Manufacturing Guidelines of Production Design
Prof. Inderdeep Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology-Roorkee

Lecture-22
Design Guidelines for Extrusion and Injection Molding

Namaskar friends, so welcome to session 22 of our course on manufacturing guidelines for product design. So, we have started now discussing the individual guidelines or the specific guidelines for specific processes. And if you go back to the previous session or maybe 2, 3 sessions maybe session maybe 19th, 20th, 21st our focus has been on specific processes. We have already discussed the specific design guidelines for sand casting process.

Specific design guidelines for die casting process, specific design guidelines for compressing moulding which is a very important as well as commercial process for manufacturing of plastic parts. Whereas sand casting and dye casting are important for giving shape or for forming the metallic parts. So, we have seen that how the product design can influence the process or how the design can help us to make the products of good quality.

Many times the designers are not aware of these guidelines and therefore they design the parts as it looks good to them. Both the functional as well as the aesthetic considerations of product design are taken care by the designers. But sometimes in many cases the designers are not able to appreciate or not able to highlight or not able to emphasize the manufacturing design guidelines. So sometimes they do not consider and they just feel that the product is looking good, it is satisfying the function for which the product has been designed.

And therefore they pass on that design to the manufacturing stage, but these days the time is a very important factor. So we do not want to waste any time and therefore the product designers must make use of these guidelines while finalizing the design of their product. So that when the product goes into the manufacturing stage it is manufactured to the best possible quality in the most efficient and effective manner and therefore this course is important.

Now coming onto the specific processes that we are discussing all of you are well aware that we have already seen as I have just now told guidelines for sand casting, guidelines for die casting, guidelines for compression moulding. And in the last session we started we thought that we will be able to cover the design guidelines for both compression moulding and the second one was the extrusion but extrusion we were not able to cover in detail.

So today we will start our discussion with extrusion and then finally jump over to injection moulding and most of the times you will see that the guidelines are majorly related to some specific design attributes. Again I will reiterate those design attributes, we are talking about the wall thickness what can be the wall thickness that can be achieved by a specific process, we are talking of ribs and bosses which are sometimes used to strengthen our part give stiffness to our part.

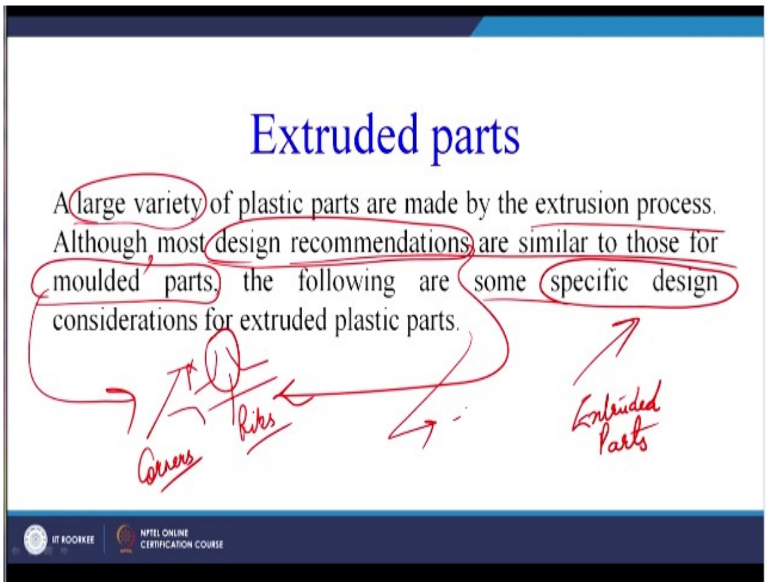
We are talking about the wall thickness as I have already told the ribs, we are talking about the tolerance that we can achieve. We are talking about the surface finish that we can achieve. We are talking about the recesses that we can create within a product, so more or less the geometrical attributes are same but specific processes have specific characteristics or the special advantages that.

For example if you see injection moulding it has a specific application that it can be use for thin walled, complex shaped plastic products. So it has a special characteristics whereas in case of other processes which are there for polymer processing, it may sometimes become difficult to process the very thin plastic parts. So, therefore the overall attributes or the design attributes more or less are same but the process is differ based on their special characteristics.

So, today our target is to first learn about the extrusion all of you know in the previous sessions, session number 21 we have focused on exclusion that what is extrusion process how the product is made. And that is not basically our target, our target is to understand the design guidelines for the extruded parts and of we will focus on those design guidelines today. So, in case of extrusion we have seen that it is use for making continuous long parts with uniform cross-sections.

The parts can be hollow or they can be solid or depending upon the profile of the part the process will become more and more complex. If it is a simple cylindrical or a tubular long part it is easy to be made, it can be easily made or it is easy remanufactured by the process of exclusions. So, that is basically our target that let us see now the part must be designed, so that it is easy to extrude, so let us quickly now start our discussion on extruded parts.

(Refer Slide Time: 05:53)

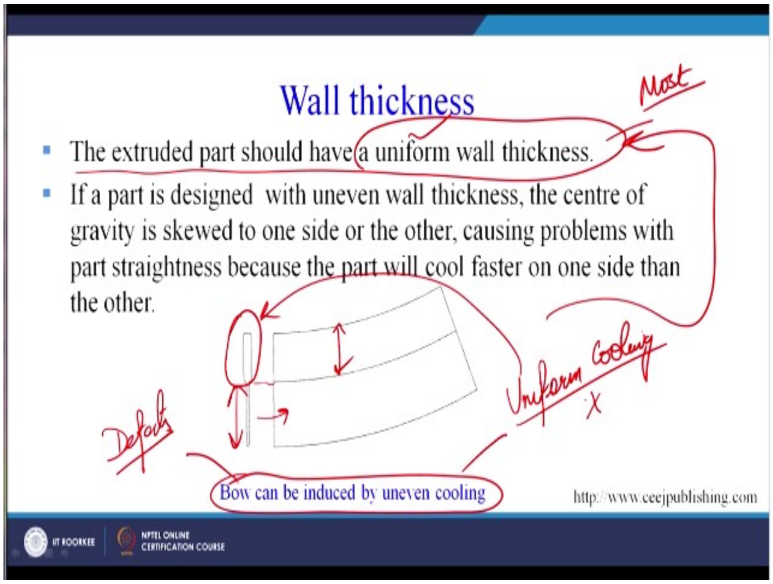


So, a large variety why variety because depending upon the cross-section of the part we can have a large variety of parts. A large variety of parts are made by the extrusion process although most design recommendations are similar to those for the moulded parts. So, this is what I have already highlighted, sometimes we say sharp corners must be avoided that must be rounded off sometimes we give the recommendations regarding the ribs, corners.

So most of the recommendations design recommendations focused on are focused on the same geometrical features. But still the processes have different capabilities, so although most designed recommendations are similar to those for moulded parts, the following are some specific only for extruded parts. So what we are going to cover now are the specific design requirements for the product which are to be made by the extrusion process.

So specific design considerations for extruded plastic parts are what we are going to see in the subsequent slides.

(Refer Slide Time: 07:16)



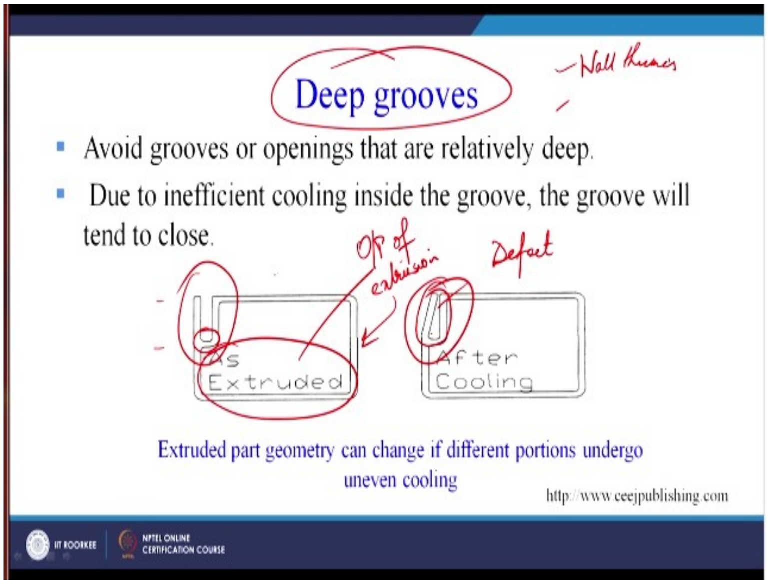
Now we can see regarding the wall thickness, very very important guideline the extruded part should have a uniform wall thickness which is common to most of the processes which are use for making the plastic parts. Now here we can see here the cross-section is not uniform, here the cross-section is different above this line and here the cross-section is different. So when the cross-sections are different we can see a bow can be induced by the uneven cooling.

So, it will not lead to uniform cooling basically we wish that uniform cooling must take place but in many cases this uniform cooling may not take place why because of the difference in the cross-section. And therefore this type of a bow is created and this is one of the defects that may take place and therefore we must ensure that uniform wall thickness is there in our plastic part that we have designed.

If a part is designed with uneven wall thickness, the center of gravity is skewed to one side or the other causing problems with parts straightness because the part will faster on one side than the other. So, already this has been highlighted that uneven cooling must be avoided and uniform cooling must be ensured. So, uneven cooling will lead to defects such as bow and may even lead to sometimes the warpage or the dimensional inaccuracies also.

So, that is one defect in regarding wall thickness we must ensure a uniform wall thickness then whenever we have to make the deep grooves.

(Refer Slide Time: 09:07)

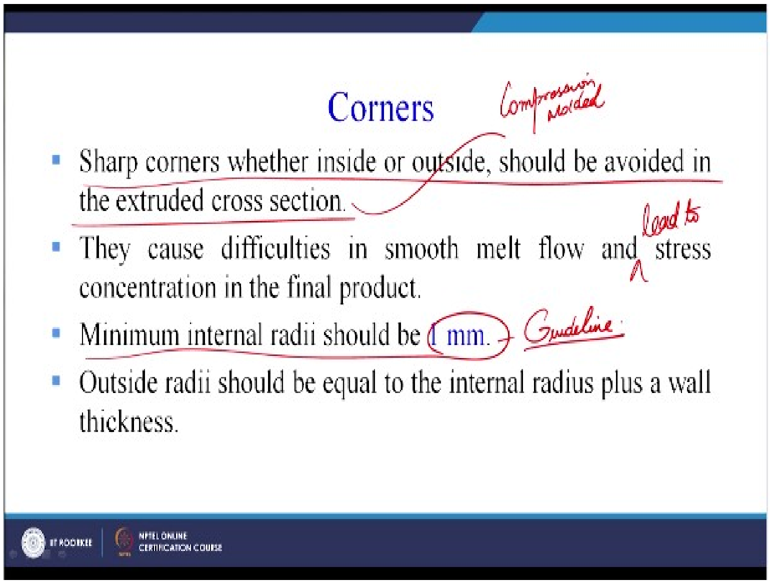


The first guideline was related to if I can just revise here wall thickness, second is related to the deep grooves, when we have to make the deep grooves like these avoid grooves are openings. That are relatively deep, so this is fairly deep groove and when such types of grooves have to be made what can be the problem. The problem can be due to inefficient cooling inside the groove may be here, the groove will tend to close.

So, this is as extruded, this is something which is the output of extrusion process, so this is the part which has been extrude after extrusion. And when it cools down you can see that this is the defect which has taken place because of the deep groove and non uniform cooling or insufficient cooling at the center or inside the groove. So the extruded part geometry can change if different portion undergo uneven cooling.

So we must design our extruded part in such a way that we ensure uniform cooling of the extruded part after the extrusion process. So, that is one standard guideline, so 2 guidelines we have seen regarding the uniform wall thickness and we must avoid deep grooves in the extruded parts. Now the corners the uniform this is one of the standard practices that we must avoid sharp corners.

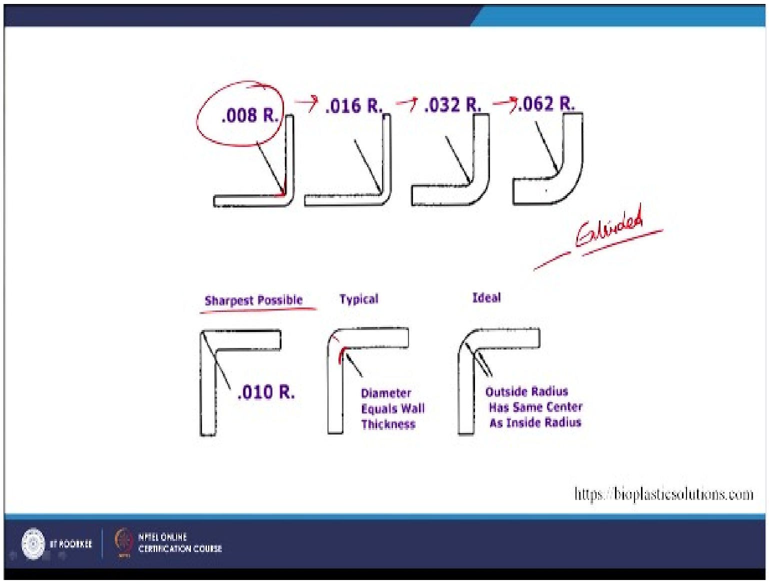
(Refer Slide Time: 10:49)



Sharp corners whether inside or outside should be avoided in the extruded cross section similar things we have seen in our previous session when we talked about the compression molded parts. So, in that case also we have seen sharp corners must avoided, if sharp corners must also be avoided in sand casting as well as in die casting why because they cause difficulties in smooth melt flow and stress concentration in the final product.

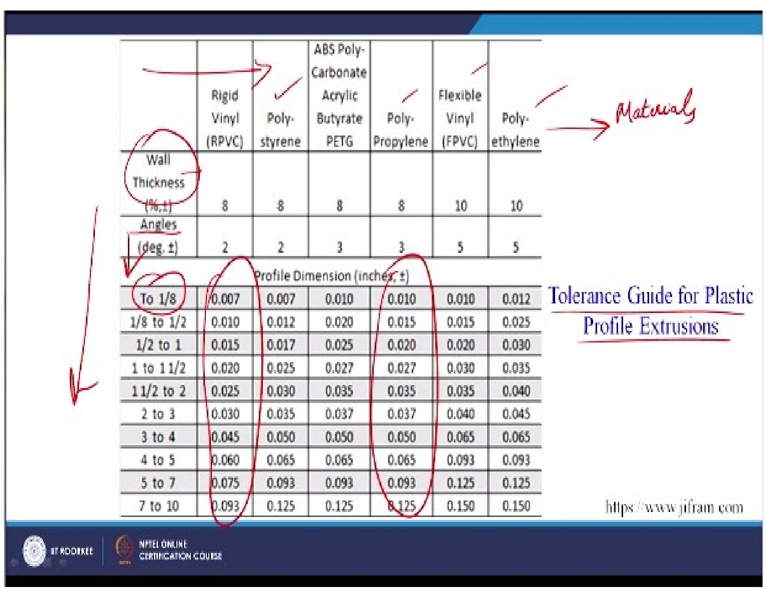
So the sharp corners often lead to stress concentration in the final product, so minimum internal radius must be 1 millimeter. This is one guideline which must be kept in mind, the minimum internal radius must be 1 millimeter outside radius must be equal to the internal radius plus a wall thickness. So if we have a wall thickness the outer radius can be decided based on the wall thickness plus the internal radius and minimum internal radius must be equal to 1 millimeter or minimum must be 1 millimeter. So in minimum is 1 millimeter, outer will be minimum plus the thickness of the part.

(Refer Slide Time: 12:03)



So, here we can see this must not be sharp, it must have a radius which is given here, so different radius are given, we can select as per our design geometry. So this is the sharpest possible is given here, the diameter equals the wall thickness, ideal outside radius has same center as well as the inside radius. So, when we are designing our extruded parts we must take care of the radius guidelines which are given here both for the internal section as well as for the external sections.

(Refer Slide Time: 12:43)

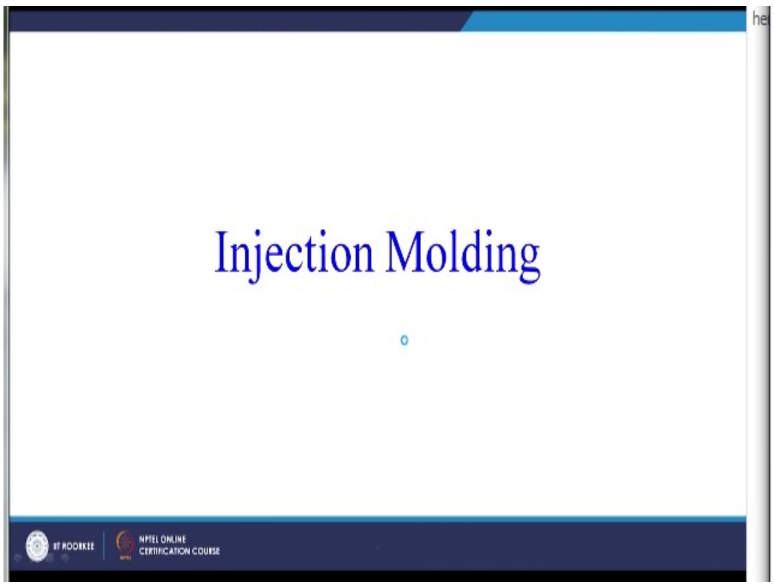


	Rigid Vinyl (RPVC)	Poly-styrene	ABS Poly-Carbonate Acrylic Butyrate PETG	Poly-Propylene	Flexible Vinyl (FPVC)	Poly-ethylene
Wall Thickness (%,±)	8	8	8	8	10	10
Angles (deg. ±)	2	2	3	3	5	5
Profile Dimension (inches, ±)						
To 1/8	0.007	0.007	0.010	0.010	0.010	0.012
1/8 to 1/2	0.010	0.012	0.020	0.015	0.015	0.025
1/2 to 1	0.015	0.017	0.025	0.020	0.020	0.030
1 to 1 1/2	0.020	0.025	0.027	0.027	0.030	0.035
1 1/2 to 2	0.025	0.030	0.035	0.035	0.035	0.040
2 to 3	0.030	0.035	0.037	0.037	0.040	0.045
3 to 4	0.045	0.050	0.050	0.050	0.065	0.065
4 to 5	0.060	0.065	0.065	0.065	0.093	0.093
5 to 7	0.075	0.093	0.093	0.093	0.125	0.125
7 to 10	0.093	0.125	0.125	0.125	0.150	0.150

Now this is the tolerance guide for plastic profile extrusions, we can see the tolerance guide for the profile extrusions. So, here we can see wall thickness with wall thickness how the tolerance will change and these are the materials in this direction. This is rigid vinyl, polystyrene, acrylonitrile butadiene styrene, polycarbonate acrylic butyrate, poly propylene, flexible vinyl, poly ethylene, so this direction is giving us the materials.

In this direction this column we have the wall thickness, now depending upon the wall thickness we can see for a material that this is angle which is given in this direction, this is a wall thickness which is given. So, depending upon the wall thickness and the angles that are given the profile dimensions the range is given for the different material. This is for poly propylene mostly we use it for making plastic parts or extruded part. So this is the tolerance that we must give to our plastic parts when we are designing the part.

(Refer Slide Time: 14:05)



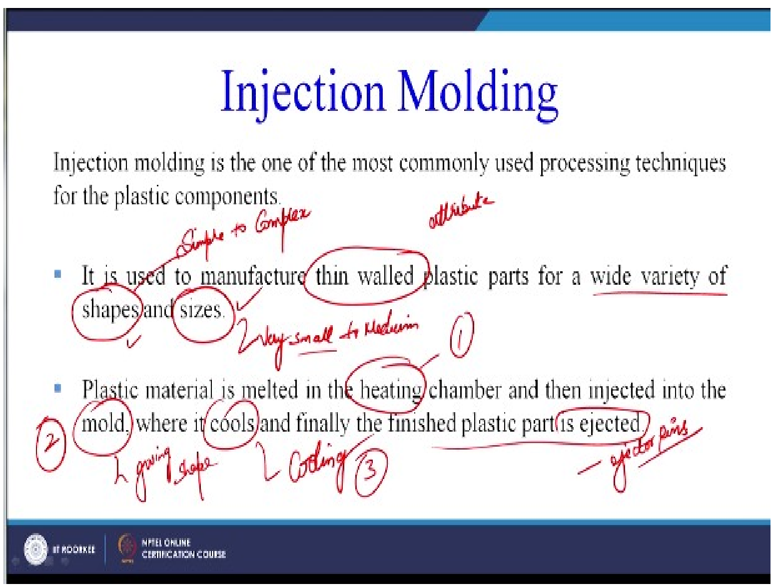
So, we can see that there are standard guidelines related to the extruded parts. Now if we can go into the much more detail we can further see for a specific material. We can take a design and see the how the part can be designed. So, if the time permits in our course towards the end we will certainly take few case studies where we will select a part, select the design of a part and see that which what all guidelines have been taken care of.

We can compare that this is a poor design we have designed it changed the profile from this to this, sharp corners have been avoided, deep grooves have been avoided, the angles have been selected properly, the tolerance has been selected properly, the shrinkage has been taken care. So, we will certainly take 1 case or 2 case study where we will show the design of a plastic part or the redesign of a plastic part for better manufacturability.

But the broad things remain same that we must take care of all these guidelines related to corners, related to ribs, related to design of deep grooves, related to uniform wall thickness. So, all these parameters must be taken into account when we are designing a part which is going to made by a plastic material or a polymer material as well as going to made by extrusion. Similarly now another commercial process which we usually see around us is the injection molding process.

And most of the plastic parts that we use are made by injection molding on a commercial scale, now what is injection molding, quickly we will like to see.

(Refer Slide Time: 15:43)



So, injection molding is one of the most commonly used processing techniques for the plastic components. It is use to manufacture thin walled, this is one of our attributes or design attribute which we are focusing again and again it is use to manufacture thin walled plastic parts for a wide variety two other attributes are coming shapes and sizes. So we can see for a wide variety of shapes which can be from simple to complex and sizes from very small to medium, will never be use for very large sizes but from small to medium sizes the process of injection molding can be used.

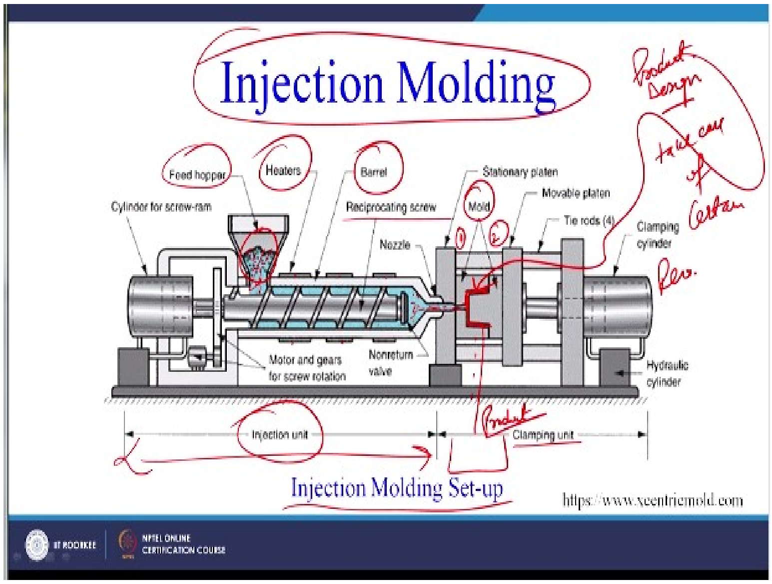
Now within one sentence we have got 3 design maybe attributes which can be characterized for the injection molding process. First thing is thin walled sections can be made, second is wide variety of shapes can be manufactured, third is from very small to medium size products can be made using the injection molding process. Now plastic material is melted in the heating chamber as I have already told for molding of plastics.

There are 3 steps only first one is heating and then injected into the mold, so second one is giving shape inside the mold giving shape and the last is the cooling process. So, for any plastic molding process these 3 steps maybe common mostly they will be common. First one is heating or melting the plastic, second one is molding or giving it the shape and third one is cooling and finally the finished part is ejected out, how it is ejected.

If you remember some of the diagrams we have already seen with the help of the ejector pins which we have already seen in the compression molding process or the compression molding diagram also. So, basically in injection molding the 2 halves of the mold or the die will close inside there will be cavity the plastic in the form of pellets mostly for thermo plastic materials, injection molding is used will go into the barrel which is heated.

So, we have to control the temperature profile within the barrel and then this material is pushed into the die cavity in the molten state and it takes the shape of the die cavity or the mold cavity and finally solidifies into the final product that we want to produce. So let us now try to see it with a help of a diagram.

(Refer Slide Time: 18:22)



This is the final product that we are trying to make this shape is the product that is going to be made. And we can see that the mold is in 2 halves this is the mold given here, this is 1 half of the mold, this is second half of the mold and this is the barrel here. I have use the word barrel just few minutes back, so there is reciprocating screw which is rotating, this is the feed hopper. So, the raw material, this is the raw material in the form of pellets or thermoplastic pellet.

So, the raw material comes into the barrel we have a reciprocating screw and there are heaters as we have seen in case of extrusion also this is heater, heating elements. So, you have heat and the material is sheared by the reciprocating action of the screw. And then this molten material from here is pushed into the through this gate is pushed into the die cavity or the mold cavity.

And once the material have solidify the 2 halves of the mold open and the material may fall down into the basket or into the maybe collecting chamber. So, you have a clamping unit then there is a injection unit here. So, all this is injection unit and this is a clamping unit where the two halves of the mold will close and will form a die cavity inside. So, this is the injection molding setup, what is our target, what we want to learn.

We do not want to learn the process of injection because this we have already covered in our course on processing of non metals or processing of polymers and polymer composites. Our target here is to see how this design of the part or the product design must take care of certain recommendations. Now what are these recommendations for the design of this part because if we do not design this part properly our injection molding process will not be able to produce the product as per the requirement.

So, let us now quickly see that what are our specific design requirements for this part which has to be made by the injection molding process.

(Refer Slide Time: 20:55)

So, here very very colorful slide, why this colorful slide is there because we see different injection molding parts in different colors. If you see the plastic buckets that we use, so many colorful buckets are there green, red, blue, so we have different types of raw material which can be used. But most of the polymers that we use or generally white or half white in those maybe white family only.

But we add the coloring agents or the coloring pigments to give the desired colors to the products. So here we see, so many products wide variety of products which can be made by the injection molding process. And if you have a closer look I think in the slide you may not be able to see there are these products have no sharp corners. If you all these products maybe this is one product, this is another product.

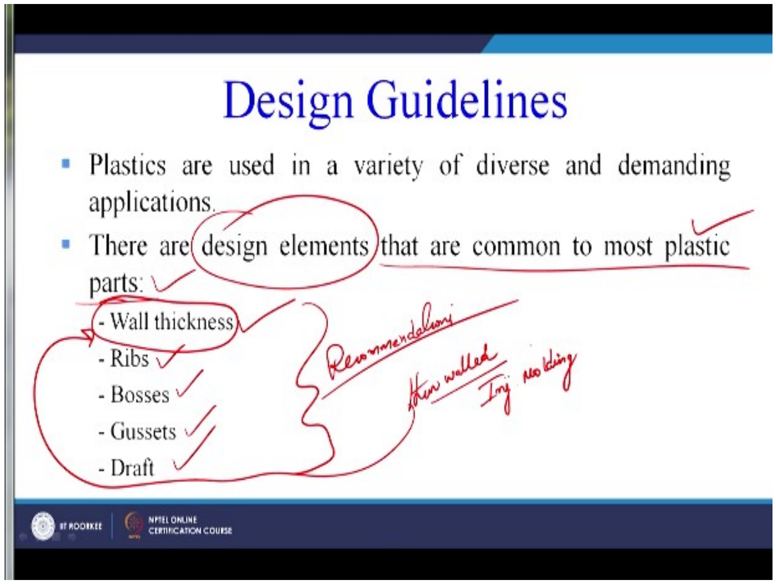
So, when you see these products we will definitely realize that these products would not have any sharp corners. Then another guideline that they will have uniform wall thickness as an engineer if you look at a injection molded part we will see that the designer has already care that the uniform wall thickness must be maintained. So, if ribs are there the thickness of the ribs will be decided based on the wall thickness.

So, basically these guidelines do exist and engineers definitely make use of these guidelines but for new designers people who want to venture into the product design field these guidelines are mandatory. So many times new designers come up with designs which are very good but are not manufacturable why because these guidelines have not been taken care of. So, for on now on if you are a learner of this course you can just look at the plastic products around you.

And see that whether guidelines which are there for designing of parts which have to be made by injection molding or compression molding or blow molding process. Whether those guidelines have been taken care of or not, if they are taken care how the product is better as compare to a product in which the guidelines have not been taken care. So, basically here we can see different types or different colors of inputs can be there and the products are also shown here and most of the actual products are shown here.

So, these are the injection molding products and what are the guidelines 3 of them already I have written regarding the sharp corners regarding the wall thickness regarding the design of the ribs. Now quickly let us see the specific guidelines for all these attributes of the design attributes that we have seen, so what are the design guidelines.

(Refer Slide Time: 24:01)

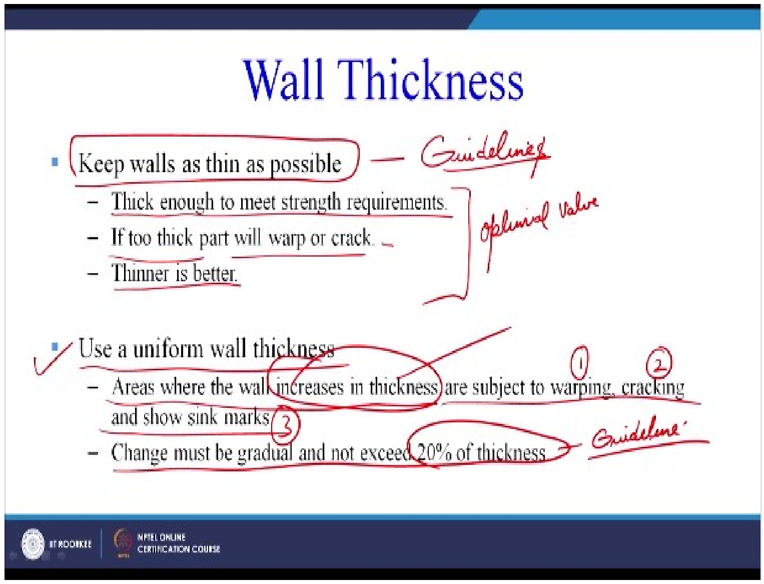


So, plastics are used in a variety of diverse and demanding applications, in the previous slide there is nothing much to say about this points, so many different applications, diverse applications we have seen. So, there are design elements that are common to most plastic parts because we have been discussing this again and again common to most of the plastic parts, what are these design elements one by one we can see wall thickness.

We have seen it for compression molding, we have see it for extrusion, ribs again we have seen, bosses, gussets, drafts. So, all these recommendations for all these may vary depending upon the process that we are choosing. So, if you remember recommendations may vary, if you remember in the start of our session on injection molding we have seen that it is used for thin walled parts which means that the first guideline regarding wall thickness is specific to injection molding.

So, wall thickness has to be thin in case of injection molding whereas it can be different in case of the other processes. So, wall thickness will vary if you choose injection molding you can use it for thinner wall sections also but if you are using compression molding the wall thickness range may vary, if you are using extrusion the wall thickness range may vary. So, let us now see for injection molding wall thickness.

(Refer Slide Time: 25:38)



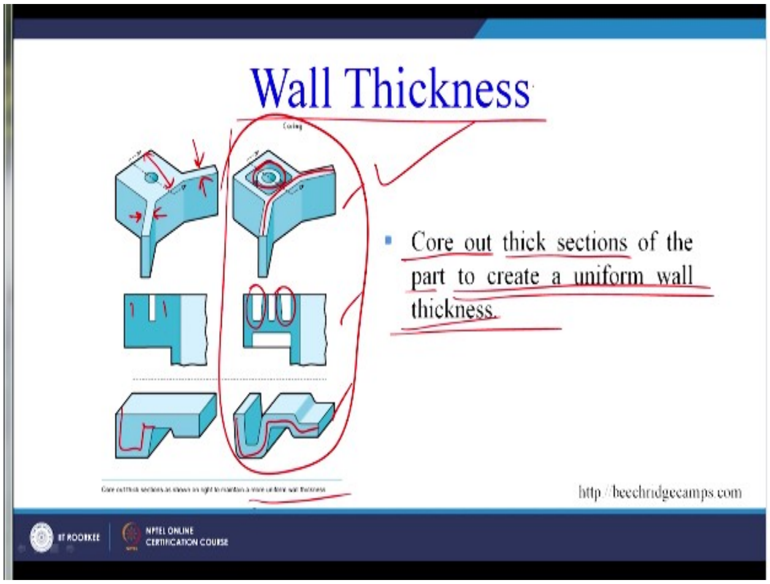
Keep the walls as thin as possible one guideline, this is a guideline for parts to be made by injection molding process one guideline sorry. Keep the walls as thin as possible thick enough, so which is the larger limit thick enough to meet the strength requirements, thinner is always better. If too thick part will warp or crack, so we have to see the optimal range or the optimal value for the thickness it must not be too thick if the wall thickness is too high or too large there are chances of warpage.

If it is too thin there are manufacturability issues, we may not be able to manufacture it to the dimensions. So, we have to optimize the wall thickness but certainly the wall thickness values for injection molding can be on the lower side because injection molding is a process which can be use for making thin walled parts. Second guideline, use a uniform wall thickness which is common to most of the plastic processes.

Areas where the wall increases in thickness are subject to warping, cracking and show sink marks. So, problem number 1 warping, problem number 2 cracking, problem number 3 sink marks why because of the increase in the thickness. So, areas where the wall increases in thickness these are the problem, so therefore we must optimize the thickness of our wall, change must be gradual and not exceed 20% of the thickness, this is another guideline.

So we must ensure gradual change in the wall thickness, so if it is abrupt change in the wall thickness it will create problems. So, let us see this with the help of an example regarding the wall thickness.

(Refer Slide Time: 27:47)

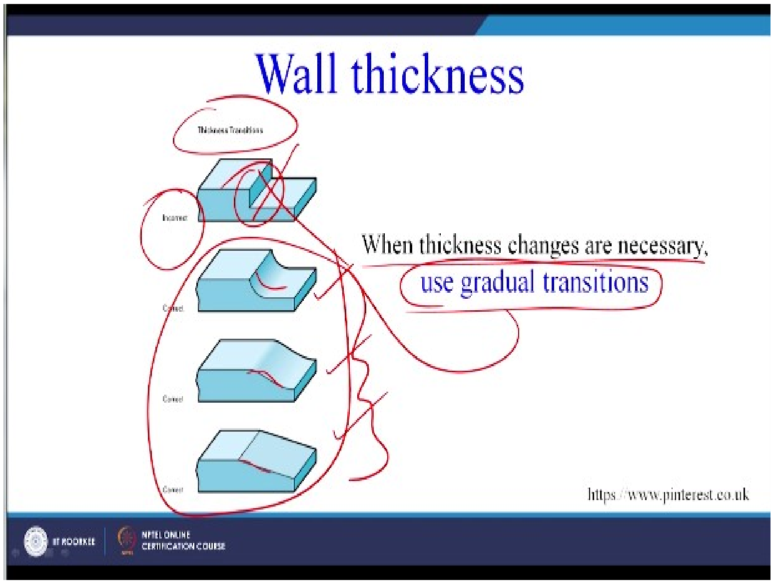


So, here we can see this is section which is cut here, so here this is the wall thickness. But if you see here the wall thickness is much more, so what we can do core out the thick sections. So, what we have done we have cored out from here thick section and now in this range we are having a uniform wall thickness. So core out the thick sections of the part why to create a uniform wall thickness, this is our target.

So, we can create a uniform wall thickness, here also you can see thicker portion we have cored it out. So, that we can ensure a uniform wall thickness similar is the case here from here we can ensure a uniform wall thickness. So, it is cored out from here, so that we can produce a uniform wall thickness here we have ensure a uniform wall thickness. So, this is the correct parts, correct designs, core out thick sections as shown on right to maintain a more uniform wall thickness.

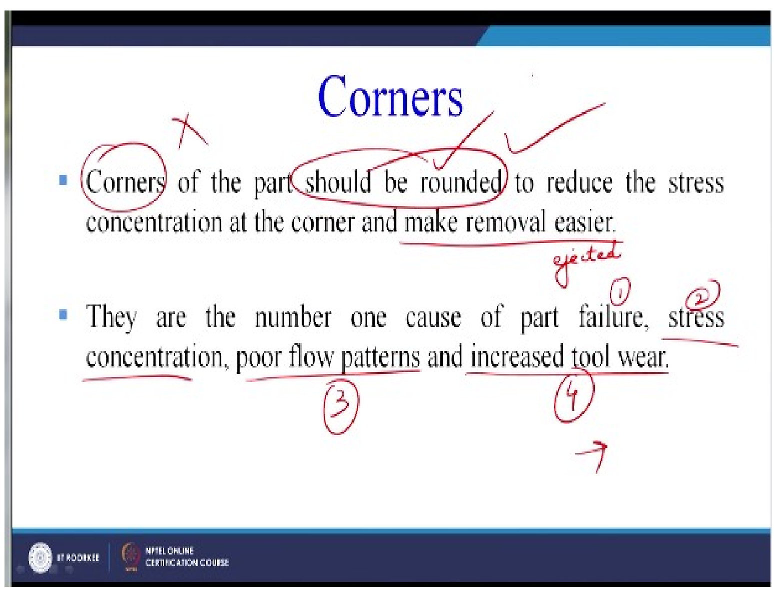
So, if we ensure a uniform wall thickness it will definitely help us to avoid the defects such as warping, such as cracking and it will lead to a good quality of the product. Now here also we can see the thickness transition which was our second point, poor design incorrect it must never be used.

(Refer Slide Time: 29:19)



But yes gradual change is always advisable here also gradual change advisable, here also to some extend it is advisable. When thickness changes are necessary you cannot avoid the thickness changes use the gradual transitions which is given here. These are the gradual transitions whereas this is not recommended a very sharp transition in the cross section.

(Refer Slide Time: 29:45)

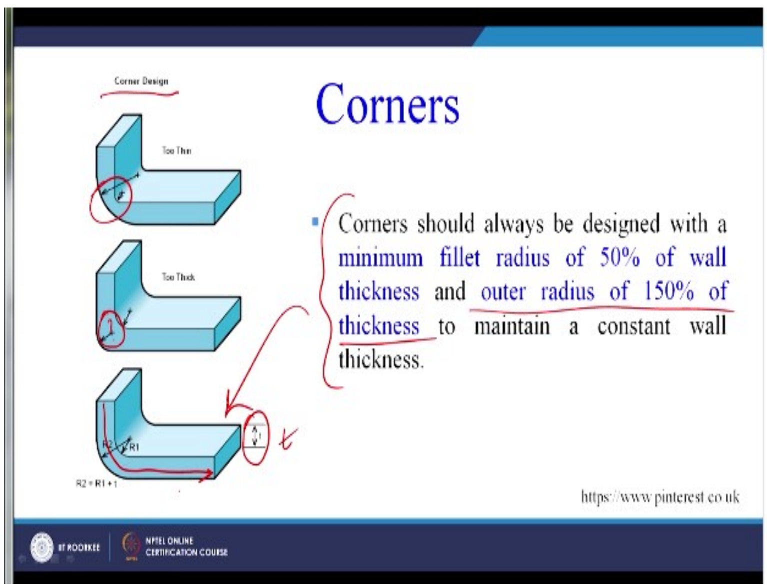


Then regarding the corners, corners of the part should be rounded very good guideline to reduce the stress concentration at the corner and make the removal easier. Because the part has to be ejected out of the mold and therefore to make the ejection process easier we must avoid the use of corners. They are the number one cause of failure, stress concentration, poor flow patterns of the melt or the molten plastic and increased tool wear.

So if you have very sharp corner the plastic is again and again come in contact with the corner or the die corner of the mold. So, there are chances of increased tool wear also, so the sharp corners are leading to so many problems like part failure, stress concentration, poor flow pattern, increased tool wears. So, these problems can easily be avoided by giving a radius at the corner.

So, the corners of the part should be rounded, so they can be made rounded to reduce all these problems or mitigate the effect of all these defects that may take place.

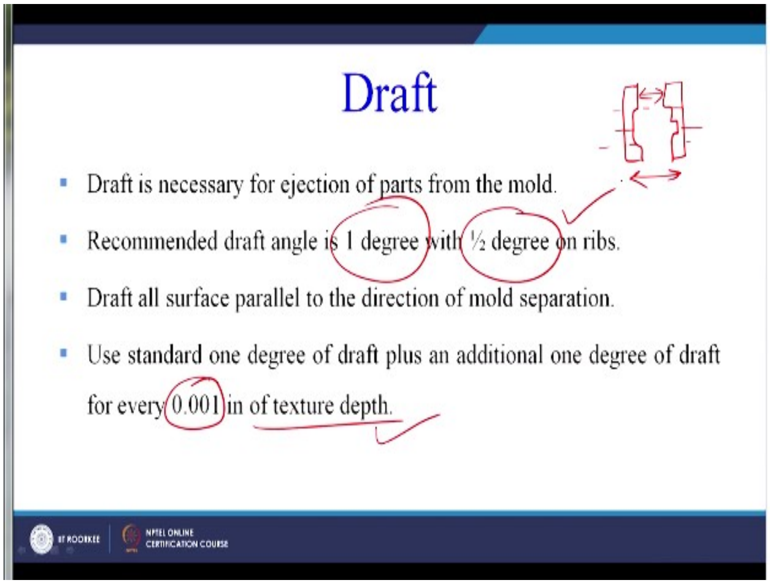
(Refer Slide Time: 31:02)



Now this is a very good example, so it is too thin must be avoided this is a corner design too thin. This is too thick there can be a problem here as we have seen in case of sand casting shrinkage cavities are form. So, here also too thick may lead to problems but here this is the right guideline which is also mentioned here you can see. Corner should always be designed with the minimum fillet radius of 50% of the wall thickness, so this is a wall thickness t here 50% of the wall thickness.

So, fillet radius must be 50% of the wall thickness and outer radius this is related to the inner radius, the outer radius of 150% of the thickness to maintain a constant wall thickness. So, this is already we have seen along this direction there is a constant wall thickness by ensuring proper rounded corners, now coming onto the draft.

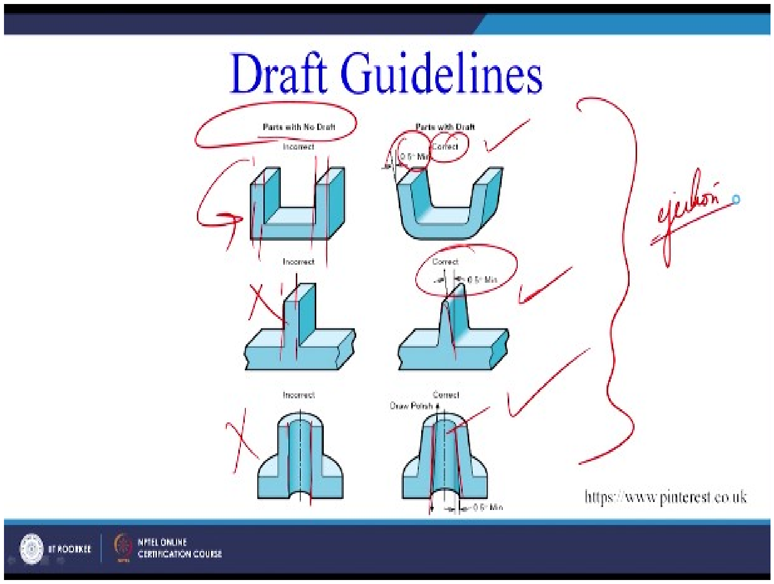
(Refer Slide Time: 32:01)



So, draft is necessary for the ejection of parts from the mold which we have already seen, recommended draft angle is 1 degree with 1/2 degree on the ribs. Draft all surface parallel to the direction of mold separation. Now suppose these are our 2 mold halves and in between we have the cavity when these will close this is a direction of our mold separation, so draft all surfaces parallel to the direction of mold separation.

So, whatever surfaces are there which are parallel to the direction of mold separation there we must provide some draft the recommended values for draft are already given. Use standard 1 degree of draft plus an additional 1 degree of draft for every 0.001 inch of the texture depth. So, if there is a texture then accordingly we have to modify our value of the draft that we have to provide well the general values are given here.

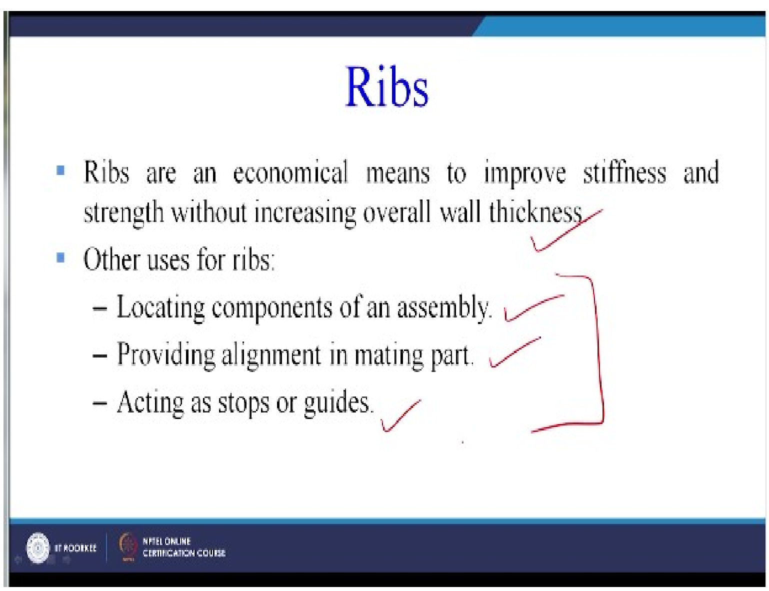
(Refer Slide Time: 33:07)



So, these are the draft guidelines which we can see parts with no draft, in this part there is no draft you have a straight surfaces. But here we have a draft this is given here 0.5 degree minimum, so this is a correct design of a part. This is again on the ribs also the draft is given, in this case straight surfaces, no draft is given here.

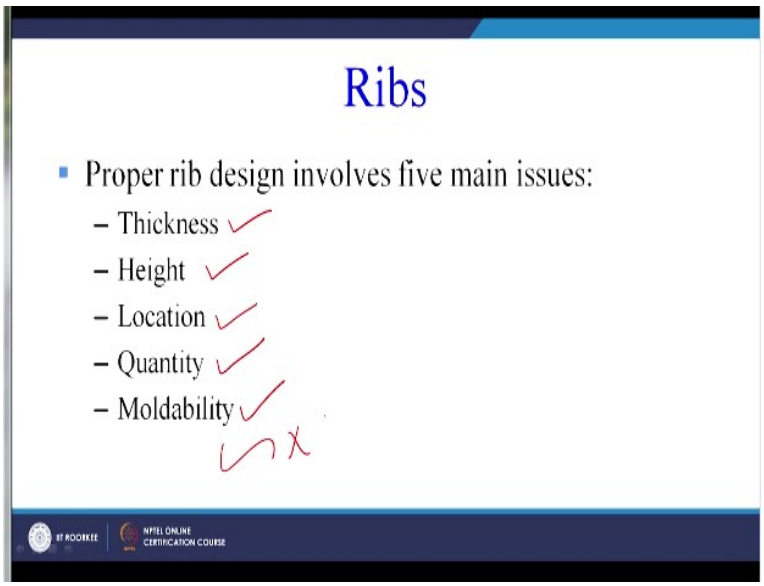
But here the draft is given here also straight surface, no draft in this case again it is made in angle, so the draft has been provided. So, this draft values will definitely help in the easy ejection of parts after the molding has been completed.

(Refer Slide Time: 33:57)



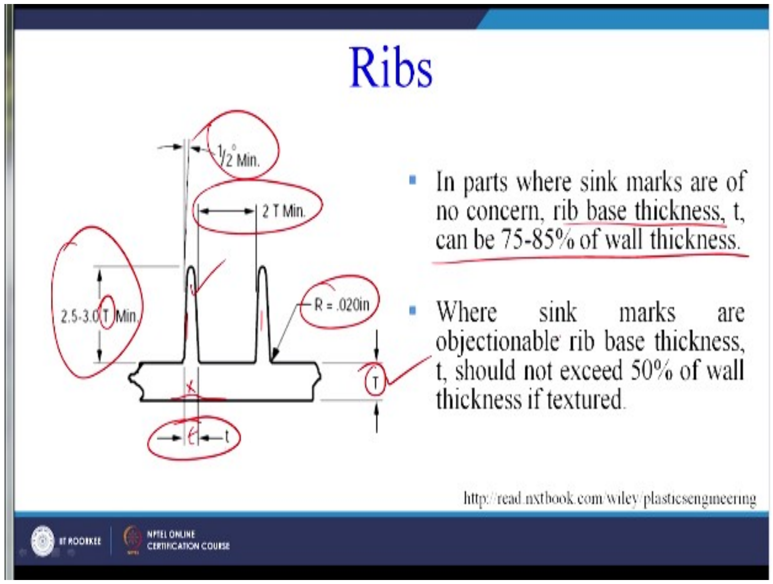
Now coming onto the ribs, ribs are an economical means to improve the stiffness and strength without increasing the overall wall thickness. Other uses for ribs, so all of us know that why do we put ribs in our product design, other uses for ribs are locating components of an assembly providing alignments in the mating part and acting as stops or guides. So, these are other uses for providing the ribs.

(Refer Slide Time: 34:19)



Now proper rib design involves 5 main issues we can see what must be the thickness of the rib, what must be the height of the rib location, quantity and moldability. Sometime we may have a rib design which is difficult to mold, so we must ensure that all these 5 parameters that is a thickness, height, location, quantity and moldability is taken care when we design ribs in our product design.

(Refer Slide Time: 34:48)

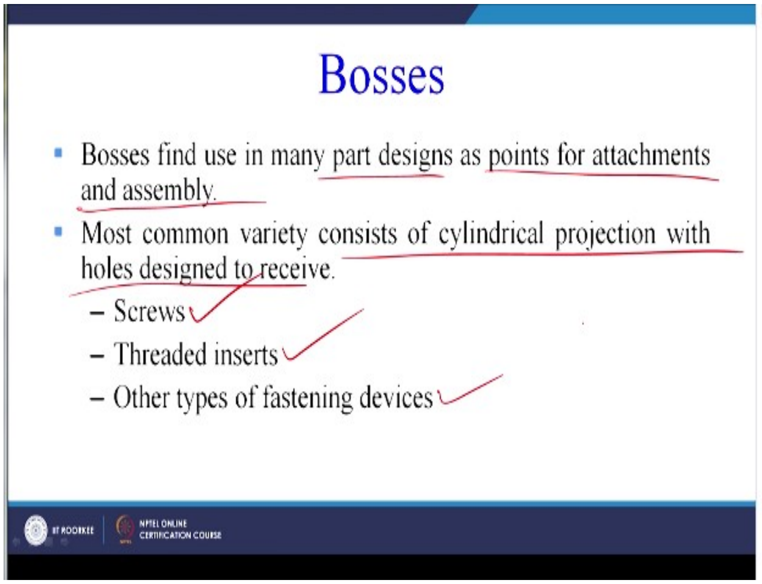


So, very easily we can see here these are the ribs shown here 2 ribs, so the height is shown in terms of the thickness T this is a thickness T, wall thickness T. In terms of T what must be the height, what must be the distance between the edges of the 2 ribs, what must be the fillet or the corner radius here is specified what must be the draft angle is specified here. So, in parts were sink marks are of no concern rib base thickness T can be 75 to 85% of the wall thickness.

So, wall thickness given here is T, so this small t is the rib thickness at the base, in parts were sink marks are of no concern. Sometimes we may have sink marks like this, so if this is not our concern then this t the small value t in parts were sink marks are of no concern rib base thickness. This is a rib, the base thickness small t this t can be 75 to 80% of the wall thickness that is the capital T where sink marks are objectionable rib base thickness small t should not exceed 50% of the wall thickness.

So, in case the sink marks are important to us then this t must not exceed 50% of the wall thickness that is a capital T. So, these are the standard recommendations guidelines which we must take into account when we are designing the plastic parts which are going to be having ribs for strength purposes.

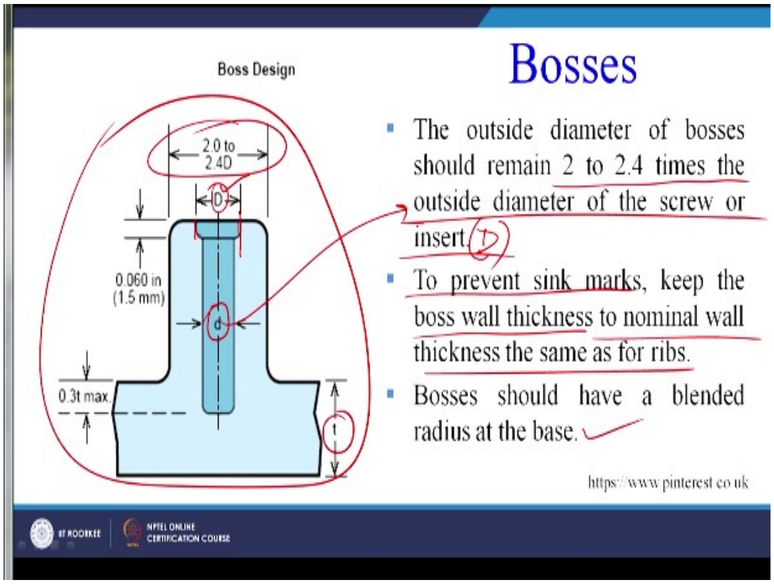
(Refer Slide Time: 36:26)



Now bosses, bosses find use in many part designs as points for attachments and assembly most common variety consists of cylindrical projections with holes designed to. So, we will see this with the help of an example please do not worry because when we see the diagram we are able to understand it in a better manner. So, what are bosses, bosses find use in many part design as points for attachment and assembly.

So, when we have to assemble the two plastic parts together on 1 side we may have a boss and the other part or a fastener will come and fit into this boss. Now how the boss would like that we will see and why it is used. Most common variety consists of cylindrical projection of the boss with holes designed to receive. So, you have a hole where you will put your screw, you can put the screw, you can put threaded inserts or other types of the fastening devices. So, you when you join the 2 parts together and one side there may be a boss.

(Refer Slide Time: 37:28)

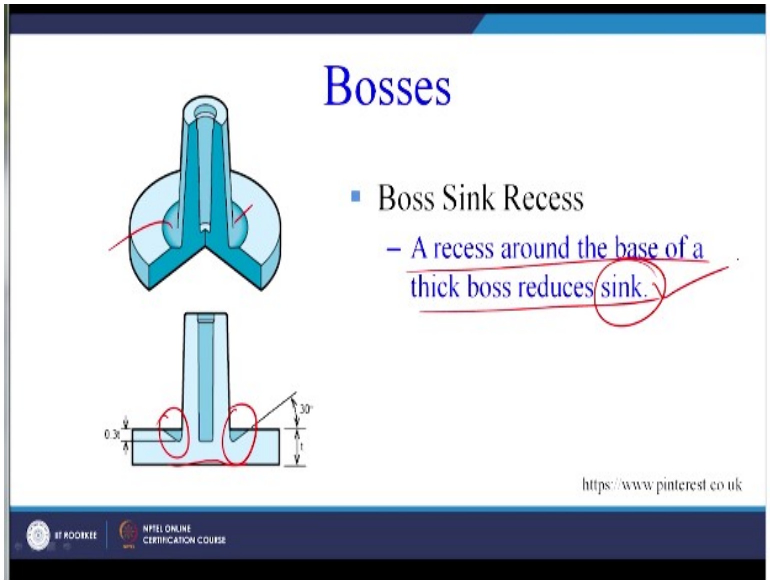


Now how the boss would look like you can see a standard boss design, so here this is a wall thickness t. So, we can see here the outside diameter of bosses should remain 2 to 2.4 times the outside diameter of the screw or the insert. Now this is basically you can see small d is the outside diameter of the screw or the insert and the outside diameter of the boss is 2 to 2.4 times of the d, this is d here.

So, this is 2 to 2.4 times t, so the outside diameter of the insert this is outside diameter of the insert that is d, so this is d here, this is the diameter of the sank of the fastener. So, the outside diameter of the bosses that is this is the outside diameter which is shown here must be 2 to 2.4 times of the outside diameter of the screw this is capital D here. To prevent the sink marks keep the boss wall thickness to nominal wall thickness the same for the ribs.

So to prevent the sink marks the boss wall thickness to the nominal wall thickness same as for the ribs. So, whatever guidelines we follow for the ribs similar guidelines we can follow for the bosses also regarding the thickness of the boss. Bosses should have a blended radius at the base, so that is also a important guideline.

(Refer Slide Time: 39:00)



Now this is another boss sink recess, these are the 2 recesses here, so a recess around the base of the thick boss reduces the problem of a sink. Sometimes the sink maybe formed here, so that sink can be avoided by providing these 2 recesses. So, boss sink recess will help us to provide the problem of a sink. So, let us now conclude the session for today, we have tried to understand the design or the product design guidelines which must be kept in mind by a product designer when the product is going to be made by the extrusion process or by the injection molding process.

So, if we keep these values. these recommendations in mind while we are designing our part, our ribs will be manufactured properly. Our wall thickness will be uniform, cooling will be uniform, warpage will be avoided and many other kinds of defects that may arise during the molding process can easily be avoided if we adhere to if we strictly follow all these guidelines. So, with this we conclude the today's session.

In our next session we will try to focus on another process with the design guidelines related to the product design, thank you.