Product Design and Development
Dr. Inderdeep Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology, Roorkee

Lecture - 18
Design guidelines for different processes

[FL] friends, welcome to lecture number 18 in our course on product design and development. As you are aware the course is the four week course with two and a half hours of discussion in every week and currently we are in week number 4 in which we are discussing the different guidelines that a product designer should keep in mind when he is designing the product.

Now, any product which is being designed if it is a tangible physical product it will be converted into its physical form using different manufacturing processes. All of you know that there are different types of manufacturing processes broadly we can classify them into primary forming processes such as casting, deformative processes such as wire drawing extrusion 4G, material removal processes such as machining, finishing processes such as grinding and joining processes such as welding.

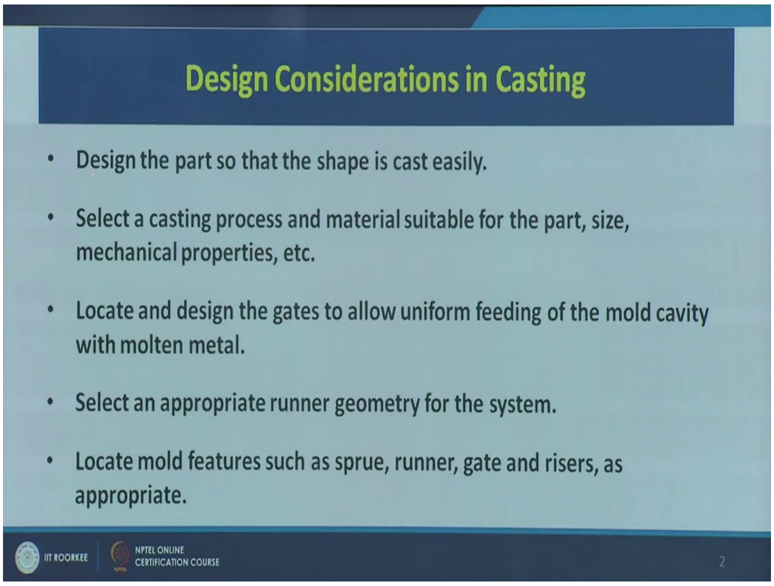
So, we have different types of manufacturing processes and any product that we design will finally, be converted into the physical form using any of these processes. So, as a product designer what is our duty, our duty is to ensure that whatever product we are designing is easy to manufacture. So, for that consideration there are few guidelines which have been laid out for various processes which if the designer keeps in mind during the design process. The product will not face any difficulty during the manufacturing stage, but most of the times designers ignore these guidelines and therefore, the product when reaches to the manufacturing facility there are number of problems associated with the manufacturing of the product. So, as a design engineer we should know that what are these guidelines and how these guidelines will affect our product design.

Now, must I tell you that it is not possible to discuss all the design guidelines within a short span of half an hour because it requires maybe a complete session of or a complete course of maybe around 20 hours in which 2 sessions should be dictated to one process only for example casting, then maybe 2 sessions only on machining. So, we cannot cover all the guidelines which are laid out for the product designed or for product designer for the product to be manufactured by various manufacturing processes. But in the short span of maybe next 25 minutes we will try to address some of the guidelines which a designer should keep in mind while designing the product. So, that the product is easily manufactured during the manufacturing stage.

So, we will cover 3 important processes we will cover machining, we will cover casting and we will cover injection molding which is an important process for manufacturing of plastic parts. And for these processes also must I tell you that we are not going to cover all the guideline, but we will just try to understand that these types of guidelines do exist and as an engineer I should keep in mind these guidelines or the detailed guidelines when I am designing the product.

So, let us first start our discussion with the design guidelines for casting process.

(Refer Slide Time: 04:04)



So, this is design considerations during casting or in casting process these are the points that we should always remember when we are designing a product which has to be made by casting process. Design the part so that the shape is cast easily a very general guideline there will be certain shapes which are difficult to make by casting, so that they are there is no problem during the manufacturing or the casting stage we should at the design stage only and sure that the shape of the product should be such that it can be made easily by casting. Select a casting process and material suitable for the part size and mechanical properties of the product.

Now, there are 3 things part material, size and mechanical properties. So, first is size of the part mechanical properties and the material of the part. So, may be if we are going to use aluminum for our product we need to select the specific process, if we are going to use may be steel for our product we need to select the appropriate process. So, depending upon the material depending upon the shape suppose it is a very big shape difficult to cast using any hot chamber cold chamber die casting process, but suppose a smaller shape can be made by permanent die casting method, but if it is a very big shape we may have to adopt sand casting process.

So, the process has to be selected accordingly based on the size of the product, based on the material of the product and based on the mechanical properties desirable in the product. So, these are the 3 important things that we have to keep in mind while selecting the casting process and we are not discussing casting in our this course otherwise in casting also we have different types of processes. Within casting we have sand casting process, we have shell mold casting process, we have CO 2 molding casting process, we have permanent molding casting processes like hot chamber, cold chamber, die casting processes. So, within casting there are. So, many we can say variants available.

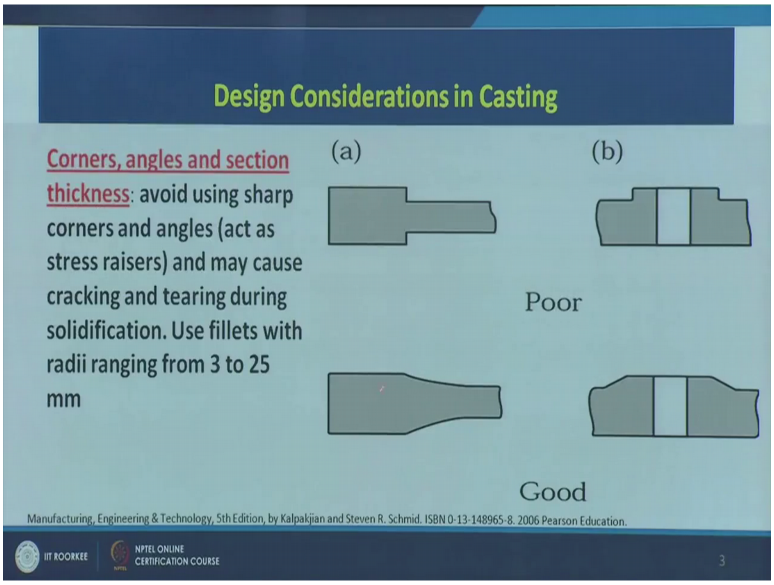
Now, when we have to select our product we have to see that which process will be suitable for our product and what are the guidelines that we should keep in mind if the product has to be made by the casting and specifically by a variant of a casting process.

Similarly, we need to locate and design the gates to allow uniform feeding of the mold cavity with the molten metal. So, third point that we need to take care is that how the design how the gates will be designed and were the gates will be located so that the product can be made easily with uniform solidification. Select an appropriate runner geometry for the system and locate mold features such as sprue runner gate risers at appropriate location.

So, we have the design our product in such a way so that we are able to optimize on all these things and we can get a good quality product if we will see with the help of example the product is not designed properly we will not be able to design our gating system that is sprue runner gate risers appropriately and finally, the product will be a defective product when we produce it using the casting process. So, all these things will depend or will be decided based on the shape of our product and this optimization is only possible if we have taken care of the design guidelines during the design stage only.

So, we will take some examples let us see.

(Refer Slide Time: 08:04)

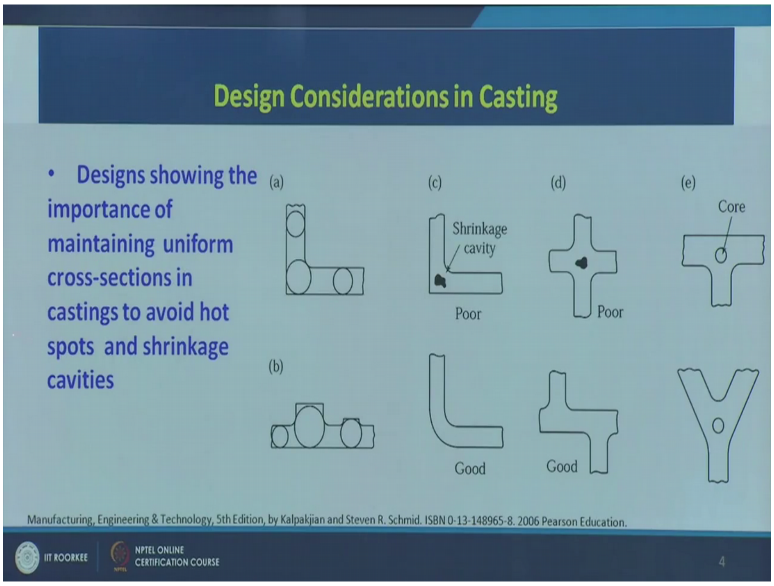


Suppose these are the two options available with me as a product designer and the product can have both the shapes without compromising with the functional requirements of the product, I should the surely go for this shape if the product has to be made by casting process. Why; it is given here corners angles and section thickness. So, we will try to understand that how these things are important during the product design stage avoid using sharp corners and angles because they act as the stress raisers and may cause cracking and tearing during the solidification.

So, this type of sharp corner should be avoided if the product has to be made by the casting process we can give you can say angle like this or a radius fillet radius like this in order to avoid the problem of stress concentration at these corners. So, the suggestion is use fillet us with radii ranging from 3 to 25 millimeter. So, this is providing the fillet in order to avoid the stress as well as producing a geometry which is simpler to cast. So, here we can see one example this is a poor design for casting this is a good design for casting.

Similarly, here we can see we have sharp corners here. So, we can avoid the sharp corners by giving this chamfered look at this corners as this corner. So, first thing is corners angles and session thickness is important when we are designing the product we should ensure that no sharp corners are there when the product has to be made by the casting process. Now, let us see the importance of section thickness.

(Refer Slide Time: 09:56)



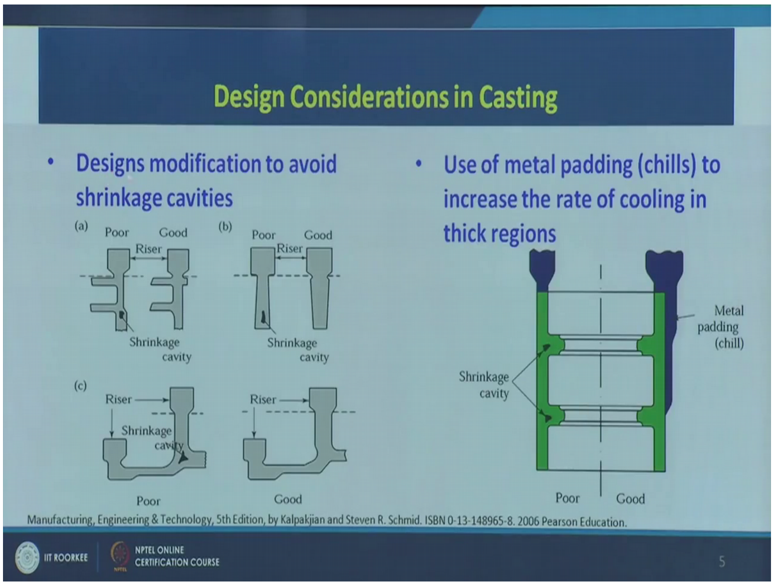
Now, here we see designs showing the important of maintaining uniform cross sections in castings to avoid hotspots and shrinkage cavities. Now here we can see this is a inscribed circle method of finding out where there is more cross section. So, here we see at this corner the cross section is more as compared to the cross section in the other two parts.

So, here there is a tendency of formation of a shrinkage cavity. So, how we can avoid the shrinkage cavity we can make this a bend like this and avoid the sharp corners. So, that we have uniform cross section throughout. So, and we are able to avoid the shrinkage cavity. So, here also we see the cross section is not uniform here we have a larger inscribed circle and there are smaller cross section and then the medium cross section here. So, this may not be a very good design from casting point of view. So, we need to redesign this part without compromising the functional capability or functional requirement of the product and try to make it as uniforms as possible.

So, here we can see shrinkage cavity can form not a very good design, here we see a very uniform cross section which can be considered as a good design. Is a poor design if we again use the inscribed circle method we make a circle here which will be of largest diameter, so tendency of forming of shrinkage cavity here. So, what we can do we can modify our design if this is not the only design that has to be used we can modify the design and make it in this manner so that this kind of shrinkage cavity can be avoided and a uniform cross section can be ensured.

Similarly, in this type of T shape we can have a core at the centre to avoid the shrinkage cavity here, similarly in Y shape we can have a shrinkage cavity here in order to avoid that we can have a circular cavity may be have we can have a circular section here or we can use other methods of avoiding this which are may be putting a riser at this location or using the chills to accelerate the rate of cooling of the molten metal. But as a product designer when we are using these shapes in our product and the product has to be made by the casting process we need to look at these cross sections or the individual cross sections of the members so that the shrinkage cavity does not take place or does not happen in our final product.

(Refer Slide Time: 12:33)



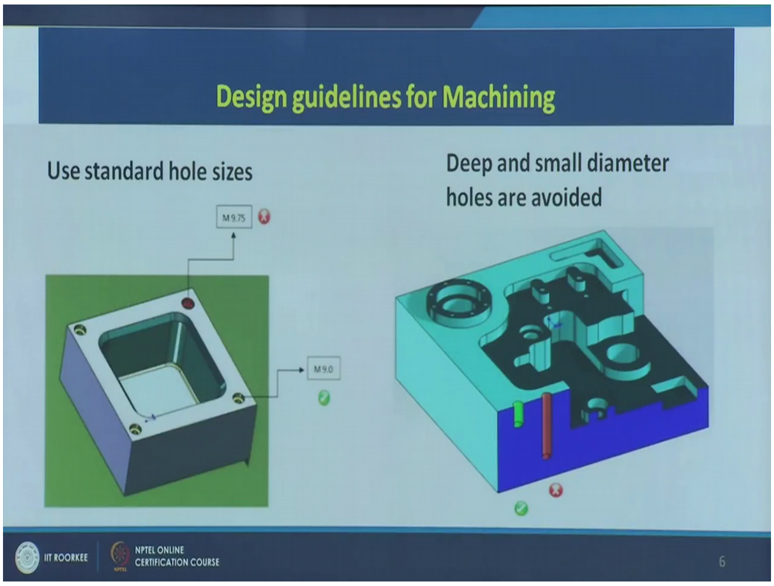
Here again we can see design modifications to avoid shrinkage cavities, here we can see this is the shrinkage cavity tendency to form a shrinkage cavity here. So, the poor design in good design we can have a uniform cross section which will avoid the shrinkage cavity. Similarly here the metal will solidify in the end at this section because the larger cross section or larger inscribed circle will be formed here. So, there is a tendency to form the shrinkage cavity.

So, we can design it in opposite manner so that the area which will solidify in the last will is fed by the riser so that there is no tendency to form the shrinkage cavity. So, we can see that if we have uniform cross sections it is difficult for the shrinkage cavity to form and we can avoid the defect defects of shrinkage cavity as well as the rejection of the parts because of the formation of shrinkage cavities by following this guidelines during the design process only.

Here we can see again this is a riser here, so there is a chance of formation of shrinkage cavity here, but if we use a metal padding or a chill which is additional part for the metal being cast we can avoid the shrinkage cavity. Why because it will excel rate the rate of cooling in this area and the metal will solidify uniformly without formation of the shrinkage cavity.

So, we can use metal paddings we can place our risers accordingly depending upon the shape, but it is better to avoid such cross sections where there is a tendency for the shrinkage cavity to form. So, as an engineer we should keep these things in mind during our design process only.

(Refer Slide Time: 14:20)



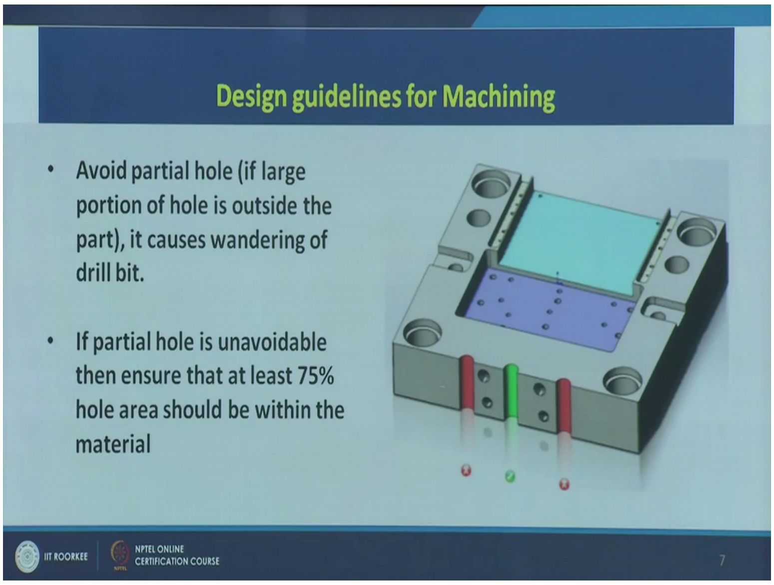
Now, I have just tried to address only one aspect that is the shrinkage cavity by changing the cross section as well as changing the corners into the fill into the radii, so that we can avoid these defects. For casting only there will be number of other guidelines which can help us to avoid these defective parts after manufacturing. So, as a designer we should focus on all these guidelines when we know that the product that I am designing will be made by the casting process, so why not to incorporate all these guidelines in the design stage only. So, that we do not face lot of rejection during the manufacturing stage because of the defects which could easily have been avoided.

Now, coming on to the second process that is machining let us take few examples of machining. So, first guideline is use standard hole sizes. So, first guidelines are that the drills that are used for making these holes are also standardized. So, depending upon the diameter of the drill we can select our hole size is because if we select a nonstandard hole size we may require to order a specific drill for that hole which would be a costly proposition for the organization.

So, depending upon the drill sizes available we need to decide on the diameters of the holes. So, it is always better to use the standard hole sizes. Then deep and small diameter holes must be avoided. So, here we see two holes are shown one is a green hole which is acceptable, but this red portion is not acceptable because it is a very deep hole and may require not only drilling, but a specific hole making operation to go to this depth for making the hole.

So, this type of holes if can be avoided must always be avoided.

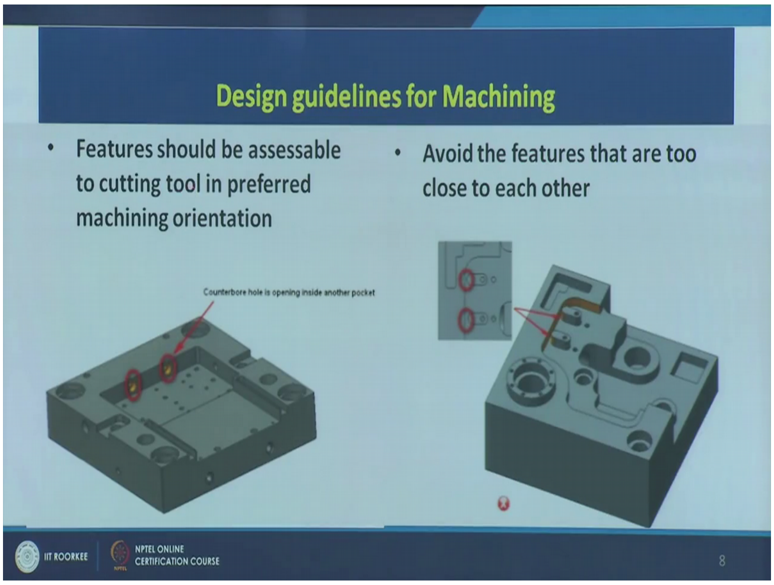
(Refer Slide Time: 16:19)



Then this is an important design guideline here you can see that avoid partial hole partial hole means here we are see it is not a complete hole it is at the edge of the component, so it is a partial hole. So, avoid partial hole if large portion of the hole is outside the part it causes wondering of the drill bit. So, we do not get a good quality hole. So, this can be avoided, but the green portion you can see that most of the hole is inside the component. So, this is acceptable can be done and this is also not acceptable most of the portion of the hole is outside the component.

So, here you can see avoid partial hole it causes wondering of the drill bit, but if partial hole is unavoidable we need to make it because maybe other part has to come and a drill has to be sorry nut and bolt assembly has to be done here, so if partial hole is unavoidable then I am sure that at least 75 percent the hole area must be within the material which is depicted in this green portion here.

(Refer Slide Time: 17:34)

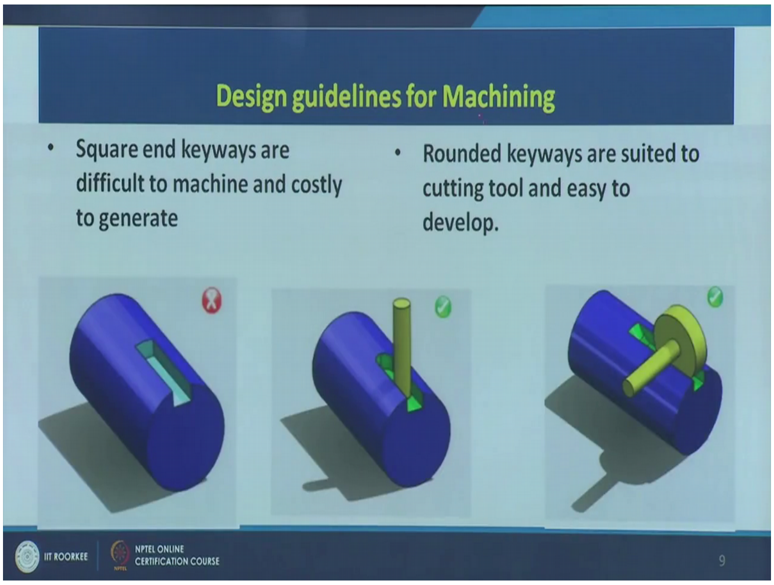


So, we should have the maximum portion of the hole area inside the part another guidelines for machining feature should be assessable to cutting tool in preferred machining orientation. Generally for drilling operation we would like to play place our work piece like this and the drill can operate like this, this is a preferred machining orientation for drilling operation, but here we see we have two holes here and circled and enlarge to just show you counter bore hole in opening inside another pocket.

So, this type of things they are opening into this pocket, so difficult to make these holes. So, these type of holes can be avoided and can be made by some other method not by machining. So, feature should be assessable to cutting tool in preferred machining orientation this is not a preferred machining orientation for drilling of holes.

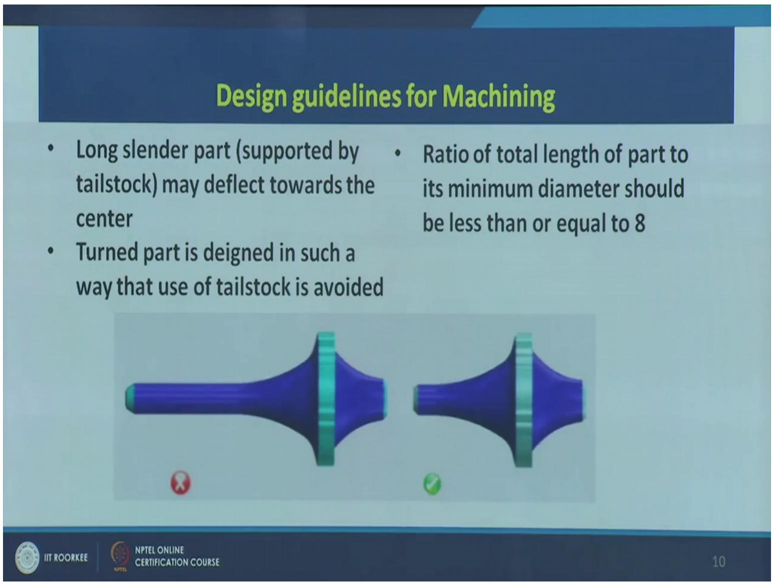
Similarly, avoid the features that are too close to each other. So, here we can see these two features are very close to each other, so difficult to perform the operation. So, this can be avoided.

(Refer Slide Time: 18:42)



Design guidelines for machining further we can see, we can see we have a keyway here square and keyway. So, here we have a square end keyway see, this is square end square end keyways are difficult to machine and costly to generate, so what can be done. Rounded keyways are suited to cutting tool and easy to develop. So, we have a rounded keyway here. So, we can we must avoid if possible we must avoid these square keyways and instead we should prefer the circular keyways here or the rounded keyways because they are easy to develop.

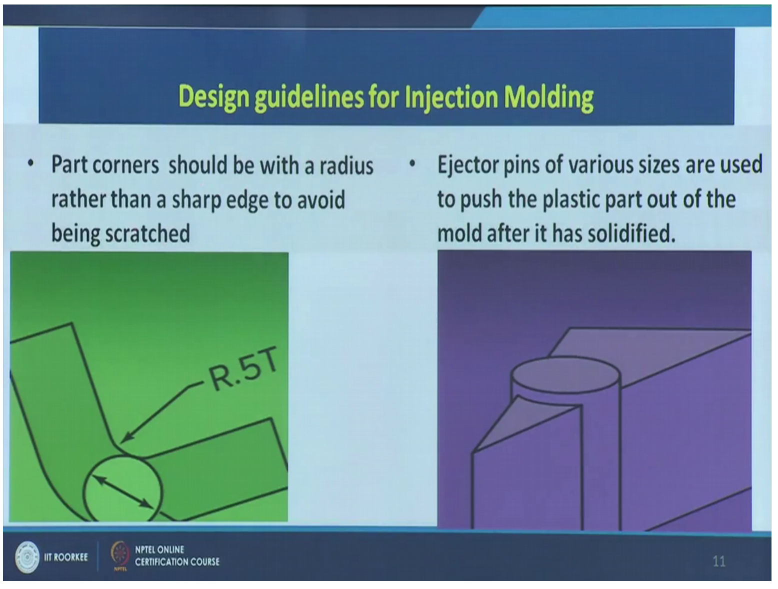
(Refer Slide Time: 19:20)



Then we can see here if we are turning a particular part we can see this is a wrong operation, why because long slender parts which are supported by the tailstock you all of you know lathe machine we have a head stock and we have a tailstock. So, in case of long slender parts which have to be supported by tailstock they may deflect towards the centre. So, long slender parts must be avoided turned part is designed in such a way that use of tailstock is avoided. So, when we are designing a part which has to be made by turning operation on a lathe machine we should avoid long slender parts why, because there may be deflection at the center.

So, one guideline that comes out of this thing that we should avoid the long slender parts is that ratio of the total length of the part to its minimum diameter should be less than or equal to 8. So, such types of guidelines are available for parts that needs turning operation or that needs operations on lathe machine.

(Refer Slide Time: 20:26)



This is another thing these are the final guidelines that we are going to consider for injection moldings process. So, we have seen by now the two types of guidelines that is guidelines for casting operation and we have seen guidelines for machining operation where we have majorly focused on drilling or making of holes. Also we have seen for turning we should not use very long slender parts why because of the problem.

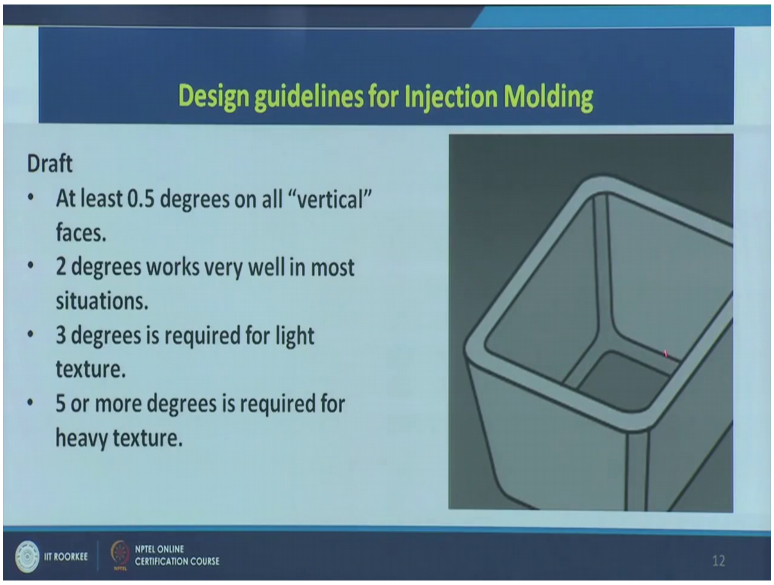
Now, we will see if the part is has to be made out in a plastic materials then injection molding will be the most common process and for injection molding what are the guidelines let us quickly go through these guidelines.

Now, part corners should be with a radius rather than sharp edge to avoid being scratched. So, the important thing you can see here in case of casting what we are doing we are melting the material or melting the metal and then pouring that molten metal into a mould cavity and allowing it to solidify and finally, the mold is broken and the part is taken out. So, that is the standard sand casting process. In injection molding what we do we take thermoplastic pallet us or polymer and then we heat it, we melt it, we inject it into the mould cavity allow it to solidify there and then we take it out or it is pushed out from the mould cavity using the ejector pins.

So, the basic principle remain same the raw material is may be melted and then it is fed into a particular mold or a die and finally, it is taken out after solidification. So, in these type of processes use of sharp corners must always be avoided and the same thing is given for injection molding also that part corners must have a radii rather than being sharp in order to avoid being scratch. So, if that is one guidelines for parts that have to be made by injection molding and you will always appreciate that if you see a plastic product in many cases you will never see very sharp edges there always there will be a crowd portion at the corners and that is very common because of this particular design guideline.

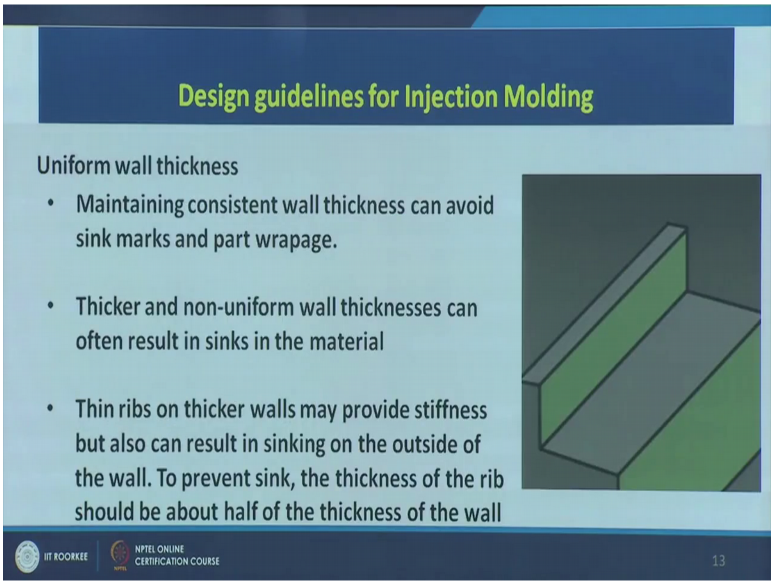
Then the ejector pins of various sizes are used to push the plastic parts out of the mold after it has solidified. So, we need to ensure that when the ejector pins will be use where they should heat the product so that those marks are not visible maybe to the customer. So, that is also a important design guidelines that our part should be, so designed that it is easy to come out from the mold cavity, so that the election is easier.

(Refer Slide Time: 23:14)



Then design guidelines for injection molding further win can and gage are draft. So, that the part is ejected out easily; no. Now let us see what is the draft at least 0.5 degree can be given as a draft on all the vertical faces we can see there are vertical faces here this is a vertical face. So, we can give 0.5 degree on vertical faces 2 degree works very well in most situations. So, it may vary from 0.5 to 2.5 degree depending upon the shape of the our product which must always give the draft 3 degree is required for light texture and 5 or more degree is required for heavy texture. So, if we need to have a texture on our surface. So, we can give more draft so that we get that adequate texture.

(Refer Slide Time: 24:02)

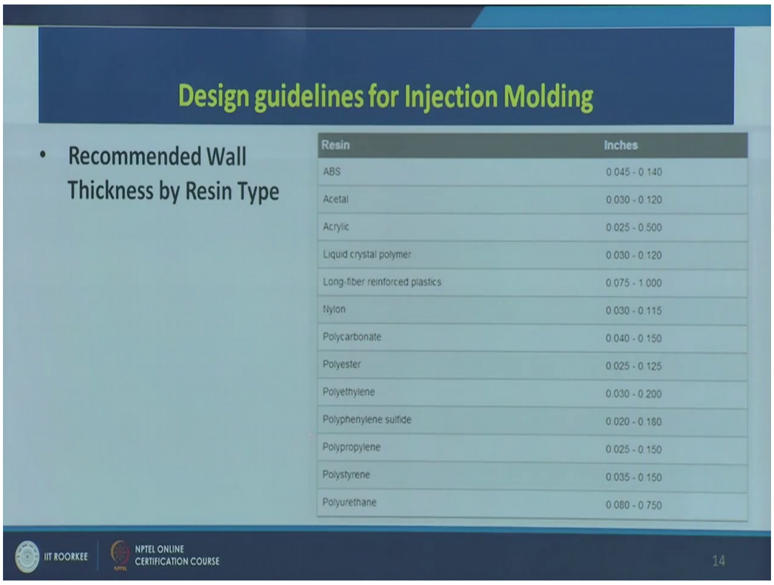


Another design guideline for injection molding is the uniform wall thickness. So, we have seen injection molding is usually used for thin walled parts and if you remember in case of casting also we have seen that the product has to be made by casting we should ensure the cross section remains more or less uniform.

Similarly, here also in injection molding we should ensure that the wall thickness does not vary too much otherwise the product will have defects. So, uniform wall thickness is a mandatory requirement for the parts or products to be made by injection molding process. So, maintaining consistent wall thickness can avoid sink marks and part wrapage. So, these are two types of defects that usually happen during injection molding. So, if we can ensure a uniform thickness are all across our product we can avoid sink marks and part wrapage thicker and non uniform wall thicknesses can often results in sinks in the materials. So, these are same points only thicker and non uniform thicknesses must be avoided.

Then sometimes we see different ribs also in plastic parts in order to give strength and stiffness to the product. So, thin ribs on thicker walls may provide stiffness, but can also result in sinking on the outside of the wall to prevent sink the thickness of the rib should be about half of the thickness of the wall. So, when the ribs are provided in order to ensure stiffness of the plastic part there is a guideline that what should be the thickness of the rib. So, the guideline is it should be about half of the thickness of the wall. So, we can see this may be the wall. So, rib if we this is the rib the thickness must be less than or about the half the thickness of the wall. So, it should be less than the thickness of the wall.

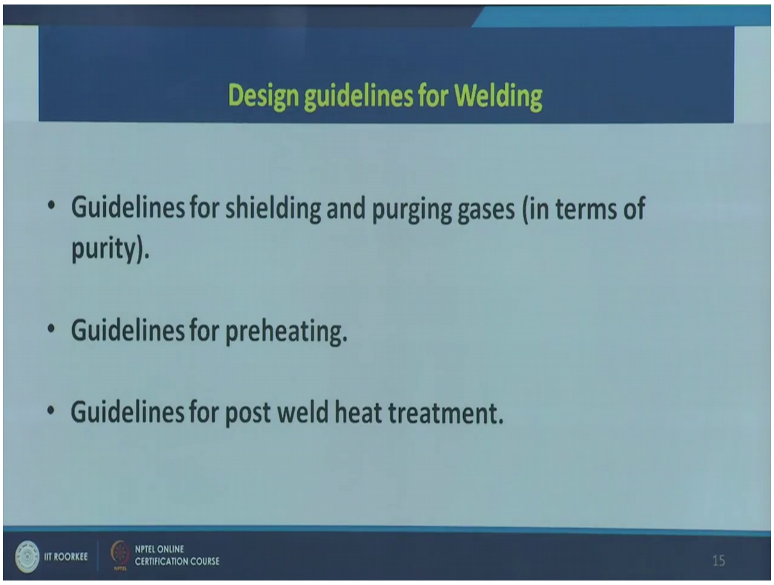
(Refer Slide Time: 26:12)



So, these are the guidelines for the parts which we are going to make out of a plastic material. So, these are the design guidelines further for injection molding this is a recommended wall thickness depending upon the type of polymer that we are using I am sorry may not be clear to all of you, but maybe I will just read for some of the names here this is ABS akronnitrile butadiene styrene, then polyethylene, polyester polycarbonate, nylon, polyphenylene, sulfide, polypropylene. So, all these polymers there is a recommended thickness of the wall that is going to lead to a good quality product if we deviate from these recommended thicknesses our product may not be of good quality.

So, if we adhere to these guidelines and we ensure the part thickness as per the guidelines the product will definitely come out as a good quality product.

(Refer Slide Time: 27:05)



So, finally, let us see just broad guidelines for welding, welding is a very vast topic we cannot cover everything in detail. But quickly let us see in welding we will have the guidelines for shielding and purging of gases in terms of their purity that when we are welding using may be in the atom gas atmosphere what should be the type of gas, what should be the purity of the gas, what should the combination of the gases that should be used in order to avoid the defects in the weldments.

Then there can be guidelines for preheating that for how much duration we should preheat our base metal should we preheat our electrode also, so guidelines exist for these things also. Then there are guidelines for post weld heat treatment also that once you have welded the two parts together what is what are the guidelines for treating or post weld heat treatment so that the residual stresses that are developed in the weldment can be easily relieved.

So, in today’s lecture or today’s discussion we have just discussed the glimpse of the guidelines, we have not gone into the details of the guidelines that are available. There are number of books that are focusing on these guidelines only that a product designer should focus on these guidelines so that the product is manufactured defect free or a good quality product can be manufactured if you take into account all these guidelines at the design stage only.

So, maybe in our next lecture our focus will be on rapid prototyping we will have two sessions on rapid prototyping, in first session we will focus on rapid prototyping basic concept and the steps involved in rapid prototyping process and finally, we will try to see the process mechanism that how a cad model can be finally, converted into a physical form of prototype using different rapid prototyping processes. At least we will try to cover three of them that is stereo lithography, selective laser sintering and laminated object manufacturing.

So, with this we come to the end of our session 18 in our course on Product Design and Development.

Thank you.