Manufacturing Guidelines of Production Design

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

Lecture-21
Product Design Guidelines: Compression molding and Extrusion

Namaskar friends, so we are today going to enter into the 2nd half of our course, the course as you all of you are well aware is a 40 session course it which 20 hours of discussion we have to do on a very important topic that is manufacturing guidelines for product design or the help desk for the product designers. So, basically what we are trying to do we are trying to summarize, we are trying to bring together under one platform different guidelines based on the manufacturing processes which every product designer can make use of.

Make use of means that he will or she will get an idea that this type of information is easily available is readily available there are different sources of information and these information can be useful during a product design process. So, sometimes some shapes have to be selected by the product designer, sometime some fillets or the corner radii has to be selected many time the material has to be selected by the product designer.

Sometimes the number of products to be made has to be selected by the product designer. So, product designer has to make a lot of decisions and this is kind of a decision support tool which can help the product designer to take the decisions in a more informed manner. Sometimes the decision taken we have seen number of designs which are wrong, wrong from the not from the creativity point of view they are wrong from the manufacturing point of view.

It is difficult or rather sometimes impossible to make those shapes using the manufacturing processes which are available. Somebody may say that a manufacturing process maybe developed to make that creative shape, yes, you can develop a manufacturing process for making a very good looking aesthetically brilliant a very very creative shape. But what about the cost, if you have to develop a process first definitely the development of the process, the testing, prototyping and then finally using that process.

So, all that will add to the cost of the product, so therefore if there is no special characteristics that are required as per the shape requirements we can modify our design in order to sewed to one or the other manufacturing processes which is readily available, which is commercialized. And which can be used for making a large number of products, so therefore this guidelines related to manufacturing process are very important and are relevant in the life of each and every product designer.

As well as the mechanical engineers who keep on coming up with new and new products. So not only from the material selection, not only from the process selection, not only from the shape selection, not only from the size selection. There are number of attributes based on which the product designer can get benefited after going through the information not only in our course but we are trying to establish the sources of information that such information or the specific information related to a specific process is available.

And when the designer has to make a choice he can just refer to that information and make a choice. If you remember for example in our 4th week of discussion we had 2 sessions, 1 selection of process is 1 and selection of process is 2. And there we have seen in session 2 towards the end that there is a complete table compiled table giving you the different processes for example sand casting, die casting, injection molding number of processes.

And then what are the various attributes which can be the special characteristics of that process or which can be the characteristic features of that process based on for example the kind of weight or the mass which can be processed using the sand casting process or the product or the heaviest product that can be made by sand casting. The lightest product that can be made by sand casting what is the range in terms of mass which can be made by sand casting.

The surface finish achievable by sand casting, similarly in the tabular form for other processes also that information is available. So, this type of databases often help the product designers to select the processes, to select the particular shapes when we are designing we must take care of

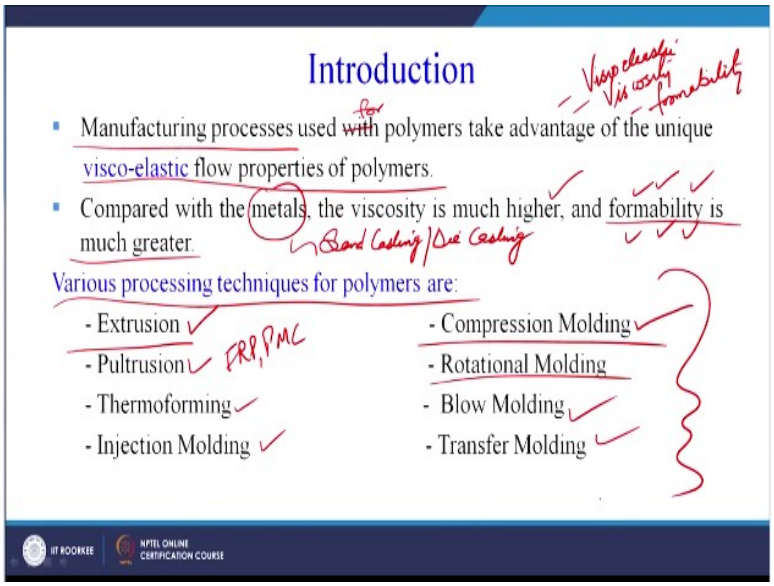
these guidelines. And if you remember in our previous session that was the last session for week 4 we were discussing the design guidelines related to the ribs, related to the section thickness.

So, there are also standard guidelines when the product has to be made by die casting product or die casting. In previous session to that we have seen sand casting. So, in die casting what is the importance of draft, what is the importance of the rib thickness, what is the importance of the wall thickness, what is the importance of the tolerance everything we have seen in the previous session related to sand casting and die casting.

Sand casting we have seen in session number 19, die casting we have seen in session number 20. Now today if you see on your screen the product design guidelines basically we are trying to focus on 2 different processes today which are very common processes for making the plastic parts. So, the processes are the compression molding process and the extrusion process, so we will try to first understand very briefly what is compression molding, what is extrusion.

And then we will jump over to the various design guidelines which are useful when we design the parts made by compression molding and extrusion. So, let us quickly have a brief overview of the product design guidelines.

(Refer Slide Time: 07:06)



So, the manufacturing processes used with polymers or we can say the manufacturing processes used for polymers take advantage of the unique visco-elastic flow properties of the polymers. We are not going to discuss the details about the visco-elasticity but we can say that it is easier to process the polymers as compared to other materials which are difficult to process by melting them and then shaping them.

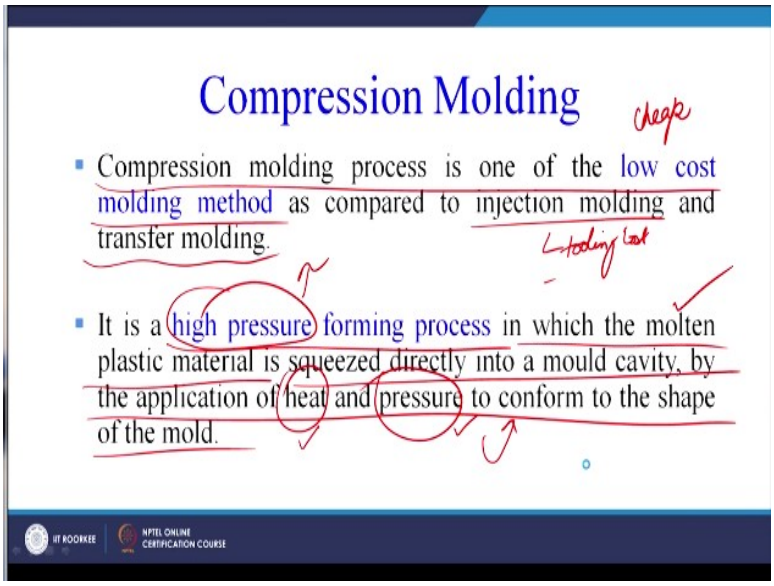
So, compared with metals which are also made we have seen already 2 processes we have seen sand casting in session number 19 as well as we have seen die casting in session number 20. So, as compared to metals the viscosity is much higher in case of plastics and the formability is much greater. So, we can see this is our target property that is formability we want to give shape to the polymers.

So, we can see that there are processing techniques for polymers why because it is easier to shape a plastic because of the 2 things, 3 important things that are listed in the first 2 points. First one are the visco-elastic properties visco-elastic second is the viscosity and third is the formability which means that it is easier to process the polymers. And therefore there are number of processes you can see here extrusion, compression molding we are going to see the guidelines for these 2 processes today.

The product design guidelines then maybe pultrusion which is used for FRP products or polymer matrix composite products. The rotational molding which is use for making large scale or large size axi-symmetric water tanks are large a hollow tanks or products thermo-forming for making sheet type or products. Blow molding for making hollow bottles, ejection molding, transfer molding and there can be additions to these processes which we have tried to list here.

So, because it is easy to melt and form the polymers there are number of processes which can be used for processing of polymer parts. That is the brief review of processing of polymers, then if we go to the next stage what is our target, our target today is to discuss 2 processes and what are the guidelines for products that have to be made by these processes. Let us first see the compression molding process.

(Refer Slide Time: 10:04)



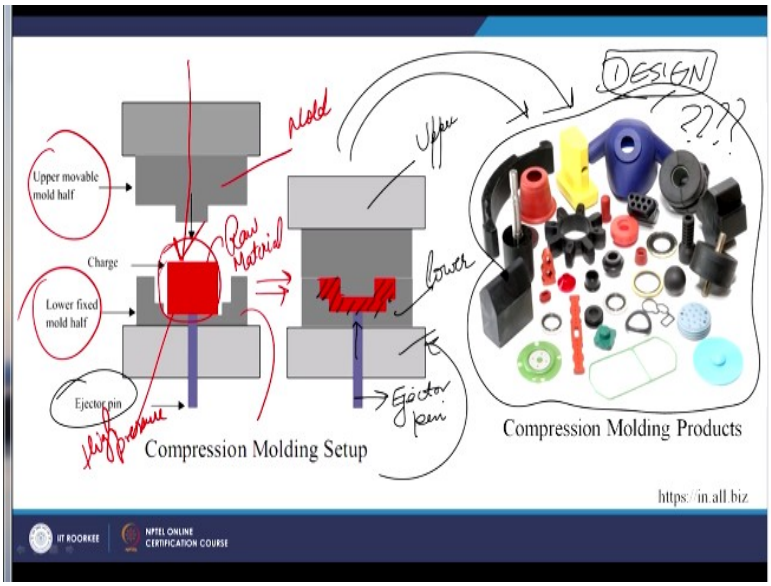
So, compression molding process is one of the low cost molding method, so it is inexpensive or cheap method of making the product as compare to injection molding and transfer molding. So, injection molding the die cast or the mold cast is high, so it is the tooling cost we can say is higher or much higher as well as the control part is little bit more electronic paste in case of injection molding.

These days we have very high and compression molding machines also but if overall we compare the injection molding is a costly process as compare to a compression molding process. Similarly the transfer molding is also relatively costly as compared to the compression molding, the compression molding the mass scale manufacturing process which makes plastic parts relatively in an inexpensive manner.

So, it is a high pressure forming process in which the molten plastic material molten is squeezed directly into mold cavity by the application of heat pressure to confirm to the mold shape. So, basically what are the control parameters here heat, pressure and it is a high pressure manufacturing process. So, we will give a shape by plastic by compressing it in the 2 halves of the mold and finally we will try to optimize the heat input as well as the pressure that we are applying to make the product, why we are applying the pressure.

Because in the mold cavity our plastic will confirm to the shape of the mold, so let us try to understand this with the help of a diagram.

(Refer Slide Time: 11:51)



So, this is an upper movable mold half, this is a upper movable mold, this is the lower half of the mold which is mentioned here, lower or fixed half of the mold. And this is the charge or we can say in simple language it is the raw material which is kept inside the 2 halves of the mold. And now this half of the mold comes down it applies pressure on this as we have seen it is a high pressure forming process, so high pressure will be applied on this raw material.

And it will finally it will take the shape of or the cavity created between the upper half of the mold and the lower half of the mold. So, you can see here this is the product that will come out from this compression molding process. So, this is the upper half of the mold, this is the sorry this is the lower half of the mold, so upper half and lower half and this is as we have seen in case of die casting process also.

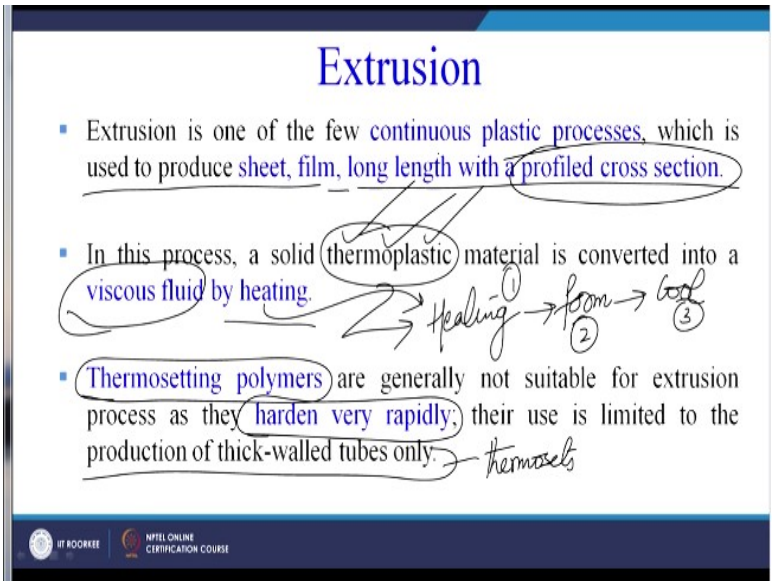
Here also we have an ejector pin which is mentioned here ejector pin, so the product will remain attached to the lower half of the mold and once the product has solidified the ejector pin will just give a slight tap here and the product will come out of the lower half of the mold after solidification process. So here you can see this process can lead to different types of product, so there are different variety of products which can be made.

Now once we are designing these products when we have to design these products what are the guidelines, what are the things that we must keep in mind. So, normally in our academic courses in our curriculum we study about the process what are the process parameters or the process mechanism how it will work or maybe explain 1 cycle of the compression molding process with a help of a suitable diagram.

So, what we do, we make a diagram and then we show that this is a raw material as I have tried to explain today. Similarly the students or the learners will explain in the examination and they will be graded accordingly but usually what we leave in our academic part is. That once the part has to be designed for to be made by the compression molding process, what are the guidelines that a product designer must keep in mind when the product is designing has to be manufactured by the compression molding process.

So, now these are the different types of products you can see, now let us see what are the general design guidelines that we must keep in mind.

(Refer Slide Time: 14:44)



Now we will come to that let us first see the other target process that we have selected for today that is extrusion. Now extrusion is one of the in this case we have seen in the case of compression molding we have seen the products will come out in a discrete manner, so one

product the 2 halves of the mold will close whatever is there are suppose this is a raw material it is placed here, so 2 halves of the mold will close and they will give a shape to the product.

The upper mold half will move, ejector pin will push this out and this is one discrete product I have been got. Then again the raw material will be again placed into the lower half of the mold the upper half of the mold will come it will close it down. It will be held for sometimes till the product solidifies upper half of the mold will go and the lower half of the mold will be attached to the product and the product will pushed out by the ejector pin, second product, this cycle will continue.

But we are getting discrete products in each cycle maybe after every closing and opening of mold we are getting one product after each closing of the mold is ensuring one product for us. Whereas in extrusion we will produce continuous long excess symmetric product or sometimes they may not axi-symmetric depending upon the shape of the die we will get our profile of the product.

So, the product in case of extrusion will be a long extruded product, so extrusion is one of the continuous plastic processes which is use for sheet, film, long length with the profiled cross section. We will see some of the examples of the profiled cross-section, so in this process a solid you can see here thermoplastic material is converted into a viscous fluid by heating. So, in mostly in case of any forming process or any molding process for polymers, there are 3 broad steps.

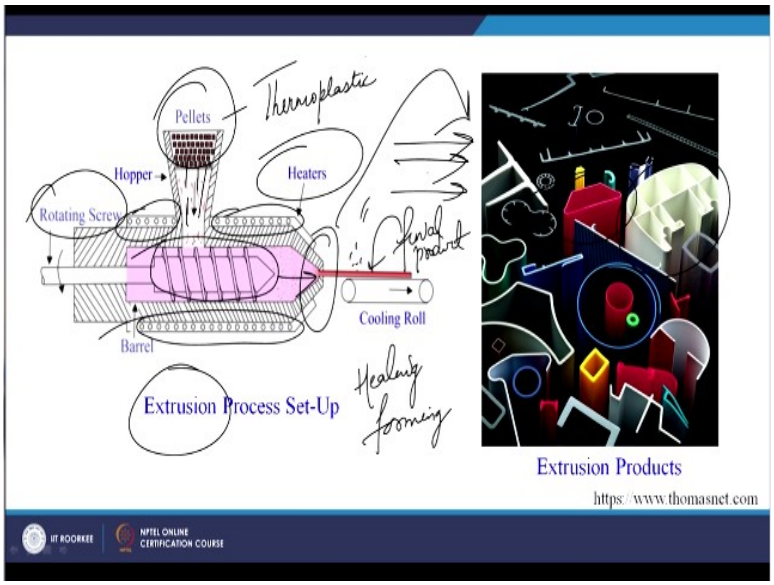
First one is heating then you have to form it or giving the shape and then we have to maybe cool it though make it solid. So, the things are there first step is heating, second step is forming, third step is cooling. So, these are the standard methodology for giving shape to any plastic material and this is specifically exclusion process is used for thermoplastic materials. So, in this process a solid thermoplastic material is converted into viscous fluid by heating.

So, the first stage we have achieved, now the second stage is forming, we have to give shape, we have to give profile to the product that we have to make or as per the design we have to give it

the shape. Then finally we have to allow it to cool, so thermosetting polymer which is another category of polymer or generally not suitable for extrusion process as they harden very rapidly, so their polymerization process or that we call as curing is different from the molding of the thermoplastics.

So, there can be maybe 1 or 2 lectures only to explain the difference between thermoplastics and thermosets. So, but here we will restrict the discussion to the understanding to the fact that in most of the cases thermoplastics are usually processed by the extrusion process. So, their uses to limited to production of thin walled tubes only based on thermosets but mostly we will be making use of thermoplastics in case of extrusion process.

(Refer Slide Time: 18:29)



So, let us try to understand it with a help of help of a diagram, so here we can see these are the pellets as we have seen in the previous slide thermoplastics are the material of choice for making the products using the extrusion process. So, here is there we have a rotating screw pellets will come down from here they will get in touch with the rotating screw here and the rotating screw will shear, they will heat the polymer pellets that are coming and converted into a fluid.

Or into a maybe molten plastic which will be pushed with the help of the rotation of the screw through this die opening here. And this is the final product that we are getting there can be a cooling arrangement here maybe a spray of water or maybe a jet of water can be used for

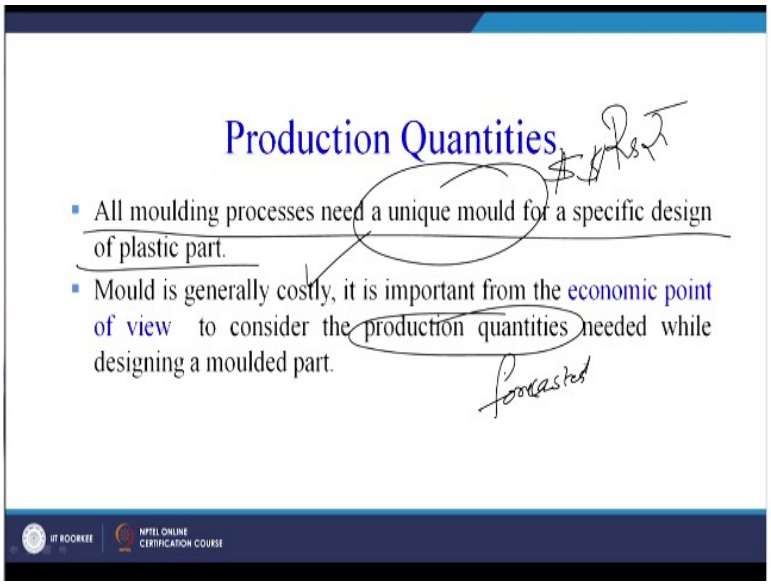
cooling. So, there are 3 things 1 is the heating and therefore there are heaters also here you can see this is representing the coils of the heater.

So, all around the barrel we have heating arrangement, so as we have seen 3 steps are there heating, forming in the die and later on cooling. So, this is a basic process of extrusion and these are the profiles that are generated, you can see different types of profiles are there which can be made by the extrusion process. So we are not going to go into the detail our focus is how to design these profiles are what are the parameters to be kept in mind while designing these profiles.

(Refer Slide Time: 20:07)

Now let us quickly go to the product design guidelines.

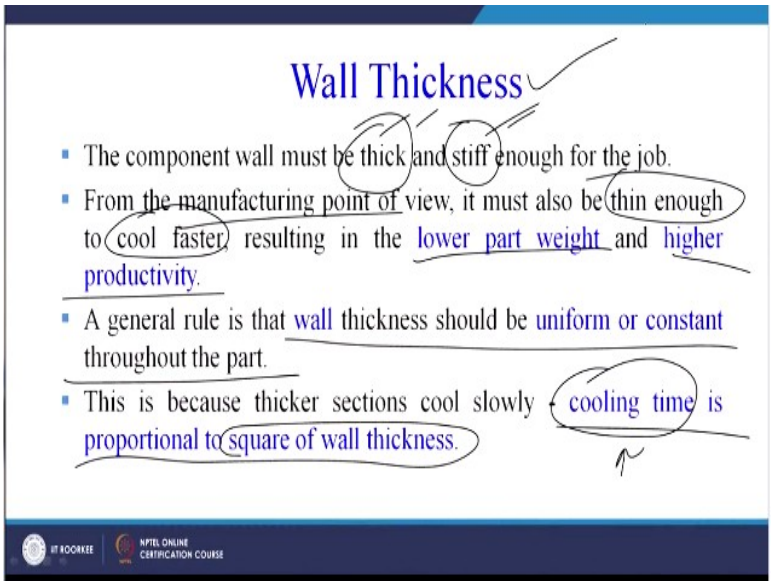
(Refer Slide Time: 20:09)



So, first thing is the production quantities, so all molding processes need a unique mold for specific design of plastic part and this unique mold is very costly**.** So, we have to take into account that the cost of the mold must be justified by the number of products that we produce. Mold is generally costly as I have already told it is important from the economic point of view to consider as I have already discussed production quantities needed while designing a molded part.

So, the quantities have to be forecasted properly when we are planning to use any of the polymer processing technique because the mold will be the costliest part and the mold will specifically adhere to our final product. So, if we are thinking that we have to make 5 or 7 or 10 less than maybe less than 100 parts. The 100 parts may not be able to justify the cost of the mold that we have to make or that we have to bear.

(Refer Slide Time: 21:19)



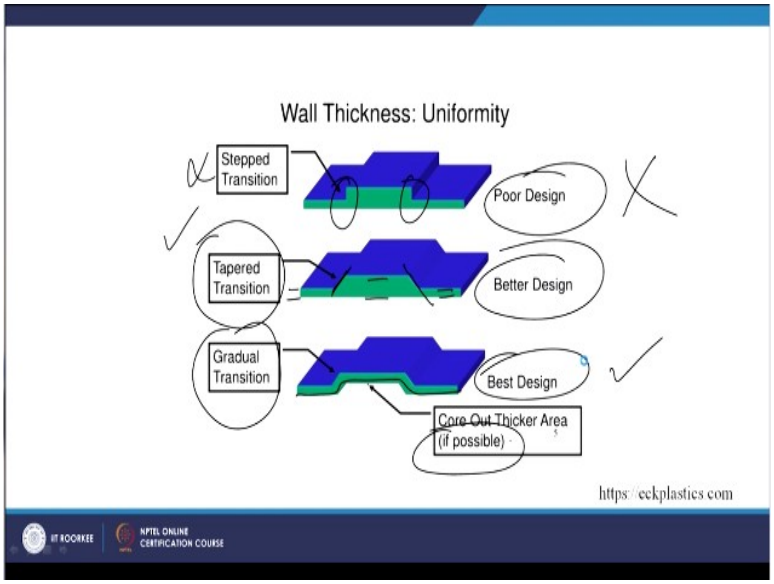
Now what about the wall thickness, the component wall must be thick and stiff enough for the job, you can see and stiff enough for the job. From the manufacturing point of view it must also be from the product design point of view stiff and thick, from the manufacturing point of view it must also be thin enough to cool faster resulting in the lower part weight and higher productivity. So, cool faster means as we have seen the process is the cyclic process, so if the product will cool faster it will solidify faster the mold will open and the product will be taken out.

But if it is takes a long time to solidify the productivity or the number of parts produced per minute or per hour will certainly be affected. So, we have to judiciously select the wall thickness and there are standard recommendations that we must make use of while designing our parts that are to be made using any of the polymer processing techniques. A general rule is that the thickness should be uniform and constant throughout the part.

We will see some examples or with the help of figure that we must have a uniform wall thickness or which must be constant throughout the part. This is because thicker sections cool slowly, cooling time is proportional to square of the wall thickness. And we have seen the importance of the cooling time, so we have to ensure that the part cools at a faster phase in order to ensure the high productivity of the manufacturing processes.

But at the same time we must not make a very thin walled part because it is certainly going to affect the rigidity or the stiffness as well as the performance of the plastic part. So, we have been judiciously select the wall thickness.

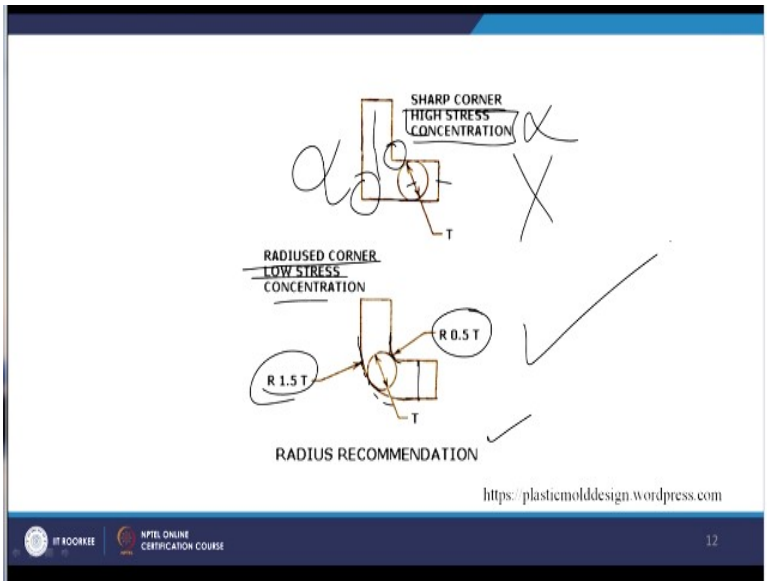
(Refer Slide Time: 23:16)



And here we can see it makes it absolutely clear, this stepped transition here which must be avoided. So, this is an example of a poor design, so we have a tapered transition which is certainly better than the stepped transitions. So, there we have a tapered transition here, here also we have tapered transition it is a better design and but still the wall thickness is not uniform we can see the wall thickness is varying.

And then we figure able to make a gradual transition and the wall thickness also is uniform this can be the best design. So, we can see core out the thicker area if possible, we cannot do it all the cases but wherever possible we can do this and ensure a uniform thickness through the design, so this is we can say the best design.

(Refer Slide Time: 24:11)



Then the corner radii and fillets maybe as we have seen in sand casting and die casting we must avoid the sharp corners. So, the wall thickness increases to 1.4 times the nominal wall thickness when 2 surfaces meet and a corner is formed. We can see the wall thickness increases to 1.4 times the nominal wall thickness when 2 surfaces meet and a corner. So, at a corner always there is a higher wall thickness which we must avoid.

The sharp corners both external and internal should be avoided because of the following causes. We have seen in case of die casting product design also that internal as well as external corners must be avoided there must be a radii that must be given there, why this sharp corners must be avoided. Because the differential shrinkage will take place as we have seen in sand casting we have to modify the product design to avoid the problem of shrinkage cavity that is there.

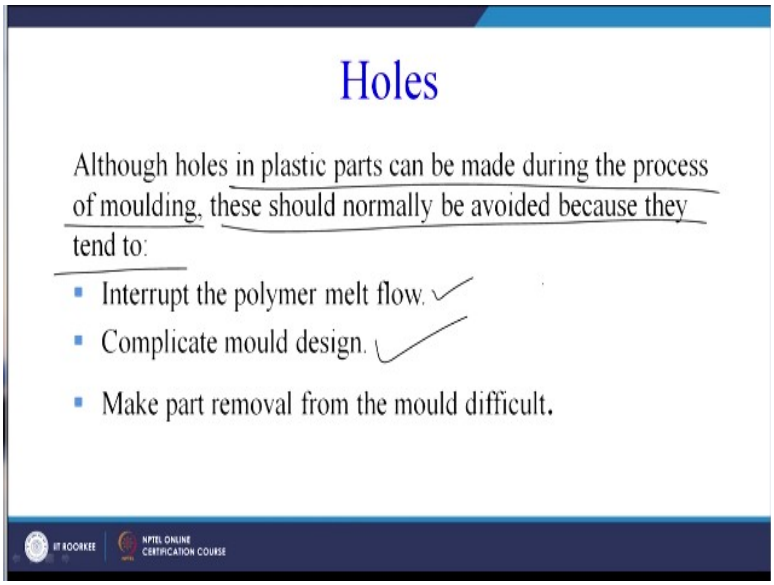
In case of plastics also then molded in stress, longer cooling time, interruption to the smooth flow of the melt, stress concentration in the finished part. So, it is recommended to have a corner radius of 0.5 to 0.6 times the wall thickness. So, we can see that how much corner radius we must give it is recommended 0.5 to 0.6 time the wall thickness. So, which means that we must avoid the sharp corners even in the design of the plastic parts.

(Refer Slide Time: 25:51)

So, here we can see sharp corner here these must be avoided, sharp corner will lead to high stress concentration which is not required. So, the radiused corner low stress concentration here we have seen, so how mush radius we must give which is given. So, this is the thickness it is some portion of the thickness, this is a inner side radius given, this is outer side radius given, so this is a radius recommendations.

So, this design wherever possible in plastics must be avoided it must be modified to a tapered section or maybe a cornered section or with a radius.

(Refer Slide Time: 26:28)

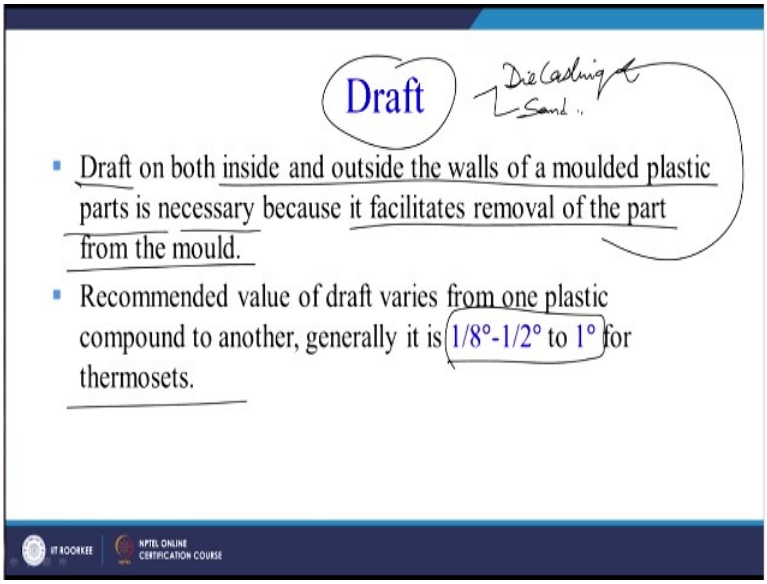


Then the coming to the holes in many plastic parts we will have holes all the holes in plastic parts can be made during the process of molding. These should normally be avoided because they tend to, so during the process of molding only when we are a making a compression molding process. We can put some inserts in the mold cavity and molten plastic will flow and settle around these inserts and later on we can push out these inserts we can generate the holes during the molding process only.

But sometimes that are not advisable why because they interrupt the polymer melt flow, complicate the mold design. Because you have to place them in the mold only and ensure that they are they do not move when the plastic melt flows into the mold cavity make part removal from the mold also sometimes difficult. So, they complicate the mold design, make the part

removal the product removal from the mold cavity difficult as well as they interrupt the smooth flow of the molten plastic inside the mold cavity, so therefore that is one issue that we must take care of.

(Refer Slide Time: 27:41)



Then coming onto the draft, draft we have seen in case of die casting guidelines it is also important in sand casting. And it is also important when we are talking about the forming of the plastic parts. So, draft on both inside and outside the walls of molded plastic parts is necessary why it is necessary. Because it facilitates the removal of the part from the mold, so same explanation we have already discussed in case of die casting.

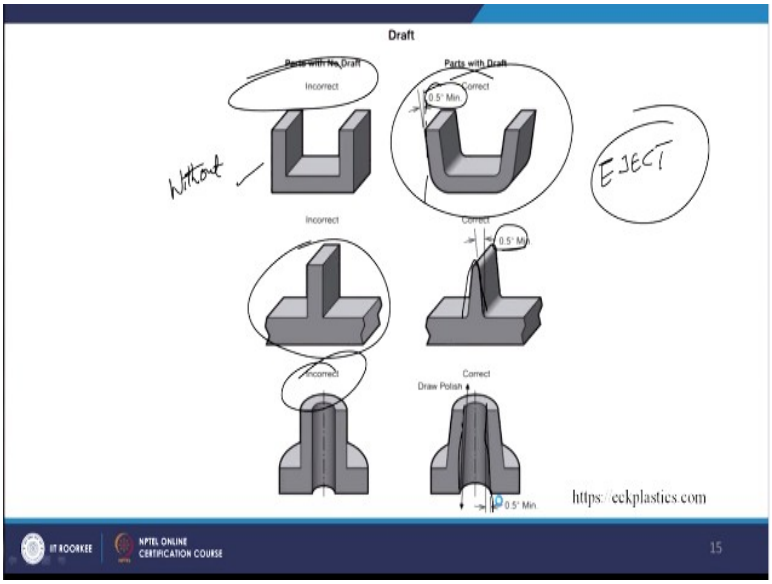
Also that we have to provide a draft at in all the sides which are in a direction perpendicular to the parting line. So, that is as product designer without going into that detailed we must remember when the part has to come out of the mold or what are cavity it must be easily possible. It must not be that we have to apply lot of force to take out our product from the mold cavity, so this draft will help us to easily extract our product from the mold cavity.

So, that is necessary as the product designer, the recommendations are there for thermosetting products 1/8 degree to 1/2 degree to 1 degree. So, that much little draft we must provide, so that it is easy to take out the product, so recommended value of draft varies from 1 plastic compound

to another. A general recommendation is given, so we need not give 25 degree, 30 degree draft because it will entirely change the design of the product.

But yes a very small angle from one 1/8th of the degree to 1/2 a degree to 1 degree must be given. So, that it is easy to withdraw the part let us try to see with a help of a diagram.

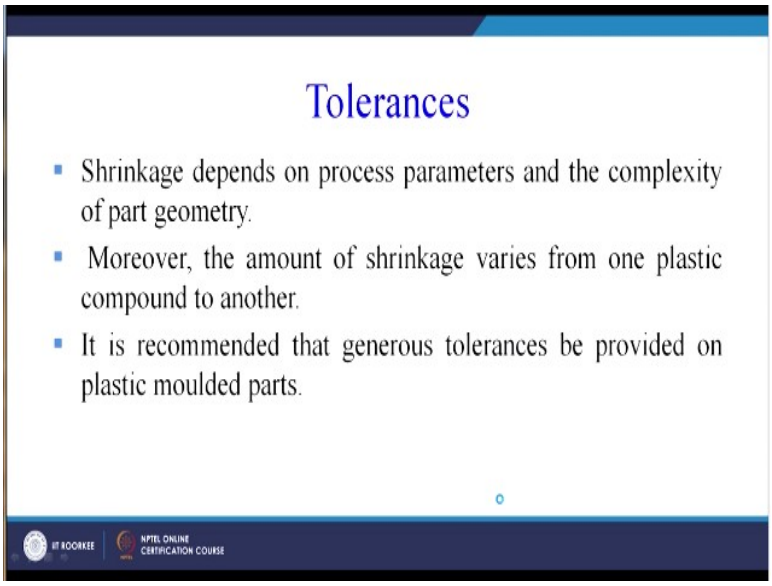
(Refer Slide Time: 29:30)



It is a incorrect design, this is a design without any draft and here we can see a draft is given if you draw up a straight line here, you will see that there is a angle 0.5 degree minimum value of draft. So, once this product is made it is easy to be extracted from the mold cavity. Here also we see we have seen a similar design earlier also during die casting, so here also little bit of draft can be provided on this product.

So, that it becomes easy to extract or eject, eject I think is the better word, eject the part from the mold cavity. Here also there is no draft given which is an incorrect design, so we can give a draft here to be ensure the easy withdrawal of the product from the mold.

(Refer Slide Time: 30:22)



Now coming onto the tolerances, so we can see that whenever the plastic product will cool down or when the molten plastic will cool down there is certainly going to be shrinkage. So the shrinkage depends on the process parameters and the complexity of the art geometry. So, depending upon the shape of the product or the product complexity in terms of shape there maybe variable shrinkage that may take place.

Also depending upon the type for which the type of polymer that we are using the shrinkage may vary. So, moreover the mount of shrinkage varies from one plastic compound to another it is recommended the generous tolerances we provide on the plastic molded parts. Normally the designers do not account for the shrinkage usually the design of the mold is done as per the exact product that we are going to make or as per the dimensions of the design, specifications of the product.

But as a person or a product designer with basic idea of the manufacturing processes also it is important to take into account the shrinkage of plastic during the solidification stage. As well as during the product design process during the design only we must plan that this much shrinkage may take place and we can make use of the tables that are available. And those tables can help us to make these decisions that how much we can make using the how much shrinkage announce or maybe how much shrinkages going to take place.

And how we must account for this in our product design. So, with this I think we will conclude the today's session we will start our discussion with we have not been able to cover the extruded parts or maybe the specific guidelines for extrusion which we leave for the next discussion. So, we will start the next session with design guidelines for the extruded parts and then carry forward our discussion for the other processes which are generally use for processing of different materials. Our focus will be on the design guidelines for the various processes, thank you.