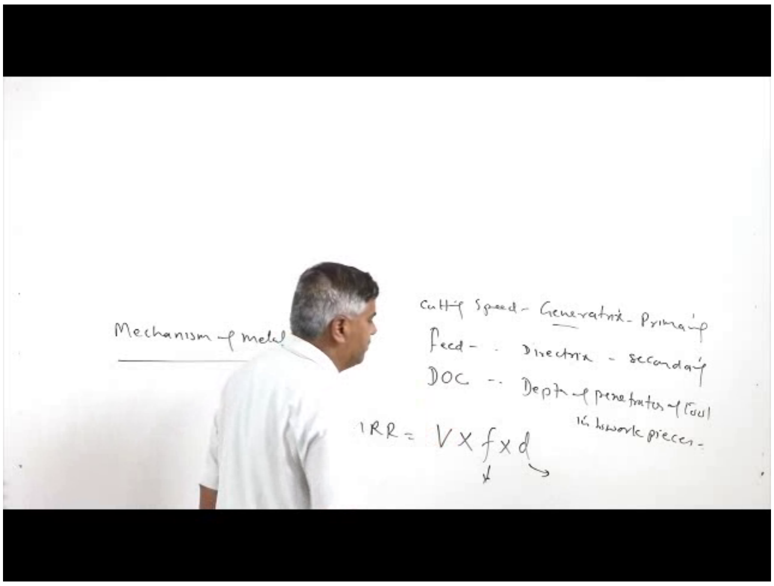
Fundamentals of Manufacturing Processes
Dr. D. K. Dwivedi
Department of Mechanical & Industrial Engineering
Indian Institute of Technology, Roorkee

Lecture – 36
Material Removal Processes: Mechanism of Metal Cutting

Hello, I welcome you all in this presentation related with the subject fundamentals of the manufacturing processes and we are talking about the material removal processes. And today we will be talking about the mechanism of metal cutting.

(Refer Slide Time: 00:31)



But before going into this mechanism aspect, as I have said there are 3 important parameters like cutting speed, which leads to the formation of the Generatrix. This is called primary motion; it determines the rate at which material will be removed.

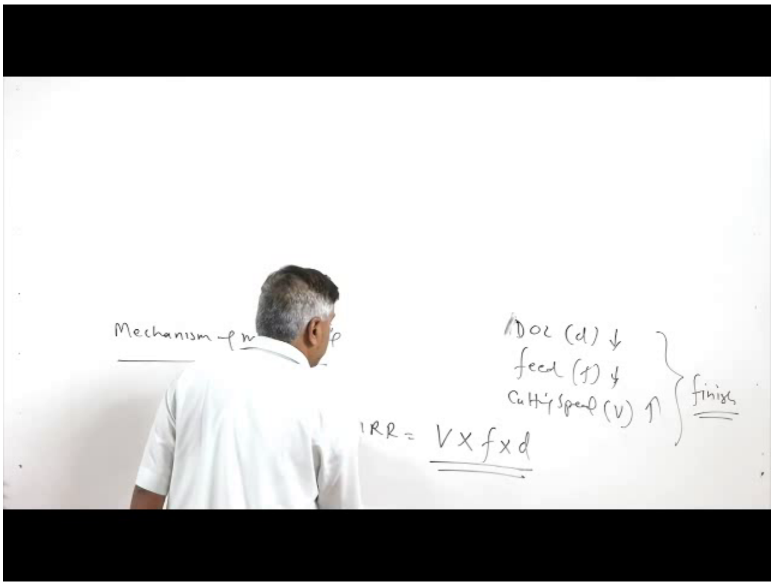
In addition to the cutting speed the feed is the another movement. This leads to the coverage of the material lateral movement; this feed is basically leads to the coverage of the entire range of the material which is to be machined and it causes that directrix, and this is called the secondary motion this leads to the secondary motion.

And the depth of cut decides determine is determined by the depth of penetration of tool into the work piece,. Actually the combination of all these 3 parameters determine the metal removal rate that metal, metal removal rate.

So, metal removal rate commonly known as MRR is expressed as in terms of the mm cube per second or it may be in mm cube per minute depending upon the units being used. And all these 3 parameters determine the MRR, which is basically the product of the cutting speed, the feed and depth of cut.

Since the feed and depth of cut are given in the limited amount, and these are also governed by the purpose of the machining process. So, depending upon the purpose that various combinations of the means the combination of the speed feed depth of cut are used. Normally the depth of cut DOC the d value is selected very low.

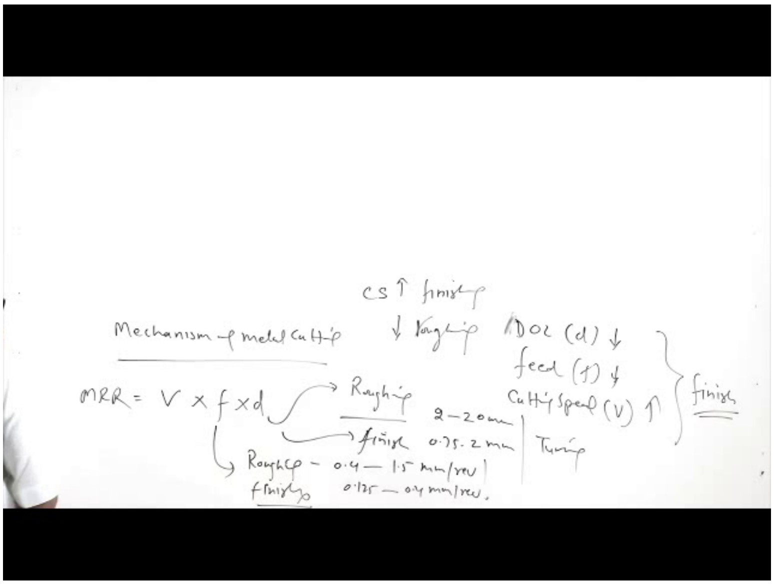
(Refer Slide Time: 03:03)



Similarly, feed f is selected very low and the cutting speed is selected high for the finishing operations.

Means at the last stage of the machining, when we want that very fine control over the damages as well as finishes to be realized, then very low value of the feed and the depth of cut is used in high feed high cutting speed is used. So but mostly we rely on the cutting speed as well as the depth of cut for governing the metal removal rate.

(Refer Slide Time: 03:55)

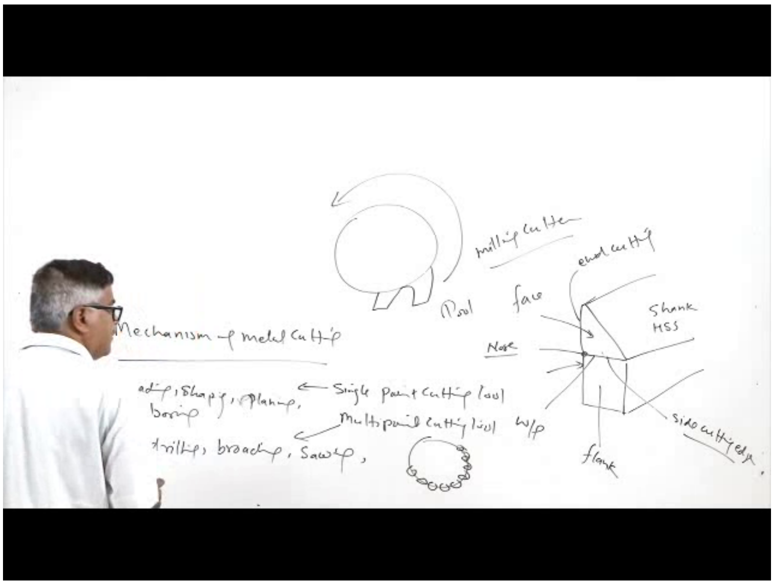


So, according to this equation increase in speed increase in feed and increase in depth of cut will be leading to the increased amount of the metal removal rate MRR. So, normally if we see the depth of cut for roughing operation when large amount of the material is to be removed. In that case the very high depth of cut may be ranging from the 2 to 20 mm depth of cut can be given, the while for the finishing purpose it is value is normally limited and 0.75 to the 2 mm depth of cut is given. So, for finishing purpose very fine depth of cut is given.

Similarly, for feed the very low feed rates are given for the finishing purpose while high feed rates are given for the roughing purpose for rough machining. Say for the rough machining feed rate ranges from 0.4 to 1.15 mm per revolution for of course, these are for the turning process.

While for the finishing purpose when the high degree of the finish is to be achieved, then it can range from the 0.125 to the 0.4 mm per revolution. So, the fine feed and fine depth of cut are used for the finishing purpose, and the high feed and high depth of cut are used for the roughing purpose. As far as the cutting speed is concerned, the cutting speed high cutting speed is used for the finishing purpose and the low is used for the roughing purpose so that the material can be efficiently removed from the stock material in order to achieve the desired size and shape at the earliest possible. So, as per the purpose of the turning or the metal cutting suitable combination of the cutting speed feed and depth of cut is used. Now we will see the how the metal is removed during the machining, and how does it help to get the desired size and shape.

(Refer Slide Time: 06:31)



So, if we see here like say this is the tool and this is the clearance so, better to understand first about the tool before getting into the mechanism like say the cutting. So, we will try to see first the single point cutting tool. There can be single point cutting tool or multi point cutting tool. Single point cutting tool basically involves the one cutting edge.

So, if we see a 3 dimensional view of the of the single point cutting tool, if we see and it may be like this yeah. So, these are this is this is basically shank it may be made of the high speed steel or like cast cobalt alloy or any other material. And this end is subjected to the tool grinding process so that the different surfaces can be generated. This top surface is called face or the rag face, and the 2 surfaces these 2 surfaces both the sides will be termed as the flank.

And there are 2 cutting edges one is this. So, this is called the side cutting edge, side cutting edge and this is end, end cutting edge end cutting edge. And the side cutting edge and end cutting edge meet at this point this is called the nose of the tool. And the nose of the tool is not pointed, but some kind of the radius is given so that it gets a desired strength.

So, the radius given to the nose is termed as nose radius, and basically the side cutting edge and this is the cutting edge mostly which interacts with the work piece. So, these are the main parts related to the single point cutting edge tool. And in case of the multi point cutting edge multi point cutting tool number of such kind of the geometries are created in the tool itself like this.

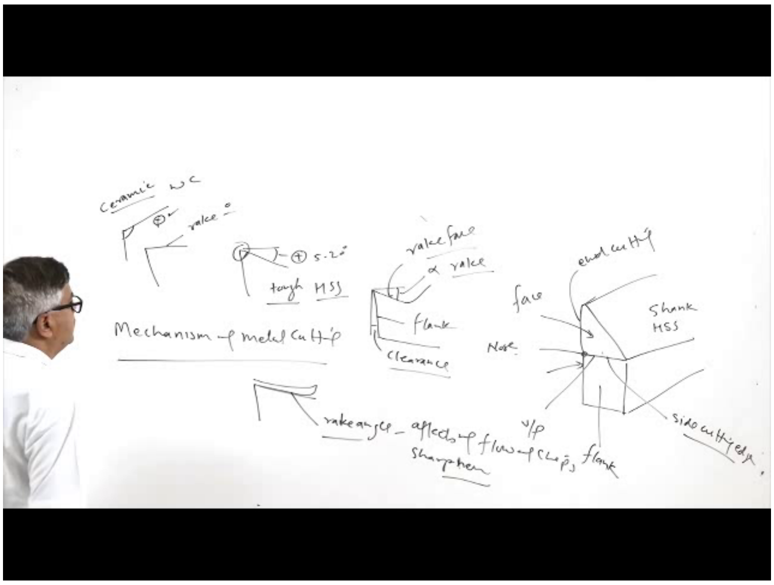
So, similarly such kinds of the geometries all around the periphery are created say in case of the milling cutter. So, we see this the kind of these kind of the features are repeated number of times in case of the multi point cutting tools. So, they are a very wide range of the processes which use single point cutting tool for example, the processes which are carried out on the lathe machine.

For example turning, threading, shaping on the shaper and then planing, boring. All these processes use the single point cutting tool. On the other hand processes which use the multi point cutting tool includes the milling, drilling basically it uses the 2 cutting edges and then broaching uses the number of cutting edges like sawing and etcetera.

And then there abrasives which are abrasive is processes like in grinding wheel that also like the grinding in the case of grinding wheel abrasive. So, are projecting from the surface. So, also these are the multi points which will be performing cutting, but the cutting edge is oriented in random manner. So, there is no specific way by which cutting edges will be performing the cutting in case of the grinding wheel.

So, these surfaces these surfaces and the angle at which the surfaces have are subjected these angles at which these are produce surfaces at which, the different surfaces are produced at a particular specific angle so that the cutting can be performed effectively and those angles are represented in very specific manner say in case of the turning. This is the work piece.

(Refer Slide Time: 11:24)



So, the cutting tool may have geometry of this kind this is the tool, and this is like say the centre of the work piece. So, after a removal of the material from the work piece like this, it flows over the face of the tool and then it passes through like this. So, so this particular angle this is the tool. So, if we see the top surface this is the rakes face or the face of the tool and this is the flank of the tool.

So, if we see there are 2 angles the with respect to the horizontal the tool face is inclined at certain angle, this is normally represented as alpha called the rake angle and this angle with respect to the vertical plane the flank is a slightly inclined, and this angle is called clearance or the relief angle. This is the simplified form of the orthogonal cutting tool; situation where in the rake angle which is sloping down facilitating the flow of the chips over the face of the tool.

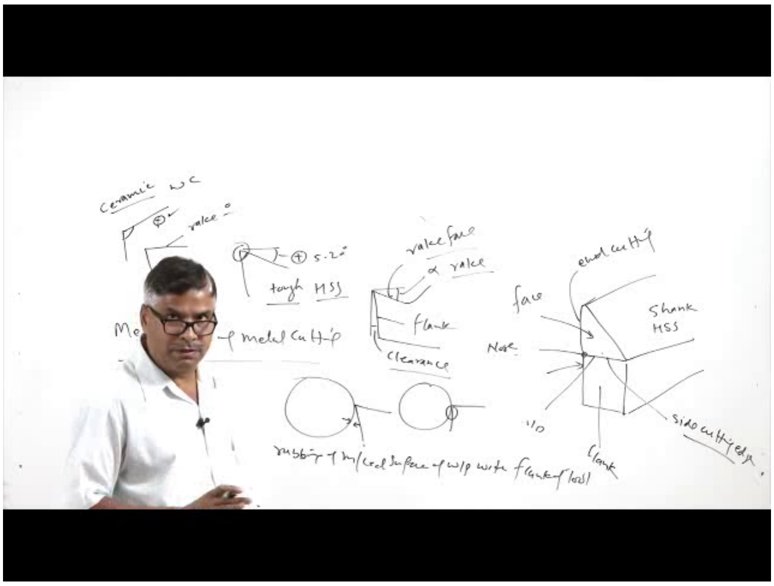
And so, the basically the rake angle affects the flow of the chips over the face after removal and the second is it also affects the sharpness of the cutting edge; positive rake angle or the negative angle depending upon the inclination of the face of the tool; like if the face of the tool is inclining back, this is termed as the positive rake. Normally given 5 to say 20 degrees, and like the tool face may not be given any angle. So, the rake is 0 rake, 0 angle or the tool face may be inclined in a forward direction.

So, this will be leading to the negative rake angle. So, negative rake positive rake is used for the tough tool materials like high speed steel, while the high the hardness and the somewhat brittle materials, the negative rake is used for example, like ceramic tool materials tungsten carbide tools the negative rake is used.

So, if we see when the negative rake is used, it increases the cutting edge cross sectional area, when the positive rake is used it reduces the cutting edge cross sectional area. So, the tough tools allow the machining by the cutting edge even with the limited cross sectional area, and in case of the ceramic and tungsten carbide tools which are of the low toughness and high hardness they can work efficiently with the negative rake only.

So, the purpose of providing rake angle is to have the proper flow of the chips after getting removed, and to have the proper sharpness of the cutting edge. At the same time it also affects the strength of the cutting edge because it determines the cutting edge cross sectional area. On the other hand the relief angle or the clearance angle, clearance angle if we see this one the clearance angle basically avoids the rubbing of the work piece with the flank of the tool.

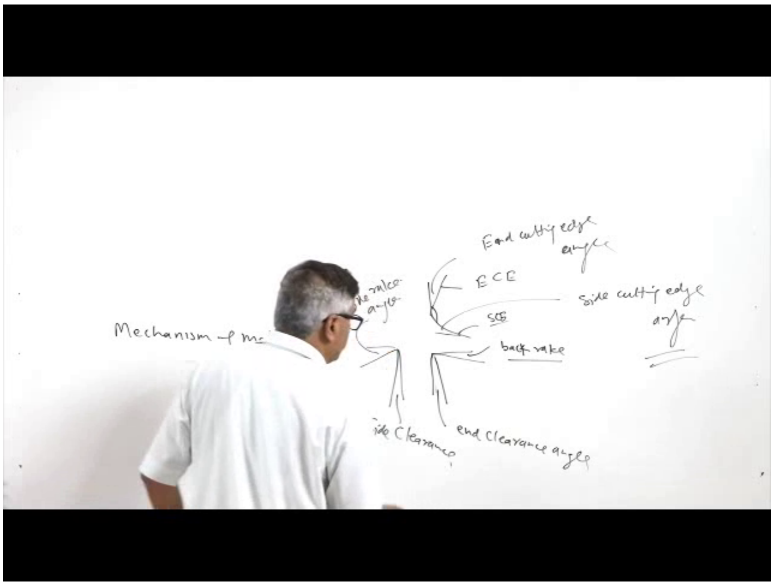
(Refer Slide Time: 15:22)



So, the to avoid the basically rubbing of the machined surface of the work piece with the flank of the tool, some inclination is given if we do not give this kind of inclination then you will see this portion of the flank will be rubbing with the machine surface of the work piece. And so, that is how it will be adversely affecting the surface finish of the machine work piece.

So, in order to avoid the rubbing of the work piece with the tool basically the clearance angle is given to the flank of the tool. So, all these tools or different parts of the tools are specified with the help of the different angles and the nose radius.

(Refer Slide Time: 16:43)

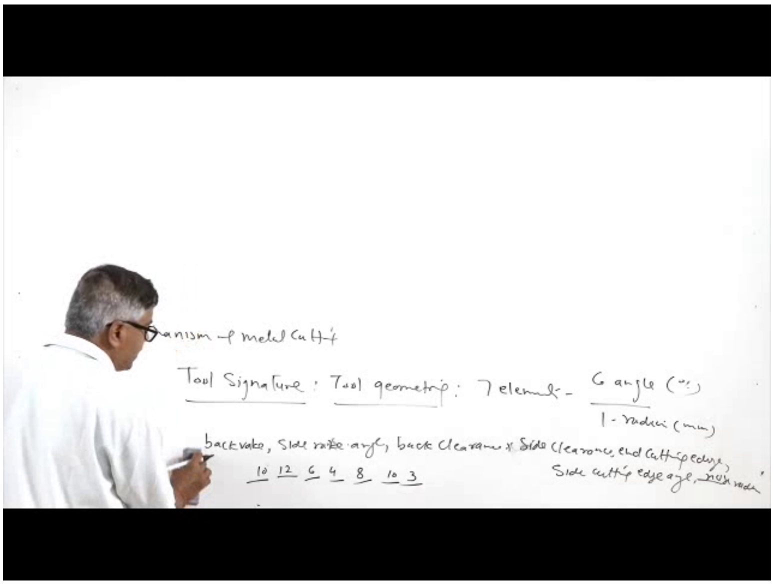


So, if we see the different parts and angles are expressed like this, here in front angle in the front view like this. So, these 2 angles as I have said back rake angle, and this is and clearance angle in the side view is we see this is side clearance angle, side clearance angle and this is side rake angle. And in the top view if we see will be able to see the different cutting edges.

So, this is the nose radius, this is the end cutting edge, and this is side cutting edge which will be basically in performing the cutting. So, this angle is termed as and cutting edge angle. And this angle is termed as side cutting edge angle edge angle. So, basically all these angles are of the tool then we will be able to see when the tool single point tool is seen from the different angles.

So, these 6 angles and the nose radius which will be nose radius which is connecting the end cutting edge and the side cutting edge are expressed in very sequential manner so that the tool features and the tool geometry can be specified with the help of the tool signature.

(Refer Slide Time: 18:49)



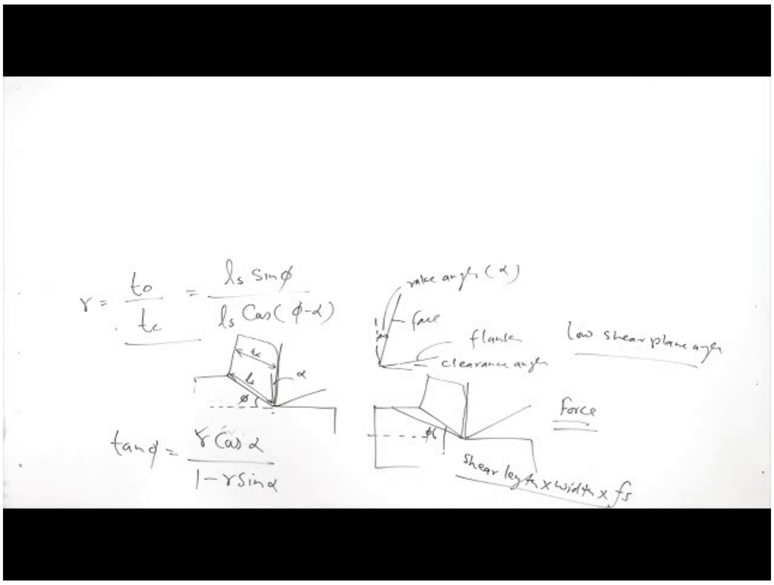
Tool signature indicates or shows the tool geometry of single point tool, where in the different angles as the nose radius are written in a specific sequence.

Which includes so, the tool signature basically comprises 7 elements, where in we have 6 angles and one radius. So, radius of course, is written in mm and angles are written in degrees. So, it so, it is starts with the back rake angle then side rake angle, and clearance angle side clearance angle. And then and cutting edge angle and side cutting, edge angle and last one is the nose radius.

So, as I have said these are the different angles which are written in this particular sequence only and at the end we have that nose radius. So, back rake angles like say these may be mentioned as 10, 10, 12, 6, 4, 8, 10 and 3. So, what it shows? This is the back rake angle. Side rake angle back clearance angle, side clearance angle and cutting edge angle side cutting edge angle and nose radius.

So, all these are in degrees and this is in mm. So, normally no sign is made with the tool signature. So, when we mentioned that these numbers in a specific sequence it indicates the specific angle of the tool. So, basically the tool signature helps to understand the geometry of the tool. Now we will see how the material is removed. So, say this is the tool single point tool and this is the so, this is a rake face of the tool and this angle is termed as rake angle.

(Refer Slide Time: 21:38)



As I have just mentioned, rake angle normally expressed using alpha and this is the clearance angle clearance angle and basically this is the flank or you can say this is face.

So, in this case when the tool is moving after certain penetration it removes the material, like say this is the machined surface this is the flank and this is the rake this is the rake angle alpha. So, a material when the tool is moving with respect to the a stationery work piece the material ahead of the cutting edge is subjected to the deformation like this. This deformation is very plastic near the cutting edge and it is elastic away from the cutting edge.

So, when the material is stressed head of the cutting edge beyond a certain limit then material along a particular plane called a shear plane this plane material gets sheared off. And when it gets sheared off it is removed and then it is starts moving over the face of the tool. So, if we see if we see this one the depth of cut and kind of the chip which is being formed in this case that will be significantly different like, yeah.

So, the depth of cut by which the tool is penetrated is this much if we see. Uncut chip thickness or the depth of cut is t naught and after getting miss this is the angle at which material will be sheared off due to the development of the stresses and then it will be coming out. So, the thickness of the chip t c you can see is this. And the angle at which the shearing is taking place is phi and the angle of the tool rake is say alpha.

So, these are the features. So, if we compare here the depth of cut of the depth of cut which was given to the tool is t naught and divided by the thickness of the chip t c, this that thickness of the chip is always greater than the thickness of the uncut chip thickness or the depth of cut which was given because the shearing is taking place at a certain angle.

So, material is strained actually a lot in course of the machining process. So, the ratio of these 2 is now termed as the uncut chip thickness, uncut chip thickness divided by cut chip thickness. Common this ratio is commonly known as chip thickness ratio. So, is small arc that chip thickness ratio is expressed as t naught divided by t c which is always this is always less than one because t naught is always smaller than the t c. So, this ratio is always less than 1. So, in order to this chip thickness ratio is expressed with the help of one equation. So, here like say if the sheering is taking place along this plane, and the length of this plane is equal to say L s. L s that the L s of the length of shear plane, length off the shear plane along which the shearing is taking place. Then the t naught and the t c can be expressed in terms of the L s and the angle phi and the alpha.

So, which we can write as a d naught as a chip thickness ratio L s sin phi divide by L s cos phi minus alpha. So, here this we can after the rearrangement of this one what we get basically 10 phi is equal to r cos alpha divided by 1 minus r sin alpha. This is one of the very commonly used equation for determining the angle at which the shearing will be taking place during the machining.

So, like say this is the tool if the shearing is taking place and this is the chip which will be coming out. So, if the shearing is taking place along this plane, then the shear plane length multiplied by the width multiplied by the shear strength of the material like say f s this combination will give us the force required for the shearing.

So, if the shear for a given for a given depth of cut, if the shear plane angle is low that is this then shear plane length will increase which in turn for a given width and shear strength it will increase the force required. So, the low shear plane angle will result in the greater area to be sheared off. And so, which in turn will be increasing the shear force which is required.

So, and this the this angle phi or the shear plane is found a function of the tool rake angle alpha and chip thickness ratio. So, if we have idea about the chip thickness ratio if we have the tool rake angle then we can determine the shear plane angle. Efforts are always made in order to increase the shear plane angle so that the shear plane length can be reduced which in turn will help in reducing the power consumption or the force required for the shearing purpose.

Similarly, when the material passes through the shear plane it is a strained severely. So, the shear strain induced in the material in course on the sheering. So, shear is strain induced in the material due to the shearing is expressed with the help of one equation which is written as tan phi minus alpha plus cat phi.

So, these equation can be used for calculating the strain which will be experienced by the material when it passes through the shear plane in course of the removal of the material by the shearing action during the machining. So, during the machining basically the cutting edge of the tool presses the material cutting edge of the tool presses the material first to the plastic deformation material which is next to the cutting edge is subject to the plastic deformation one, which is away from the cutting edge is subject to the elastic deformation.

And when the stresses in the material and of the cutting edge exceeds the particular limit of the shearing then it gets sheared off. Now I will summarize this presentation. In this presentation I have talked how can we get the metal removal rate, and how the metal is removed in course of the shearing, and how the shear plane angle can be obtained, and how does it affect the machining process.

Thank you for your attention.