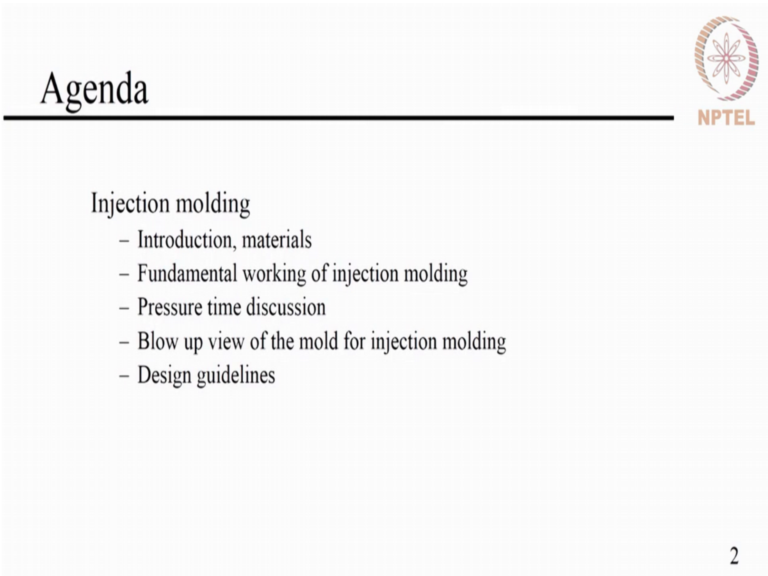
# Design for Quality, Manufacturing and Assembly

**Prof. G. Saravana Kumar**
**Department of Engineering Design**
**Indian Institute of Technology, Madras**

## Lecture-19
## Introduction to Injection Molding Process, Materials Terminologies Related to Plastic Parts & Design Guidelines

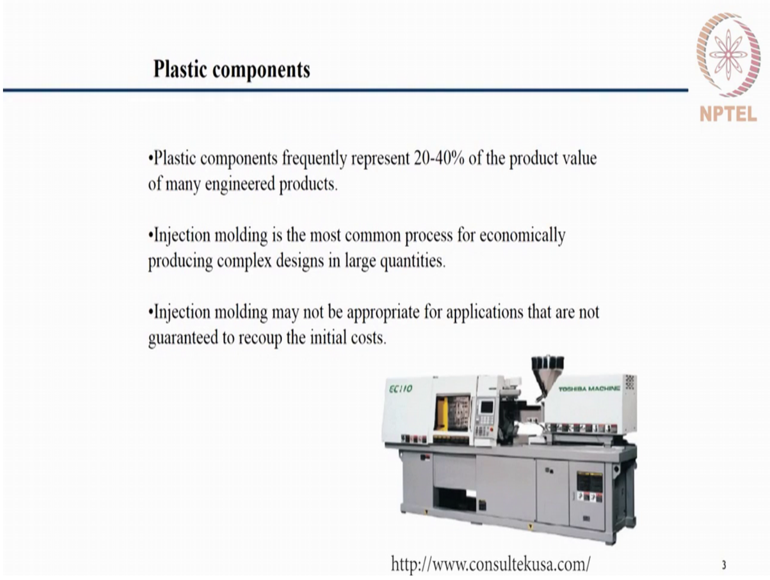
Good afternoon everyone. This a course is on Design for x and as part of this course I am going to discuss Design for Manufacturability specifically that of plastic parts in this lecture. So, I am Saravana Kumar from Department of Engineering design IIT, Madras. So, this is agenda for today's lecture.

(Refer Slide Time: 00:47)



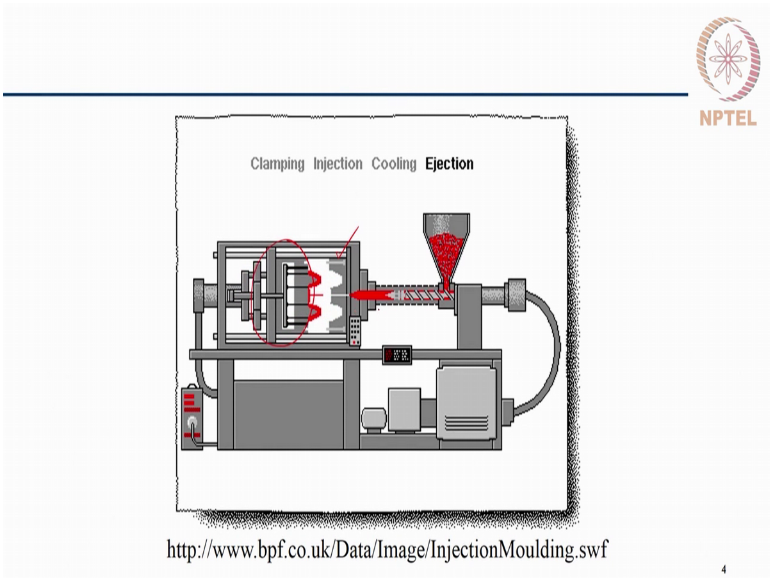

So, we will see the injection molding process, which is used predominantly for manufacturing plastic components. We will see the process the material and the design considerations for plastic parts. So, plastic components are majorly used in most domains.

(Refer Slide Time: 01:21)




About 50 percentage to 60 percentage of the components used in various sectors whether it be automotive or biomedical or household appliances. They are made from plastics as plastic materials are used extensively the manufacturability and the cost of the product is significantly affected by the manufacturability and the process processing of the plastics. So, injection molding is one of the processes that is predominantly used for making the plastic components, because for large volume production the cost per piece is the least for the injection molder components.

(Refer Slide Time: 02:22)

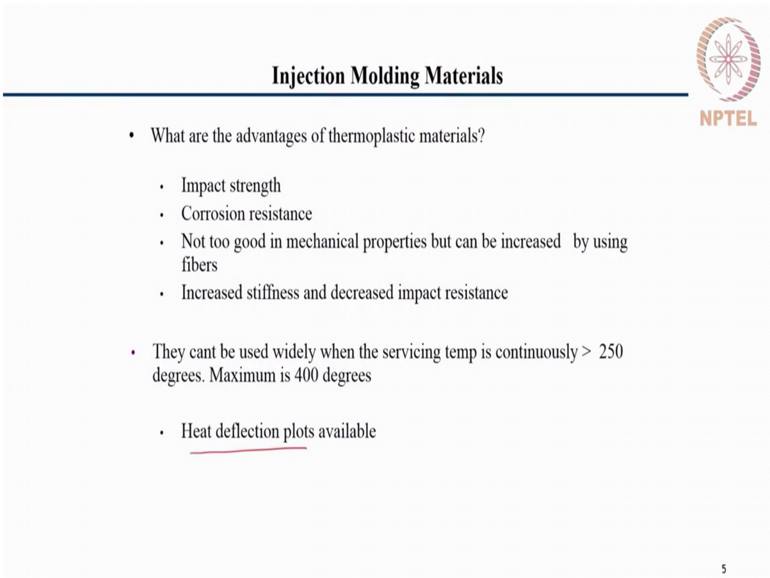


So, injection molding process consists of about four stages the mold which confines the material to the part geometry. It is made from the part drawing and mold is manufactured and the mold is clamped using the machine.

So, there is a movable part of the mold and there is a fixed point, on the side from which the injection takes place. So, you can see that this is the moving component and this is the fixed part of the mold and once the mold is clamped, the injection takes place the plastic material is injected by the injector. Once the material is filled in the mold, it is allowed to cool and finally, the part is ejected. So, this constitutes one cycle of molding and the time that is required to complete one cycle determines the cost of the part. So, the optimization for the process involves are reducing the cycle time. One of the important parameters is the cycle time.

So, as thermoplastic materials, they are quite versatile and they are lot of engineering grade thermoplastic materials that are used.

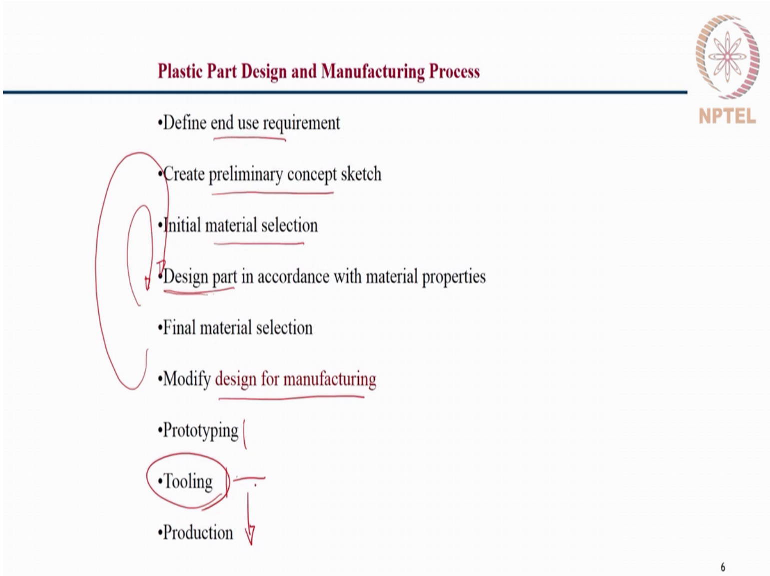
(Refer Slide Time: 03:59)




It can be used wherever there is chemical and corrosion resistance is required, where we need good strength, we can use it. And the limitations in terms of the usage comes from the fact, they cannot be used for very high mechanical loads and also for very high temperature applications. Typical usage temperatures are up to about 200 degree Celsius and there are some grades of plastics which can serve at slightly higher temperatures.

So, the heat deflection plots for different materials are available as design guidelines which are used for making the choice of the material if there is a requirement for service temperatures at higher degrees.

(Refer Slide Time: 05:07)

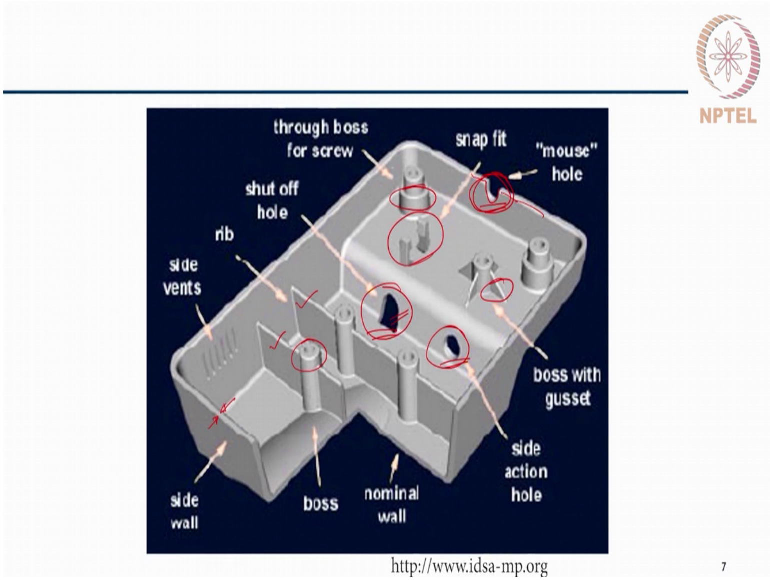



So, a plastic part design and manufacturing process starts with defining the end use requirement, because that decides the type of material that needs to be used and the process for manufacturing the same. Based on the functional requirement, preliminary concept is generated design concept is generated and material is selected and the part design is made according to that.

So, this is an iterative process because once the material is selected and a design is made, various functional testing either in computer simulations are by using proto type and verification is done; before the material as well as the design can be finalized. So, this step itself is highly iterative in nature and also we consider manufacturability and the cost and this step is also iterative.

So, we will see what are the design considerations for manufacturability for the injection molded components and once the design is fairly matured, we go ahead with prototyping and then developing the final tooling because here we are making the tool and thereby we are in curing the cost. So, this step is very critical, you cannot typically go back to design, when we are here and then we release the mold for production.

(Refer Slide Time: 07:05)




Before we see the process, let us understand certain terminologies associated with plastic components. Typically plastic components are thin components shell like structures. And for stiffening the structure, we use ribs and webs and bosses wherever we need to have some interfacing elements, wherever this plastic component has to interface with other components like placement of say some other component on to the structure. So, the part is typically of constant thickness or what we call as this side wall in most places.

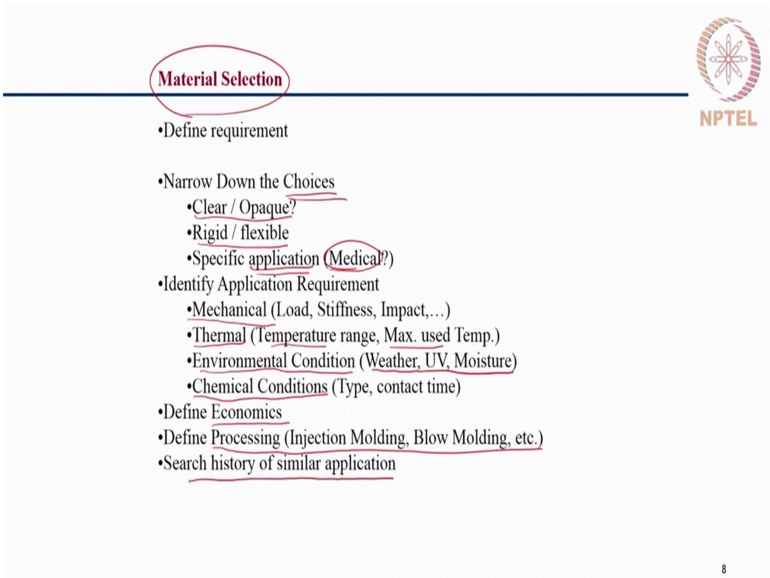
And it may have these kind of structures which are the ribs which are used to increase the stiffness of the component without adding too much weight. And we have the bosses which can take in elements like screws. The bosses may also have sometimes these gussets to provide additional stiffness and sometimes the bosses may be made thicker or they may be through bosses to accommodate screws. There may be holes like this, as shut off hole that you see over here or these mouse holes. So, these holes as you can see, they are on the transverse direction. The injection molding direction for this component is from top to bottom that is the mold action takes place from top to bottom and the part is taken out.

So, all the features that are along this mold opening and closing direction. So, they are the side walls the ribs, that you see in the picture as well as the buses. The features like this mouse hole and the shut off hole and the side action hole, they are transverse to the mold. So, that should be some special structures in the mold like what we call as a side action pin. So, that these features can be created in the mold and the part will have these features. Once we see the whole process will be able to appreciate the need for this side action pins.

So, these holes like the shut off hole can be made using a mold feature in the mold opening and closing direction, but this side action hole needs a side action pin. Even this mouse hole can be made without side action because that can be placed just along the parting line; this is a parting line. So, this feature can be obtained without having a side action pin in the mold.

So, these are some of the terminologies that we may we referring to as we discussed. We also see one feature here this is called as a snap fit feature most of the times in plastic components. We will try to avoid to the extent possible the screws and snap which I have encouraged because that reduces the assembly time when these components are assembled with other structures.

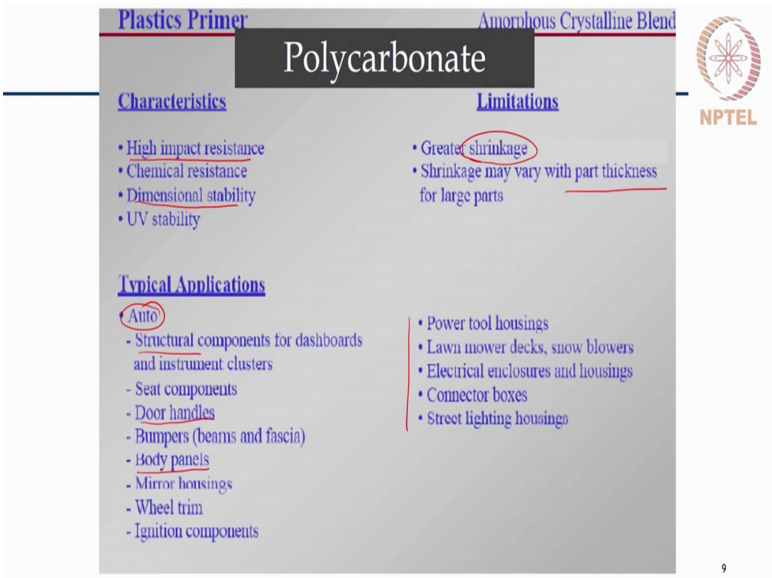
(Refer Slide Time: 11:25)




So, in this process the material section selection is the key and the first process. So, there are several choices to make based on the functional requirements such as whether the component that is being made should be clear or opaque or whether it has to be rigid or flexible, what is application, applications like medical needs special types of plastics. There are FDA approved grades of materials used for medical applications and based on the functional requirement, what kind of loads and impact will come on the component, what kind of thermal loads will be there like the working temperature, maximum temperature. The environment in which this component will be servicing like the humidity weather, UV light and so forth also the chemical nature of the environment is to be considered for doing the material selection. Material selection also should consider economics because one of the key characteristics or one of the significant parameters associated with plastic part design is the final cost.

So, economics is important and based on this, we have to also look at the processing condition and take help from history in terms of applicability of the material or similar applications. So, this a guideline for doing a material selection. So, there are lot of materials that are available.

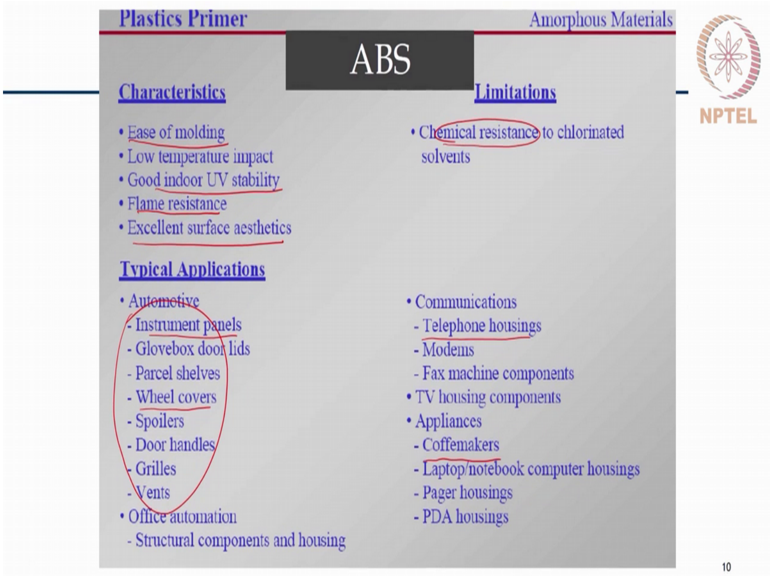
(Refer Slide Time: 13:33)




There are guidelines available for choosing the materials based on the various characteristics few sample materials, we will see. So, one of the most used materials is polycarbonate are or in short we call it as pc. It has very good impact resistance and dimensional stability apart from various other positive attributes is predominantly used in automotive structures wherever strength and impact resistance is important like door handles body panels etcetera.

Some limitations in terms of the shrinkage, the part shrinkage needs to be considered while designing the mold. So, that the final components that are manufactured, they have dimensional they are dimensionally accurate and also do not exhibit warpage because of non uniform shrinkage. Like when the part thickness varies, the shrinkage will cause dimensional instability and warpage. So, there are lot of applications where this material can be used.

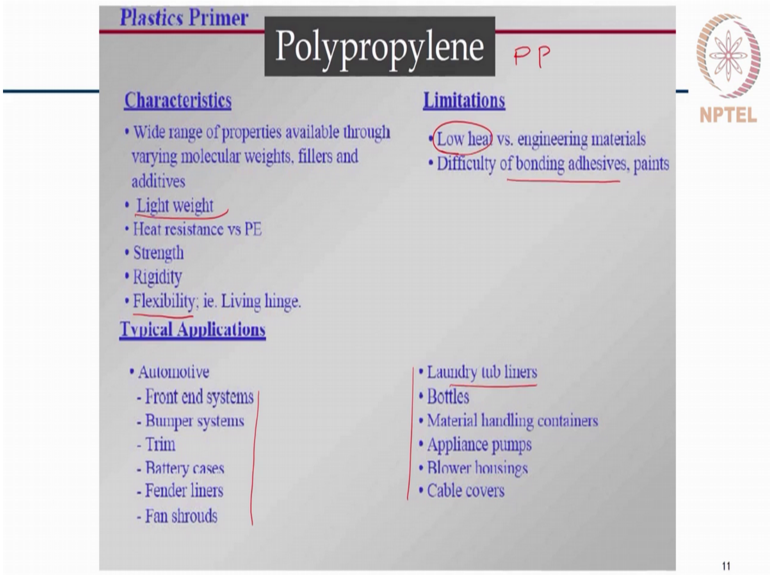
(Refer Slide Time: 15:09)




The other important materials is ABS. So, this is used primarily because of the excellent surface characteristics and ease of molding apart from several other characteristics like flame resistance and UV stability.

This material is again used for several applications like in automotive domain for making several instrument panels to wheel covers and in other applications like telephone housing and household appliances like coffeemakers and so forth. In terms of limitations, the chemical resistance to chlorinated solvents is less. So, based on these considerations, one can choose the material.

(Refer Slide Time: 16:15)

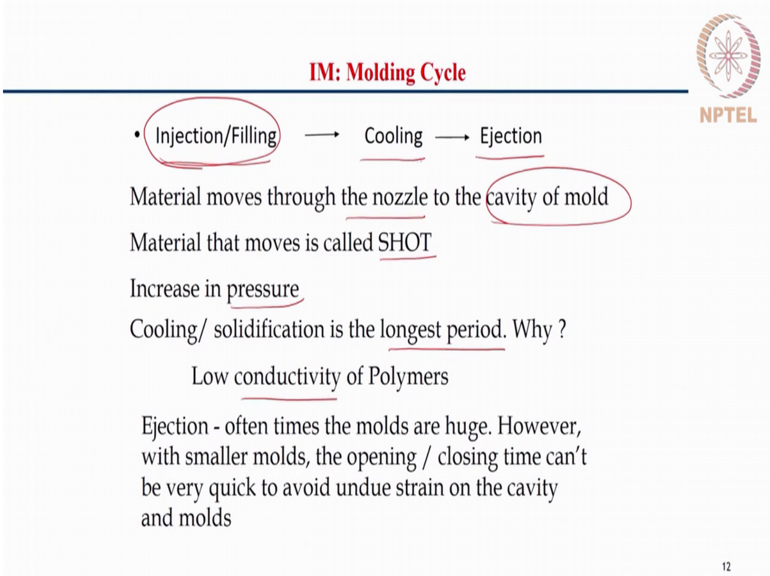



One of the other important material is polypropylene or what we called as PP. It has lots of applications particularly because of the requirement wherever there is a lightweight requirement and flexibility.

Lot of casings and liners, shrouds and all are made using this material. In terms of limitation, it cannot be used in high temperature or whenever there is lot of you know use of materials for bonding and difficulty in painting. So, lot of other applications like bottles, water bottles to laundry tub liners are made using this material. So, this is just a sample of the spectrum of materials; the thermoplastic materials that are available. And there are good material selection guides available to help engineers make the good initial choice of material and then iterate upon based on the design and the requirement.

So, once the material is selected, we need to make the mold and in order to design the mold, we need to have a understanding of the mold cycle injection molding cycle.

(Refer Slide Time: 18:07)

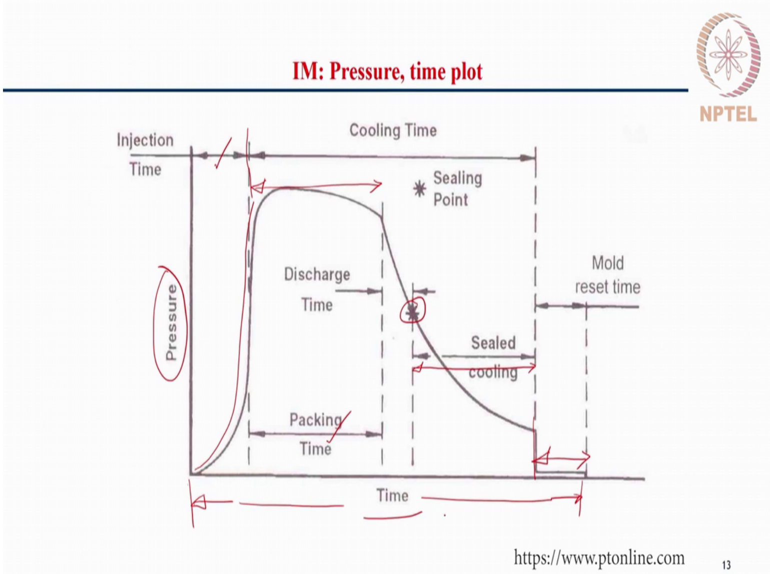



So, typically it consists of the filling stage or what we call as the injection stage and then, the part cools inside the mold and then the part is ejected. So, the material that is injected through the nozzle into the cavity of the mold is called as a short. And to make the material move into the mold, we need to increase the pressure.

So, typically the injection pressure is the maximum while the material is injected and then gradually it is reduced while it is cooling and it is not there in cool part of cooling as well as in the ejection stage. So, cooling is typically the longest period in one mold cycle because, we need the material to cool and polymers being low conductive in nature. We need to give more time for them to cool though heat sinks can be used to increase the rate of cooling.

Ejection is the final stage where the part is ejected from the mold by opening the mold and there are also some ejection pins that are provided to help the part come out of the mold. Drafts are also provided and the part surfaces; so, that the part can be ejected easily from the mold.

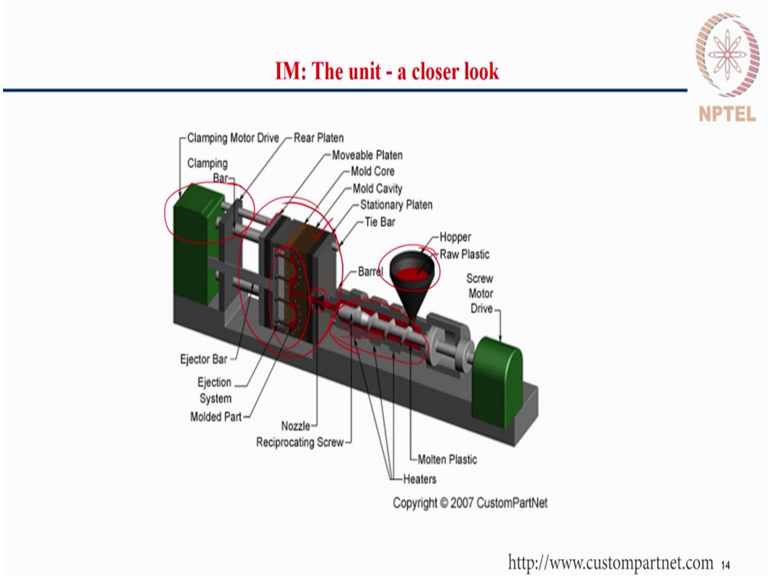
(Refer Slide Time: 20:17)




So, what we see here is the injection pressure versus the time associated with the molding. So, what we see here is one full mold cycle. It is consists of the first part which is the injection time where the pressure gradually increases and then, the material fills the mold during this time which is called as the packing time. And once the material fills the mold, the pressure is reduced. There is a point called as sealing point where there is no further flow of material and the part is allowed to cool in this sealed condition.

Once the part is cooled, the part is ejected and it is reset for the next cycle. So, this graph which shows the injection pressure versus the time is very important from the perspective of optimizing the mold cycle and thereby the cost of the component. This cycle decides the mold filling characteristics and thereby the part quality as well as the cost of the part. So, this picture shows the injection molding unit.

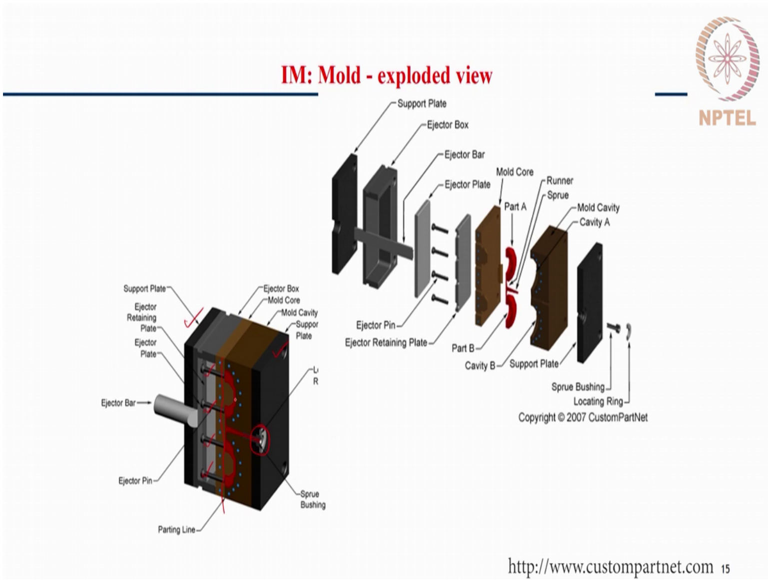
(Refer Slide Time: 22:34)




This is the mold and this is the injector. We have a nozzle here through which the material is injected this is a hopper from which the material is fed. So, there are heaters available in the injector, which heat the material and the material flows through this nozzle into the mold cavity.

So, the this side of the machine shows the clamping system which applies the clamping force that is required to keep the two halves of the mold; this half and the other half in the closed con[dition]- condition when the material is injected into the mold cavity.

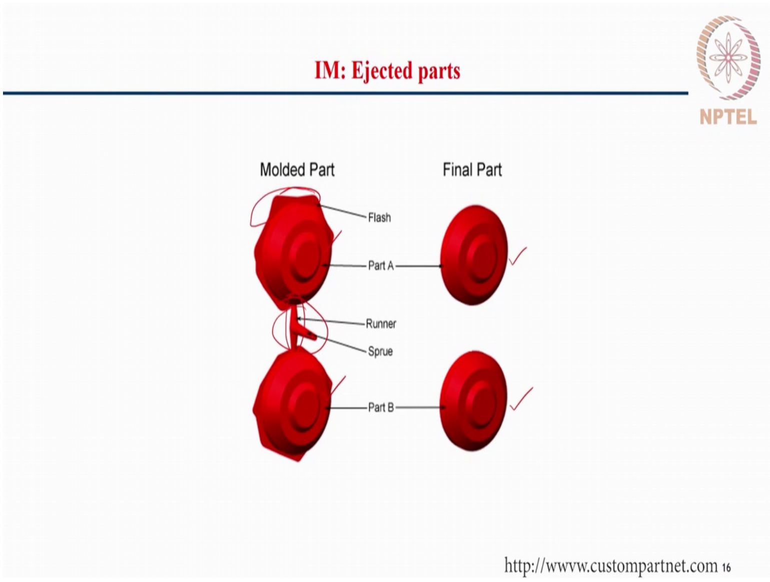
(Refer Slide Time: 23:37)



A closer view of the mold shows the details of the various components. So, there are supporting plates which are clamped to the machine and which help in clamping the mold plates. The surface that divides the two molds is called the parting line or the parting surface.

And on the injection side, we have screws hole for filling the material and these are the ejector pins which are used to eject the component once the injection is injection molding is completed.

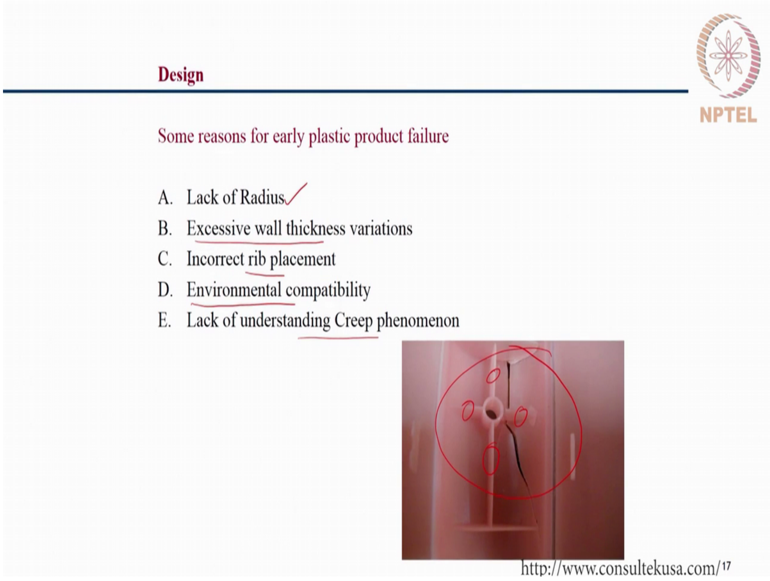
(Refer Slide Time: 24:37)




In the part comes out of the mold or when the part is ejected, there are few unwanted the elements associated with the part like the flash. A flash is required because flash ensures that the mold has completely filled and the material actually tries to come through the parting surface. This ensures that all parts of the mold are filled. So, flash is an essential component that can be minimized, but cannot be completely removed.

Once a part is taken out, this flash is removed as well as other elements like the runner the spruces. And once these are removed, we have the final component to reduce the part cast typically in a single mold there are multiple cavities. So, each of this is a cavity; this is one cavity, this is an another cavity. And to make the material flow to all the cavity, we have this runners on the mold which help in distributing the material from the the injections prove to the different cavities in the mold. Multiple cavities help in reducing the cost of the parts.

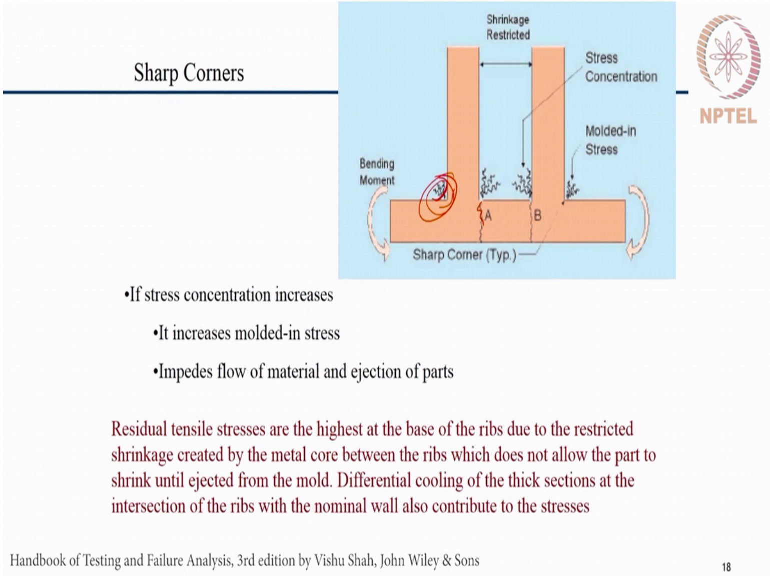
(Refer Slide Time: 26:27)




While designing the mold and as well as the selection of material and the design of the plastic component, we need to consider the typical failure that happens during service. In this slide, you see a picture of a failed plastic component where the part has cracked because of the load that they part have to take. Even though in the presence of stiffening elements like these webs are stiffness that are available. So, some of the reasons for such failure could be that there is not sufficient radius of the mold surface in the corners which could cause the stresses to build up or what we call as the building up of the stress or stress concentration because of lack of radius.

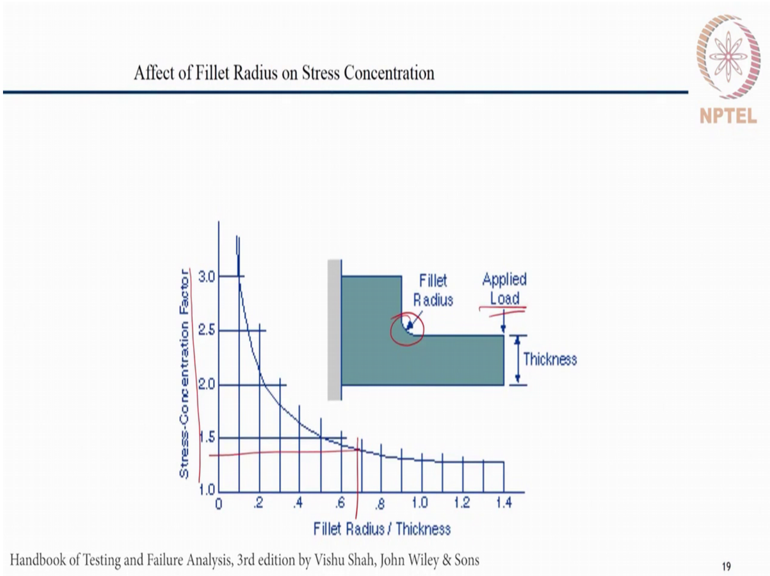
It could also be because of excessive wall thickness variation because of the change in the wall thickness, the material may not flow completely into some parts or there could be differences in the stress which could lead to the failure. The replacement of ribs are important also the environment in which the component is serving. If the material is not chosen considering the the various environmental conditions under which it has to serve, then also the component can fail. Also material can fail because of creep which is a long term failure phenomenon. To avoid failures, one has to consider some design thumb rules while designing the plastic components.

(Refer Slide Time: 28:53)



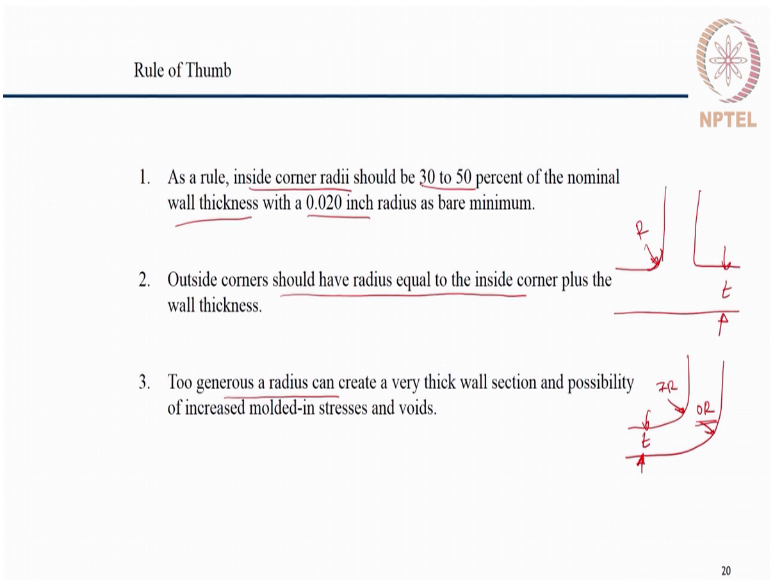

We will see one by one first thing is to avoid sharp corners or provide sufficient radius; so, that the stress concentrations can be minimized. So, wherever such stress concentration happens, failure will happen; the cracks can initiate at this points and the the material or the part can fail. So, at these points, we have what we called as the mold in stress because the material has resistance in flowing at these corners and that could also be a potential cause of increase in the failure because of the forces that are involved in ejecting this component. Because of sharp corner radiuses, the part ejection from the mold can also be a problem. So, one of the important design consideration is to avoid such a corners.

(Refer Slide Time: 30:20).




Studies which reflect upon how this corner radius will have an effect on this stress concentration. So, one can choose an appropriate radius and know what kind of a stress concentration factor, one has to use in the design for any given applied load.

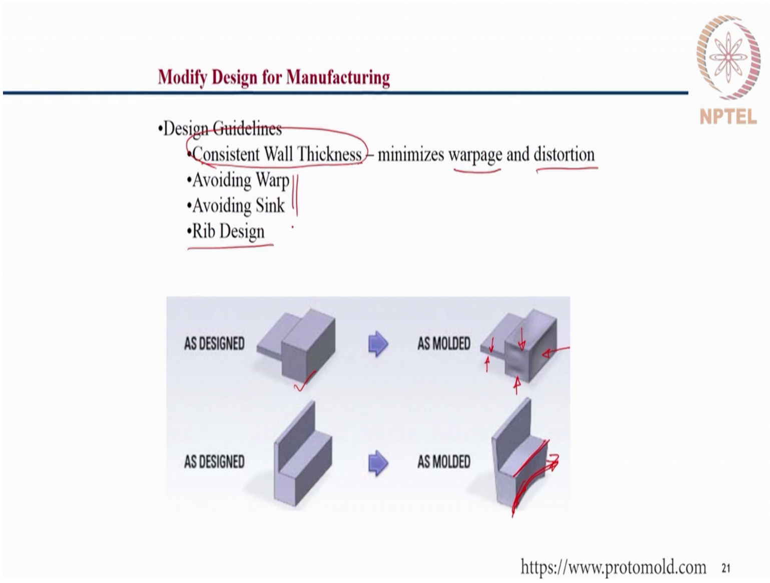
(Refer Slide Time: 30:49)




The general rule of thumb is to have an inner corner radius about 30 to 50 percent of the normal wall thickness say, the normal wall thickness is t a corner radius here R has to be about 50 percent of this value.

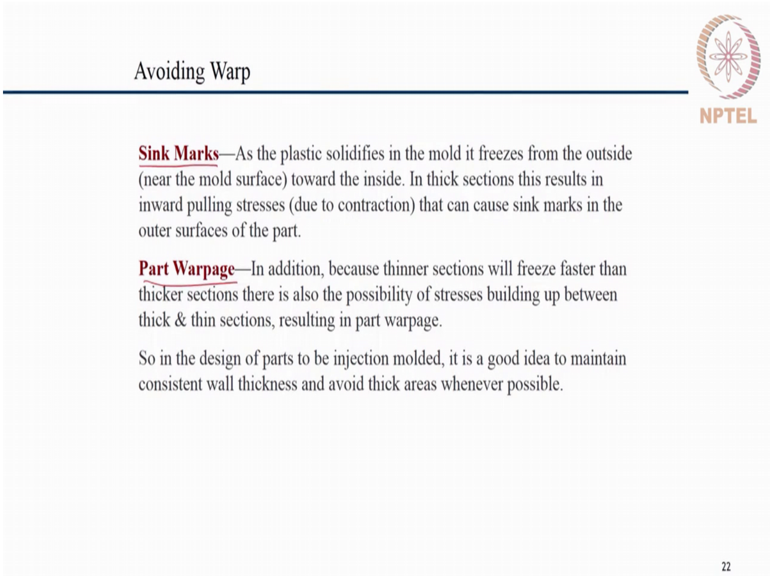
There are some suggested minimum value. Outside corners should have radius equal to the inside corner plus the wall thickness. So, if there is a component like this. So, this is the inner radius and this is the outer radius. So, once the inner radius is decided the outer radius is taken based on the wall thickness and the inner radius. Too larger radius is also a problem because it will lead to lot of material and thereby shrinkage and warping.

(Refer Slide Time: 32:12)



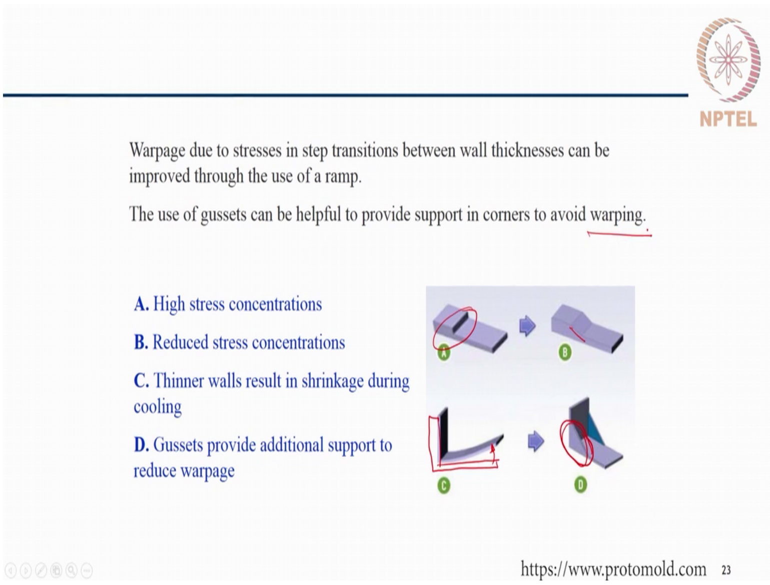

So, when there are problems associated with changing wall thickness the part source warpage and distortion as can be seen in this picture. So, this is the designed component or what is expected from the mold, but because of the change in thickness. So, here the wall thickness is this much and there is a sudden change in the wall thickness. And because of which because of this changed material, so there is a warpage and distortion. So, we can see the sink marks here and the warpage. So, this is supposed to be a straight surface instead it has wart. So, by proper rib design and by either having constant wall thickness or gradually changing the wall thickness, these can be avoided that is warping and the sink marks.

(Refer Slide Time: 33:37)



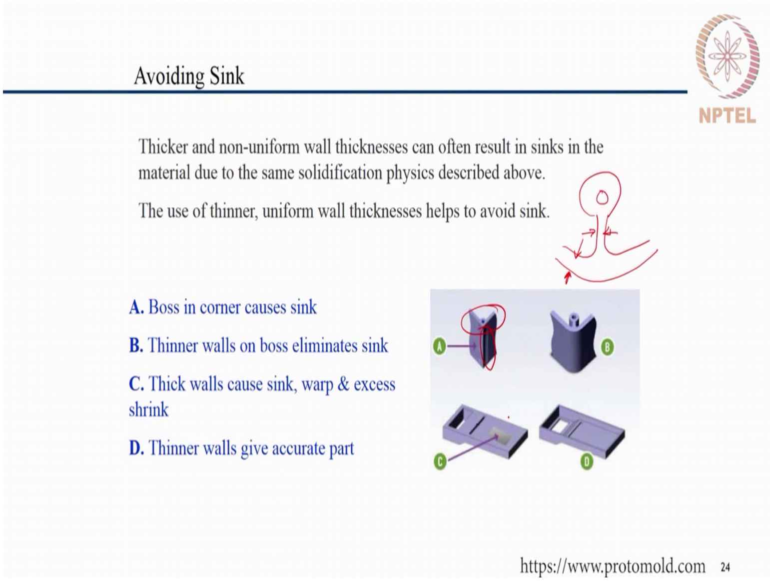

So, we as we saw sink marks are caused because when the plastic solidifies on the surface material is pulled inside and thereby the surface shows a sink mark. And part warpage is again due to shrinkage because largest section is shrinking it pulls a thinner section or deforms the thinner section because of the shrinkage of the larger section. So, these are typical problems associated with the plastic component.

(Refer Slide Time: 34:25).




So, warpage can be avoided by providing gussets. So, these are the gussets without gussets the as designed component is supposed to be like this. But once the mold is done and the part is taken out, the part works due to the stresses and that can be avoided by providing the gussets. And one can also avoid these steps are what we call as change in the thickness by providing a gradual change in the thickness. So, these things can help avoid warping. So, these are general guidelines one can follow while designing the mold.

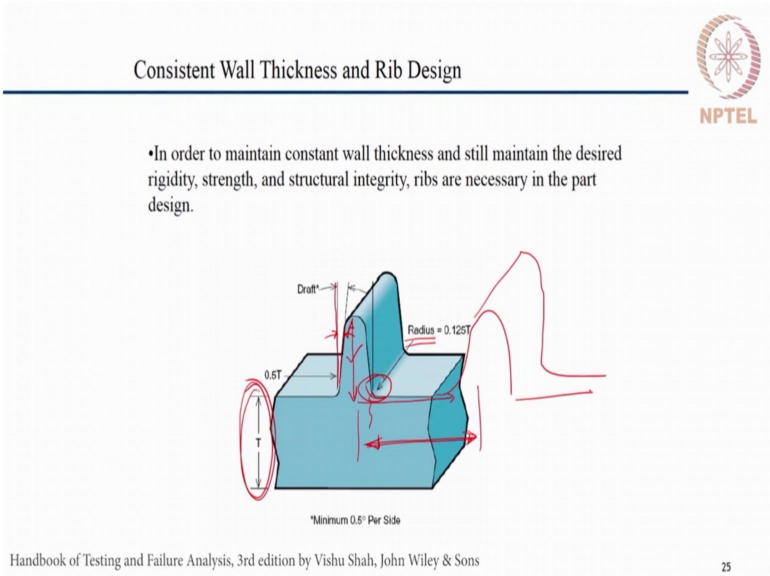
(Refer Slide Time: 35:29)




To avoid sink marks one can use uniform wall thickness as far as possible wherever it is not possible, one has to put design features that will help in maintaining uniform thickness. As we can see over here, there is a corner feature because of which there is a change in the thickness and causing the sink mark.

So, we can change the design. So, that there is a constant wall thickness. I am just redrawing this for clarity. So, this thickness is kept uniform everywhere to avoid the sink marks. Thicker sections can be avoided wherever possible.

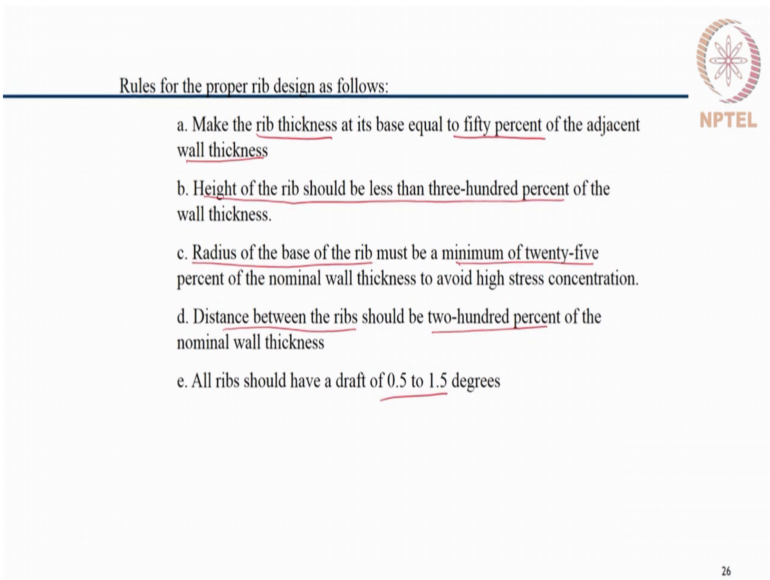
(Refer Slide Time: 36:36)




Ribs are to be designed properly. So, that one can avoid failures at these corners. So, one of the important feature dimension is this corner radius. Based on this nominal wall thickness T, one has to make appropriate choice of this radius the height of the rib and even the placement of the successive rib. Let us say if an another rib has to come here, what is the distance between the placement of two ribs?

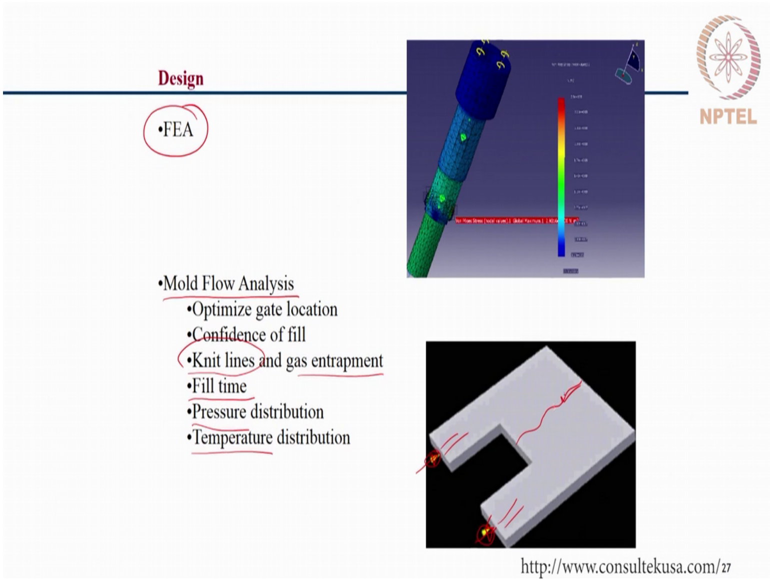
The height of the rib the corner radius and placement of successive ribs, they need to be determined based on the nominal wall thickness. So, there are guidelines to this.

(Refer Slide Time: 37:40)



Like the rib thickness is 50 percent of the adjacent wall thickness. The height of the rib should be less than 300 percent of the wall thickness. The radius of the base should be minimum 25 percentage and distance between the ribs should be 200 percentage and so forth. And there is also a draft. So, this is the draft angle, this angle is the draft angle. This helps in removing the component from the mold.

(Refer Slide Time: 38:27)



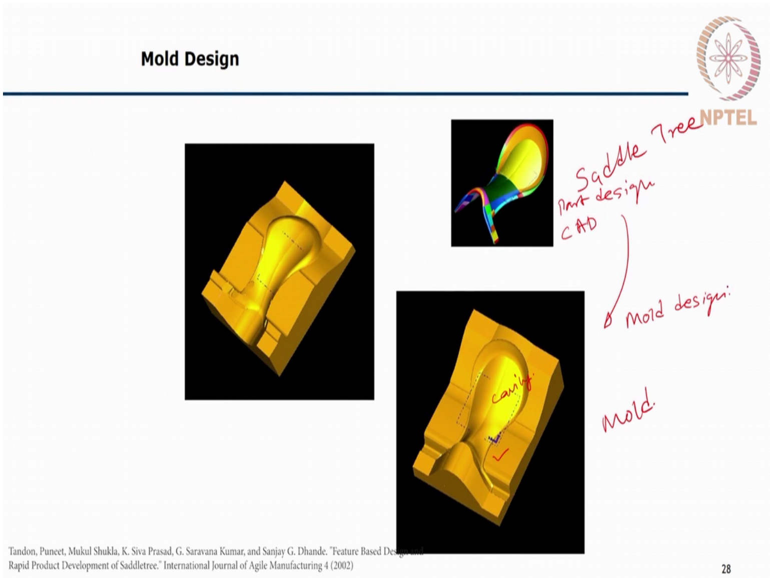

Another design tool which helps in proper design of the plastic components is using the numerical modelling and simulation using finite element method. One can also do what we call as a mold flow analysis. Analyzing how the mold flows into the material flows into the mold, the temperature distribution, the pressure distribution. All these can be studied numerically by computer simulation models.

So, finite element analysis structural analysis can be done to understand the load versus deflection behavior of the designed part and one can change the design based on that. And for the design component, one can do then a mold flow analysis to understand the process variables and its effect on the quality of the final component.

So, one can understand the field time because this is critical in terms of the mold cycle time and the pressure and temperature distribution. Also the formation of what is called as knit lines and gas entrapment like as an example in this picture, there are two gates through which the material flows into this cavity. So, the materials start filling this cavity and once the material fills they will finally, come and join there by forming these knit lines. So, appropriate placement of these gates can help in deciding where these knit lines will happen.

Knit line should be avoided wherever there are high stress concentration region. So, by suitably placing the gates, one can decide or one can get the knit lines at places where there are no high concentration of stresses.

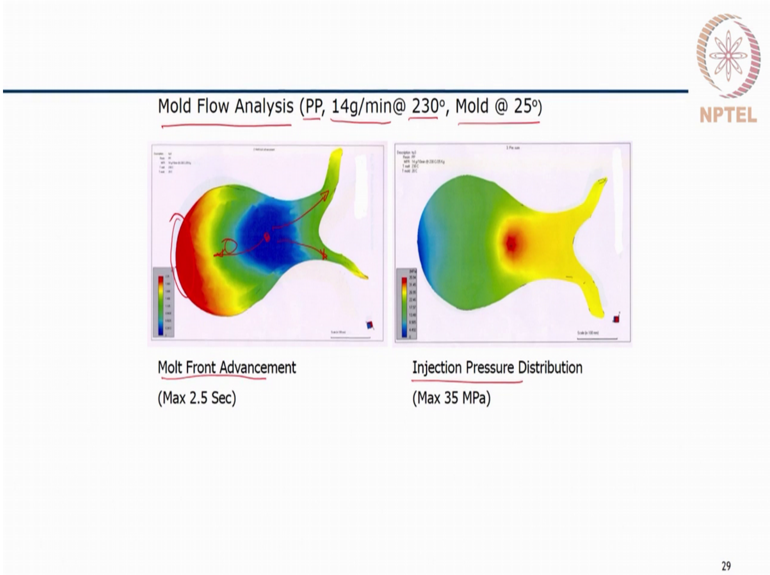
(Refer Slide Time: 40:49)




So, this an example design process, which I am showing using one product. This is called as a saddle tree which is used in the saddle which is main ha you know leather component used to help jockey sit on a horse while riding. So, this is a saddle tree which is a injection molded component what you see is a cad drawing. Based on the cad, one designs the mold.

There two halves of the mold are shown, this is a parting surface that you see and this is a cavity. So, there are software's available to help do this mould design, once we have the part design  from part design we get this mold design.

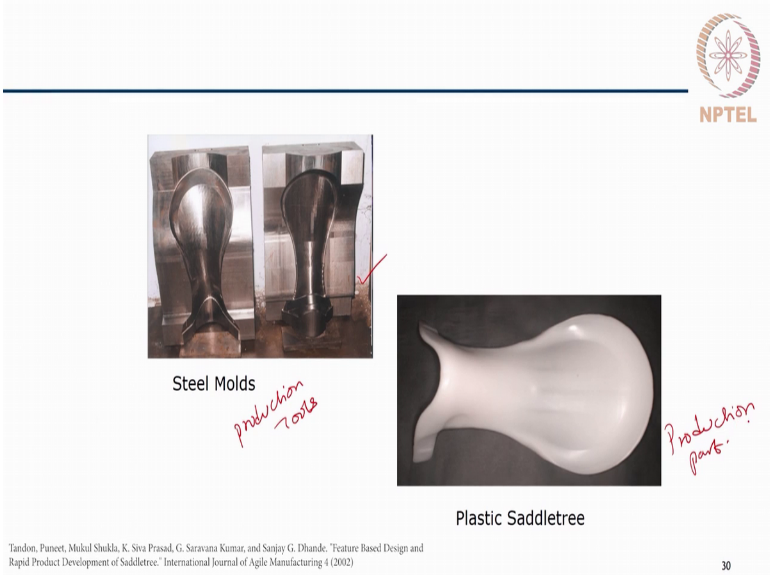
(Refer Slide Time: 42:08)




Once we have the mold design, we can use suitable software's to do the mold flow analysis like here you see a simulation of polypropylene being injected at 14 grams per minute rate at this temperature mole being kept at 25 degrees. It shows the mold front advancement how the material flows from the injection point.

This is over here to the different regions. So, one can see at different times how the material flows into the cavity, what are the regions which gets filled up at the last like here. We say that it takes almost 2.5 seconds to for the material to reach here whereas, its only about one second somewhere over here. One can also analyze the injection pressure that is required for filling this cavity and its distribution. Injection pressure decides also the capacity of the mold holding force that is required and there by the capacity of the machine itself to hold the, to provide the clamping force that is required to hold the molds together.

(Refer Slide Time: 43:45)




These are pictures showing the finally, manufactured tools. Once this simulations are done and the design of the mold is verified, one can go for production tools and we have the production parts coming from that. So, we see that the design of the mold is critical from the perspective that the material has to flow, we need to optimize the filling time or what is called as the mold cycle time. So, that the cast is kept at the minimum.

One of the key parameters for the choice of plastic components is the cost of the component itself. So, economics plays a vital role and thereby the designers have to consider this while doing the design. So, we will continue with the design considerations in the next lecture.