Processing of Polymers and Polymer Composites
Dr. Inderdeep Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology, Roorkee

Lecture - 10
Injection Molding-II

Friends, welcome to lecture 10, in our course on processing of polymers and polymer composites. Just to have a brief review of what we have covered in our previous session, we have covered the basic fundamentals of injection molding process, which is one of the closed mould forming processes for plastics. We have covered the basic principle of the process, and then we have seen that how the process operates with the help of a video. In that simulation we have seen that the 3 basic steps for forming any plastic material are followed in injection molding also. That is, we have pellets in the hopper first we bring them into our barrel or injection chamber, where we have a heating arrangement, all around the periphery. Because of the heat the polymer melts, and then it is forced through the nozzle into the mould cavity.

Now, the mould is a 2-part mould, one is a fixed part, another one is a movable part. When the 2 parts of the mould close, there is a cavity inside which is the exact replica of the final product that we want to make. Now our raw material which is already melted is pushed through with the help of a screw arrangement, or with the help of a piston cylinder arrangement. So, it is injected through the nozzle into the mould cavity where it takes the shape. That is the forming step.

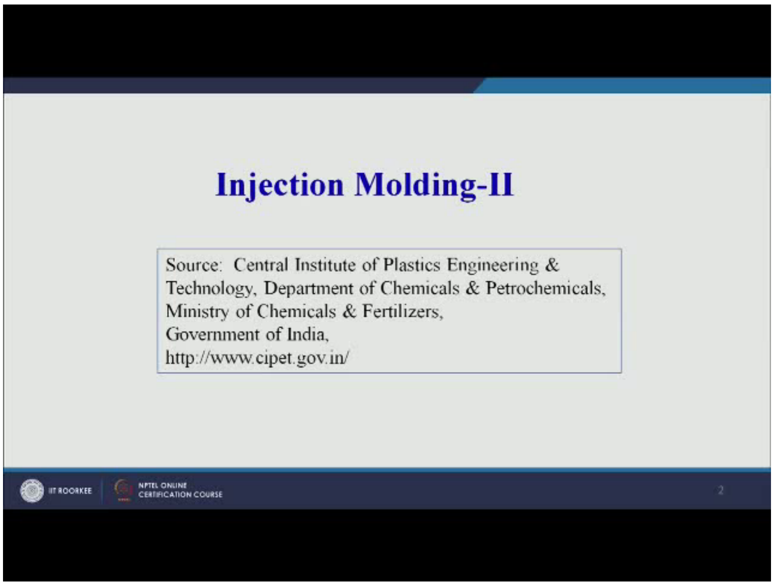
So, the first step is heating and melting of the plastic. The second step is deforming or changing the shape of the molten plastic as per our required product. Once it is formed, it has taken the solid form we cool the mould. Now cooling process can be natural cooling it can be forced cooling through air or through water circulation all around the mould. So, there will be passages or cooling coils that will help in the cooling process. That is the basic working principle of the injection molding process. So, 3 basic steps, feeding and melting through the heating arrangement, then forming changing shape in the mould cavity, that is a split mould type of arrangement, then the cooling with the help of water cooling or air cooling.

So, that that is the basic principle of our injection molding. Now we will try to understand today that what are the important parameters or the process parameters or the operating variables that we need to control in order to make a very good quality injection molded part. So, after that we will see that although we take all possible precautions we optimize all the operating parameters or the operating variables, but still we get number of defects in the injection molded parts.

So, with the help of diagrams we will try to understand that what are the defects that take place in injection molding. And finally, we will see that what are the reasons for these defects so that we can we can address those reasons in order to improve the quality of the injection molded part. So, let us start our discussion for today on injection molding. So, we have already discussed injection molding in our previous session. Today also we will continue our discussion on injection molding. And finally, we will switch over to a next process that is resin transfer molding. And there we will see what are the process intricacies of resin transfer molding.

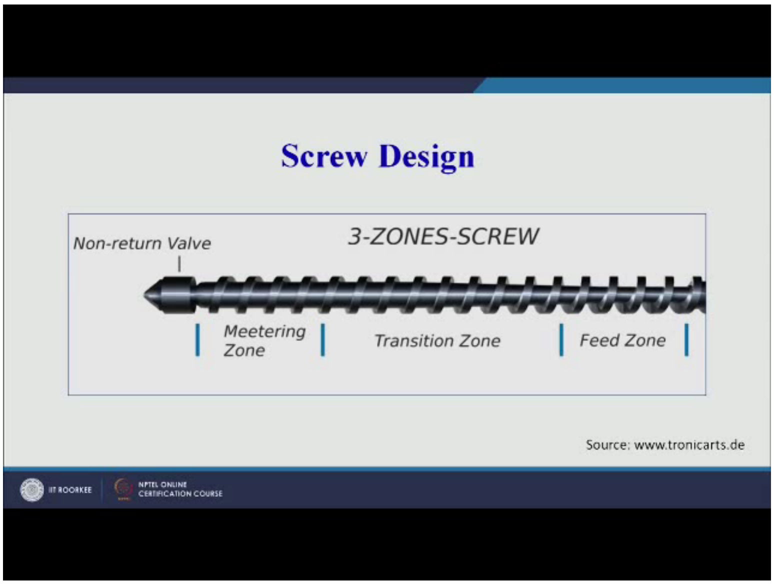
So, to start with; we can first see the source for today's session.

(Refer Slide Time: 04:02)



The source is central institute of plastics engineering and technology. That is; that comes under department of chemicals and petrochemicals. So, the data that we have taken for today's presentation is available at this website. The website is also given. It is a government of India organization, www.cipet.gov.in. So, all the information is available at this website. So, you can refer to this website also other source of information regarding processing of polymers.

(Refer Slide Time: 04:39)



On the screen there is a diagram which was shown in the previous session also, that is the screw design. Why we have put this diagram here? Because there is one important parameter for any injection molding process that is the L by D ratio. And we will try to understand what is L and what is d, with the help of this diagram. There are basically 3 zones of a screw, in case of injection molding.

Similarly, in case of extrusion process also we have seen, a similar diagram of the screw and we have tried to understand that what are the different zones of the screw length. Here also if you see the screw length which is starting from one end to the other end, we can see the length of the screw will be in the longitudinal direction. So, this is the length if you can see on your screen this is a length of the screw along this longitudinal direction or the axial direction of the screw. There are 3 zones one is a feed zone, second one is a transition zone, and the last one is the metering zone.

So, here as per the requirement that screw is designed in such a way; that the purpose of this particular screw is met or the objective is fully satisfied. Now what is the objective of the screw? If you refer to the our last session why we are using this screw? The screw is used for feeding the molten pellets or the molten material into the mould cavity.

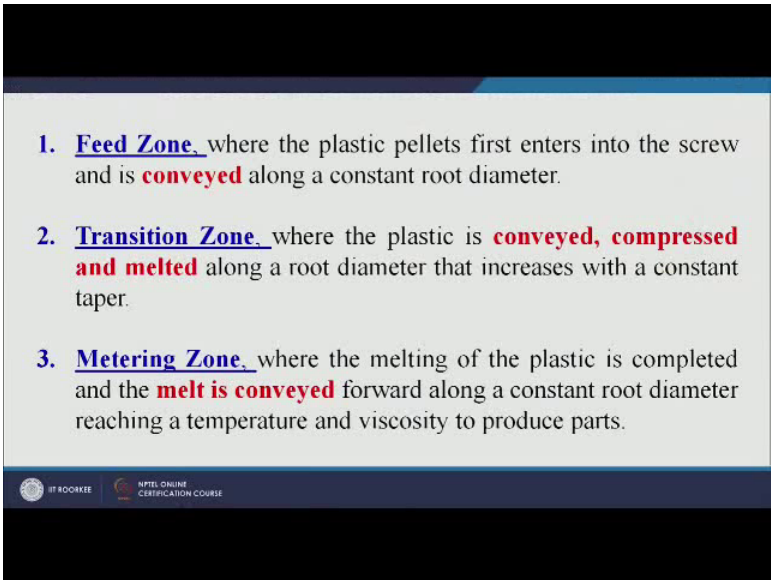
Now, initially the pellets in the solid form or in the spherical granular form are fed from the hopper into the barrel. The screw is rotating inside the barrel. So, there is one feed zone where the screw receives the solid pellets from the hopper; that is the first zone. The second zone is a transition zone. If you remember all around the barrel or the injection chamber there are heaters. Concentric or circumferential heaters, all around the periphery, all around the circumference your heating arrangement.

Now, these heaters will melt the solid pellets into the molten form, they will bring these solid pellets or granules into the molten form. So, in the transition zone there will be this mixing and melting of the pellets in the next zone we have to feed this as the name suggests injection molding. So, we have to inject this molten plastic into the mould cavity. So, that injection will be done in the metering zone. And we will see that how much quantity has to be pushed and last session we have refer to one term that is called the short volume.

So, here that shot volume will be decided and that much amount of material will be pushed through the nozzle into the mould cavity. So, the design of the screw is very, very important. And the objectives for which the screw has to satisfy are also explained. Now because the length is a variable the diameter of the screw is a variable, there is a scientific or logical parameter that is L by D ratio. That is the ratio of the length to the diameter. So, that we have to see we have to optimize. So, that our objectives of the screw that is feeding melting and mixing and feeding all these objectives are met by the screw, and none of these objectives is left un addressed unaddressed so that our product that we get is spoiled or is damaged or is defective.

So, if our screw length L by D ratio is accurate or is optimal, it will lead to proper mixing of the melt, proper melting of the melt, proper feeding of the melt, proper we can say metering of the melt. Therefore, if everything goes on well our final product will be of good quality, but that does not happen too often we will see in today's session towards the end that what are the various types of defects that happen during the injection molding process, and how we can avoid those effects are what are the parameters leading to those defects. This is the basic purpose of providing, but this is one of the most important. After the mould design the screw is the next important parameter in the injection molding process, because this is the place we are the actual we can say process is happening the polymer is coming, it is melting, it is being fed through the nozzle into the mould cavity. So, it is also an important part of the complete injection molding machine.

(Refer Slide Time: 09:36)



Now, just I have tried to explain what each section of screw will do, but let us read first for better understanding. Now feed zone; where the plastic pellets as I have told plastic pellets or granules, first enters into the screw and is conveyed along the constant root diameter. So, the feed zone is the zone where the screw will receive the pellets from the hopper.

Transition zone where the plastic is conveyed compressed and melted along the root diameter; that increases with the constant taper. So, we will see in the diagram that there is a constant taper in the screw so that we will see in the diagram, but what is the basic objective purpose of the transition zone it will convey the pellets that have been received from the hopper, further towards the metering zone it and the pellets will be compressed and because of the heaters all around they will be melted also. And the last zone is the metering zone where the melting of the plastic is completed.

So, it means the melting is a 2-stage process. The melting process starts in the transition zone and is completed in the metering zone. So, the melting of the plastic is completed, and the melt is conveyed forward along a constant root diameter. So, that is the thing. We will convey it through the mould cavity through the nozzle into the mould cavity.

So, again I will read the metering zone, where the melting of the plastic is completed and the melt is conveyed that is all the screw will be rotating at an RPM, and it will be constantly pushing the molten plastic towards the nozzle. So, that will be conveyed forward along a constant root diameter reaching a temperature and viscosity to produce the parts.

Now, if you see the screw length, initially solid pellets are received. Transition zone melting started, metering zone melting completed. But all through the screw is rotating the melt or the charge is pushed forward. And at the end point at the end of the screw it has reach the temperature as well as the viscosity at which we have to feed it into the mould cavity.

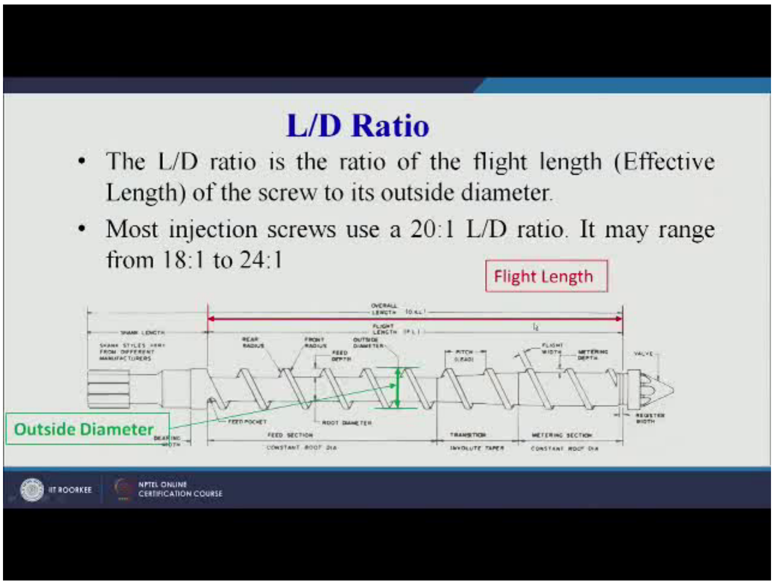
Now, we should keep these do important things in mind that is the viscosity and the temperature at the end of the screw is the temperature and viscosity, that is our aim; that is our objective that we have to supply the molten plastic into the mould cavity at this temperature. And at this much of viscosity, why? Because the melt has to flow through the runners and the gates and enter into the mould cavity and reach to each and every corner of the mould cavity so that we get our final product. If that does not happen, our part will be defective.

So, the screw therefore, is the most important maybe after the mould design screw design is another important parameter, which I have already emphasize. And therefore, we are studying screw design in slightly much more detail as compared to the other parameter. So, if we know that what is screw what are the various sections of screw what is the L by D ratio, then we can say that we can if there is a defect in the final product we can try to relate that defect that what we need to change in our process. What we need to change in our setup or machine in order to avoid those defects. And for that purpose, we need to understand that how the process is operating or working.

So, if I think this point is absolutely clear to all of you; that what is a screw? What are the various zones of the screw? And what is the purpose or objective of the various zones of the screw? Now let us try to see that how do we define the L parameter, and how do we define the D parameters, and what are the typical L by D ratios for whichever injection molding machines are available.

So, the L by D ratio is the ratio of the flight length that is the effective length of the screw to it is outside diameter. So, L basically; as the alphabet goes L is for length and D is for diameter. So, the L by D ratio is very, very important and normally L by D ratios are in 20 is to 1 and many a times it can be it ranges from 18 to 24.

(Refer Slide Time: 14:19).

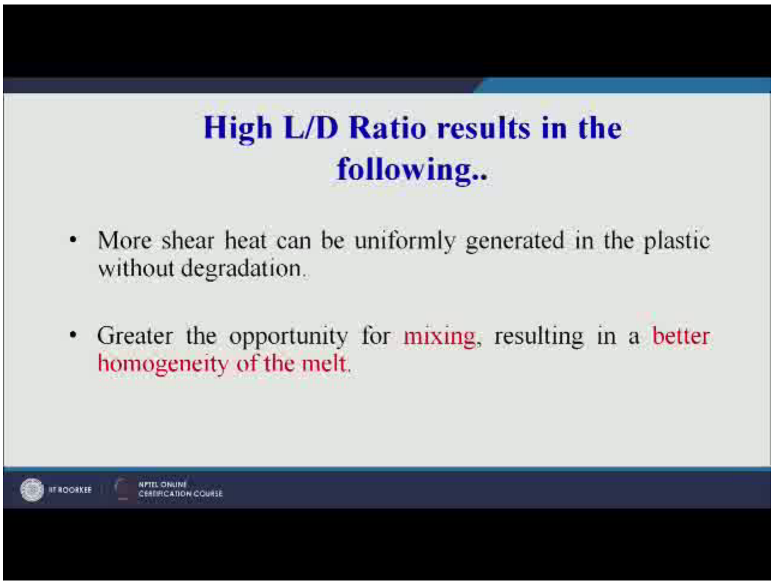


So, means that the length as is specified in the diameter, sorry in the figure also you can see the length is 20 times that of the diameter.

So, it can range from 18 times of the diameter to 24 times of that of the diameter. Now L by D ratio will have a effect on the final product that we are producing using the injection molding machine. And here you can see the diameter is changing. So, we have taken average diameter here. From this side there is a taper you can see diameter is more here and diameter is slightly less here.

So, that was the taper that was discussed in the previous slide, but majorly we should keep in mind that L is the length or the we can say effective length of the screw and D is the outside diameter average outside diameter of the screw. So, this L by D ratio is an important parameter, and it ranges that we should remember from 18. The length being 18 times that of the diameter to 24 times that of the diameter.

(Refer Slide Time: 15:19)

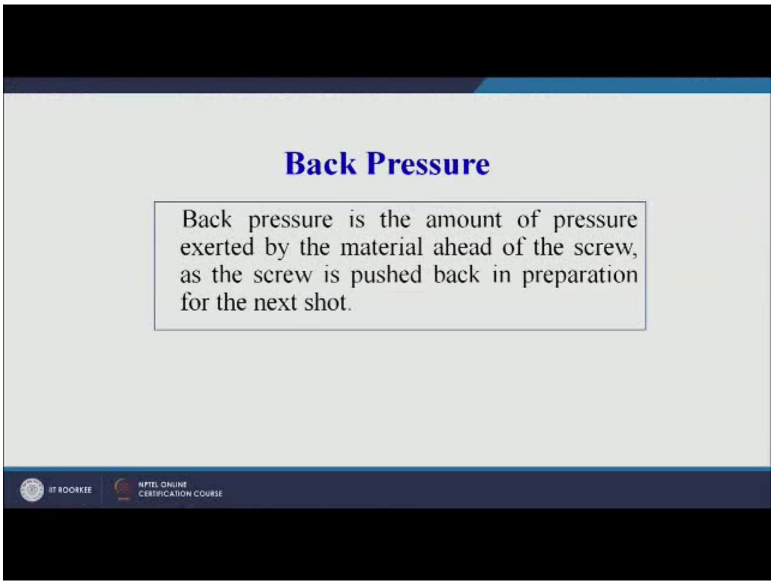


Now, how the L by D ratio will influence? The process that we will try to understand now. So, high L by D ratio results in the following now suppose L by D ratio is higher that is the length is 24 times that of the diameter. More shear heat can be uniformly generated in the plastic without degradation. So, if you if we have a L by D ratio of 24 is to 1 more shear heat maybe 24 the length being 24 times that of the diameter. The more shear heat can be uniformly generated greater the opportunity for mixing resulting in a better homogeneity of the melt.

So, basically if you remember we have seen yesterday also in our previous session. Again, we are seeing today that the screw length what are the major objectives it has to ensure the melting and mixing of the plastic. So, if more is the length that is through the diameter it will have the plastic melt will have more chances of melting shearing and mixing leading to a more homogeneous plastic melt. So, that can be advantageous, but there can be limitations of a larger L by D ratio also. As we have seen here without degradation.

So, more shear heat can be uniformly generated in the plastic without degradation, but many a times depending upon the melting temperature of the plastic depending upon type depending upon the chemical nature of the plastic, it may so happen that little bit of degradation of plastic may take place because of the extended length of the screw. Now the next parameter that we want to consider today is the back pressure.

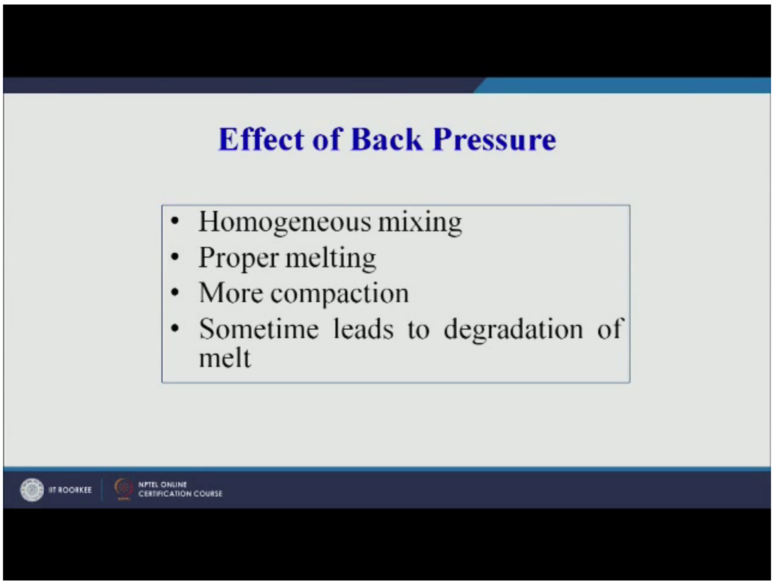
(Refer Slide Time: 17:06)



Now, what is back pressure back? Pressure is the amount of pressure exerted by the material ahead of the screw. Now then the screw will rotate it will certainly have some material in front of the screw after you have injected this back pressure will apply pressure or a force on the screw and a screw will be pushed back.

So, the back pressure is the amount of pressure exerted by the material ahead of the screw as the screw is pushed back in preparation for the next shot. So, after one single shot the screw has to move back. So, that it can bring more material to be pushed in the next cycle. So, that pressure that is exerted by this material on the screw, during it is backward movement is called as the back pressure. Now how this can affect the process? Effect of back pressure it will lead to homogeneous mixing, proper melting, more compaction, sometimes it may lead to degradation of the melt.

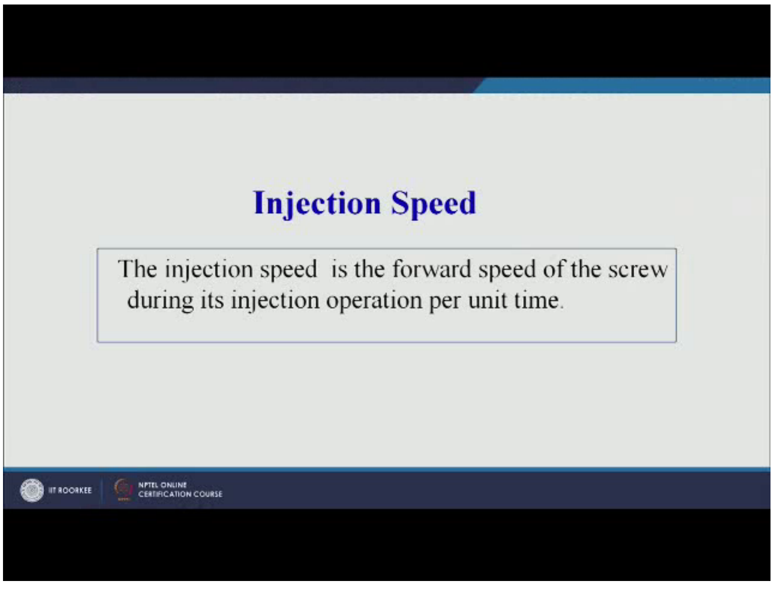
(Refer Slide Time: 18:02)



So, if the back pressure is too high, it will force the screw maybe slightly more backward as it is expected or the optimal movement what is expected up to a particular length, but it may more back pressure and push it is further and that may lead to degradation of the material. Because the material is already if you remember towards the end of the screw length our material is already ready. It has attained the temperature it has attained the viscosity at which we want to push it through the nozzle into the mould cavity. But if again it goes back even slightly into the injection chamber, it may get overheated and may lead to the degradation of the property.

So, basically, we have to optimize each and every parameter in order to make a good quality part using the injection molding process. And 2 important parameters we have already seen. One is the selection of the L by D ratio, second one is the optimal back pressure that we need to select. Third is the injection speed.

(Refer Slide Time: 19:13)

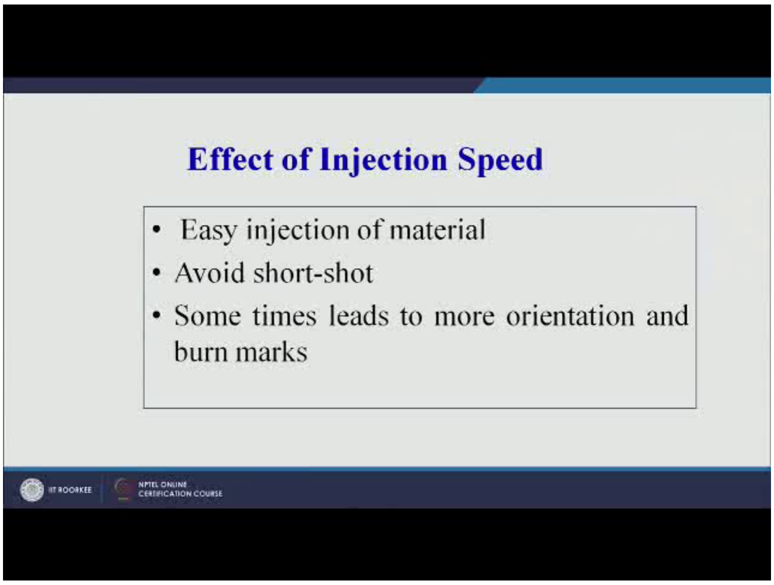


Now, injection speed is the forward speed of the screw during it is injection operation. So, the screw will be rotating. Then it will be moving forward to push the material into the mould cavity. Not during that movement whatever what is a distance it travels per unit time that is called the injection speed.

Now, what can be the effects of injection speed. If it is optimal it will lead to easy injection of the material. If it is optimal it will avoid the short-short. Short-short we will see with the help of a diagram; how do we define this term short shot. So, basically, we are trying to emphasize that if we do not select our variables or operating parameters judiciously or optimally it will lead to a disastrous product or it will lead to a defective product that we need to avoid.

Now, injection speed also can lead to number of defects. And if we have not optimally selected our injection speed it may lead to short-shot. What is short shot, we will see in the subsequent slides.

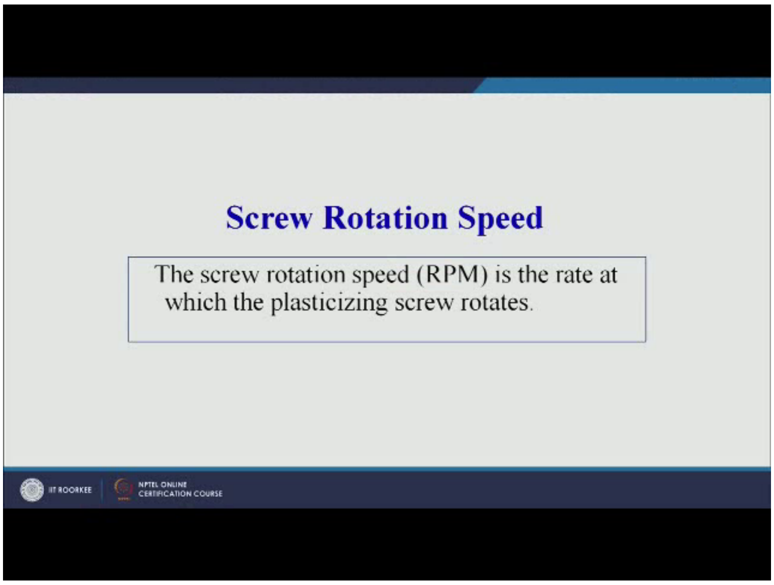
(Refer Slide Time: 20:24)



Sometime leads to more orientation and burn marks. Now more orientation is related to the composite materials. When the fibers are also put in the hopper and we are making a short fiber reinforced polymer based composite. So, the orientation of the fibers play a very important role in defining the properties of the composite. And if the injection speed is not optimal we will not get proper orientation of these fibers, which may lead to a defective product. Or we will get a product where the properties will not be as we are desiring or the desirable properties. And sometimes the injection speed may also lead to the burn marks in the final product the.

The next parameter that we want to consider today is the screw rotation speed. Now the screw is rotating and the units will be revolutions per minute that is RPM.

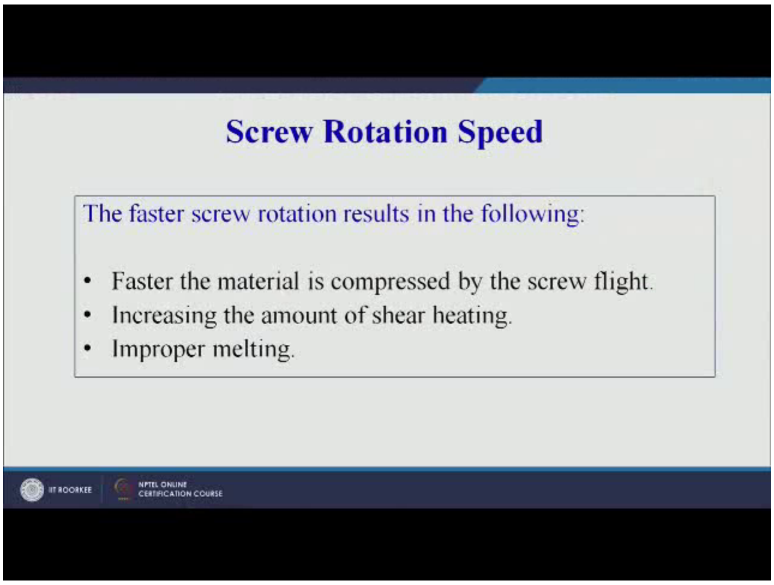
(Refer Slide Time: 21:20)



So, screw rotation speed that is RPM is the rate at which the plasticizing screw rotates. So, that is the rotational speed of the screw, and that also needs to be optimized. Now what are the purposes or the objectives of the screw. As all of us know it is feeding it is mixing, it is melting, it is metering.

Now, the different objectives of the screw have to be satisfied, and the main controlling parameter there is one is a we have already discussed L by D ratio which is fixed once you have the machine your L by D ratio is fixed. Now what you can change is the screw rotation speed. And if the screw rotation speed is not selected judiciously, then proper mixing may not take place proper melting may not take place; that is given in the slide on your screen.

(Refer Slide Time: 22:13)



The faster screw rotation results in the following if it is too fast. Faster the material is compressed by the screw flight. So, the confection or the compression of the material is more, amount of shear heating is more. And sometimes may lead to improper melting. Shear heating may be there, but the uniform melting may not be there if the screw is rotating at a very fast speed or a very higher RPM value. So, that we need to check and we need to optimally select the RPM of the screw rotation speed.

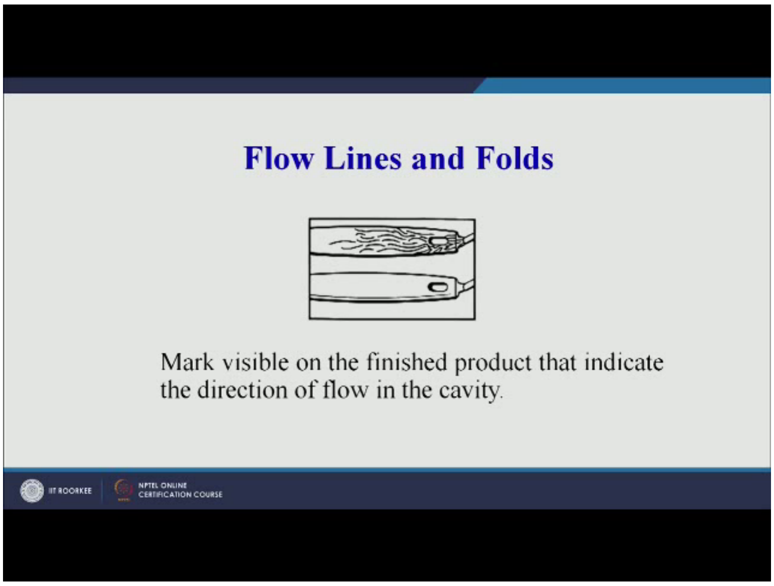
So, just to summarize what we have covered till now in today's session. We have tried to see that what are the important parameters of the injection molding machine that we need to take into account. We started our discussion with the definition of the length, the diameter, and the L by D ratio. And we have seen that what are the range what is the range of the L by D ratio that is usually used in the injection molding machines. Then we have seen the effect of back pressure, we have seen the effect of screw speed or the speed of the screw, then we have see seen the effect of the rotational speed of the screw. We have not seen the direct effect, but we have seen what it can do or what it can lead to if the if the parameters are not selected optimally. Or what are the effect of these parameters on the overall process, that we have try to understand.

Now, I think till now if you have already gone through our discussion in the last session that is the basic of basics of fundamental or basic or fundamentals of injection molding process. You have seen the video, you have seen today's session on the various parameters that influence the process, I think the process fundamentals are absolutely clear to all the learners.

Now, let us see that if we do not control our process properly, what can be the type of defect that may happen and what can be the reasons. Now why he we have try to address all these parameters that is back pressure L by D ratio screw speed, rotation speed, because our these are the reasons for the defects that happen during the injection molding process or the products are rejected based on the non-proper or non-optimal selection of these parameters, because if we do not control our process properly our product will be faulty or defective.

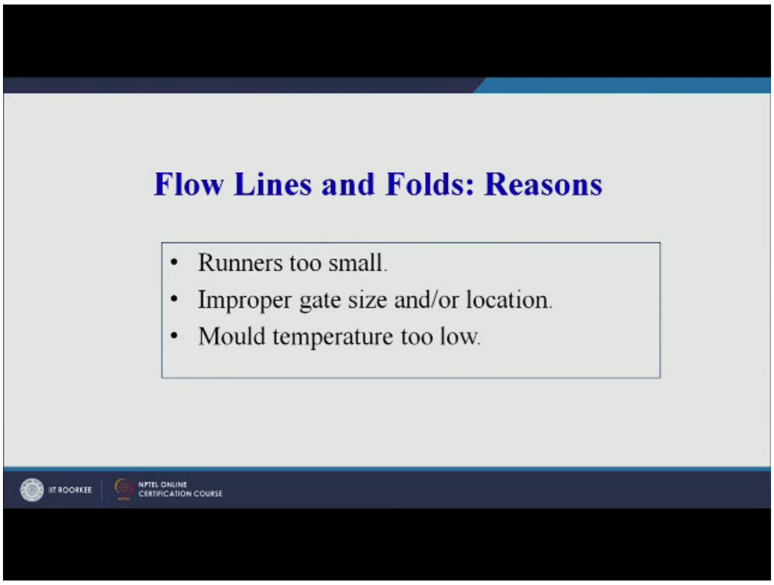
Now, what can we the defects that we will try to understand.

(Refer Slide Time: 25:06)



On your screen you can see the first defect; that is flow lines and folds; we can see there are flow lines. These are the flow lines, and here we can see a fold that has happened. Now these are the marks visible on the finished product that indicate the direction of the flow in the cavity.

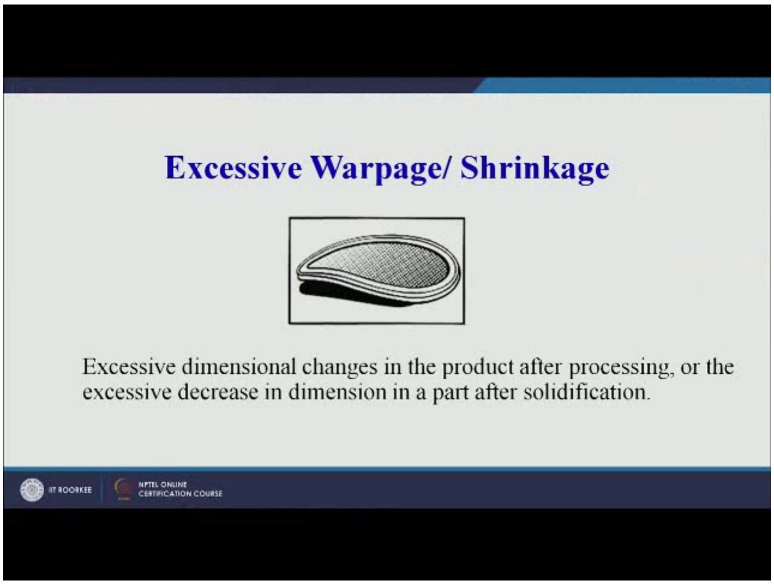
(Refer Slide Time: 25:37)



Now, in the mould cavity the plastic will flow because it has by optimal or adequate viscosity. And sometimes the flow lines are visible. Now what can be the reasons for that? That the runners are too small; the you can say openings through which the plastic molten plastic flows are not up to the optimal diameter or up to the optimal cross section improper gate size and or location. So, the gate through which the plastic enters into the mould is not of the adequate size. Mould temperature is too low. So, these can be the possible or plausible reasons for the lines to appear on the surface of the final product.

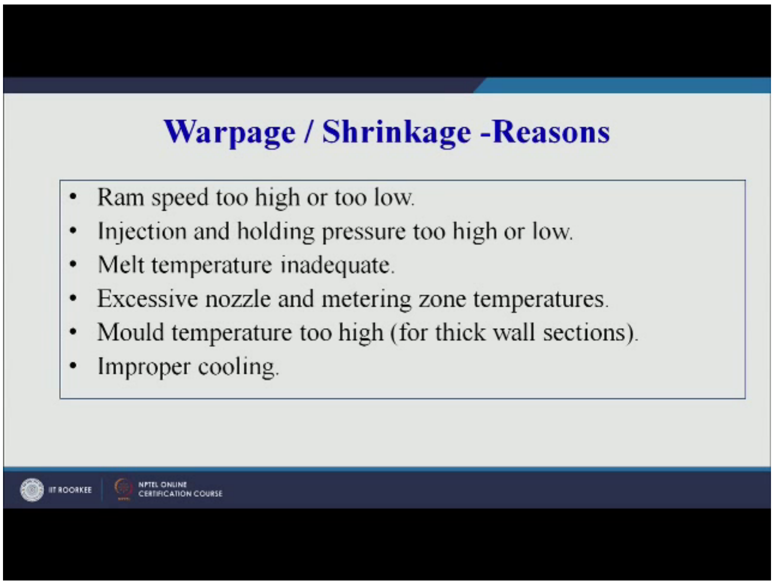
Excessive warpage or shrinkage. So, we want to make a flat product. Sometimes we the ends may get lifted or the warpage may take place. Many times, the product may shrink because of a number of parameters, now excessive dimensional changes in the product after processing or the excessive decrease in the dimension in a part after solidification.

(Refer Slide Time: 26:24)



So, the first part is warpage; that is excessive dimensional changes in the product after processing; that is warpage and shrinkage is the excessive decrease in the dimension of the part after solidification. So, we are addressing both together the warpage and the shrinkage.

(Refer Slide Time: 26:54).

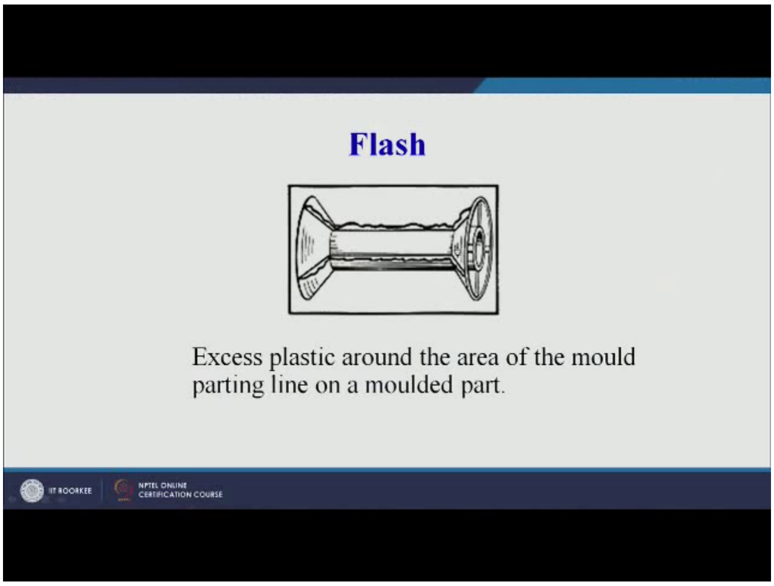


Now, what can be the reasons for warpage and shrinkage? Ram speed too high or too low ram speed, we can say if we are using a plunger type of arrangement for pushing the molten plastic into the mould cavity. So, that is speed if it is too high, it can lead to warpage and shrinkage and if it is too low the again we can get warpage and shrinkage. Therefore, it is important to judiciously select the ram speed. Injection and holding pressures too high or low.

Now, both ends are important. We cannot have a too high injection pressure, we cannot have a too low injection pressure, because both will have or will lead to defective products. Melt temperature is inadequate. So, L by D ratio will come into picture here. So, if you our L by D ratio is considerably less our melt may not be in at a temperature at which we want to inject our plastic into the mould cavity. Excessive nozzle and metering zone temperature.

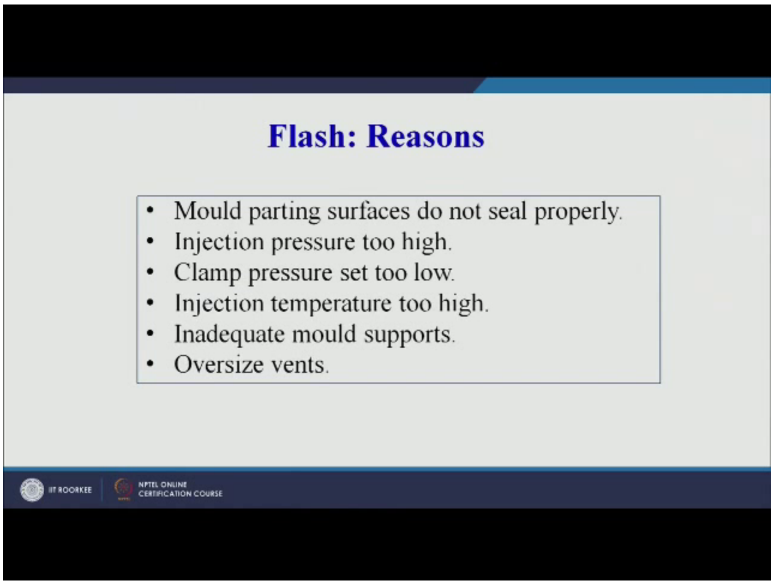
So, excessive temperatures are also not desirable. So, mould temperature is too high, we always have mould at a relatively higher temperature. So, that also is undesirable. And sometimes our cooling that is a third stage of any plastic processing technique that is improper means our product may have warpage or it may have shrinkage.

(Refer Slide Time: 28:19)



The next defect is flash, we can see flash around the product this is a flash, all around the product. Now why flash will happen? That we will try to see in the next slide. Now what is flash it is defined as the excess plastic around the area of the mould. Mould parting line on a molded part. So, and the periphery if you see a plastic bucket we will see around the 2 corner or 2 edges of the bucket, there will be a slight line that is visible. So, that is basically kind of flash and may have been finished to remove the excessive flash.

(Refer Slide Time: 28:57)

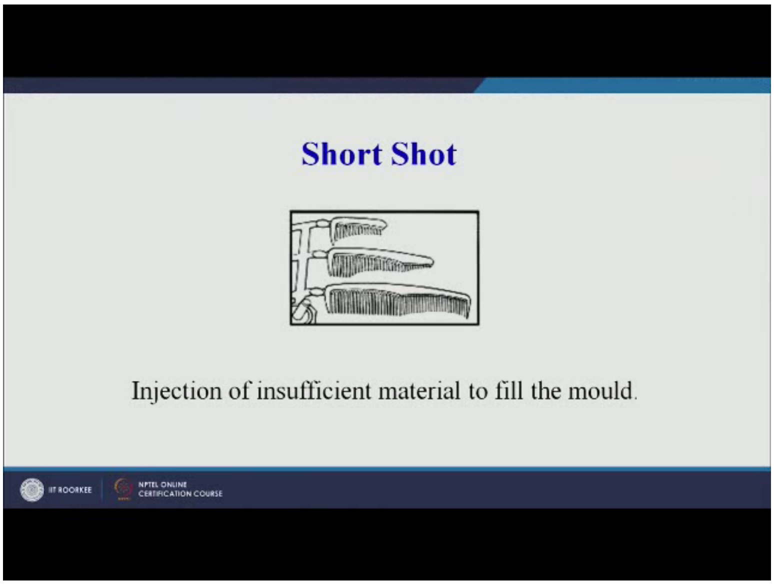


So, that flash why does that flash happens. The reasons for flash or mould parting surfaces do not seal properly. So, as you know in the previous session we have seen, one part of the mould is fixed another is movable which moves over the tie rod. Now when the 2 parts close in between we have the mould cavity. Sometimes the mould parting surfaces these surfaces parting surfaces do not seal properly, and when you inject the plastic at a very high pressure the plastic material flows out of these parting lines and forms the flash; that is undesirable. Injection pressure is too high already I have told.

Clamp pressure is set too low. So, we have to clamp these 2 mould halves at a relatively higher pressure. So, that because of the pressure of the plastic the mould does not open up or turn it to open. So, the clamp pressure if it is less, then also it can lead to flash. Injection temperature too high can also lead to flash. Sometimes which is the mould is supported there are additional supports for the mould, if the mould supports are not adequate it may also lead to even sometimes movement of the mould 2 halves of the mould which may also lead to flash. And sometimes oversize vents may lead to flash.

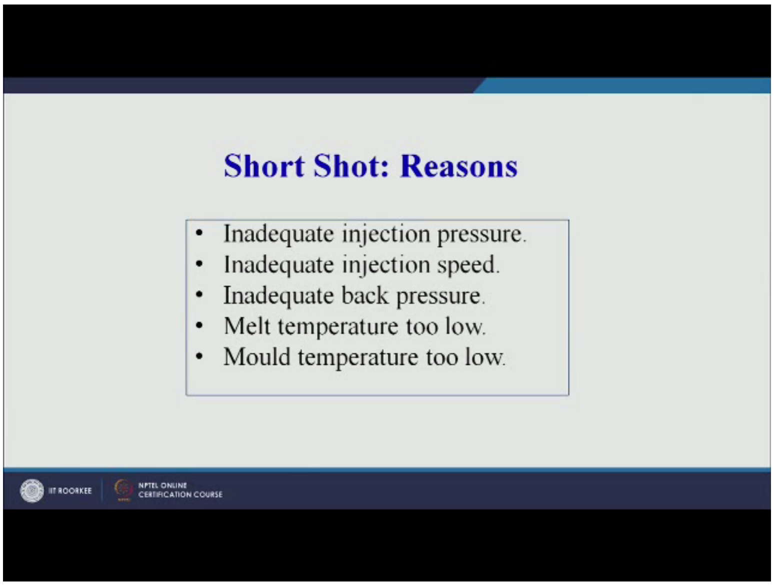
So, we have different reasons for the flash to happen, but the bottom line is that all these reasons have to be addressed in order to avoid this type of defect. Now the last type of defect that we want to cover today is the short which we have covered earlier also.

(Refer Slide Time: 30:34)



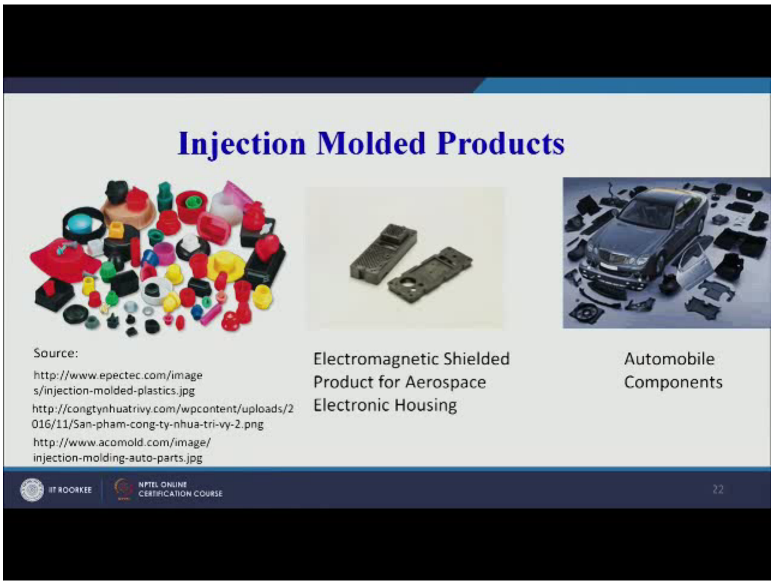
So, injection of insufficient material to fill the mould, now what can be the reasons? Inadequate injection pressure, Inadequate injection speed, inadequate back pressure, melt temperature too low, mould temperature too low.

(Refer Slide Time: 30:37)



So, if you have understood the process, you can take it as a assignment to just figure out that why the mould will not be filled properly. What can be the reason? These are the reasons given, but just where thinking you can jot down or you can write down the reason; that these can be the reasons that the mould cavity may not be filled properly. On your screen now, you have the injection molded parts the source is also given.

(Refer Slide Time: 31:14)



So, in an automobile you can see all of us travel in cars so many parts are made by the injection molding process. Similarly, on the left-hand side this colored parts so many household domestic parts can be made by injection molding process. So, injection molding is a very, very important plastic processing technique which I as a teacher believe that all engineers must know. So, that they are able to figure out that which part has been made by which process, or at least this part may have been made by injection molding process.

So, that much information all engineers must have. So, with this although there are so many other topics related to injection molding; that we can cover, but I believe that in the last 2 sessions today's session and the last session we have tried to address this issue of injection molding and all learners must have got the fundamental knowledge related to the process of injection molding. In our next session we will the start our discussion on the another plastic processing technique transfer molding.

Thank you very much.