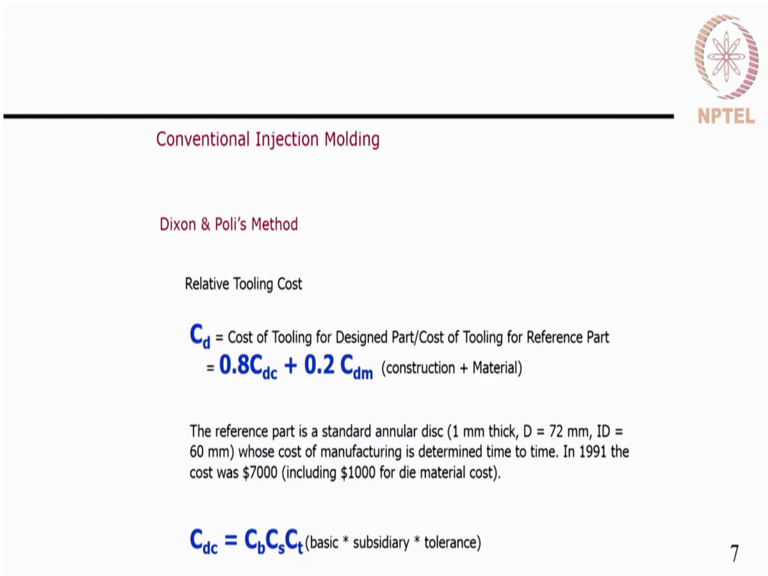
Design for Quality, Manufacturing and Assembly
Prof. G. Saravana Kumar
Department of Engineering Design
Indian Institute of Technology, Madras

Lecture – 21
Estimation of Mold Cost for Injection Molding (Dixon & Poli's Method) – Continued

Good afternoon everyone. So, today I will give a quick recap of what we discussed in the previous class before continuing with the you know discussion. So, in injection molding which is the predominant manufacturing process for plastic components the cost of the you know end part is primarily determined by the tool cost, ok. The operating cost is fairly very low in terms of materials or in terms of the costs. So, the tool costs gets distributed over the number of parts if you are making let us say 100 of 1000 of parts the tool cost is going to distribute on, on that right.

And injection molding will be cheaper only go when you go for mass production ok. So, the tool cost being one of the important cost component, the design for manufacturing should consider cost of making the tool. How difficult it is to machine the die the tool that will be used for doing the injection molding, ok. So, there is several methods, but Dixon and Poli's method is one you know very good method to estimate the cost. So, in this cost costing method we compute what we called as a relative tooling cost that is called as the C d.

(Refer Slide Time: 01:43)



So, there is a standard component which is typically a circular disc as described a 72 mm outer dia and 62 mm inner dia disc with 1 mm thick. The tooling cost of that is considered as a reference tooling cost and the numbers that you get as a relative tooling cost is multiplied to that cost to get the cost of the tool. So, this is to take care of you know the fact that you know with time you know the costing change, right how material cost the labor cost everything changes, but as a method this is remaining this needs kind of normalized the method is normalized over the current you know costs that will be.

So, the relative tool cost is divided further into die construction cost and die material cost and it is roughly 80 percent of deconstruction cost and 20 percent of the die material cost. The die construction cost itself is computed or it is decomposed into a basic cost a subsidiary cost and a tolerance cost. The basic is primarily based on the size of the component; how much you know machining needs to be done to take care of features such as the undercuts in the component right. The undercuts are as an example if I would like to show.

(Refer Slide Time: 03:15)

If I am going to keep this part in the as, this is the mold closing and opening direction, let us say if I am going to place the component like this then the undercuts are all features that are transfers to this which cannot be obtained in the mold opening and closing direction. Because, this whole if I have to obtain in the mold opening direction, I cannot have a pin in the transverse direction because when it will come out the part will still be get stuck in the tool.

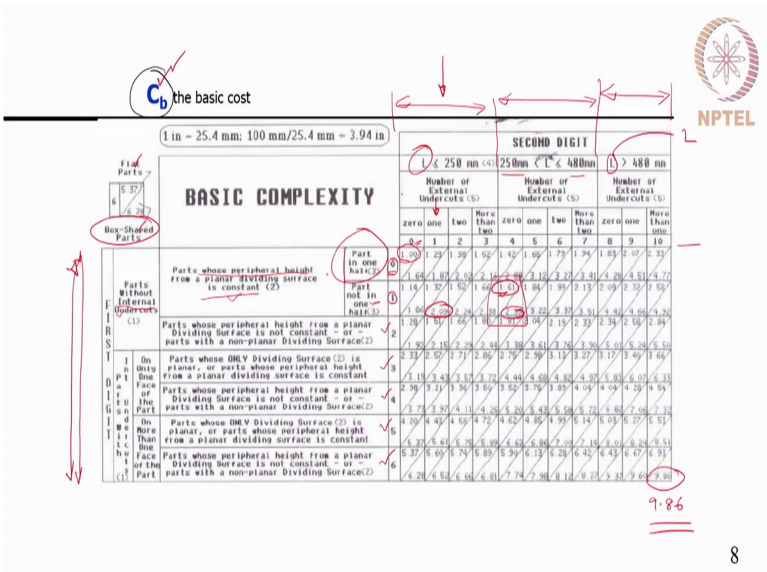
So, you need 1 side action right, you need to have features which can be you know retracted when I need to eject part out of the mold block once the injection process is complete. So, such features are called as undercut. So, something can be internal undercut, something can be external. External are those that are available on the cavity side. If this is a part the cavity side and there is a core side, a because side and cavity side as determined based on the location of your injection port from which the material is being sent. The port through which you are injection is being sent that is called as the core.

And the other side is called the cavity, ok. Typically the part will be in the cavity, once the mold is opened out and it will be pushed out. So, there will be some ejector pins new and you will even see such kind of marks on the part in the plastic component and if we closely watch, you will see some marks which are essentially the marks of that ejector pin ejecting the components, still the material being slightly know you know not molten stage I would say, but you know slightly lower, but it still deforming because of the ejection pins action that you know the it will be that impression will be there on the component.

So, the basic cost takes care of the part size how complicated is your external and internal undercuts and so, forth. And on top of that you have something called as a subsidiary cost is mainly the cavity machining. So, in the mold opening and closing direction how much machining or details of the features you will have to introduced. Let us say I have this component in this orientation and I have to open and close it in this direction, you will see that if these features are there then I will have to have them machined on that core plates or the cavity plates. So, these features have to come in the cavity plate, if the mold is being opened in this direction, right.

So, such features which are mentioned on the core plate and the cavity plate they constitute what is called as the subsidiary cost. And finally, depending upon how tight you want the tolerances what is the surface finish that you want that is a tolerancing cost or what is called as C t. So, multiplying all this is gives you the relative die construction cost and then the material cost will depend upon the size of the mold block and you know the other the basically the dimension of the component, ok. So, this is this is the basic. So, quickly we will see the method itself which we saw in the last class.

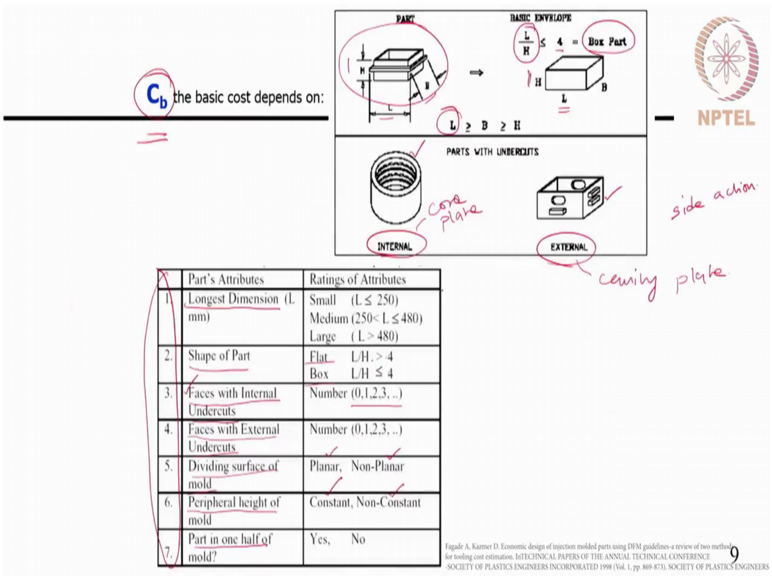
(Refer Slide Time: 06:40)



The basic complexity is determined by this L which is the largest dimension of the component. So, when I go through the example it will become more clear and then whether it is a box type part or it is flat part right depending upon that you choose either the top number in a cell or the bottom number in a cell.

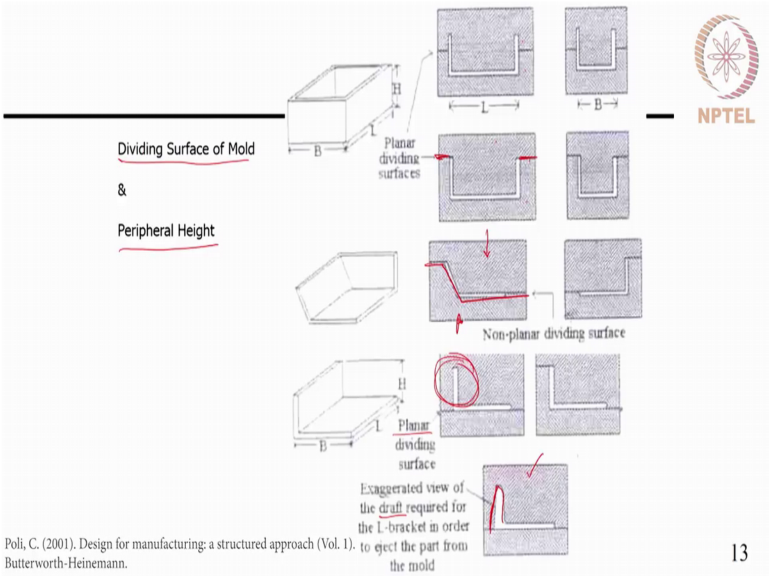
And based on whether you have internal undercuts and the type of you know the parting surface whether you know the parting surface is a planar 1 or a non planar 1, you will choose appropriate rose and, nah determined the basic, cause basic cost is like 1 to 9.86 the meaning is either it can be at the same cost as that of the reference part or it could be as largest approximately 10 times that of the reference part that is 9.86, right.

(Refer Slide Time: 07:44)



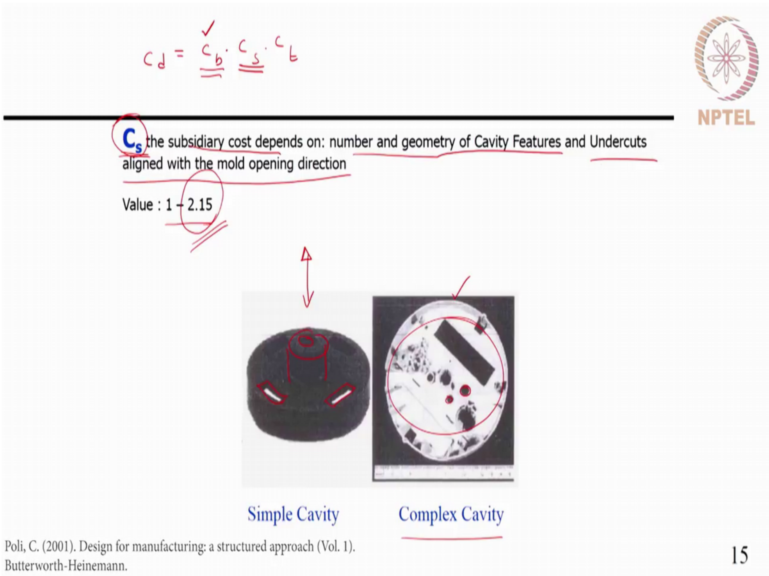
So, the slide helps you understand how to determine those numbers that are useful for determining the basic cost. So, the envelope of the part is used to find Lb and H and L being the largest dimension is taken as the to, to determine the column in which you will see the numbers. And to determine whether it is a box of flat type you will find L by H ratio and internal and external undercuts are pictorially shown here like internal threadings and internal undercuts those that help to me machined in the core. And, or you need side action in the core side and external undercuts you needs side action on the cavity side like whether you know from outside you need to retract or from inside you need to retract; so, that you can take out the component, right.

(Refer Slide Time: 08:55)



So, these things I will skip. So, these are just to illustrate you part in 1 half part in 2 halves where is the parting line whether it is non planar parting line or planar parting line and so forth.

(Refer Slide Time: 09:05)



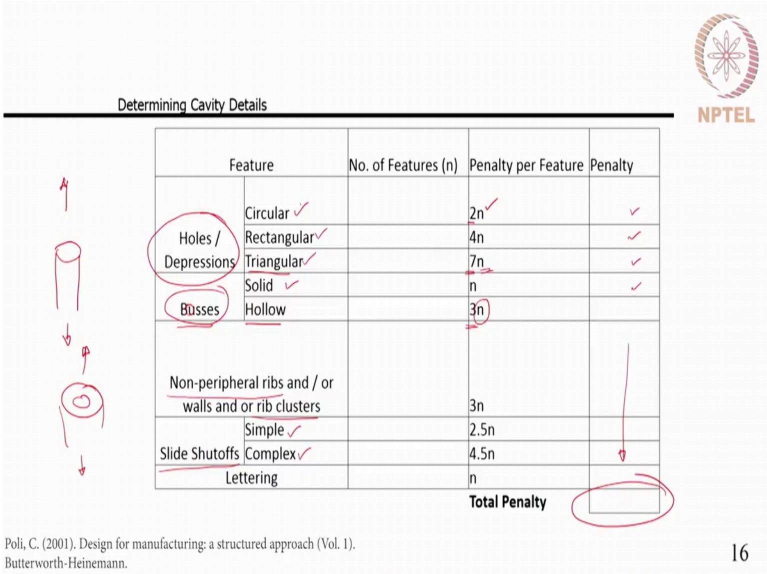
So, second one is the subsidiary cost it can take a number from 1 to 2.15 and as I said the subsidiary cost is mainly due to the machining on the core plate or the cavity plate. Lot of these type of bosses like if you this example component if this is the mold opening and closing direction as I can see that there are no parting lines here, the parting line is actually the top surfaces itself.

(Refer Slide Time: 09:32)

So; that means, this part is actually in 1 half in the C b computation this part is actually in 1 half, it is not on both halves after and this is your the core side and this is your cavity side. And, all these features that you are saying, this ribs, this bosses right they all have to be machined on the core plate, the one that comes from the top right, this is the mold opening and closing direction.

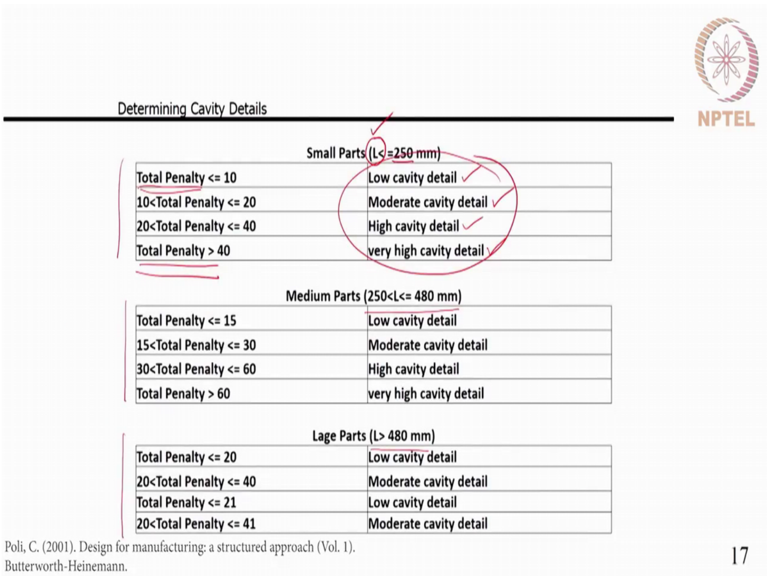
So, all these features that you are seeing that have to be mentioned on the core plate, on the cavity plate its fairly very simple maybe you know some little bit of machining is required, but all of this constitutes the subsidiary cost or what we call as the Cs, right. So, that could increase your mold cost from that is 1 to 2.15 times of the basic cost of making the mold that is: what is the meaning. So, how do we determined? So, we determine what is the complexity of these features that needs to be machine like how many rib C n clusters are there, how many bosses are there, whether they are circular or rectangular depending upon that we introduce what is called as a penalty.

(Refer Slide Time: 10:59)



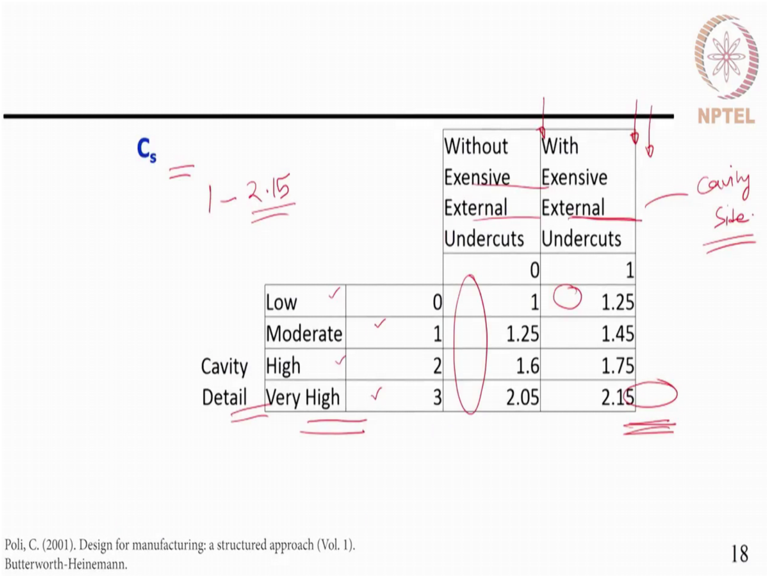
So, this is a table that gets the penalty information required for you to compute how difficult is, is to machine the feature. So, you have lot of types of features like buses, holds, depression you know rib clusters and you know side shutoffs and so forth and depending upon that you get the total penalty.

(Refer Slide Time: 11:21)



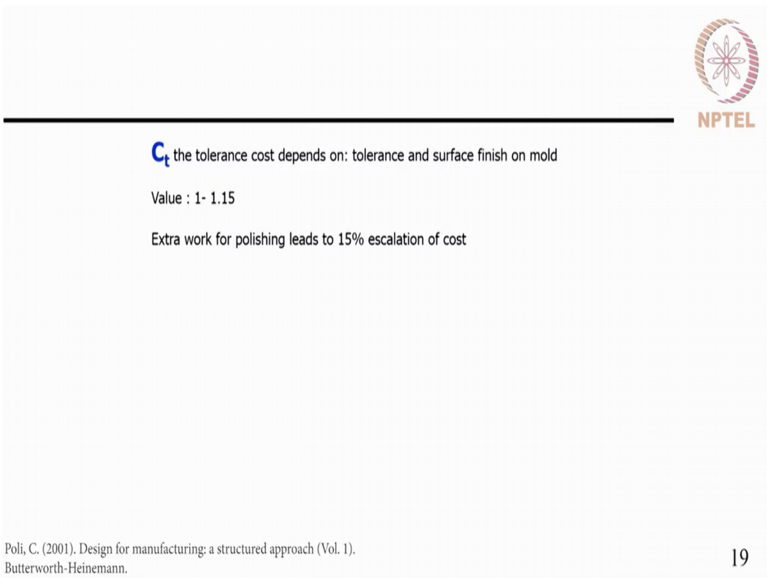
And based on the size of the component the largest dimension L you determine based on the penalty value whether it is low cavity detail moderate are high cavity detail.

(Refer Slide Time: 11:33)



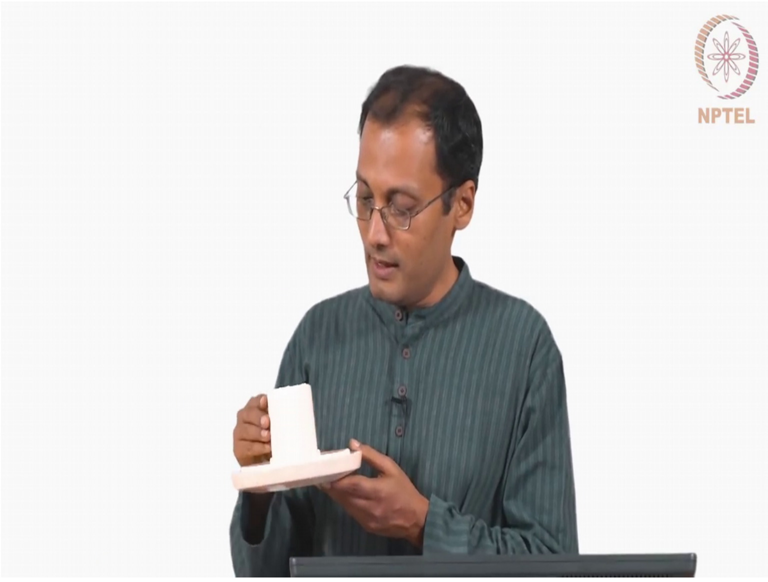
And then you determine along with the external undercuts, how many external undercuts you are having you can determine what should be the n multiplying factor for the subsidiary cost. So, 1 to 2.15. So, this is what we discussed in the last class. So, we will continue with the tolerance costing, ok.

(Refer Slide Time: 11:57)



See the done costing is N the cost that is incurred to get certain type of tolerance and finish that is required on the part.

(Refer Slide Time: 12:11)



The finish on the component is both determined by the material that is whether it is ABS or PP and. So forth like ABS gives you very good finish that is why all these kind of components which are mostly used in appliances and where you know esthetics is important, you try to use like housings of you know very many components you use ABS as a material. So, finish is a property of both material as well as the finish on the mold surface, right. So, first thing you need to have the finish on the mold surface, next the material also should help you achieve that finish on the final component, right.

So, depending upon the level of accuracy that is needed the tolerances that are needed and the finish that is needed, it can increase the mold manufacturing cost because several times manual buffing or you knows operations may have to be done. So, that you can achieve those finishes that will increase the cost; so, it approximately increases the cost by about 15 percent that is what if the shows.

(Refer Slide Time: 13:31)

So, how do we determine this? So, there are some standards. So, the specified by SPI, what is called as society for plastics industry. They have classified the finisher into several categories as you can see in this picture. So, you have these categories. So, A 1 to D 3 so, A 1 means it is grade 3 diamond buff finishing like that you have for each type what is the process that is required to achieve that kind of a finish.

So, if you do a dry blast with glass bead, you will get D 1 finish and so forth. So, the finish quality increases as you go for the processes from D 3 to A 1 and; that means, the cost of the process increases if you specify A 1 and if you specify say some C 3, then the cost of your mold will be less. So, these are some example surface finishes to show you how you know each surface finish will look like.

(Refer Slide Time: 15:00)

So, to understand the surface finish they have given a small um, you know sample machined surface. So, you can see that this is A 3. So, this is A 1, you can see this is the diamond finish, you can see how you know reflective based on the reflection of the surface you can see and this is say by some sand you know blasting with certain you know particle size you will get you know this kind of D 3 finish. So, the quality of the surface finish increases as you can see from D 3 to A 1, ok. So, this is just a sample to see how you know the surface finish differs if you change the type of finishing process.

(Refer Slide Time: 16:00)

C_t

			Commercial Tolerance	Tight Tolerance
			0	1
	SPI 5-6	0	-	-
	SPI 3-4	1	1	1.05
Surface	Texture	2	1.05	1.1
Finish	SPI 1-2	3	1.1	1.15

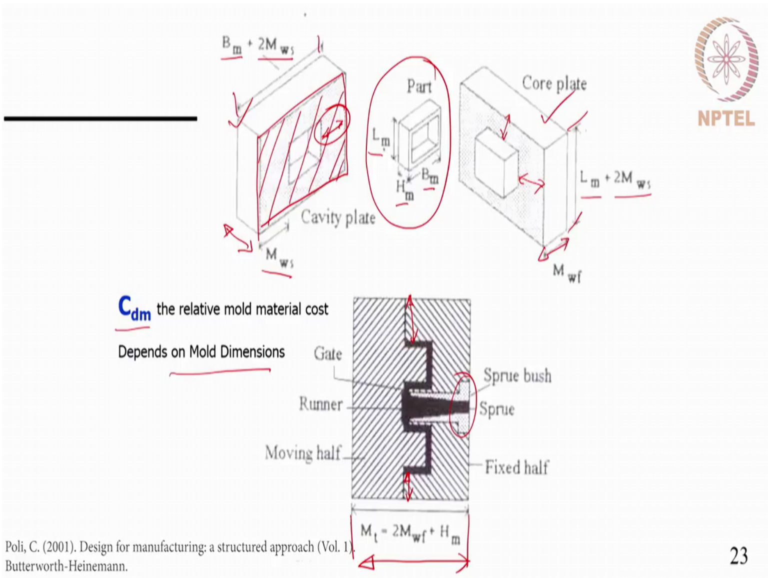
Poli, C. (2001). Design for manufacturing: a structured approach (Vol. 1). Butterworth-Heinemann.

22

So, from 1 to 6 the surface finish is classified SPI 1 to 6, all these processes are map the to the surface finish 1 to 6 and then on the column you can see the tolerances. So, if you need commercial tolerance you choose the first column, if you need tight tolerance you choose the second call column. Again you get a 2 digit number for Ct. So, 0, 0 means you need SPI finish 5 to 6 with commercial tolerance and. So, it will not increase the costing, ok. So, the costing can increase by 15 percent, the largest being you specify a tight tolerance and you specify SPI grade 1 2 on the surface finish of the model. So, once we have these 3 you can multiply to get relative die construction cost. Commercial tolerance is what you generally if you do not specify a tolerance. So, there is a machining tolerance that can be obtained by finish machining process.

So, those are the commercial tolerance in the sense you do not need very tight tolerances, but let you know you have some components where we know there are some precision requirement in terms of assembly or you know other considerations. So, you specify tighter tolerances on that. The general tolerances are of the order of say sub millimeter orders say plus or minus 0.5 something like that, but if you specify tighter tolerances then what is toy know normally achieved then those are classified as tight tolerances. This is just the classification, rough classification to understand: what is the degree of difficulty that you are introducing in the mold manufacturing process.

(Refer Slide Time: 18:24)



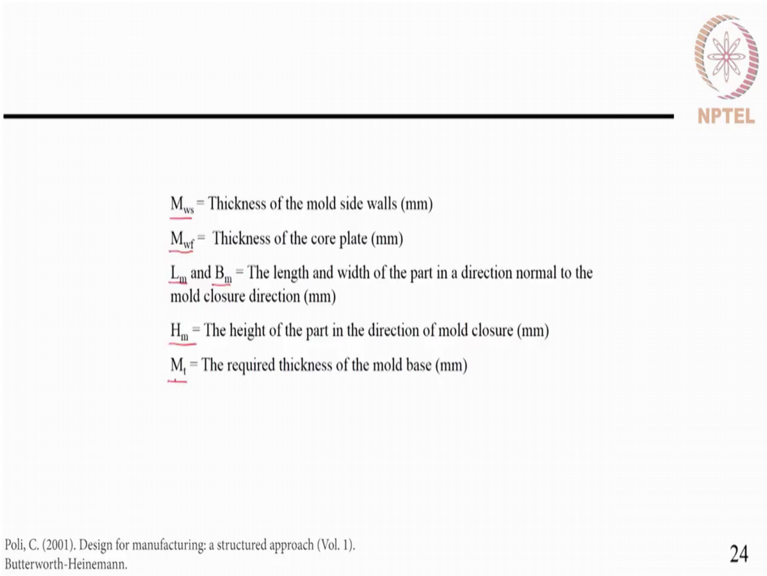
The next cost component is the C dm or the relative cost for the die material, dm means die material, or the mold material, ok. It primarily depends on the mold dimension which in turn depends on the part dimension. So, L H and B are the part dimensions which we have already seen, ok. This is the core plates and this is the cavity plate now it will be very clear this is the, the part is let us say an opened box, ok. So, this is the core plates and this is the cavity plates. Now it will be very clear this is the, the part is let us say an open box. So, this is the core plate side and this is the cavity plate on the cavity plate you have the injection port through which the material is sent into the cavity right.

You need the side wall dimensions so, that sufficient mold holding force can be applied, why that is required? Because you are going to use injection pressure to inject material into the component right into the cavity. So, you do not want the material to come outside, it has to go into the intricates smaller because typically these parts are very thin, plastic components are like you know 1 mm 2 mm thickness. So, through those gaps you have to push through the plastic you know you have to inject that so; that means, you need to develop injection pressure.

So, you need to also put sufficient mold holding force are you know you need to be applying that force. And in order that you know the mold will withstand that you need to give certain dimensions otherwise over time what will happen it may start cracking you know it may start wearing out very quickly because of you know inexpressive contact pressures that are developing in the process, right. It has to withstand several you know million cycles of this you know the operation.

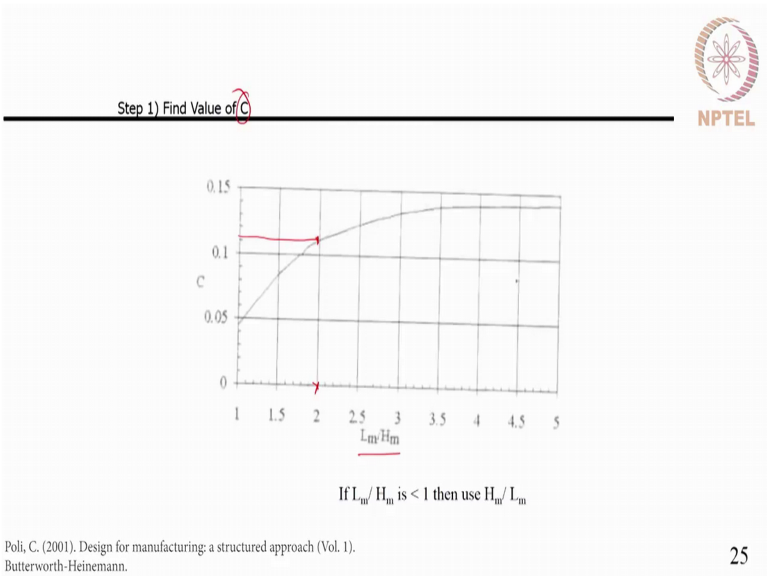
So, considering that fact you need to provide sufficient wall dimension whatever I am marking these from the cavity to the outside. So, there are some guidelines to understand how much I need to give these dimensions, ok. This is the mold side wall thickness. So, 2 times of that needs to be added to be to get this dimension and same way to get this dimension I will have to add to the length of the part 2 times the side wall thickness and I need to also know the thickness of the core plates and the thickness of the cavity plates to determine the mold thickness. This will give me an rough idea about the size of the mould block that I need to start machining and thereby the material cost associated with making the tool, ok.

(Refer Slide Time: 21:45)



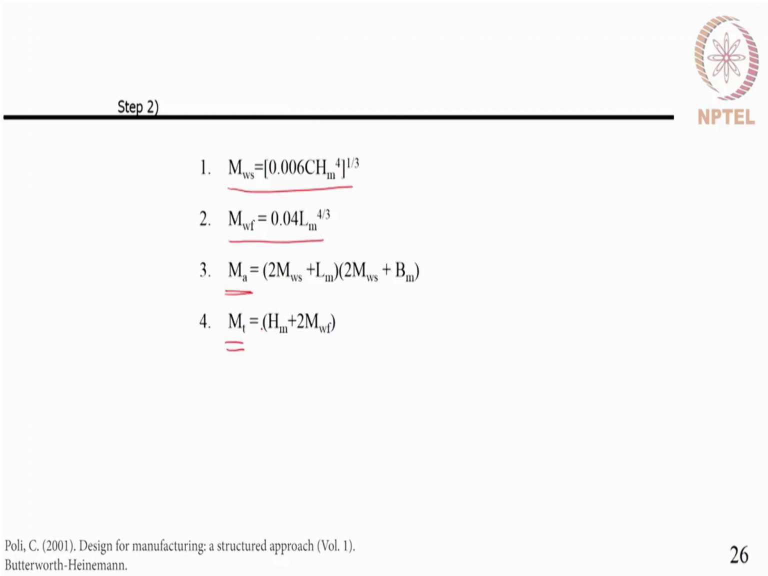
So, M ws in the picture is the thickness of the mold side walls M wf is the thickness of the core plate Lb are the length and width of the part and height of the part and Mt is the required thickness of the mold base, ok.

(Refer Slide Time: 22:03)



So, the guidelines to get that so, first we determine this constant C which is using this graph; so, first we get this ratio L by H which is length of the part by height of the part and based on this ratio we choose this value of C, ok. So, this is aspect ratio, if it is less than 1 then we use H by L ok. So, let us say just as an example if the value is 2 then my corresponding value of C is point 1 1 ok. So, we determined this.

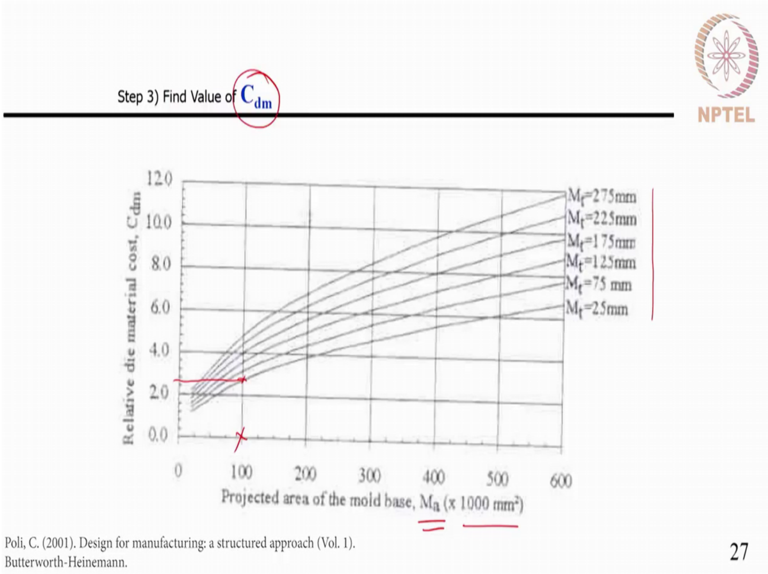
(Refer Slide Time: 22:57)



So, that this can be used in my computation, then these are some guidelines for choosing the side wall dimension M ws is given by 0.006 C H m to the power of 4 cube root. So, how these numbers have been come have been derived is over experience lot of data shows that these kind of dimensions will prevent damage to the mold because of the forces that needs to be applied during the mold you know the cycle.

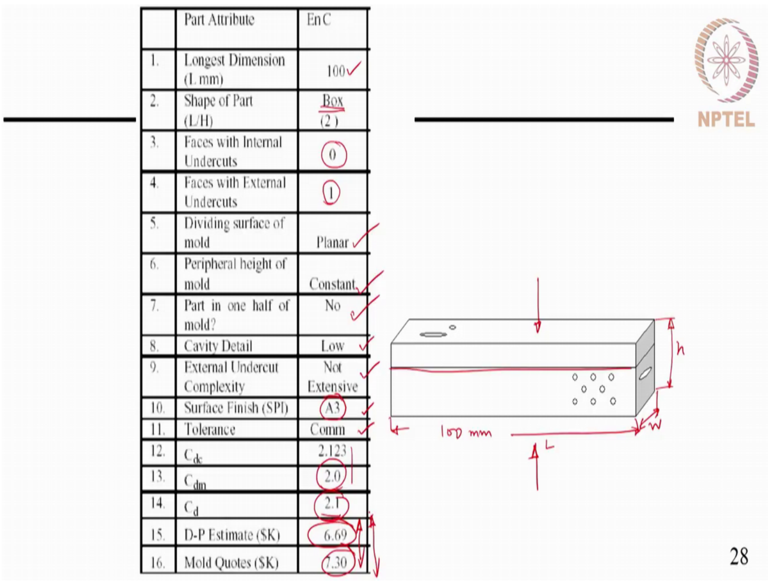
And this is the expression for getting M wf the other dimension that is 0.4 times Lm 4 by 3 and. So, this is the mold area which is this cross section. And then Mt is the thickness of the mold. So, M a into M t will give you the volume of the mold block that is required.

(Refer Slide Time: 24:25)



So, based on this M a the relative die material cost is determined using these graphs. Depending upon the thickness of the mold you can choose 1 of these graphs. If your thickness is lying between any 2 values, you could choose appropriately by interpolating this graph, Say if my projected area is 100 so, this unit is 10000 mm square, ok. So, if it is 100 into 1000 mm square and your mold thickness is 25 mm then your relative material cost is say roughly 2.8 or so. So, these graphs help in determining the relative die material cost.

(Refer Slide Time: 25:30)



	Part Attribute	EnC
1.	Longest Dimension (L mm)	100
2.	Shape of Part (L/H)	Box (2)
3.	Faces with Internal Undercuts	0
4.	Faces with External Undercuts	1
5.	Dividing surface of mold	Planar
6.	Peripheral height of mold	Constant
7.	Part in one half of mold?	No
8.	Cavity Detail	Low
9.	External Undercut Complexity	Not Extensive
10.	Surface Finish (SPI)	A3
11.	Tolerance	Comm
12.	C_{dc}	2.123
13.	C_{dm}	2.0
14.	C_d	2.1
15.	D-P Estimate ($K)	6.69
16.	Mold Quotes ($K)	7.30

So, I will take one small example, it is a very simple example to quickly understand say in this example the longest dimension of this part is 100 mm. So, other 2 dimensions as of now you know is not necessary for me so, relatively. So, this is my H, this is my W, this is my L. So, the longest dimension is 100 mm. And, you can see that this is not a flat part it is a box type of plant part. So, I choose that attribute. So, they about some 9 to 10 attributes that I need to understand or get the values before I could employee this method.

Phases with internal undercuts; so, is there are no internal undercuts and there are some external undercuts. So, like here I have some holes to be made or some you know slots. So, these are the phases with external undercuts. It is a planar dividing surface and peripheral height is also constant part is not in 1 half because if part is in 1 half the dept may be too large. So, sometimes I will place part in both the halves. So, in this example I am placing the part in both the halves, cavity details are very low, I do not see much cavity details that is cavity details are the details in the mold opening and closing direction.

And the undercut complexity is not extensive and surface finish requirement is normal SPI this A 3, there is also this grade, grade A 3, if you remember is not very tight surface finish requirement it is, it is finish ID required, but not you know too high tolerance commercial. So, for these values, if you see the basic cost if I try to determine the basic costs. So, it is a box type of part with L equal 100 mm with internal undercuts, no internal undercuts and external undercuts are only 1,ok.

So, with this details if I see: what is the basic cost I will just go to the basic cost table. So, length 100 so, I will choose this column and number of external undercuts was.

Student: 1 (Refer Time: 29:08).

How many?

Student: 1 (Refer Time: 29:09).

1. So, I will choose this column n and then on the row side and it is a box type of parts. So, I will choose the lower half value, part is there is no internal undercuts, part without internal undercut, parts where peripheral height from the planar dividing surface is

constant and part is not in 1 half. So, I will choose this value agreed 2.09 because this is a box part with maximum dimension 100 mm number of external undercuts 1 and parts without internal undercuts peripheral height is you know constant and part is in 1 half.

So, my basic cost is 2.09, ok. So, this gets multiplied by the subsidiary costs. So, we do not see that there is too much of cavity details. So, the length is less than 100 mm and low cavity detail. So, I will choose low cavity detail without you know extensive external undercuts. So, it will be 1. So, it is does not gets multiplied too much. The C t which is the tolerance cost. So, depending upon the finish and I think I am asking for a commercial tolerance and SPI.

So, it could be say 1.1 because the finish is on the higher side. So, my basic cost gets multiplied by 1.1. So, that gives me this 2.123 which is the die know die constructor curve. Then in a same way my die material cost is obtained as 2 and these 2 determined the total relative die making charge which is 2.1 and based on the cost of making the standard part, the mold for standard point the mold for standard point that is multiplied by this factor 2.1 to get this value ok. So, this is an example just illustrated in some work, we will see one actual case now or with 1 example we will take and illustrate this. This just to compare how the estimate by this method and the actual mold quote given by some mold maker, right.

That will is; obviously, be some differences, but one can see that at the design stage, we are able to reasonably estimate the cost of the tooling and you can make the changes if let us say this casting that you are forcing is too much. Then you can do some modification to your design, you can see whether some features that you have put in are absolutely necessary are not whether the tolerance that you are asking or the finished that is that is being you know asked is required or not you can make some decision on those and you can justify the cost or if you say no I need it then you can justify the cost. So, we will see one example in the by taking one plastic component, we will try to understand this method.