Introduction to Mechanical Micro Machining

Prof. Ajay M Sidpara
Department of Mechanical Engineering
Indian Institute of Technology, Kharagpur

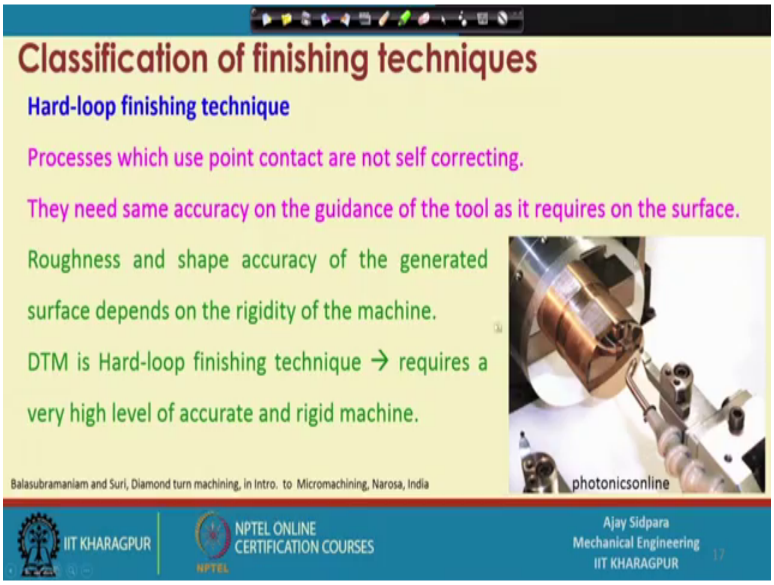
Lecture – 58
Diamond turning (Contd.)

Start.

Start ok. Good morning everybody and welcome again to our course on Introduction to Mechanical Micro Machining. In the last class we have seen some of the aspect of diamond turning machining and we have seen that there are different ways we can do machining of different type of components; like a concave type concave or convex type and some freeform surface is also by utilizing x z and c or b axis machining.

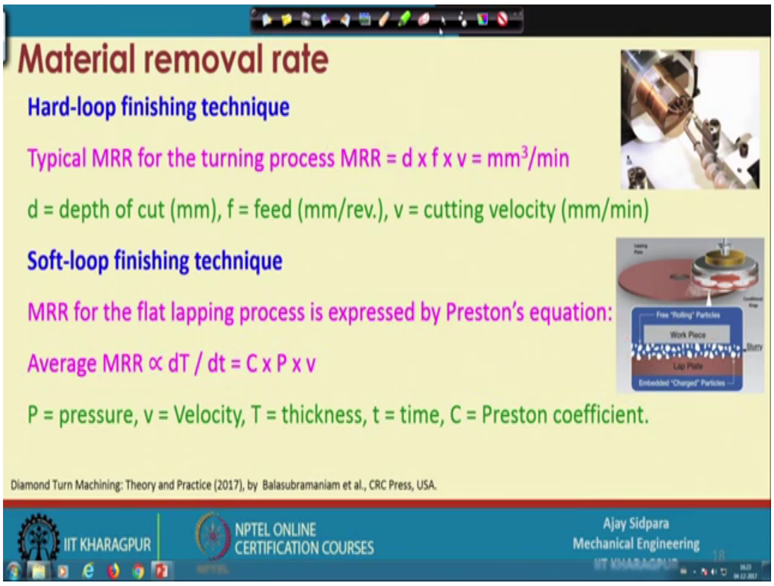
And we have seen two classification one is the Hard-loop machining and another thing is the Soft-loop polishing or Hard-loop polishing. And it was found that diamond turning is considered as a hard-loop finishing process because we have to maintain the tool path with respect to the workpiece or it is not like a multiple point contact machining, but it is a single point contact machining. So, let us continue further our discussion on diamond turn machining.

(Refer Slide Time: 01:17)



So, this thing what we have seen in the last class is the hard-loop finishing technique. So, here what we require that it is a single point contact. So, it is not self correct income similar compared to the soft-loop polishing.

(Refer Slide Time: 01:29)



So, let us see that how the material removal takes place in both the cases. So, first let us take about the hard-loop finishing. So, here we are talking about diamond turning machining and this is what we are doing in the diamond turning machining. An equation is similar to what we have used in the conventional machining process like a depth feed and the speed. And where the d is the depth of cut mostly it is in millimeter when we talk about conventional processes feed rate is also millimeter per revolution and cutting velocity, we consider as a meter per millimeter per minute; and when we get all this value you can find out material removal rate in terms of millimeter cube per minute.

So, here what is important? That here we know that only single point is coming into contact. So, that is the reason that depth of cut you can actually predict very easily the how much you are going in the z direction, feed rate you can set it and velocity also you can set by that mean. But if you consider soft-loop polishing this equation cannot be used here. So, here what is happening that there are multiple abrasive particles which comes into contact with the surface. So, depth will be different because some bigger size particle indent more.

So, depth of cut will be more for that cases and some particles small particles which will penetrate less in the workpiece so, that that time depth of cut will be less. So, here depth of cut cannot be used as their something like that feed rate here what is important? That how you are moving. So, that is actually the cutting velocity not the feed rate. So, feed rate it is not coming into picture here; velocity is playing important role; other than velocity what is important? How much you are pressurizing from the top; so, that is that cutting velocity and pressure play important role in the soft-loop polishing or the loose abrasive machining.

So, here for material removal rate actually there is one classical equation available that is called Preston's equation. So, what this equation tells average MRR is proportional dT by d capital T by d small d equal to C P and v. So, let us see what are these parameters; so P is the pressure. So, this pressure is how much pressure you are putting from the top; that means, suppose you this is the abrasive particle slurry at this line and then you have to put some pressure from the top.

Then only this particle will indent into the surface. So, right; so that is related to the pressure part and then you have to give a relative motion between the workpiece in the lap part so, that is called the velocity. Now what is this thickness? Now initially what we are looking here right. So, suppose what is happening that suppose this is your initial surface on the workpiece; and this is the tool or lap which you are moving in this direction these are the small small abrasive particles.

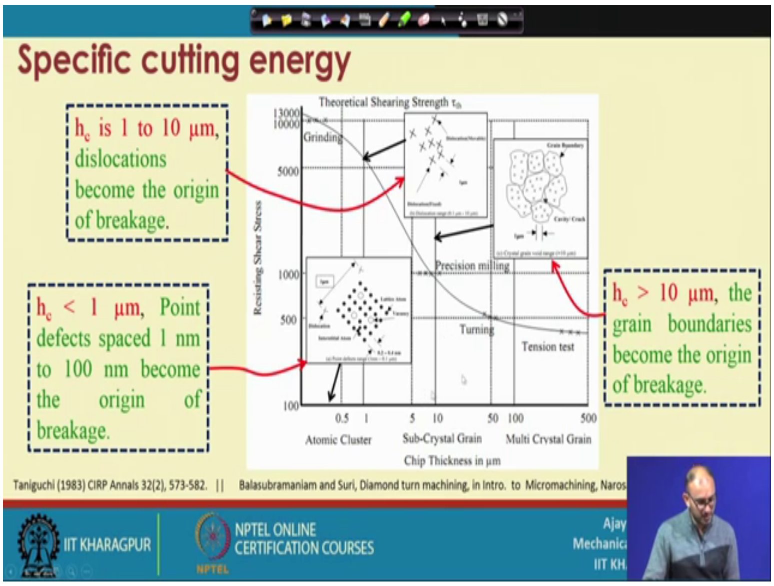
So, when it is moving, at that time; what is happening that it is removing some of the material from the workpiece surface? So, that z height is that how much amount of material is removed here 1 minute or 1 hour or that is called the z height. So, this is the T. So, that is that dT by capital D by small d is equal to the rate of reduction; in thickness of the workpiece right.

So, by that way you can actually find out how much is the reduction is the thickness with respect to time. So, pressure is more important, because when if you do not press the water; it will not indent inside and velocity is required because after indentation of this particle. So, this is the particle it is indented into the surface. So, this is the surface and once it is indented by normal force then it should translate in this direction. So, this much amount of material it can remove; so, that is a velocity is also important.

Now, there is one thing that is called Preston coefficient. Now, we know that many things we are playing important role other than pressure and velocity, if you change the particle size then things will behave differently, if you change the particle type also because suppose you are using a silicon carbide and then you use diamond particles then things will change. At that time this all these parameters other than pressure velocity you can actually fit into the Preston coefficient.

So, now, you can see here. So, this is actually not more predictable. So, this is more we are talking about the average of MRR and here what is them the it is predictable because we are actually fixing this all three parameter and which is not possible here. So, in our hand what is our hand that is the pressure how much load we can put from there and then the velocity; rest of the things are random because some particles will indent some particles will not at all indented. And these are the things which makes this particular soft-loop finishing process more unpredictable.

(Refer Slide Time: 06:32)

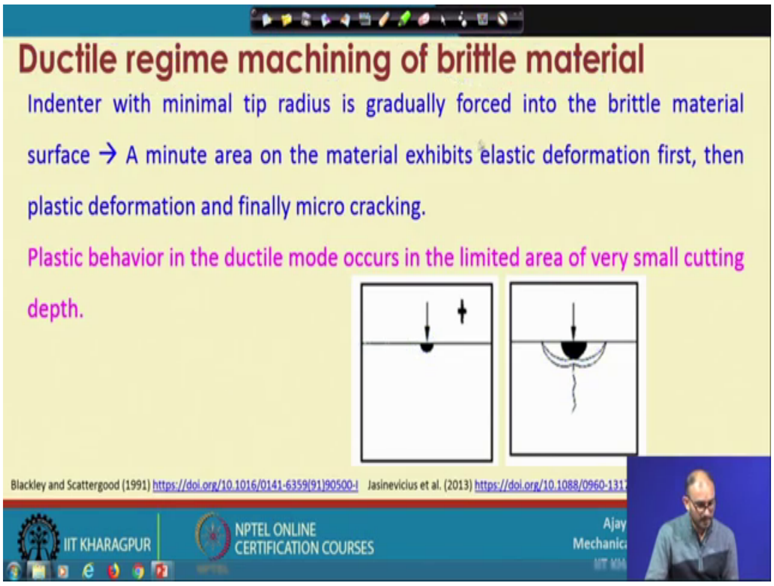


This graph we have seen in the micro cutting operation also, but still it is relevant to this particular process because here what we are doing? we are actually doing machining at this particular level. So, this is the normal thing what is happening that here you have to find out the grain boundaries through which the material removal takes place.

So, wherever your tool is coming into contact or it is passing through the grain boundary then there is a removal of that material in form of chip, but if you reduce this particular uncut chip thickness then what is happening then it will around the 10 micron or something then it is looking at the defects within the grain. So, now, you have to look around the this location or some other things which will create a or which will allow this tool to pass through the cutting edge.

So, this is the second thing. And if you go further inside it; when your uncut chip thickness is less than 1 micron then this is what is happening. So, then you are actually looking at the atomic scale; where you have to find out the what are the where can see and the dislocation something within the this location part atoms and everything. So, here this what we can say defects at the; within the dislocation you have to find out in such a way that that particular location will become the origin of the breakage. Because our objective is to get a uniform chip thickness so that you can get a material removal very efficiently without any problem. So, here the material removal takes place at that level. So, this graph is also relevant to the diamond turning.

(Refer Slide Time: 08:07)



Now, let us talk about the ductile region machining of the brittle material. Now we can see here the brittle material mostly actually ah; we can get the discontinuous chip in the brittle material, but you can actually cut this brittle material in the ductile region. Now take this example, now here what is happening that indenter is gradually indenting the workpiece surface.

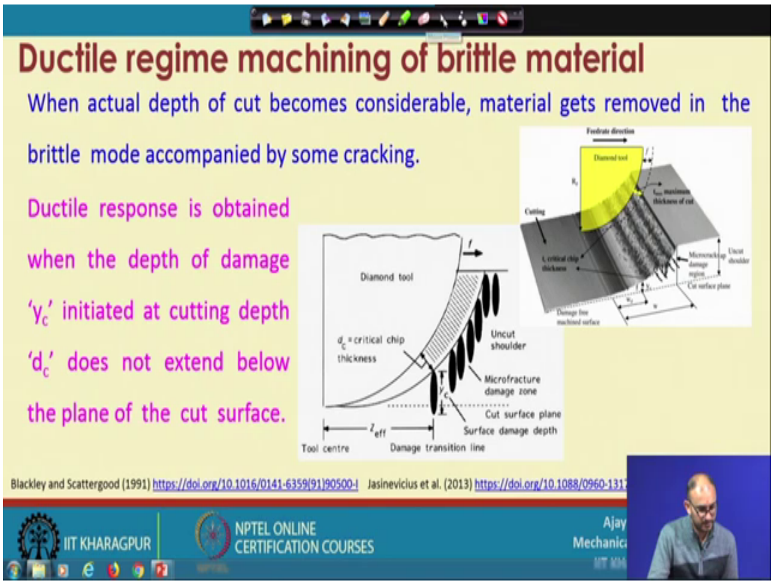
So, this is the initial [vocalize-noise] contact between the indenter in the surface. And you can find out a small amount of dimple will be created and if you continue further indentation then what happens the lateral crack and the this crack particularly will start forming and then it will remove the material in terms of some of the craters.

So, indenter with a minimum tip radius is gradually forced into the brittle material surface a minute area on the workpiece exhibit the elastic deformation first; so this is that area what we are talking right now. And then; plastic deformation finally, the micro cracking. So, this micro cracking will takes place, if you further continue your loading. So, plastic the behavior in ductile mode occurs in the limited area or a small cutting depth. So, now, our objective that if we cut any brittle material within this depth of cut. So, let us consider this we considered as a depth of cut; correct.

So, if you work within the depth of cut then what we can do the actually cut a brittle material in the form of a ductile machining. So, that is why it is called the ductile regime machining of a brittle material. We are not actually going into the brittle model we are more restricting ourselves within the ductile regime only. So, let us see the how this can be done in the diamond turning.

So, our objective or our bore focused is on the depth of cut now because now, we know that if you maintain your depth of cut less than or equal to this particular phenomena that also depends on the material and the type of indenter, but if you maintain that thing. Then you can actually do cutting of a brittle material in a ductile mode; right.

(Refer Slide Time: 10:16)



So, when actual depth of cut becomes considerable; considerable means when it is higher, then material gets removed in the brittle mode accompanied by the some cracking. So, we have to work within that area. So, now, this is the very classical figure and mostly it is used for explaining this ductile regime machining of a brittle material; what it tells us. So, here this is the diamond cutting tool.

And then this is the feed rate. So, when you give a feed rate in this direction this is the chip thickness whatever it is a variable 0 at this particular location and maximum at the top surface of the workpiece. Now when you cut a brittle material, at the time what happens there are lot of cracks generated just before the deformation or the for this uncut chip thickness. So, what is our objective? That our we know that our surface is this one this is the surface which we require. So, this is our required surface.

So, if you, if we do not allow this particular indentation or the damage to cross this particular line then we can and we can sure that there is no defect or something generated because of the machining. So, now, this y c what is this y c y c is the surface damage depth. Now right now it is entered in this. So, this is the cut surfaced plane. So, this is the surface what we required and now this is the things which we do not require correct.

So, now, this is called the critical chip thickness. Now you can see the chip thickness is variable 0 at this location and maximum at this location right, but this is the one location which is very critical that is why that; if you do machining below this level; that means, anywhere here at that location at that time this y c; this y c will be within this particular zone that is it is on the surface it will not cross this cut surface plane. Then you continue this machine because whatever these things are generated here, these will be actually removed in a subsequent feed because feed we are giving in this direction; anything generated ahead of this tool that will be removed in the machining.

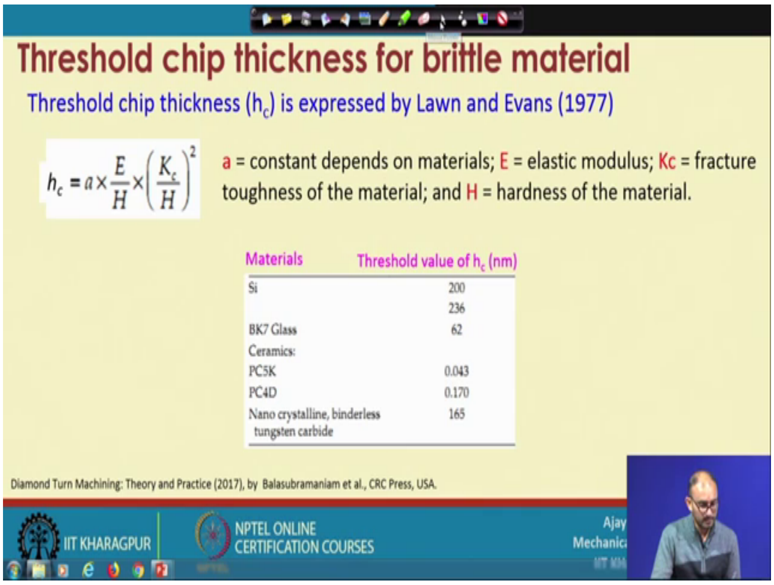
But if anything is penetrated this cut surface plane then this cannot be removed. So, this will be a permanent damage to the; or permanent defect to the workpiece surface. So, our objective is that this y c should not cross this particular surface this line. So, that is our objective. And this particular d c will make sure that if you operate your machine or if you operate your or if you set to d c that is chip thickness uncut chip thickness below this level, you are making sure that this y c will not cross the plane surface right. So, ductile response is obtained when the depth of damage y c initiated at the cutting depth d c.

So, this is y c that is initiated d c does not extend below the plane of the cut surface. So, that is our objective. So, over if you cut a ductile within the ductile regime then there is a high chance that this y c will not cross the surface and now this is the three d view of this image. Now it is also showing the same thing now these are the crack generated at this location and now you can see that this is the machine surface which is generated after machining of by diamond cutting tool.

And you can see this y c if you maintain this and here you can see this is the maximum uncut chip thickening this that is exactly at the top surface of the machine component. And this is somewhere here it is showing the critical chip thickness and that is showing at this particular y c. So, anything generated here you can see that there are some more cracks are here, but we know that our surface is here at this location. So, anything which comes at this location that will be removed by the diamond cutting tool at the later stage.

So, our objective is to do cutting at this particular d c or lower than d c and we know that the d c will change depending on which type of material you cut. So, let us see there; how this d c will be calculated for different regimes, right?

(Refer Slide Time: 14:16)



So, this is the threshold chip thickness that is what we are talking h c right now because we earlier in that micro machining of mechanical methods we use h c as a term, but that is why we let us continue hc is in earlier case what would d c that is the same as h c. So, that threshold chip thickness is given expressed by this particular formula.

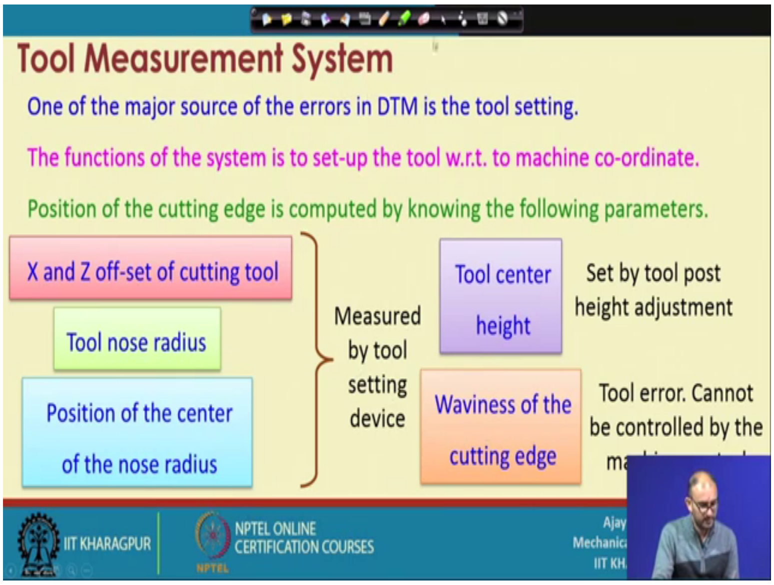
So, here a is the constant of the material because we know that, but this particular threshold chip thickness is variable depending on the material which we are using. Then elastic modulus is the E H is the hardness of the material K c is the fracture toughness of the material. And by that way actually you can find out h c of a different; different materials.

And these are the h c value in terms of nano meter, now you can see that if you want to cut silicon in a ductile mode then you have to main you have to be you have to select h c value lower than; 200 nanometer. And similarly there that there are 2 or 3 because earlier some other researchers also suggested 236 h c value for silicon. Then b k glass, then there are some of different above ceramics material available then Nano crystalline.

You can see that how much depth of cut you have to maintain. Now if you want to maintain this depth of cut; and if your machine is not capable to give this much amount of depth of cut that means; by means of penetration of the tool into the workpiece. So, your penetration should not cross the 200 nanometer. Now which particular system will give you this type of resolution that is we you have to use the linear motors and some type of aerostatic bearings.

Then only it is possible to work within the nanometer regime. Otherwise whatever ball screw type of things and some other type of where mechanical contexts are more and friction and wear is more. At that time you cannot get this type of resolution at the nanometer scale. So, that is the why diamond turning machines are very very costly and similar the case for the micro machining center. Because the technology which is being used for making this machine more and more robust and more and more accurate. That is the reason that you can get the accuracy at the nanometer level.

(Refer Slide Time: 16:25)



Now, coming to the tool measurement system because everything is depend on how accurate is your tool. Once you get the required tool accurate then what is the second thing is the how you are fixing the tool on the tool post. So, that is the thing. So, one of the major sources of source of errors in the DTM is a tool setting. So, what we are looking here? We are looking at the nanometer scale depth of cut and if nanometer scale in the depth of cut. Then your requirement of the rigidity of the tool setting also very very important; so if you do not set your tool perfectly at the x 0 and z 0 then there are lot of problems related to the machine surface.

So, function of the system is to set up the tool with respect to the machine coordinates because we have machine our workpiece is located and the headstock. And then what you have to do that you have to find out that, at which location you are a starting the machining. So, the because in normal turning operation what we do there we try to align our what we say the nose radius we through at the center of the workpiece.

But that is not the case always in the DTM because we know that we want to create some freeform surfaces of concave and convex surfaces by different type of setups; we have seen. So, at the sometimes what happens we do not require to go to the center, but we have to find some location onto the facing surface; where we can start the machining for the different different operations.

So, position of the cutting edge is computed by knowing the following parameters. So, what we have to find out? First we have to find out x and z offset of the cutting tool right. So, because mostly our machine is related to x and z only. So, this is our workpiece right and this is our cutting tool which is located at this part location and this is the center. Now when you are moving in this direction this is the z direction; and when we are moving in this direction; this is the x direction right.

So, mostly in the x and z at the configuration mostly this is normal similar to, but normal turning oppression or facing oppression. So, at that time we have to make sure that these things are actually aligning; that means, this is coinciding with the center of the tool workpiece is coinciding with the edge of the cutting tool. Then you can do actually facing oppression conveniently, there will not be any type of (Refer Time: 18:43) material whether it is on the top surface or this surface.

So, that is the one of the ways of setting this part. second way is the tool nose radius. Because tool nose radius is playing important because now, right now you whatever you are setting you are setting from this particular plan, but if you see from the top surface. Now when you see from the top surface then your tool has a nose radius right. So, we have to make sure that it is touching at this location not at this location or this location otherwise your program control will be different than the; what you have to achieve. So, nose radius you have to maintain right; position of the center of the nose radius that is what we discussed just now right.

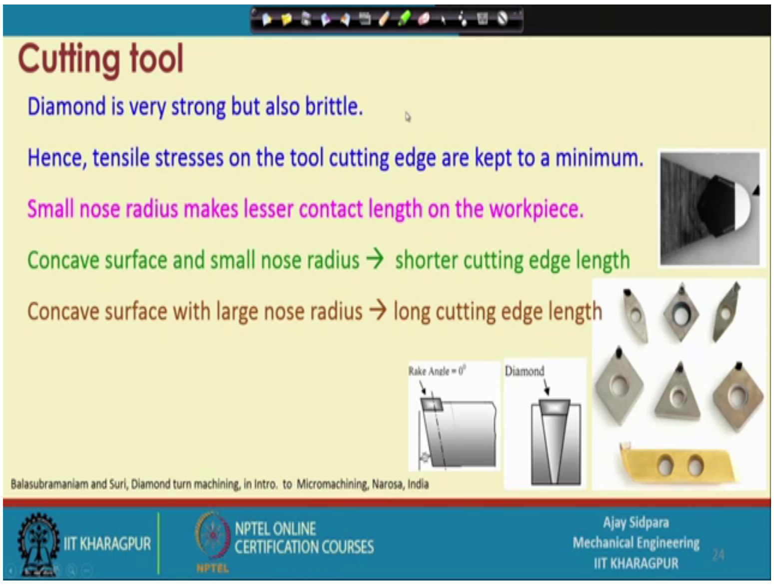
So, all this three parameter this measured by the tool setting device. So, there are tool setting device available; some of them are optical microscope which are attached over the cutting tool and by some of the things which you can get this thing done by the some type of manual or some type of sensory elements. Then the tool center height adjustment because we want to make sure that we are our tool is exactly lying at the center of the workpiece at that time a minor change in that suppose it is a few micron here and there. up and down then it is very difficult to get the center done.

Because of when we are talking about the optical quality of surface finish then even a 1 micron or 2 micron of adjustment is also very very important even we are looking at the sub nanometer level of adjustments. So, this tool height adjustment is performed by the one of the things which is attached with the cutting tool that we will see that there is one slide available for discussion of this part.

And another thing is the waviness of the cutting edge right. So, whatever we are cutting edges you whatever you are getting mostly we will see something like this is you consider this is the nose cutting nose radius; nose radius of the cutting tool; correct. So, now, what happen by look wise it look it is ah; it looks very sad, but if you magnify it then what you can see here that you will find lot of irregularities on the top surface. So, here you will get some of the irregularities. So, this irregularity is also play important role. Because if at this particular location if some of the peaks are very very large then the contact point will actually happen at that localized point only; not to the wall nose radius. So, that plays very important role.

But this is the tool error, this cannot be controlled by the machine controller because for making the or correcting the waviness; you have to fabricate your cutting tool in such a way that there is a less variation on the waviness of the cutting tool. So, this is not controllable parameter, but it is a tool parameter, but not related to the machine.

(Refer Slide Time: 21:36)



So, now, cutting tool mostly we our name is the diamond turn machining. So, mostly we use the diamond only because diamond is very very hard material and it will not actually create any problem when you do machining on of a ferrous material or non ferrous materials; so, that is the advantage. So, we use diamond it is hard, but it is also brittle material; so, that is the big problem. So, hence the tensile stresses on the cutting tool edge are kept to a minimum because we do not want to actually stress the diamond, but if you put a compressive loading we will not create it will not create any problem.

So, small to nose radius makes the lesser contact length on the workpiece surface. Now if you see this is the rake surface and this is the clearance surface; clearance angle this is the rake angle and this is the diamond cutting tool and this is for looking from the top surface. So, this is the nose radius and so, this is the nose radius what we are talking here. So, small nose radius makes the laser contact on to the surface. So, now, what we can see here that suppose our workpiece is something like this is our workpiece right and we have a two tool with a one is the nose radius is something like this and second is the nose radius something like this much.

Now, you penetrate this particular workpiece; tool into the workpiece. So, now, let us see that we are putting this thing. We are drawing one line here; now this is the line now our work is located at this location right. So, now, smaller tool makes the lesser context. So, now, see there were contact area is this much only for small tool small nose radius, but for large nose radius or contact area is more; why this is important? Because we know that we have we have to work within a very very small amount of depth of cut that is not only thing, but we have to maintain the force is acting on to the surface of force playing will be very very high.

Here because area is more, area is more; so, force is more. So, similarly here area is small and your force is small. So, you can retain this particular geometry of the cutting edge for a longer time if you want to go with a smaller nose radius, but smaller nose radius also play important because you have to make it more and more sharp, but your life will be degraded.

So, you have to actually do a compromise between the because whether you want to go with a smaller nose radius or the larger nose radius right. So, concave surface and the small nature if you have machining with a concave surface. Now, suppose let us take that example; so, this is your concave surface; right. And you are using a small nose radius; so let us consider this is the small nose radius; right. And what happens because of that? That because of it is a concave surface.

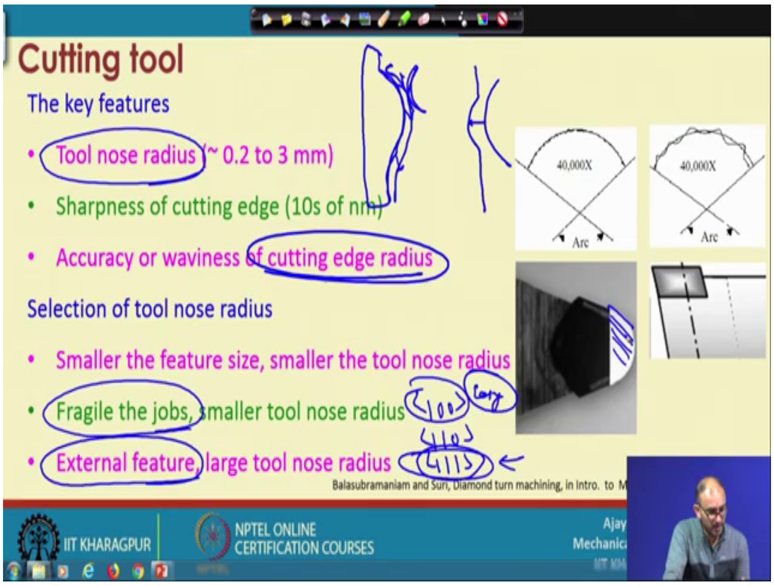
So, it contact will be very very small at this location, but now if you say that if you want to use the same thing, but if you have a larger nose radius. So now there is another nose radius let us see that. So, this is the another nose radius; this is larger than this one. So, at that time cutting length will be very very large. So, wherever it is going to touch the surface at that time we know that this is the cutting length. So, this cutting length will be very very large at this location.

So, sometimes what happen the when your curvature is very very large; at that time you can actually go with a large nose radius because if you are ah; if you want to do machining very quickly. Then you have to go with this one because this way you can actually increase the cutting area here and within a shorter time you can complete the complete you can complete this particular path, but if you go with this thing; it will take more time.

But there is another disadvantage here because we have seen here that if your contact is more your forces will be more and the life of the; particular or the sharpness of this particular nose radius will be or the cutting large radius will be very very less here. So, that is the reason that you have to see that which material you are machining. And then you have to set a parameter in such a way that you actually play around the cutting edge length, that which particular cutting edge length is enough to increase the production it also and that is enough to reduce the viewer of the cutting edge radius; cutting edge radius. So, that is the; that is way you can actually play around the parameters right.

These are the different photographs of the actual cutting tool. Now, you can see that all most of this; diamond cutting tools are the braised one here. So, you have to braise it with the; standard cutting tool or the standard shape. And then you can actually use this particular cutting tool for a micro machining operation; machining of a ferrous materials.

(Refer Slide Time: 26:29)



So, what are the key features of this particular process? So, it is at nose radius is around 0.2 to 3 millimeter even larger is also available for machining of a very very large amount of surfaces. Sharpness of the cutting edge is 10s of nanometer because sharpness is very important. Because we know that we have to give a depth of cut of a less than 1 micron; and we have seen some of the material where we have to maintain that uncut chip thickness or less than 100 nanometer or something [vocalize-noise] to make sure that we can cut those brittle material into a ductile mode.

So, that is why the sharpness of the edge; edge is 10s of nanometer accuracy or waviness of the cutting edge that will be play important role here because now see in actual case what we you are, what we are looking at. So, in the normally what we see is this one, but if you magnify your cutting edge at a 40000 then you will see this type of irregularity.

So, this may be considered as a acceptable because you cannot make something like at this thing that is impossible, but you have to; you have to maintain this unevenness or the roughness in such a way that, it will not create any problem, but if you see at 40000 and if you find out this particular thing; then it is a problematic.

Now what is happening? Now see this is the center of your cutting edge radius that that is nose radius. Now what is happening? That this particular point actually it is inside, but this particular coating cutting two cutting points which are actually projected away from this center point. So, now, what is happening that your whole program or whatever path you are giving to the cutting tool or the workpiece, then actually it is reference with respect to this point; not to this point. And when your workpiece comes to this location actually it will come in contact with this particular location; not to this location because this is more projected.

So, this is the material mode takes place you are not at this location; this will take place here. So, you are intended to location is actually shifted to the different other location and you are actually not able to control that particular location and that will create a problem on the surface geometry or whatever features you want to create. So, you have to also find out what is the accuracy of the waviness of the cutting edge radius that is the; right now what we are talking about the nose radius, but if you see on these particular cross section.

So, if you see this is the nose radius and if you cut down this thing then you will find this is the surface. So, this is the cutting edge radius. This is the cutting edge radius and this nose radius is this one is the nose radius; so, both things are important. So, we have to maintain the waviness and accuracy of the cutting edge radius also as well as the nose radius; right.

Now, how do you select the nose radius here? So now, let us see that this is the nose radius; if you are machining a smaller feature size it is better to go with a nose a small nose radius. Because we know that smaller the nose radius actually you can reduce or you can actually control the contact point very easily. So, smaller feature you do not want to come across this type of problem because now you know that there are; lot of problem if the contact point is different than the reference point.

So, it is better to go with a smaller nose radius; Fragile jobs better to go with the nose smaller nose because we know that if your nose radius increases, your contact point or the contact area increases. And we know that for Fragile job even a small amount of additional load is enough to break those component. So, better to go with a small and more amount of loading; and to make sure that loading is small; then your tool nose radius will be small.

External feature better to go with the large nose radius because now what is happening in external feature; now you know that this is a convex surface right you want to do machining at this location. Now with going with a smaller convex radius what is happening that you have to spend lot of time here and you since it is external surface. So, your nose radius has no connection with the what are the features available on the surface.

Now you can say that this is the features available and your nose radius is this much; then what is happening that you will be actually not able to penetrate to the extreme position; that is going to happen if the features are the internal features, but these are the external features. So, it will not make much difference in this case. So, larger or external features, but again it depends on which type of merc piece you are cutting, but mostly if in general statement the larger the; or external the feature it is better to go with the larger nose radius so, that you can actually do more amount of cutting operation and you can finish your tool path very quickly.

So, these are the different ways you have to select the cutting tool and these are the parameter our cutting tool material is fixed, but within the material we have to play with the different type of parameters. One is these two are the very important one is the cutting edge radius and another is the nose radius.

Then we have to find out because diamond you can actually do machining with a different because different different type of planes available; 1 0 0 plane and 1 0 0 1 1 0 and 1 1 1 plane we have seen in terms of silicon right in the; some lectures long time before. And we have seen that this is very difficult to cut and this is very easy to cut. If you cutting is easy; that means, (Refer Time: 31:48) is also very very high.

So, we have to also see that which particular plane is coming into contact with the workpiece; that means, what is this particular plane? And which plane you are making the cutting edge radius and the nose radius. So, depending on there are lot of process available by which you can actually manipulate this particular plane and you make sure that the hardest plane is coming onto the surface and you can do machining for a longer period of time. So, let me finish this lecture here; we will continue this topic in the next class.

Thank you very much.