**Production Technology: Theory and Practice**
**Prof. Sounak Kumar Choudhury**
**Department of Mechanical Engineering**
**Indian Institute of Technology Kanpur**

**Lecture – 13**
# Tool Wear and Tool Life

Hello and welcome to the discussion series of production technology theory and practice course. Let me remind you that in our last session, we started discussing the tool life and the effect of various variables on the tool life. I said that there are 5 variables which affect the tool life. First is the cutting parameter that includes cutting speed, feed and the depth of cut.

Then there are tool angles, particularly the rake angle, flank angle and the cutting-edge angle. Then the tool materials, workpiece materials and the cutting fluid. These are the 5 basic variables used in the case of the machining. All of them affect the tool life in a different way. We discussed that the cutting speed has the greatest effect that has been already found out from the experiments.

First initiative was taken by FW Taylor who has invented the Taylor's tool life equation which famously gives us the V that is the cutting speed, tool life to the power constant n is equal to some constant. I said that in this Taylor's tool life equation, Taylor could consider only the cutting speed because in that equation, $VT^n = C$ you can see that only the cutting velocity has been considered.

However, further research shows that not only the cutting velocity but the feed and the depth of cut will also have the effect on the tool life and it has been seen that the cutting speed has the greatest effect on the tool life followed by the feed and the depth of cut. Next we started discussing the effect of the cutting tool geometry and in that case, the rake angle has great effect on the tool life.

As we have seen earlier that as the rake angle is increased, the cutting force decreases. If the cutting force decreases then the temperature will be decreased and as the temperature is decreasing the thermal deformation of the cutting tool or the thermal softening of the cutting tool will be less. So, the tool wear will be less and the tool life will increase.

This is an overall conclusion that overall if we increase the rake angle then the tool life will be increased. But here what we have shown is that if the rake angle is increased infinitely, then after a certain value, you get the maximum tool life after that the tool life decreases with the further increase in the rake angle.

This is because as we further increase the rake angle, the cutting angle of the tool decreases that means the tool becomes weaker, and less material is available for the heat conduction. So, the tool becomes mechanically weak that means it can break and can be thermally affected because the heat is accumulated.

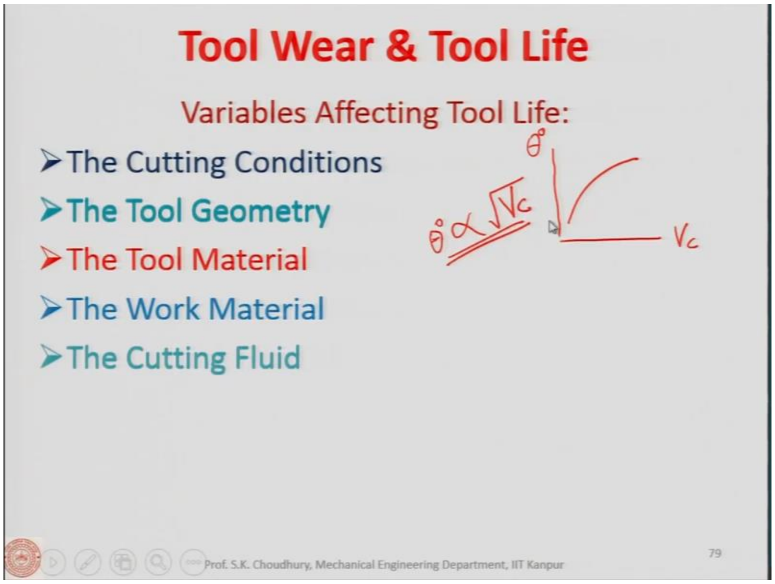
And because of the cutting process the heat is increasing that cannot be conducted from the tool. Therefore, the tool will have more heat and the tool life will be less. After further increase in the rake angle, the tool life will be decreased. I also said that if the workpiece material has more strength this optimum value of the rake angle should be low.

Because the tool has to be fatter and the strength of the tool has to be more for removing the material from the workpiece which has more strength. Then, we said about the flank angle the same thing that as the flank angle increases, the rubbing between the flank face and the already machined surface will be less. Therefore, the temperature occurring will be less in the cutting zone and the thermal deformation of the tool will be less.

Again, if we further increase the flank angle after obtaining the maximum tool life then the tool cutting angle will again decrease. Tool will be weaker and the less material will be available for the heat conduction then again, the tool life decreases. Therefore, we have a maximum tool life for a certain value of the flank angle as well as the rake angle which we are calling as the optimum rake angle or optimum flank angle.

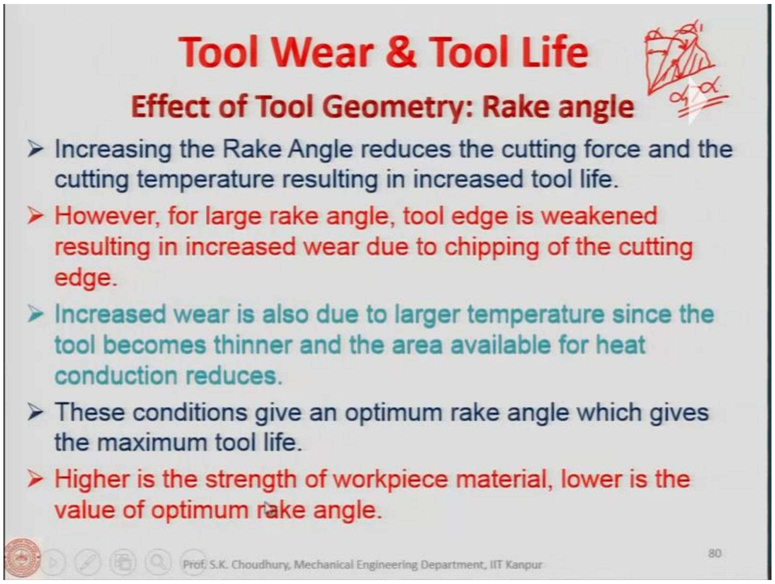
Now, in case of flank angle, if the feed is more, then, the optimum angle that we are obtaining for the maximum tool life that has to be less because for more feed we need to have more strength of the tool and we have to have the more material for the heat conduction from the tool.

**(Refer Slide Time: 06:45)**



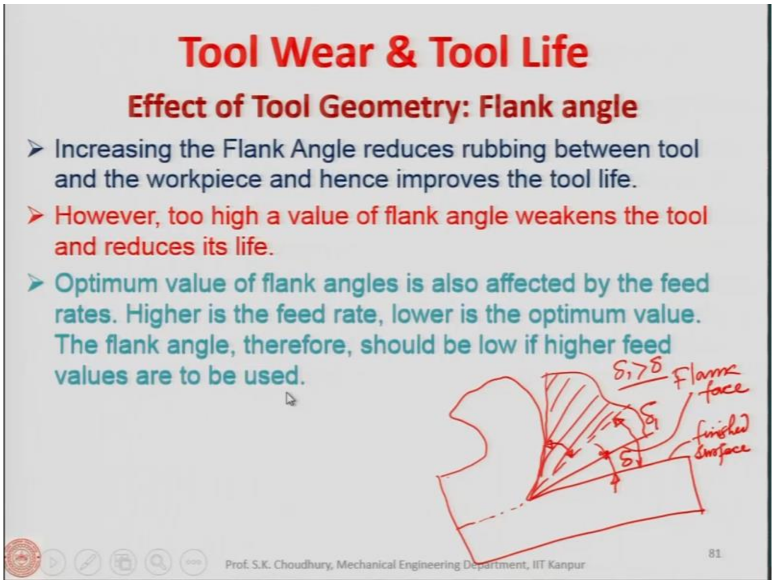

These are the 5 variables which affect the tool life. The temperature occurring during the machining process is proportional to $\sqrt{V_c}$ .

**(Refer Slide Time: 06:57)**



Then we said that increasing the rake angle reduces the cutting force. And then in the final conclusion, we said that higher the strength of workpiece material, lower is the value of the optimum rake angle.

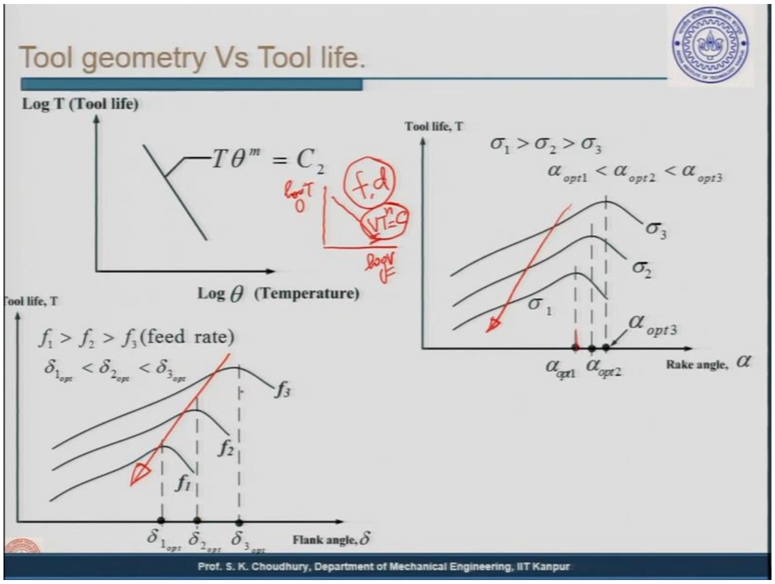
**(Refer Slide Time: 07:13)**




About the flank angle, I have shown it to you that as the flank angle is increasing then cutting angle of the tool is decreasing and the tool becomes weaker and the material available for the heat conduction will be less. We also said that the optimum value that we are getting for flank angle also affected by the feed rates. That means higher is the feed rate, lower is the optimum value the flank angle. Therefore, it should be low if higher feed values had to be used.

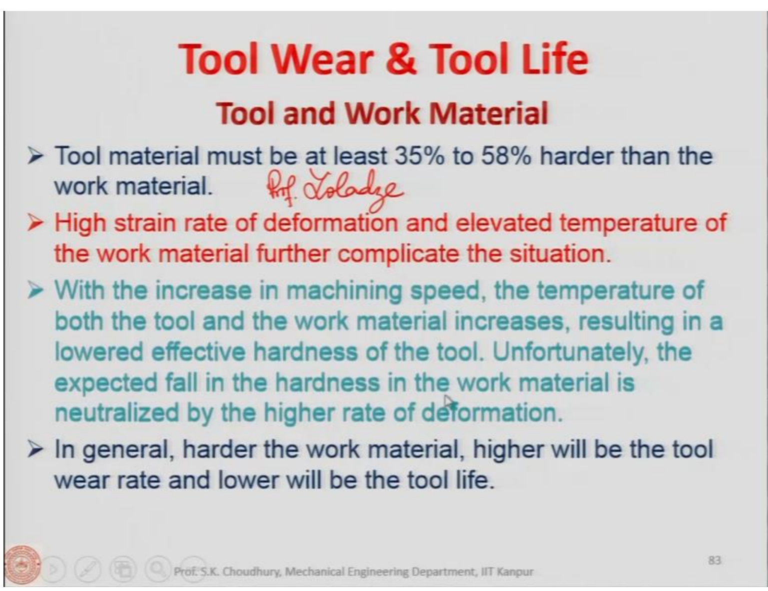
In case of rake angle, we said that is the hardness or the strength of the workpiece material. And in this case for the flank angle, we are saying that if the feed is more then, our optimum value of the flank angle should be lower. These are shown in the curves below.

**(Refer Slide Time: 08:14)**




See here that the $\sigma_1$ is more than the $\sigma_2$ than the $\sigma_3$. It is in the increasing order. $\sigma_1$ is maximum in that case the optimum value that we will get for the rake angle will be less than the other values of the strength of workpiece material. Similarly, here also feed is in the increasing order that means $f_1$ is more than $f_2$ is more than $f_3$. In this case the optimum value that we are getting for the maximum tool life has to be less than in case of the other values for the other feed rates which are less than the one that we have discussed here.

**(Refer Slide Time: 09:08)**




Now, next is tool and work material. The overall research says that the tool material must be at least 35% to 58% harder than the work material. This conclusion has been made by the research done by Professor Loladze. Prof. Loladze is a Russian scientist who has actually shown that the tool material must be at least 35% to 58% harder than the work material.

Now the high strain rate of deformation and elevated temperature of the work material further complicate the situation. There are two things that happen during the machining process; the strain rate of deformation is very high. At high strain rate of deformation, you can have the plastic deformation and segregation of the workpiece material from the blank, as well as there is a very high elevated temperature.

These two phenomena, that is high strain rate of deformation and the elevated temperature, affect in the following way:
With the increase in machining speed the temperature of both the tool and the work material increases that have been discussed many times earlier that as we are increasing the speed the temperature increases and that I have already shown you through the curve also that the temperature is proportional to root over *V c*.

*Vc* is the cutting velocity. These results in lowered effective hardness of the tool because the temperature increases means the tool is thermally becoming softer. Thermal deformation of the tool happens and the tool material becomes soft. Unfortunately, the expected fall in the hardness of the work material is neutralized by the higher rate of deformation. Let me explain this to you what does it mean that tool and the workpiece are in contact with each other.

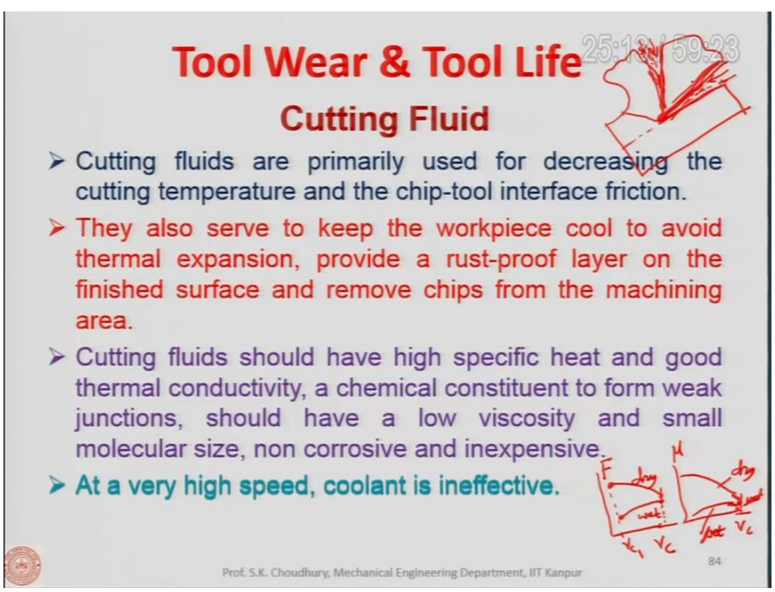
Now we are saying that in this process when the material is removed because of the interaction between the tool and the workpiece, 2 things are happening. One is the very high strain rate of deformation given by the force which we are imparting on the workpiece through the tool. Second thing is that when particularly the speed is very high the high temperature is occurring. Now because of the high temperature which is occurring the tool is getting thermally soften.

Actually, the same temperature should have gone to the workpiece as well and the workpiece as well should have been thermally deformed but what is said is that the tool is becoming thermally softened. But workpiece does not because the high strain rate of deformation is imparted on the workpiece. And as you understand that with the high strain rate of deformation the resistance will also be higher.

Then the strain hardening takes place in the workpiece material and as if it has been given an added hardness because of the high strain rate of deformation. Therefore, the elevated temperature which was supposed to soften the workpiece material does not do that because that is compensated by the strain hardening process by the extra hardness which is imparted on the workpiece because of the high strain rate of deformation. High strain rate of deformation is imparted to the workpiece not to the tool.

The tool is imparting the high strain rate of deformation to the workpiece. So, the additional hardness is added to the workpiece material. This is what happens and because of that the hardness of the tool has to be more than the workpiece material. Once again because of the high strain rate of deformation and because of the elevated temperature; in general, harder the work material higher will be the tool wear rate and lower will be the tool life. This is obvious as the temperature is increasing then the tool is getting thermally softened and the tool wear is more and the tool life decreases. So, it can be concluded that as the temperature becomes more, the tool life becomes less.

**(Refer Slide Time: 14:03)**




Cutting fluids are primarily used for decreasing the cutting temperature. I told you that and the chip tool interface friction. So, there are two actions of the cutting fluid, one that is when the cutting fluid is applied basically the temperature which is occurring in the machining zone, part of it can be taken away. Why I said part of it because part of it goes by the chip because the chip is segregated and chip goes away. Chip takes away most of the heat or a large portion of the heat, but still the lot of heat remains in the machining zone and that should be removed because otherwise the tool life will be less that we have seen and that is done by the cutting fluid. Another action of the cutting fluid is that cutting fluid reduces the friction between the tool and the workpiece particularly between the chip and the tool on the rake face.

Because as it is moving along the rake face of the tool. If the friction between the chip and rake face of the tool is more then, the more temperature will be produced. So, if we can reduce the friction, in that case the temperature occurring will also be less, they also serve to keep the workpiece cool to avoid the thermal expansion and provide a rust proof layer. Rust proof layer is provided because in the cutting fluids there are different kinds of additives.

Those additives actually put a layer on the freshly removed workpiece surface. When the workpiece surface is freshly removed it is prone to the oxygen of the atmosphere and it can get oxidized and the rusting can happen. The rusting has bad effect on the surface of the workpiece, it can spoil the workpiece, it can spoil the surface finish and so on.
It actually gives a coating so that the rusting does not happen on the nascent surface of the workpiece immediately after the fresh layer is removed from the workpiece and remove the chips from the machining area. This is another problem which are faced often and I have already discussed this that the removal of chip can be a problem, otherwise if particularly when the ductile material is machined then the continuous chip will be formed.

If the chip is not removed from the machining zone in that case the chip can get entangled with the workpiece. Chip can get entangled with the tool, it will rub the already machined surface, it will rub the tool faces, rake face and the flank face and can spoil the geometry. There are chip breakers which have to be used particularly when the ductile material is machined because those continuous chips have to be removed.

But apart from that, the chips can also be removed, particularly when they are small chips, from the machining zone by high flow of the cutting fluid, this is also another action of the cutting fluid. Cutting fluids should have high specific heat so that a large amount of heat can be conducted from the machining zone and good thermal conductivity, so that the heat is conducted away.

A chemical constituent to form weak junctions should have a low viscosity and small molecular size, non-corrosive and inexpensive. Let me explain it to you that why they should have a chemical constituent to form the weaker junction. There are two reasons. Weaker junction means that the chip which is being formed with the freshly removed surface it should not have the tendency to get welded to the rake face of the tool or the to the tool tip.

It causes the buildup edge formation. So, if there is a coating made by the cutting fluid in that case the buildup edge formation can be prevented to a large extent, even if the speed range is that in which the buildup edge normally is formed. I will remind you that in case of mild steel and high-speed steel as the tool and the mild steel as the workpiece in that combination of materials that cutting speed is about at 80 to 110 *m/min*. So, within that range of the cutting speed, the temperature is such that the built-up edge formed. Now when the built-up edge forms, then they are getting welded so they will get welded only when the two surfaces are nascent surfaces because it is a very thin layer of chip. A very thin layer of the material is removed from the chip surface as well as from the rake face of the tool.

So, they get welded because they are nascent and they will have the affinity to each other. If both the surfaces are coated, by the cutting fluid and the cutting fluid for that reason should have some chemical constituent so that the coating can be made and then another purpose is to resist the corrosion, so it will not be corroded. It should have the low viscosity and small molecular size so that the cutting fluid can penetrate easily to the machining zone because cutting process happens at high speed. So, there should be enough time for the cutting fluid to react to the surfaces which are being machined or the freshly removed surface of the workpiece. Therefore, the cutting fluid should have the low viscosity so that it can flow easily and flow up to the tool tip because at the tool tip the temperature is maximum. The cutting fluids should reach there so that it could be cool down and small molecular size, for the same reason. It should be non-corrosive and

inexpensive i.e. the cutting fluid itself should not corrode the workpiece. And it is easily available because the cutting fluid is flown to the working zone in a very large quantity. At a very high speed the coolant is ineffective because at a high speed there is no time for the coolant to reach to the machining zone.
If we draw the curve between let us say $F$ and $\mu$, so here is the cutting velocity let us say. What I am saying is that at very high speed the $\mu$ cannot be reduced to that extent what is desired from the coolant. Similarly, the cutting force cannot be reduced to that extent.

If it is for example $\mu$, coefficient of friction and let us say this is the force. So, if it is dry cutting and this is the cutting velocity if it is wet cutting and this is the dry this is also let us say dry and this is wet, here are the curves. So, through this I wanted to show you that when the cutting fluid is used overall, the force is reduced.

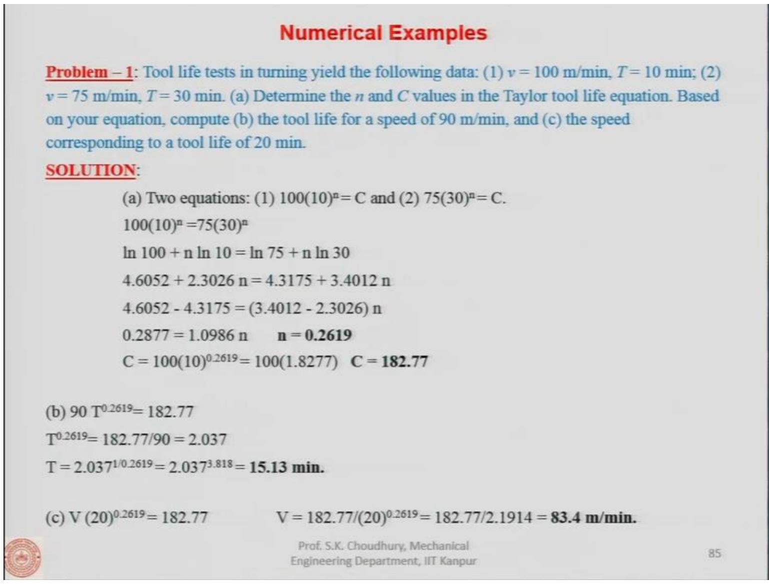
You can see that from here that in the dry cutting the force is here for a certain value of the cutting speed and as it is increasing it is going like this and at a higher speed if you see this the dry cutting value and the wet cutting value are becoming very close. The $\mu$ cannot be reduced to that extent at a higher speed. This is the experimental result.

That at higher cutting speeds, the wet cutting and the dry cutting are basically almost close to each other and there is no effect almost it is only because that there is no time for this to get it. Another reason why this happens is like suppose if we said that this is the machining zone, this is the tool, this is the workpiece now here we are supplying the cutting fluid.

So, what happens is that at a high speed the coolant will come at a higher speed and here there is a wedge, this wedge will not allow the coolant to come to the tool point at this edge or the cutting tool point where the temperature is maximum. This is because there is a hydrodynamic force and that hydrodynamic force happens when it is in wedge shaped. This is the wedge and here also there is a wedge.

The fluid is being passed at a high speed, so these are the two reasons that this is because of the hydrodynamic force which is created in the wedge between the flank face and the already machined surface and between the rake face and the chip, these are the two places. Also because of the very high speed, the time is insufficient for the coolant to reach the tool point and cool down the workpiece. Now let us discuss some of the applications through some problems.

**(Refer Slide Time: 25:36)**




Numerical problems: How we can implement one of the tool life equation, the tool life theory that we have learned. Let us say tool life test in turning. In the following data, cutting speed is 100 *m/min* when the tool life has shown to be 10 *min* and as the cutting speed decreases to 75 *m/min* that tool life increases to 30 *min* from this data we have to determine the constants *n* and *C* values of the Taylor tool life equation.

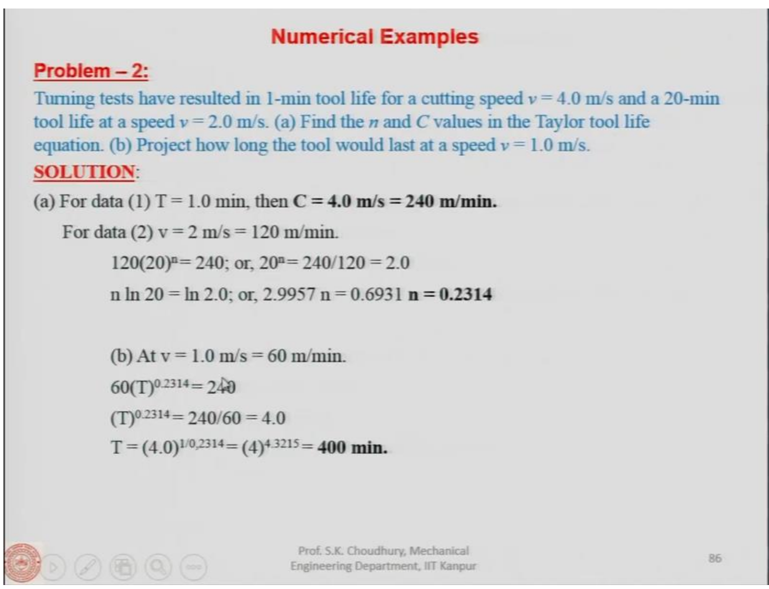
Based on the equation, compute the tool life for a speed of 90 meter per minute, so if the tool life is in between what should be the speed, what should be the tool life and the speed corresponding to a tool life have 20 *min*. So, 20 *min* will be in between the 10 *min* and 30 *min* and find out what should be the speed here. Now how to solve it. we have $VT^n = C$, *n* and *C* unknown. For the first machining and for the second machining process we have the $VT^n = C$. So, we have two equations and two unknown *n* and *C*. So, *n* and *C* could be found if we simultaneously solve these two equations, here is these two equations how they are simultaneously being solved this is shown here. So, *n* we can find out as 0.2619 put in one of these equations the value of the *n* you can find out what is the *C*.

So, the constants have been found out in that case the second problem that is the tool life or a speed of 90 m/min that can be easily found out because in that case the tool life is not known, V is known, $VT^n = C$. So, it is one equation and one unknown which is T it can be solved and found out the T is equal to tool life is 15.13 you can solve that.

Now the third problem was that the speed corresponding the tool life of 20 minute here The T is is given but the V is not given again this is the same process that $VT^n = C$ except V all other parameters are known. So, you can solve it and find out the value of the speed when the tool life will be 20 min. So, you can see that the implementation is easy when you know the concept when you know that what is the tool life?

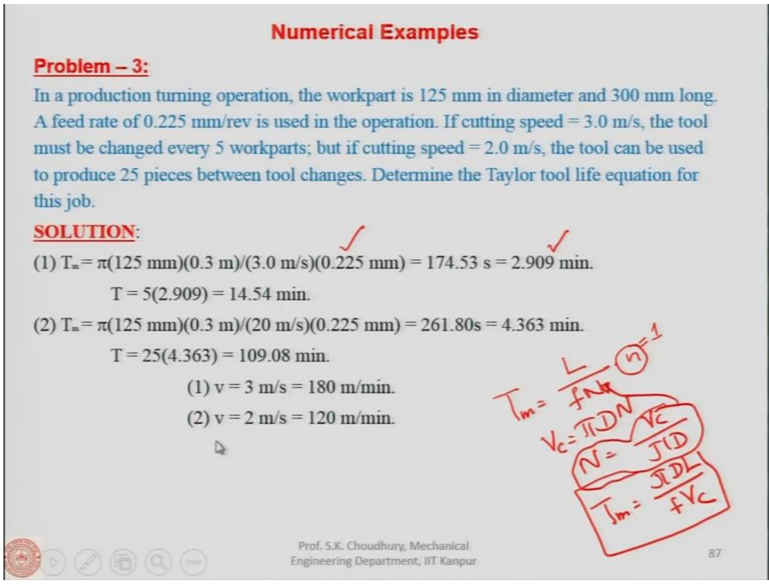
How it actually varies with respect to cutting velocity then you can find out and you can implement them to solve any problem. Let us say another problem will be the turning tests have resulted in 1 min tool life for cutting speed of this and 20 min at the speed of this.

**(Refer Slide Time: 28:38)**




Find the $n$ and $C$ values. So, probably these two are similar to that for data 1. $T$ is given and the $C$ is given. So, $C$ is to be taken in meters per minute it is given in meters per second; so you have to convert it to *m/min* and for data 2 also $V$ is given. So, $V$ has to be taken in *m/min*. So, you can find out what is the value of $n$ from here using again the Taylor's to life equation. And from here the constant $n$ can be found out and here in the *b* part, we have to determine how long the tool would last at speed of *1 m/sec*, *T* has to be found out one equation *T* is unknown you can find out the *T*.

**(Refer Slide Time: 29:29)**




Similarly, in turning operation the work part is *125 mm* in diameter and *300 mm* long. Here physical parameters are given that is what is the diameter of the workpiece, what is the length of the workpiece then the feed rate is given as *0.225 mm/rev* for the operation if the cutting speed is *3 m/sec* the tool must be changed every 5 work parts.

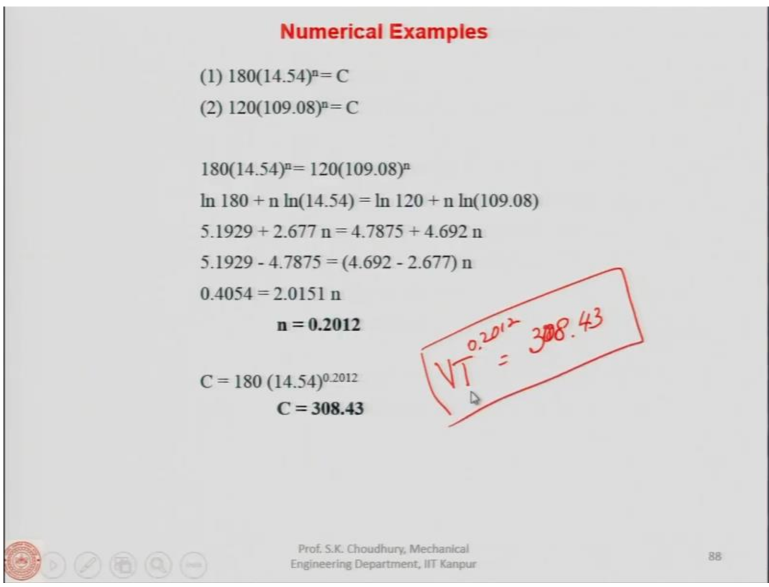
But if cutting speed is *2 m/sec*, the tool can be used to produce 25 pieces between the tool changes. You have understood that how the tool life can be measured or tool life can be calculated from this data. This is different from the other problems like here what is given is that cutting speed is given and the tool must be changed for 5 work parts. The life is not given but 5 work parts can be machined and so on, determine the Taylor's tool life equation for this job that is *n* and *C* have to be found out. Now, the cutting velocity, *V* is given by $\pi DN$ . We can find out easily what is the $T_m$? That is the machining time. The tool life that is 5 work parts you have to change the tool after each 5 work parts, so 5 into this machining time will be this 14.54 min which is the tool life.

You find out the machining time which I have already discussed with you and that is *L/fN*. You can now have number of passes, let us say 1. We are taking the number of passes 1. So, this is L, length of the workpiece and here we have taken $V_c = \pi DN$ and $N = \frac{V_c}{\pi D}$.

Therefore, the $T_m$ can be taken as $T_m = \frac{\pi DL}{fV_c}$. we have put this value of the *N* here. This is the equation from where you can find out the machining time. 5 parts can be machined. After which you have to change the tool. So, that gives the tool life as 14.54 minute.

That solves the first part, then you have the second condition, from the second condition the similar way you find out the machining time and then tool must be changed after 25 pieces, so 25 into this time is the tool life and then you have this equation. So, in one case it is 3 *m/sec* which is this and in second case it is this.

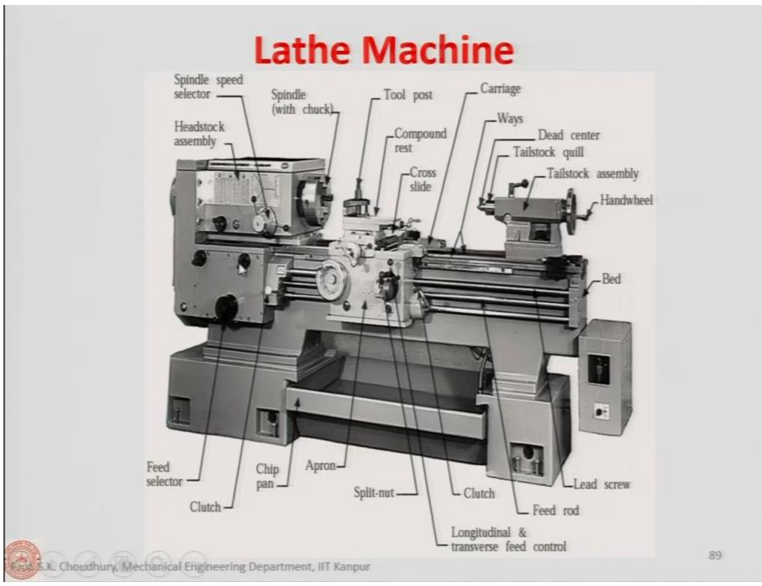
**(Refer Slide Time: 32:59)**




So, you can find out what is the $VT^n = C$ in first and second cases. In both cases, *V* and *T* are different we have found out from these conditions that if the cutting speed is 3 *m/sec* then tool must be changed after 5 work parts and the second equation is if cutting speed is 2 *m/sec* then the tool must be producing 25 pieces, from here we can get these 2 equations.

2 equations 2 unknown n and $C$ you can solve like we have seen it earlier and we can find out the value of the $n$ and the value of the $C$. The tool life equation has been asked so $VT^{0.2012} = 318.43$. This is the tool life equation and the answer. You can see that these are also not so difficult and first of all you have to understand what is asked here. And then you have to scrupulously apply the theory that you have learned that is the tool life equation and from the tool life equation you can solve the problem.

**(Refer Slide Time: 34:20)**



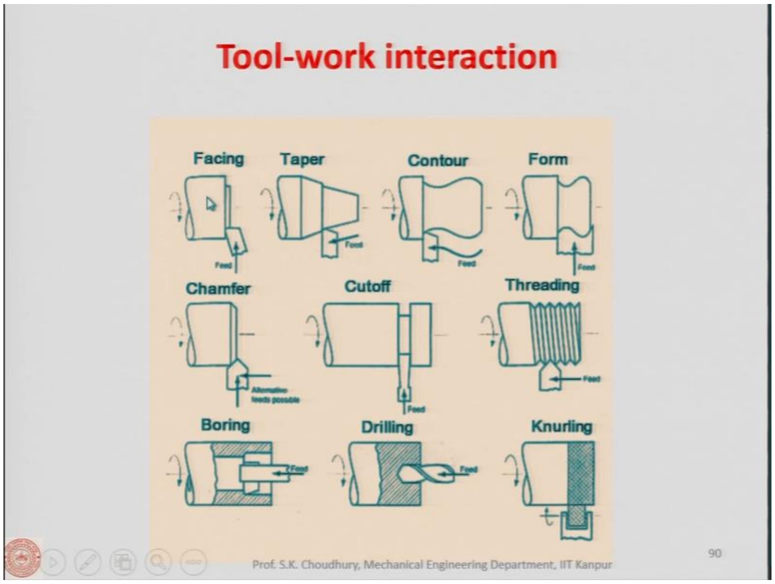

In the above slide, here is a complete pictorial view of lathe machine. In the lathe machine that we will see in the lab also in details how these machines work, here is the headstock. In the headstock we have basically the gear train which provides the rotation to the spindle, on the spindle there is a chuck to mount the workpiece. So, the chuck can be 3-jaw chuck or it can be 4-jaw chuck that I will explain in details later in the lab.

Apart from that, there is a tailstock and this tailstock assembly has the tailstock quill. And in the quill, we have the dead center, so the dead center means the center does not rotate and there is a live center. We rotate the handwheel by which the quill can be forwarded. So, apart from the headstock and the tailstock we have the tool post, in the tool post we also have the apron. Apron moves on the lead screw; this is the lead screw. And through this the feed rod also goes in which the Apron is supported when the feed is given. Now here if you see that these are the selectors for the speed. By selecting this or by rotating that you can have a particular combination of the gears in mesh so that you can have a different kind of transmission ratio and the different RPM given to the spindle of the machine. This is the feed box.

Inside the feed box we have the feed selector, that helps to select the feed that we are willing to supply and this is the clutch. When the clutch is on then the feed is on. This is another clutch for the feed. We have seen now this is the longitudinal and transverse feed control. These are the handles either you rotate this so that you can have the apron moving along with the tool post towards the headstock or towards the tailstock.

By rotating this you can provide a cross feed. Cross feed is for the facing operation. This is the cross slide for that and on the lathe bed we have the guide ways on which the apron, the tool post moves on the lead screw. This is a lathe bed. So, these are the details of the lathe machine.

**(Refer Slide Time: 37:47)**




Here are the different kinds of surfaces that can be made using the lathe machine. I told you about the cross feed, so this is the cross feed that is given here along this. And particularly here what is given here is a tool. Look at the tool, how the tool can be moved along the direction which is perpendicular to the axis of the workpiece. If the feed is given not parallel to the workpiece axis, then a taper surface is formed. The feed is given at an angle inclined to the axis of the workpiece. If it is a cylindrical job then the 3-jaw chuck is used so that 3 jaw chuck can self-center this kind of a cylindrical workpiece. Here is a contour turning process in which accordingly the feed given is neither straight nor inclined; it will be along this curve which will be pre-determined.

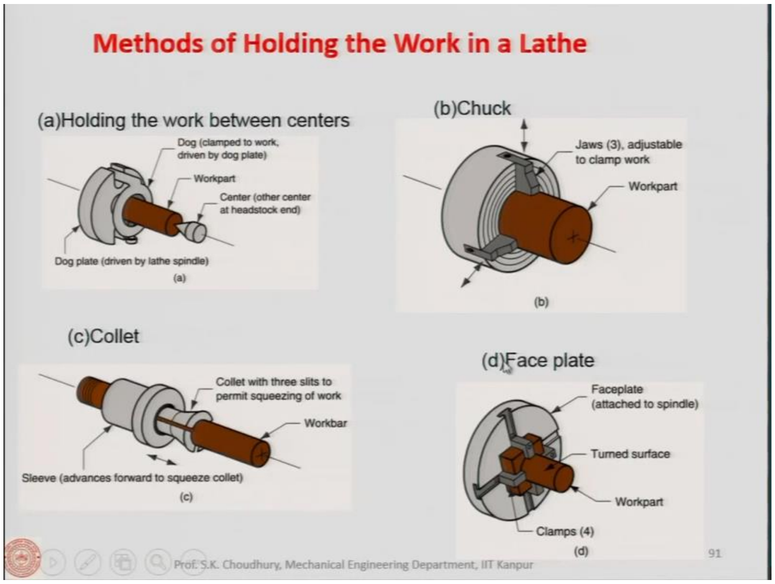
Feed is given like this in case of CNC machine, there we will show you how this feed is given for the contour machining, for the contour turning and how this can be programmed so that automatically the tool moves along the contour of the workpiece and the entire surface is made. This is the form tool. This tool has a form or a particular shape which is conformed to the workpiece to make the form and the same kind of curvature. This is chamfering. The chamfer is made for the guidance so that this could go well in the hole. This is the cutoff that means again the cross feed is given similar to the facing so that the two parts are separated. This feed is given towards the axis perpendicular to the axis of the workpiece.

This is threading, threading we will show you in details in the lab, how the feed can be given for the threading and how this threading can be finished particularly in different passes and in different passes when the tool comes back to the initial position it should go along the same path otherwise the thread will not be properly machined. This is boring which is done for getting the internal hole. to enlarge the diameter of the internal hole. So, this diameter of the internal hole was existing let us say it was obtained by drilling. And this can be enlarged to this diameter by a tool which is called the boring tool and this also can be done by the turning lathe. So, all these operations are performed on the turning lathe that we have already seen here.

Drilling also can be done. In case of boring and drilling in the turning, boring tool and the drills are fixed at the tailstock and they do not rotate. This is the difference between the drilling machine and the drilling of a hole in the turning. In that case the job will be rotated job will be fixed here and in the tailstock the tool is the drill or the boring tool, the boring tool is fixed in the tool post.

The drill is fixed in the tailstock. A hole can be drilled and that hole can be enlarged by the boring tool. Knurling can also be done; this is the knurled surface. Knurling can be made by a special tool which is called the knurling tool. This also will show you in the lab in more details. So, you can see that with the help of different kind of relative movements between the tool and the workpiece different kinds of surfaces are formed. Each of the relative movements between the tool and the workpiece is different.

**(Refer Slide Time: 42:39)**



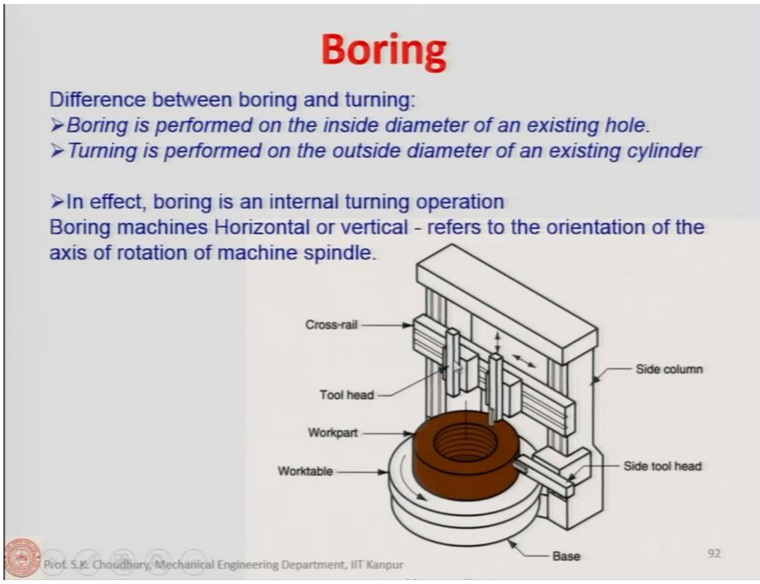

Here you can see the methods of holding the workpiece in a Lathe. Here is the holding between the centers. This is the center in the tailstock and this is the other side which is held on another center on the headstock. The rotation is given by a dog carrier. Here is the dog plate and this is called the dog and the dog carrier, this is the carrier dog plate which is fixed here with the workpiece.

The rotation is given from the chuck as the chuck rotates, this is inserted in that groove therefore this dog plate will rotate along with the workpiece because it is fixed here and this is supported by the center and the other center at the headstock side. This is the chuck. If it is 3 jaw chuck, this is self-centered. There could be a 4-jaw chuck also and the 4-jaw chuck is not self-centered where the cylindrical jobs are not clamped.

But for the 3-jaw chuck since it is a self-centered so the center of the workpiece has to be exactly coaxial with the center of the spindle. For that purpose, the 3-jaw chuck is used which is adjustable to clamp the work. This is the collet chuck. The collet chuck with the 3 slits permits the squeezing of the work. I will show it in the lab as well that this workpiece can be inserted here and that whole collet goes into the spindle or the collet holder and it will press this so that workpieces clamp.

So, this is very similar to the clutch pencil, you must be having that mechanical pencil. This is a clutch pencil and you move that head and the lead of the pencil will come out from this collet. That is also collet and the principle is same as that is a collet chuck. This is called the face plate. The face plate is attached to the spindle and this is the work part which is turned in the face plate. These are also the clamps which are supporting this. So, this is a special kind of fixture which is called the face plate.

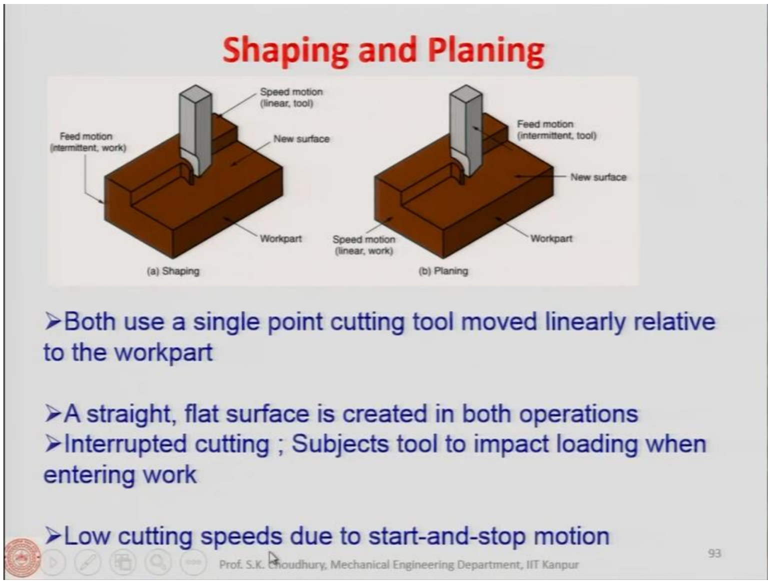
**(Refer Slide Time: 45:20)**




Now as I said about the boring, I will go quickly through this that the difference between the boring and turning is that boring is performed on the inside diameter of an existing hole, the hole already exists and boring is performed so that the internal hole is enlarged. Turning is performed on the outside diameter this is a turning and inside this the boring tool. The boring is made to enlarge that. In effect boring is an internal turning operation.

Horizontal or vertical boring machines refer to the orientation of the axis of rotation of the machines spindle. If the machines spindle rotates horizontal it is a horizontal boring machine or otherwise it is vertical boring machine. This is a boring machine with the base worktable, work part is clamped in here, this is the side tool with the turning process can be made and this is the tool head where you have a cross rail in which this tool heads can be moved with the tool which are the boring tool which will make the internal hole enlarged.

**(Refer Slide Time: 46:31)**



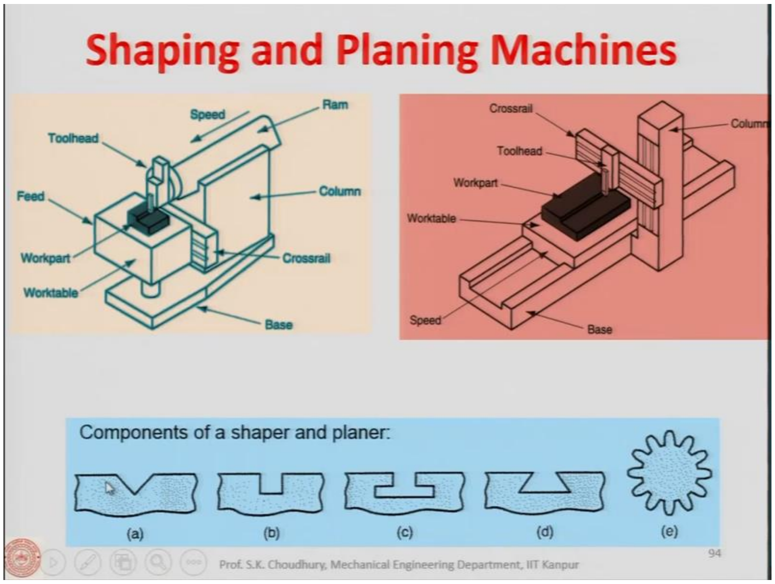

Shaping and planing both use a single point cutting tool and moved linearly relative to the workpiece. The shaping operation, we will show you in the lab also in more details. This is the shaping where the tool reciprocates and with each reciprocation that is going forward and going back. Once the feed is given to the workpiece and here it is opposite in case of planing when the workpiece reciprocates and the tool is given the feed.

This is the feed motion, an intermittent motion, given to the tool. The feed motion given to the workpiece is also intermittent because in one stroke and stroke is going forward and coming back. This is one stroke. During one stroke the feed motion is given along the perpendicular to the motion of the tool directions, the feed is given perpendicular to the direction of the workpiece.

Because here the workpiece reciprocates and there the tool reciprocates. Straight flat surfaces created in both operations. This is the new work surface having interrupted cutting because after the one stroke it will move and then again subjects tool to impact loading when entering the workpiece. Because each time it is entering the workpiece in case of shaping or in case of planing there is an impact loading.

Therefore, what happens is that the tool has to be stronger. Any brittle material that is used may not withstand this kind of impact loading. Low cutting speeds; due to start and stop motion, cutting speed used is lower.

**(Refer Slide Time: 48:28)**

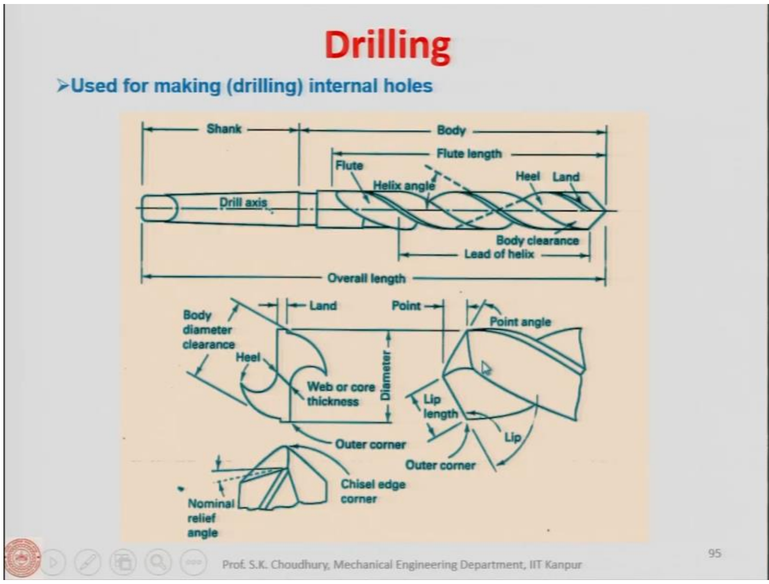



Shaping and planing machines are like this, this the shaping machine where the tool reciprocates and this is the planing machine where the workpiece is longer than in case of shaping machine and the workpiece reciprocates. This is the base of it. This is the worktable on which the workpiece is mounted and this is the cross slide, cross rail and it is the tool head. Similarly, this is the tool head and this ram reciprocates with the tool.

The tool when it is going back it will actually not remove the material. Therefore, going back at a higher speed. Cross rail is used to give the feed to the workpiece after a stroke of the tool. These are the components that can be fabricated in shaping and planing. So, these are the internal dove tails.

That can be machined by a shaping machine easily. These kinds of slots are used in the milling machine table here for clamping the tooling, the fixture or the workpiece on the table of the machine tool that is the milling machine. This kind of a shape can also be cut within the same slant surface using the shaping or the planing operation.

**(Refer Slide Time: 49:54)**




Drilling: Here are the details of a drill. Starting from the shank, this is the shank where the drill is held in the drill chuck. This is the drill axis and this surface is the most tapered surface, this has a taper and this taper is of particular type and this taper is called the Morse taper. This is the shank and the shank is for removing the drill from the drill chuck. How to remove that we will show you in the lab further.

Here we have the flute, this is a spiral and this goes all along the body of the drill and through this the chips are removed and also through this the cutting fluid can be passed to the machining zone. Here when the drilling is made it is enclosed whereas in case of turning the chips are coming out easily because it is in the open. But in case of drilling, the drill goes inside the workpiece and as the drilling is made or the hole is made, the drill penetrates to a certain depth then it is enclosed.

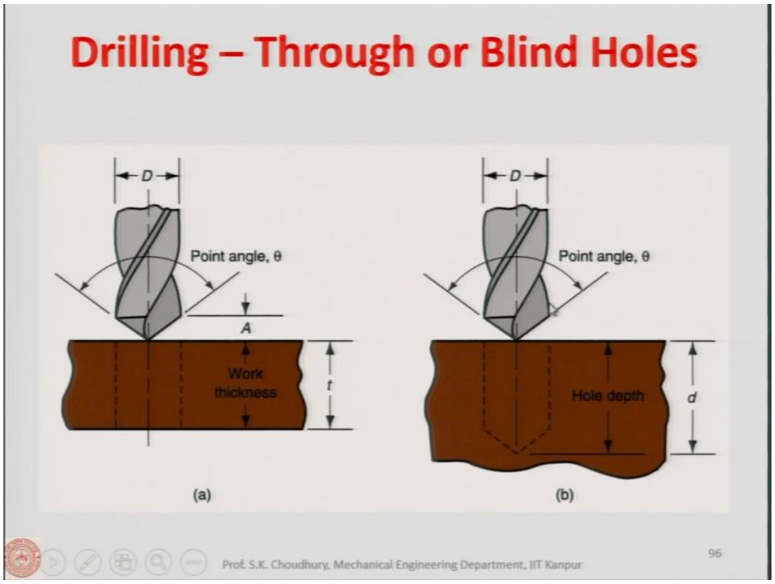
It is very difficult for the chip to come out therefore this kind of a flute is given so that the chip can come out through this flute. There is a helix angle here, this angle with respect to the axis of the drill is called the helix angle, this is the body of the drill, this is the lead of helix and this is the body clearance. Body clearances is that diameter is different along the length of the drill.

The drill can be guided and drill can be penetrated easily to the workpiece and here are the geometrical parameters. If you look at the drill from this side, this will be visible, this is the drill again the same thing here and this angle is the point angle,  like in case of turning point tool we have shown you that is the cutting angle. So, this is equivalent to the point angle.

This is the lip, one lip here and one lip here. From here to here this is called the distance is called the point and therefore this is called the point angle. Lip length is here, this is the tool diameter from here to here and this portion is called the land. This is the web or the core thickness, body diameter or the body clearance is here. You can see that this is the chisel edge.

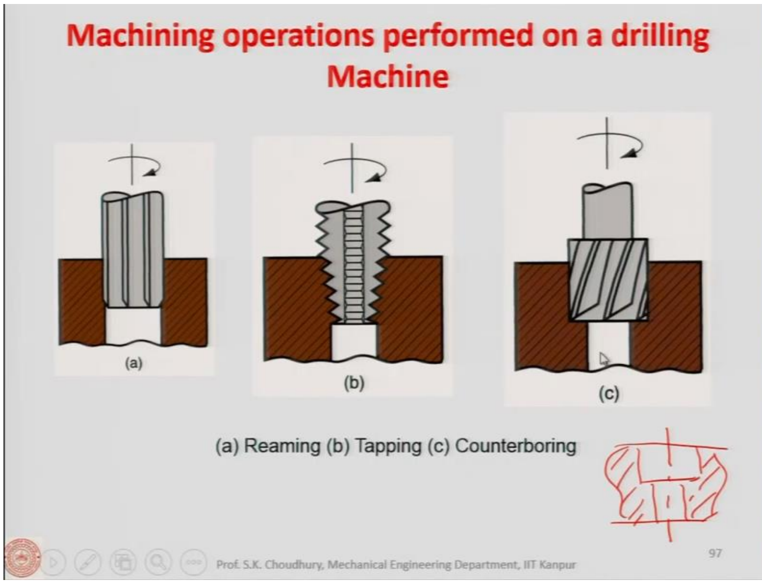
This is the nominal relief angle in the lip and in the face which is the rake face here through this and the flute from where the chip flows. You can show the rake angle here. The rake angle will in the drill and drill we have already shown it to you earlier that if you take a section which is perpendicular to this during the cutting process then the rake angle and the flank angle can be visible along with the chip removal process.

**(Refer Slide Time: 53:43)**




Here is how the drilling is made. When the drill is made throughout the workpiece thickness then it is called the through drilling. Here is the point angle, this is the diameter of the drill and if it is drilled up to a certain depth this is the hole depth this is called the blind hole that means it does not go all through the width of the workpiece. Now here one thing that you will notice that; any drilled hole, suppose this is the hole which has been already drilled. It will always have an impression of the drill obviously although this is not required. Suppose you drill the hole, tap that drilled hole, make an internal thread and then you pass a bolt through this. Bolt will be up to this and this much is not required but you cannot help because this is the impression of the drill which will remain here. But in case of through hole of course it will not come because it was out.

**(Refer Slide Time: 54:53)**




Here is what I was telling you about the reaming. The reaming process is done after the hole is drilled to increase the surface finish of the hole because after the drilling, the drill always will have the vibration, drilling machine will be having the vibration and normally the drilling operation is the rough operation because the entire material is to be removed and there was no hole the entire hole is made but in case of reaming the hole is already there.

Since after the drilling, the hole diameter is not perfect, not to the desired diameter and surface finish is not according to the drawing. So, the reaming is done so that a very small layer of material is removed, thin layer of material is removed. Material removal is not the basic purpose of the reaming but basically it is the increase in the surface finish that is required. Therefore, the reaming tool is called the reamer.

Reamers will always have the even number of teeth, these are the teeth. This is because you have to get the diameter of the reamer properly measured and if it is not even number in that case the Vernier caliper may not sit well to find out the exact diameter That is measured so that the exact diameter of the hole can be obtained.

This operation is called the tapping operation, to make the internal thread.
This process is called the counterboring, counterboring is when we have the two holes. I mean internal hole with two different diameters. See this hole is made and after that what you will have is this hole. Here is a bigger one and this is the smaller hole.

So, this is the workpiece and this is the hole that is made, this is called the counterboring. So, the process is called the counterboring. In case of tapping, you can see that here the diameter of the tap is increasingly more because this part is actually making the rough operation and the subsequent teeth up the tap is making the finish operation. Finally, what you are getting is the tapped hole which is of the maximum diameter.

The drill that is the tap or the thread diameter that thread diameter will be according to the maximum diameter of the tap and the rest of it is for guiding purposes. This is the tool that is a unique tool because it can have two drills one drill which will have of this diameter and one drill which may have the bigger diameter.

So, at a time you can have actually such a hole drilled with the two different diameters one is bigger, one is smaller. This is made just to put up a special bolt where the bolt head will be sitting inside and it will not be protruding out of the surface. Rest of the materials we will discuss during our next session. Thank you for your attention.