver something in terms specifications.

So, basically one thing that you can talk about is the Press Bed Size, how big it is the l times w and what is the press force? So, basically for a shearing operation that we are talking about. So, we saw some holding sheet metal and then this happening right. So, that what is happening in all these cases is it is a shearing operation, it is shearing, it is happening, it is doing that.

For a shearing operation what are all the factors that should be taken into account to get the press force basically the force is a function of the geometry and your material property; obviously.

So, during shearing the strains build up rapidly, in such a case the strains build up and especially if you have a corners right corners or sharp corners it is going to there is going to be residual strains that are building up. So, your ultimate tensile strength is the better choice than your yield strength ok. So, you will take your U as a quality and then you estimate your F that is required.

(Refer Slide Time: 18:16)

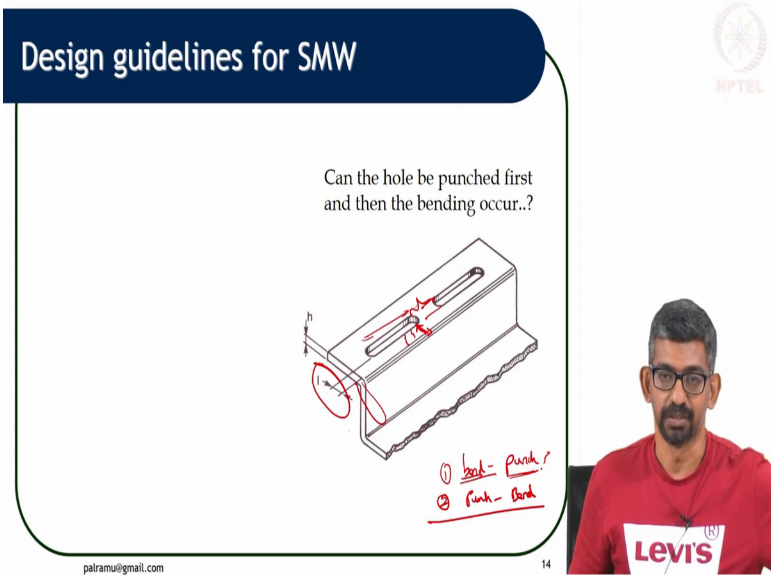


So, this is some design guidelines for sheet metal working. So, the point is the end profile meet straight edges only at greater than 15 degrees. So, the other one is there is no narrow projection or notches. So, you cannot have very narrow notches or projections that are very narrow.

Here it is broad, but you cannot let us say you cannot have a small like this. There is some guideline a for that, there is some minimum value that you need to maintain. Then internal small holes to be avoided such kind of small holes internally not on the external part like this internally some small holes needs to be avoided.

2 holes the difference between them, irrespective of the shape of the hole the distance between them should at least be twice the thickness. If this thickness was h it should at least be 2 h here. Small corner radii you can talk about this small corner radii problem with the die plate because, the die plate cannot give you the corner radius that is an issue with that ok. If you have a small corner radii I mean I am not sure whether you are able to see it, it is asking for this one. There is a problem to generate such kind of stuff ok.

(Refer Slide Time: 19:42)



There is a food for thought; there are 2 operations in a sheet metal to come to this component. So, one is bending here and the other one is the hole.

So, the question is do you punch the hole first and then bend it? Or you bend it first and then punch the hole?

There is no specific unique answer in this case ok, but we can kind of understand what the problems are. Let us assume the first case where I bend and then I punch. And the second case is I punch and then I bend.

So, let us take the second case, what will happen if you punch and then you bend because you see this distance that becomes important. If I am going to because the material has undergone a deformation here ok. So, soon after that if you kind of bend it here there is a likely chance that it might crack ok. And this side of the component is weaker because of the hole. So, you need to have some distance where you can bend it that is one thing.

On the other hand, if I am going to bend it first and then punch what could be the problem? So, the one usual case is your spring back ok. So, you bend it and then you believe this is the dimension and then you punch the hole, but then this guy comes back which will create a whole lot of issue.

So, there are issues on both ways, but what you just need to take care is you need to worry about these relationship on h and l and what is this dimension, what are these dimension, what is even the holes next to each other you will have to be careful ok.

So, basically this is the guidelines for the sheet metal stuff that we discussed, this is very similar to the injection moulding if you see like you know you cannot have bosses standing by themselves without rib support, fins. And if you are going to have a fin like this then you need to have supports in the perpendicular direction to them. Very similar to that there are some discussions that we have this is not comprehensive as I pointed out. So, that is that is all about sheet metal working.