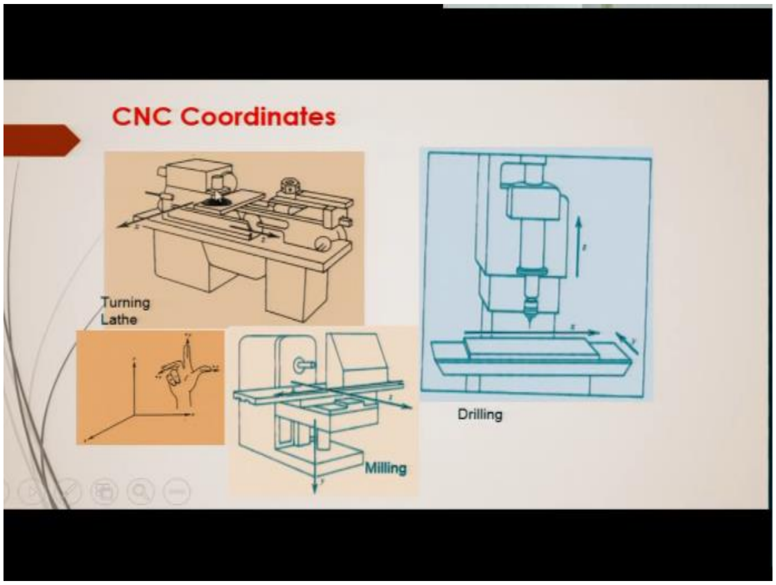
Production Technology: Theory and Practice
Prof. Sounak Kumar Choudhury
Department of Mechanical Engineering
Indian Institute of Technology Kanpur

Lecture – 26
Lab- 03

Hello and welcome to the course on manufacturing technology theory and practice. So, in our previous session we have looked into the conventional machining processes.

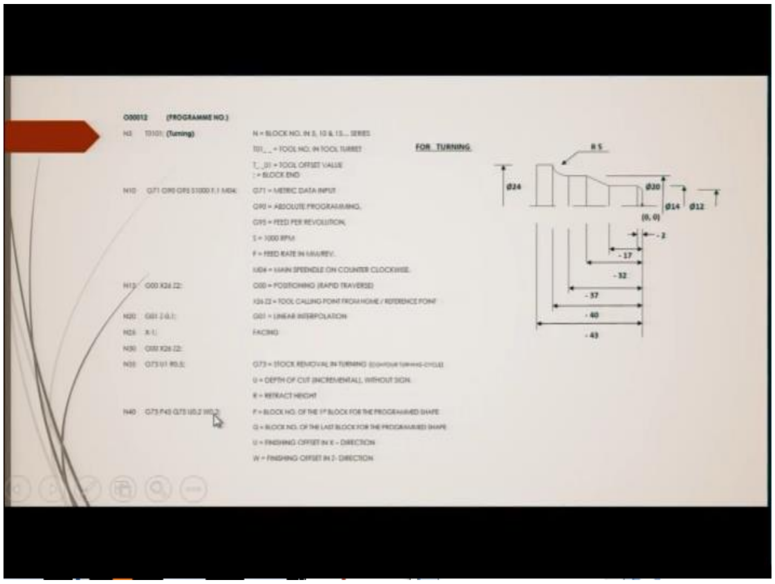
(Refer Slide Time: 00:28)



Let us start making the program, again I am telling you that for the lathe, this is the x, this is the coordinate which is x, this is the coordinate with coordinate z. Similarly, in case of milling if you see this is z, z is in the cross-slide direction and x is in the feed direction that is the direction of the feed velocity and y is vertically located with respect to the spindle if it is horizontal milling.

In case of drilling, it is the x in this direction, y in the cross-slide direction and z in the vertical direction. For some reason, this is internationally accepted and everywhere, wherever there is a CNC machine and the NC machine in service. In practice, this is the axis or the system of axes that is used for turning for milling and for drilling. Now let us come to this keeping this in mindthat we are going to make the program for turning. So, this is x in the feed direction and y in the cross feed direction and z in the vertical direction,

(Refer Slide Time: 01:55)



Keeping that in mind, let us see how to program manually. Well, first of all the programs are made with the blocks, each block has some information about the tool, about the machine, about the cutting parameters and how the machining has to be performed including the coolant on, off, turning of the spindle in the clockwise direction or anti clockwise direction and so on.

So, all details will be given in the blocks and the program is made block wise. So, the block number is given by a letter N as you can see here and we will start a series that is 5, 10, 15it could be 5 next block is N6, next block N7 and so on, but we are using N5, N10, N15, N20 and normally also this kind of a series is used because after the program is made in the simulation if you find that something is wrong and some steps have to be inserted then it can be done in between the lines.

For example, if you have taken N5 and the next is N10. So, in between still you have provision to put N6, N7, N8, N9 but if you have N5 and N6 then of course you cannot insert in between because there cannot be N5 A or N5 B that is the reason why we have taken N5, N10 and so on. So, this is a block number. First, we will take T0101 0101 means the 01 is the tool number. I will show you how these tools are located in a turret head when we will demonstrate the CNC machine.

Turret head contains a bank of tools where different kind of tools are located. Let us say for turning on a turning lathe we can use different kinds of turning tools, single point cutting tools, grooving tools, parting tools and we can also use the drill, boring tool, chamfering tool etc. So, those tools are different tools meaning that each of them serves a particular purpose.

For example, with a parting tool you cannot remove the material on a cylindrical surface. Or for example with a threading tool, you cannot really part or make a facing operation. So, all these tools have to be made ready for the operation and all these tools are kept in a turret head and in the turret head, there could be 10, 15 even more number of tools depending on the size of the machine.

Different kinds of tools along with the tool holder for example, if it is a drill, it will be inside the drill holder or the drill chuck and the entire drill along with the drill chuck will be taken out from there it will be put it in the drill chuck of the machine, we will show you that. So, that can be taken out and inserted in the machine by using a tool changer and that tool changer should be programmed.

So, whenever we are telling that this is T01 that is tool number 1 in the turret, we are also giving the command to the tool changer that it has to be activated to take the tool number 1 from the turret head and then mount it to the spindle. And next 01 you can see that here it is T0101. So, next 01 is the offset value. Offset value is the tool will not immediately touch the workpiece but it will be offset from the workpiece to a certain distance.

So, we are now preparing the machine for making the process here. Next, after that there is a semicolon if you can see. So, this semicolon means that this is the end of the block, here semicolon is the block end, next block starts after that. So, you have to have a number. So, this number is N10 and the commands are given as the G71, G90, G95, S1000, F.1 and M04.
So, these are the codes.

G is the G code, there is a series of G codes from G00 to G100. So, 101 different G codes are there and each code means something. I will show you the table where these G codes are tabulated. You do not have to remember all of these, that is not possible you will always have a handbook while making the program for a particular part for the NC, CNC machine.

So, that table can be readily used. A particular data input system has to be mentioned in the program. For example, here it is G71. So, in this case, it means that we have to use the metric data input, the data that we will be giving all these data will be in metric system, meaning that we do not have to put the unit after the value and it will be assumed that unit will be in the metric system.

So, that is indicated by the G71. Now in the NC and the CNC machine after the program is input that program is read and translated by the machine control unit MCU which is the brain of the machine which will try to understand for example, what G71 is or what G90 it is all embedded there, as soon as the MCU or the machine control unit looks at G71 or some such similar code, it will automatically understand that.

All commands in the program is read and interpreted by the MCU and accordingly this MCU will translate and it will give the signal to the actual machine to the hardware what to do. So, here it is the metric data input. Next is G90, G90 is the absolute programming. So, as soon as the G90 program or G90 code is given, the MCU should understand that this is the absolute programming, absolute programming means the following.

That if we look at this dimensioning in the part, the dimensions are made from one side, you can see that from this line all the dimensions are given like from here to here it is 2 millimeter. From here to here, it is 17 millimeter, from here to here; it is 32 millimeter and so on. Meaning that here all the dimensions are from the base and the base taken as the line which is here.

Why it is that because we have selected this point on that base on that surface as the machine 0 that is 00 coordinate. We have assumed that this is 00 coordinate and that we are telling the machine that we have taken this as a 00. Now, let us see how we are telling that to the machine because in the program so, for we have not seen this, only thing is that the G90 the program the MCU will understand that all this will be based on the absolute programming.

From a certain base and that base it will try to find out where is the 00 where is the machine 0. Now, the next code if you see is G95. G95 is the feed per revolution. So, feed per revolution in the sense that here it is this feed is millimeter per revolution, this is the feed value G95. And this is this feed will be in millimeter per revolution S is the RPM that is the rotation of the spindle rotational frequency of the spindle, this is given by S is the speed.

And speed given is the 1000 RPM, please note that the RPM we are not writing but we are giving the G95 because G95 signifies that this is the feed and feed will be in mm/rev. So, then that means this S will be in revolution although we have already said that this is in the metric data input and therefore, the program will assume it. F stands for the feed rate, this will be in mm/rev and the value is here we are giving the value as 0.1 millimeter.

And then it is M04 M is another code like the G code, M code stands for the miscellaneous code. Here the M04 for example, it is a main spindle on counter-clockwise. So, in the table you can find out what is M01 what is M02, M03 for example, in the clockwise direction M04 is in the counter-clockwise direction and so on. So, each code tells the MCU that this means that the hardware has to do something in a certain fashion that is all.

And all of these codes are available in the table and what each code means is given in the table. You have to just find out that how the tool has to move initially, that will be your decision and accordingly you put that path or the pattern of movement of the tool in terms of those codes which are already tabulated. Let us take the further example in the next code next block we have the N15 which is G00 X26 Z2.

Now, this means G00 this is the rapid travers rapid movement of the tool, since the tool is somewhere parked. And we are not bothered about where it is parked because it is decided by the manufacturer, the tool is safely parked somewhere it is away from the workpiece away from the spindle so that it is not harmed, then our task is to bring the tool to a position where it is ready for making the park you see that we have given the program.

We have given in the program G00 this is a rapid travers of the tool and the coordinates we have given as X26 that means in this axis 26 millimeter and z axis along this z axis it is a 2 millimeter. So that will be with respect to the 0.00. So, if you see that it is 26, 26 means it will be just above the park here it is 20 millimeter diameter, we are putting that safely in that 26.

So, the tool will come from the parking to the safe position. At a point where the tool is still not touching the workpiece but it is close to that. So, it will be at 26 and above this part by 2 millimeters. So, away from the park by 2 millimeters z is 2, z is in this direction this understood here we are saying that this will be the reference point where we have to which point we are we have called the tool from the parking.

Next, we have to give the we have to start making that we are giving the next program next code is G01 and Z-0.1 So, we are saying that this is G01 means the linear interpolation, linear interpolation in one word I will tell you linear interpolation in little more details during the lecture sessions; right now you should understand the linear interpolation when the tool has to move in a linear way.

So, it will go just linear either along the x axis or along the y axis or with both axes together in an inclined path, but the path will be in a linear path. Now, we are saying that Z is minus 1 with this so, it will come from that at a distance which is very close to this that is one millimeter away from here because it is in this direction. Next, we are saying that here it is x we are giving now, minus point 1 that means it will take a depth of cut and it will try to move on X-0.1.

So, at this distance it will remove the material from here and the tool will be moving at x because, if we put the x and z together it will be a tapered surface because both on x axis and z axis the tool will start moving as a result you will see that this is not a straight but it is a tapered surface. Therefore, z is given separately 0.1. So, the tool is brought rapidly to this place and then we are giving a value which is 0.1 in the along the x direction and then the tool will start moving by that also the tool will be removing the material.

So, it will be facing tool will move perpendicular to the axis of the workpiece, it will face the surface. So, this is the facing. Next, we are saying the G00 X26 Z2. So, you can see that this is the same comment as here, meaning that after it completes the facing, it will go back to our reference point. And the reference point will be using as earlier, 26X and Z2 this is a safe position for the tool that we have brought the tool to the reference point from the parking point, next code that is next block is the G73 U1 R0.5.

So, here the 73 is the stock removal in turning and stock removal in turning this is the somehow it is used in the official word of the CNC manufacturer, but this is the contour turning, cycle contour turning cycle is that when the tool will be moving like this, this here along a contour, this is when the tool is moving along a certain contour the MCU has to understand that this would be G73 or rather if we are giving G73 the MCU will understand that the tool has to move along a contour; what is that contour? it has to take U1, U is the depth of cut.

And depth of cut will be that incrementally the tool will take always one millimeter of depth of cut incrementally that means one millimeter it will move along this profile and then it will go up and again come back taking another 1 millimeter and move along the profile and so on. I will show this movement in the machine as well as in the simulator after this discussion. R is 0.5 that means R is the retract height as I said that it will take the one millimeter depth of cut move along the profile retract go back by 0.5 millimeter.

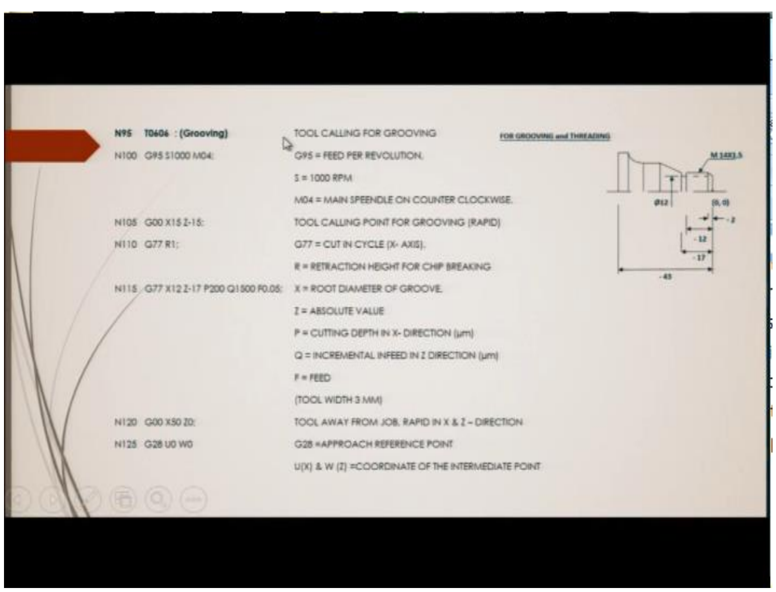
And then again take one millimeter depth of cut. So, all these movements are the different passes as you understand these are the rough passes and in each pass, it is taking a depth of cut of one millimeter incrementally, each time it is moving along this profile, it will take a depth of cut of 1 and R 0.5 which is the retract of the tool. In the next block interestingly just look at it, this is also G73.

Then we are saying P45, which means block number of the first block of the program shape let us go to 45 this is N40, P45 means you go to 45, whatever is the program and whatever is the action given you do that here. Similarly, after that it is a Q75, which means that 75 block of the last block for the program shape meaning that how this shape has to be made these each of these shapes will be given in different programs 45, 75.

And each time this is made, it is supposed to take a finishing offset of 0.2 in the x direction this is you and the finishing offset in the z direction is 0.2, mind one thing this is the finishing offset in x direction given us you this is not the same us in here. Now, of course it can be a little confusing that both are U but U in this case means the depth of cut and here that U means that finishing offset.

Now mind it that when this U will be associated with G73 particularly, it will mean that this is the depth of cut. And if the U is associated with the G73 in the blocks when the blocks are being recalled, then it has to be the finishing offset. So, finishing offset is 0.2 millimeter is taking for the finished cut and W0.2 is the finishing offset in the z direction. So, it means that when the profile is made and the finishing cut will be with the depth of cut of 0.2 in the x direction and 0.2 in the z direction. In both the perpendicular directions that is this direction is the x direction, this direction is the z direction in both the directions 0.2 millimeter has to be taken and the entire profile has to be made as a final shape.

(Refer Side Time: 22:50)



So, the N45 you understand that N45 will be first point coordinate for turning here and here the coordinates are X12 and Z0, Z0 is here and X12 is taken, see here for example, it is X12. So, this is 12 mm diameter you can see that, this diameter is 12 millimeter therefore, this is 12 mm and z is 0 from here we are starting. So, in this direction, there is no movement, but in this direction, it is making 12 mm of the facing. Second point coordinate for turning is coordinate given in the block number N50 as X14 and Z-2.

X14 here Z-2, now it is a movement here because when this was made, Z was 0 when this is made minus 2 has to go and diameter is 14 millimeter. Similarly, the next third point will be Z -17, Z that is in this direction in this direction that is minus 17 millimeters the tool has to move and while moving it is assumed that this movement is for removing the material. So, it will go through the material of the workpiece.

Next command is X20 and Z-32 you can see this minus 32 and 20 so and so on. So that means all these profiles up to this N65 this will be the points coordinates of each of these points. So that the tool can move along this profile, along this profile with certain allowance initially for making the rough paths and then final path with the final 0.2 millimeter of the depth of cut it will give the make the final pass.

Now, next in the 70, it is the GS02 that is X24 Z- 45, it is this we have to give the radius and G02 is a circular interpolation, if you remember, we had the G01 it was the linear interpolation because tool was moving in all these cases up to this point linearly, but from this point to this point the tool has to move in a circular path. So, this is the circular interpolation we have to give the command to the tool that no more it has to move linearly.

But now, it has to go in a semi-circular way, that is the circular interpolation. Now, particularly G02 stands for the circular interpolation in the clockwise direction. So, it will move like this, because as you can see, the tool is here and it has to make a movement like this. So, it will be in the clockwise direction coordinates are given X24. So, you can see that 24 millimeter is the diameter here.

This diameter is 24 millimeter and this distance will be Z-40. So, this is the distance right this is 40 millimeter 40 millimeter is the distance from here to here in the absolute dimensioning and the diameter is 24 millimeters. So, this is the comment that is we have tool has to move a circle make circular interpolation and the coordinates given us 24 40 in the Z and the R5 is the radius of this circular interpolation. Next comment is the G01 this is the last point coordinate for turning here 43 here given, this the Z43 and then G72 is the finishing cycle.

So, from first point to last point, this is the finished G00 and the coordinates given X50 Z50 tool away from the job after it does it the tool will go away from the job and it will move up to 40 millimeters which is more than 43. So, away from the workpiece and similarly, Z also will be 50 millimeter, now, the G28 is approaching the coordinates of the interpret intermediate point is U0 and the W0 these are the intermediate points where the tool will be going.

Next is the tool calling for grooving. So, now we have to change the tool because so for with the same tool we have made this circular interpolation linear interpolation chamfering, but for making the groove we have to have another tool. So, command goes that we are taking the tool as T0606. So, these are tool for the grooving next block is 100 N100 here the comments G code is 95, speed is 1000 RPM, M is 04 is the main speendle in that counterclockwise.

S I have already explained to you earlier, this is the RPM. And the value given as 1000 like it was given earlier also it was S 1000. Next block is G00 that is the rapid movement in the direction in the in the coordinate X15 and Z-15 it is coming some distance away from here because it is 14 millimeters, it is coming up to 15 little more and z also minus 15 in the Z in this direction and X also in the 15 that is in the diameter, diameter 15.

And here if you see distance is 17 and here it is 12. So, it is somewhere in the 15 that is what it says. Next is G77 R1 this is caught in cycle in the x axis, x axis is in here. So, this is to be these are grooving tool we have already taken GT06, this grooving tool has to move to make that groove in the direction perpendicular to the axis of the workpiece that is in the direction of the x axis.

Therefore, we are giving the value as X12 because as you can see the diameter is 12 millimeter and Z is minus 17. Here is the value of the Z P200 Q1500 F0.05. Now this P and Q associated with the G77 which is the cut in cycle, G77 means along the x axis the cutting process is in cycle, x axis is here, that means the tool will move like this only. So, in this we have to give the cutting depth and for the parting tool depth is very small because otherwise the tool will break you understand that. And it is a parting tool as in case of parting the workpiece.

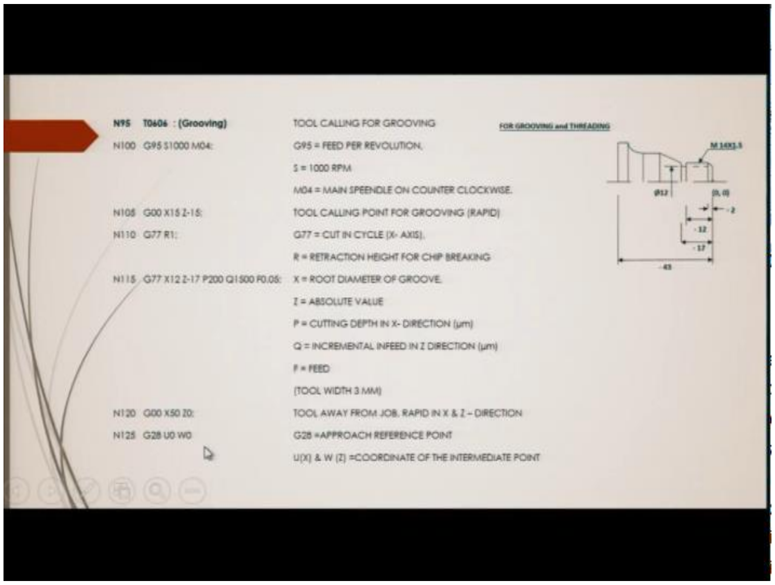
So, tool will move like this and each time it is taking the depth of in the X direction as 200 micron and incremental feed in the Z direction as it is moving it will also move in the Z direction to make the width because here there is a certain width of the groove. To make the width it has to have the incremental infeed in the Z direction that will be in micron also. So, each time it will move at 1500 micron and then F is the feed taken as 0.5 for the tool, tool width is 3 millimeters that has been assumed here.

So, exactly 3 millimeter will be made here in case there is no feed given but depending on the width given for this groove, you have to have the feed supplied to the tool. Next is tool away from the job which is given us G00 rapid movement, this is in the block number 120 and we are keeping it to the X50 and the Z0 this is the approaching the reference point. then we are next is G28 approach the reference point.

And here it is the coordinates and these coordinates are the same as we have taken the tool here for example, if you see the N90 this is also U0 W0 the same way we are taking the tool to the reference point. Now next is the threading. So, this threading is M14 into 1.5 now we have made the groove and the diameters of 12 millimeter then you have to make the thread. So, you understand that nuance that why these threading is done after we make the groove.

See if we did not have the groove in that case making the thread would have been difficult because tool that is used for making the thread needs an exit from here after it is cutting the thread it needs an exit so without that groove that exit of the tool we could not get. So, therefore first the grooving is made groove is made and then that thread is made. So that the tool can have an exit. So, for threading we are taking another tool which is 03 0303 will be the trading and here we are taking in the next block G95, 5S200 and M03.

(Refer Slide Time: 33:16)



So, G95 we are using it here that is the feed per revolution the same will be here feed per revolution and speed is 200 RPM and M03 is the main speendle in the clockwise direction that means the right-hand thread for clockwise direction when the main speendle is moving, we are having the right-hand thread cut in the along this length of 12 millimeter. Next is G00 rapid movement in X14 and Z2 tool calling point for the thread cutting.

We are calling the tool but it is rapid because tool at that point when it is coming to this point while coming from another point it is not removing the material so the it can come at a rapid movement very fast so that the time taken is less. Next point next comment is G78, G78 is comment means according to the table, it is multiple threading cycle that means it will cut the thread. And then it will again cut the thread going back again coming back so it is multiple threading.

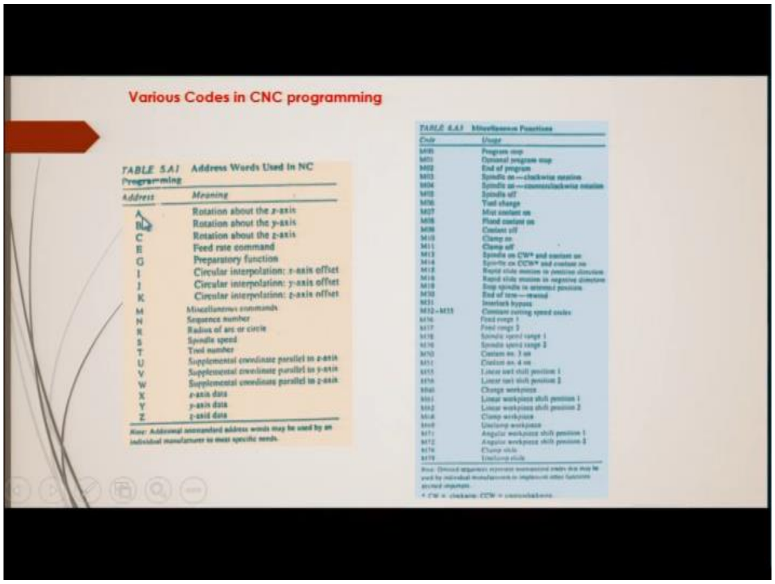
So that each pass will be a rough pass and finally the depth of the thread can be achieved, that is given as the G78 and P 030660 is you can see P means P03 is the finishing cut. There is a comment. Now it is 06, 06 is the chamfer value, when the tool to be retracted so the tool is cutting and then it has to be retracted then again it will come back and again go back to cutting the same way with the different depth with an additional depth of cut.

So, this is given here, flank angle of the tread is given as 60^0 here flank angle of the thread is this. So, this the angle which is the flank angle of the thread equivalent to the cutting angle of a tool see the tool shape will be conformed to the thread.
So, if the tool is having for example, a shape like this, the thread also will have a shape like this meaning that this tool is moving and it is moving towards this direction as well as moving towards this direction along this profile.

So, this angle is the flank of the thread. Now, this angle is specified. Q50 stands for minimum cutting depth which is incremental in micron that means how much depth it will be taking minimum, not more than that, incrementally. Incremental system I already told you that each time it will take once and then again and then again and it is incremental. So, then R is 0.01 which is the finishing offset that is also incremental, the last cut will have with an offset of 0.01 then we will have the G78.

Again, the multiple threading with the root diameter of the thread, thread length is given minus 14. So, because it is the M14. And R0 is the incremental taper value we have already said that here, Q value we have already said cutting depth for the first cut that is in micron then we have the P805, this is the thread depth 805 will be the micron and F is the pitch of this of this thread from one pitch to another pitch is the pitch of the thread in one for 55 block after the 150 it is a rapid movement along the coordinate X50 Z50 that we are taking that in the shape and direct shape position.

Then the next block we have the approach reference point we have taken that here like in 125 like in 90. So, then it is coming to the reference point and then it is the end of program is M35. So, M30 stands for the program ends that means now the tool has to stop and the machine will be switched off according to this program as M30 machine will understand or MCU will understand that the comment is given that after that no other operation will be performed.
(Refer Slide Time: 38:17)



Well, as I was telling that there are different kinds of codes here you can see the M codes. So, M codes are also many which are the miscellaneous function. And each one of them stands for something for example, M00 is the program stop. M01 is optional programs is optional program stuff. So, miscellaneous program is that you are preparing the machine each of them you can see that for preparation of the machine how the machine will start making the process.

Here the address words used in the NC you can see that these are 26 alphabets of English and from A to Z each one of them mean something for example, A is the rotation about the x axis, B rotation about the y axis, C rotation about the z axis. Now, these are the tables which are standard tables which are international standard and any machine used in any country of the world will understand the same codes if the system used is the same.

There are different systems used and the one that we will show it to you is the FANUC system which is a system made by the Japanese and in that these codes which are embedded meaning that if you give the A for example, the machine will automatically understand that this is the rotation about the x axis and so, on. So, here you can see we have used that S you remember S we had the spindle speed, we have used the U which is the supplement coordinate parallel to the X axis and so on.

Here it is X14. So, along the X axis it will be 14 millimeter, X axis is here along this and so, on. If it is Z meaning that along the Z axis, it has to be the movement as per the value given. So, these are the values which are given here that are the X axis data and so on. Similarly, you can see the M codes. So, what I mean to say by showing you these tables is that all these are available with you and you do not have to really remember each of them.

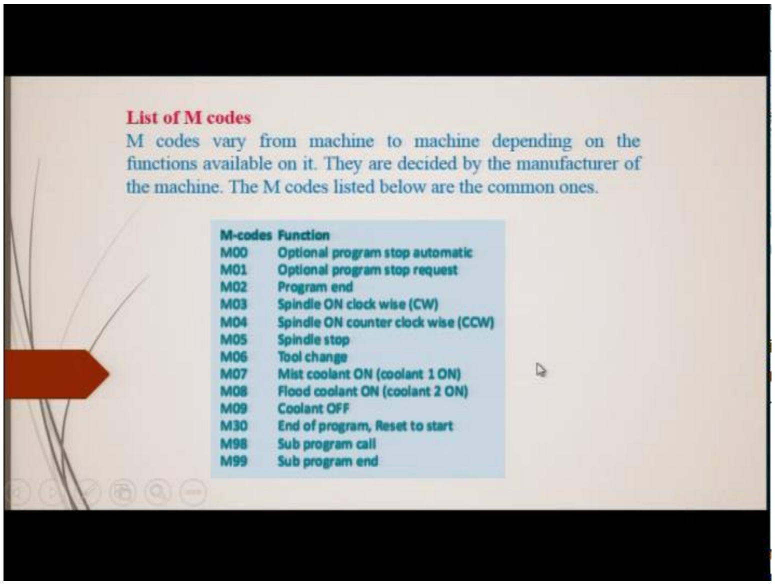
But you can always use these tables while making a program, suppose you have to move the, you have to rotate the tool rotate the workpiece along the X axis. So, you have to use the program in the program A and so on. So, you can find out when you will make the program very frequently you will get used to some of the codes and you will remember some of the codes; I mean G00 probably you already remember which is rapid movement.

Because we have used it so many times; here are the G codes. As I was telling that G00 this is a rapid linear positioning. So, rapid movement of the tool and similarly from G00 to G99 these are the 100 codes these are popularly known as the G codes and these G codes are used for some functioning or other, for example, clockwise circular interpolation will be G03, clockwise circular anti clockwise, clockwise circular interpolation with G02 linear feed interpolation will be G01, and so on, so, forth. So, all these are given by the G code and these codes are embedded in the program.

(Refer Slide Time: 42:11)

As the program is given using one of these codes, MCU which is the machine control unit will read and understand what it is required from here from them and accordingly it will send the signal to the hardware to the machine that what to do. How to move along the x axis or z axis? So, accordingly the signal or the voltage will be given to that particular prime mover which is connected to either x axis or z axis to move the x axis and z axis according to the program or moving both the axis together to get a taper surface or inclined surface and so on.

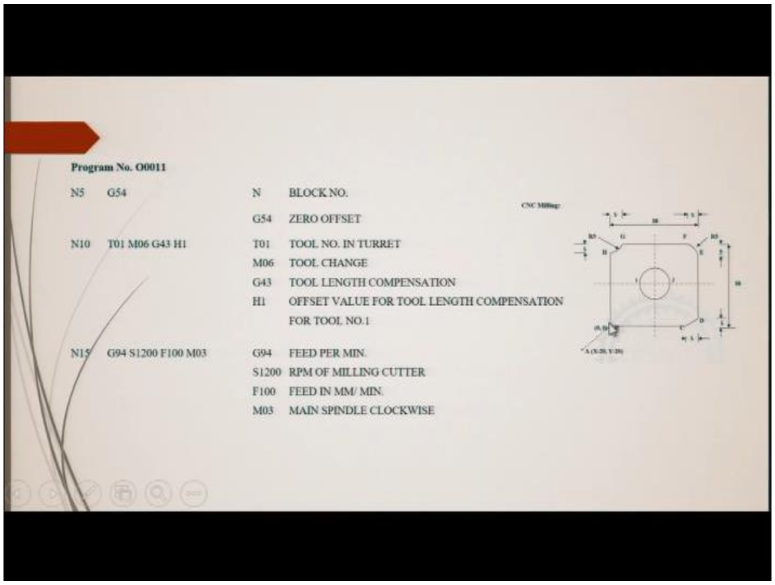
(Refer Slide Time: 42:54)



List of M codes vary from machine to machine depending on the functions available on it. Here are some of the popular M codes. Functions M, as I said, that these are the preparatory functions they are decided by the manufacturer of the machine. The M codes listed below are very common for example, spindle tool changes M06. If you have to stop the spindle it will be M05, program end is M98 sub program call M99 is a sub program end and so on, so if you have to end the program, it is the M99 or M98.

End the program is M30 that we have already seen that is the end the program, reset to start, coolant you can put on or off for example, if you have to put the mist coolant this is M07, if you have put the flood coolant it will be M08. So, you understand that different kinds of coolants can also be used to switch on or switch off by using one of these codes and these codes are understood by the machine control unit.

(Refer Slide Time: 44:10)



So, whatever we have seen so far is the example of a turning, that means, we are using the turning lathe for making this. So, it is the CNC turning lathe; similarly, we can have the CNC milling; if you see this slide, here we have one part which is a very simple part for that matter, which has the A tool goes to B, then C, then D, E, F, G and H. So, this is the movement of the tool which has to be made which has to be programmed with a circle in the middle.

So, if you see now, I will go a little quickly because this will be to a large extent repetition of some of the codes which you have seen in the case of the turning for example G54. So, block number and it is the 0 offset, earlier we have given some offsets, some value was given, a tool number turret is 0.1 we are calling here in this case it will be milling cutter and M06 is the tool change.

So, tool has to be changed to 01 G43 is the tool length compensation, this is something which I have to explain it to you it was not there in case of the turning that is the tool in this case it is a milling cutter. In the case of the turning, it was a single point tool. So, the tool point was exactly touching the workpiece; in this case what happens is that milling cutter has a diameter.

And all the movements of the milling cutter along the profile, for example here the milling cutter will move from here to here, let us say from B to C then here, then here and so on. So, this movements of the milling cutter is, as far as the axis is concerned, this is the movement of the axis that means, what the periphery will be moving, it is the radius of the milling cutter - that much compensation we have to give.

We have to give for the movements that we are planning. So, at the periphery away from the axis of the milling cutter, that much to be compensated, that is the along the diameter. Similarly, we have to give the compensation along the length, along the length means suppose, we have again the tool which is the end mill cutter. So, when this tool is touching, it is the periphery.

What is touching is again not the center and to have the center, assuming that it is touching this point, we have to give some length compensation. So, that is given by the G43 either it is a length compensation or we will see later on that it could be a diameter compensation as well in this case it is the length compensation that we are giving which is given by G43. The length compensation I will explain in a different way which is popularly explained in our lab to the students.

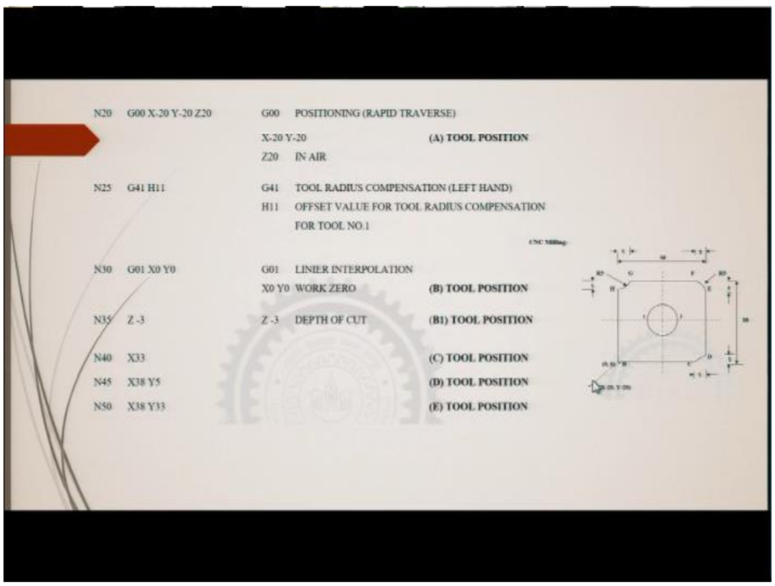
Let us say we have different kinds of lengths in the milling cutter, this is one length which is bigger, this is little smaller, but it is still bigger than this and so on they are all different. Now, when these tools will be touching the workpiece surface, suppose we take the reference point when this tool which is the largest length is touching. In that case for this tool, it will not be sufficient, it will be less and so on.

Therefore, each tool is touched and that point for that particular tool is recorded as 0. So, that will be expressed as the tool compensation or the offset. So, this is the tool length compensation and offset value for to length compensation is H1. So, this is some kind of a value which is given to the tool offset. So, that the tool length so that the length can be compensated. In the next module next block N15. This is G94.G94 is feed per minute, if you remember in case of turning it was feed, it was per revolution, here it is milling.

And as you understand that feed is given in feed per minute millimeter per minute. Therefore, the code is G94 in case of the turning, the code was different it was millimeter per revolution, this RPM is given for the milling cutter; feed is given 100 for the millimeter per minute. So, as you say F100 it will understand that 100 will be mm/min and then M03 is the main spindle moving clockwise.

Further I will discuss in our next discussion session. However, you understand that the programming principle will be similar to the one that we have discussed in case of the turning that is we have to ultimately move the tool along all these points and come back here, so that the entire profile can be made.

(Refer Slide Time: 49:50)



We have the N20 block, N20 block says that, we have to have the rapid positioning of the tool, rapid movement is given by G00 and coordinate is given that is X20 Y-20 and Z- 20. Mind it this is milling cutter movement. So, it is on the milling machine and in the milling machine, the z direction is this x direction is this and y direction is the across like coordinate system. Do not get confused with the x y z of the turning, in turning we have the x and z, there is no Y movement.

And here we have the x y z because the table movement with the workpiece can be along the X and along the Y and along the Z axis, the tool will be moving that is a spindle with the milling cutter we will be moving, so here it will be all x y z. Now, Z20 in here and this is the X20 Y20 the tool is positioning here at this point. So, this point coordinate is A and along the Z it is 20.

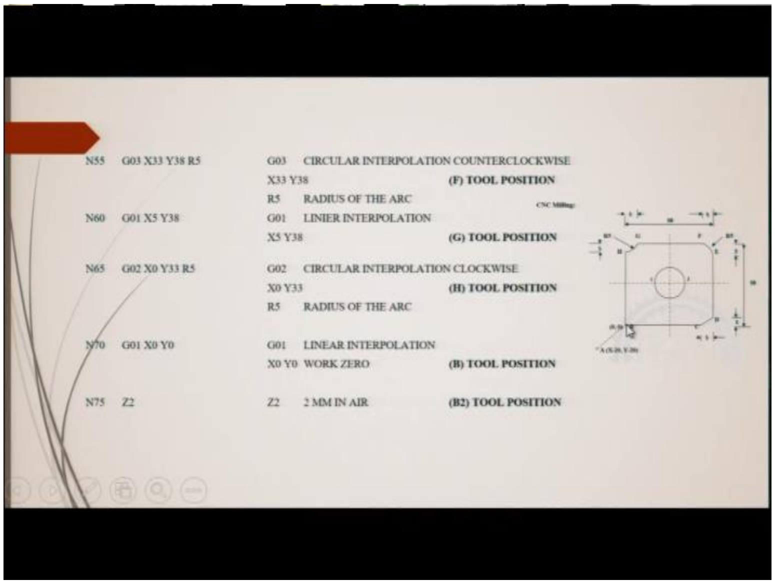
So, it will be just above the point above the workpiece. It will not be touching it is on the air and it is away from the workpiece by 20 and 20 in the x and y coordinates accordingly. Next block is G41 H11. G41 stands for the tool radius compensation. And the tool radius compensation is I already told you that the periphery is cutting, but the dimensions are given with respect to the axis.

So, the compensation has to be given and H11 is stands for the offset value for the tool radius compensation what is the value for tool number one. In block 30 given G01 which is the linear interpolation we have seen that earlier and this is X0 Y0 this is positioning B. So, let us say we have made this point as the machine 0.

If you see the diagram here this is the point on the workpiece which you have made as the machine 0 that is the 0 for us whereas manufacturing 0 or the manufacturers point reference point and then intermediate point then the parking point and so on, that is, there are differences. We have made a point, although of we could have made it here, that depends on you, in that case you have to make the machine understand that this is the 0 point then with respect to this all the dimensions should have been given.

You understand the difference that you are made this is made by you the user, user is saying that I will make for me it is convenient that this point should have been 0, in that case, based on that line, or this let us say here also you can see that this is 00. So, based on this line, which is this surface, all the dimensions are given in the absolute you know dimensioning similarly in case of milling, here we have made as a 0 ? And this is the tool position X0 Y0.

(Refer Slide Time: 54:22)



Now next block says move Z-3 is the depth of cut and this is in here, it will go to the go little deeper again with the 3 millimeter of the depth, this is the depth of cut given this is the tool position which will be let us say B one that means it has gone below the B with a 3 millimeter of depth. X33 in the 48 block this is the position C, so you can see that this 33 is the dimension 38 - 5. So, this will move from here to here, it will be 33 millimeter.

So, this is what it is given as X direction, as I said earlier also that this direction is plus. So, as the tool is moving from here to here and mind it this is the axis of the tool, we have already given the radius compensation and the length compensation. So, according to that, we can assume now, that the periphery of the tool that is the tool cutting point in the milling cutter is removing the material and it is moving while moving from B to C and this distance is given as 33 millimeters.

So, now, the position of the tool is C. Now, from C to D to move it X38, this is the tool position here with the linear interpolation X38 and y = 5. So, this is the 5 millimeter it will move in here and this will be the D position of the tool. Next is the X38, again the linear interpolation, it will go from here to here and the distance is again 33 38 -5 from here to here, from here to here 33. So, then the next position from here to E, which will be 38-5.

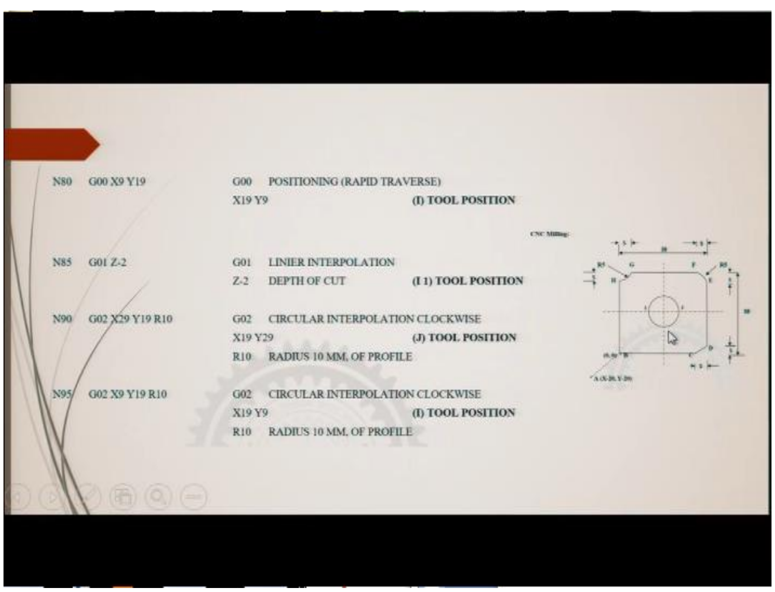
So, this is the 33. Now, next block is N55 and the command given is G03 which is the circular interpolation because from E to F you can see that there is a radius given which is 5 millimeter.

So, the tool needs to have a circular interpolation, like we have seen in the case of the turning where we had the semi-circular part of the tool. And the tool position will be in F when the coordinates will be given as X 33 and Y38.

That means you remember that all these dimensions will be taken with respect to this point, when it is going to F position then this distance will be from this point. And from this point, this distance will be 38 -5 this is 33 and y is 38 given here, so, this is the position of the F after the circular interpolation, then it will have a linear interpolation with X5 and Y38, it will come up to this.

And then we will have the circular interpolation here with the R5 and the coordinates will be X0 because if it is coming here, then X will be 0 and then Y will be 33 this is the 33 and then the tool position will be H, R5 is given already for the radius of the arc and in the next block we have the linear interpolation again from H to B this will be when it is coming to this, this is the final point which will be giving from the H, so it will be 00 because 00 is coming back to the initial position. Next is the Z2 that means from here the tool is moving up at the air and it is tool position is somewhere the B2 which is away from the workpiece.

(Refer Slide Time: 58:01)

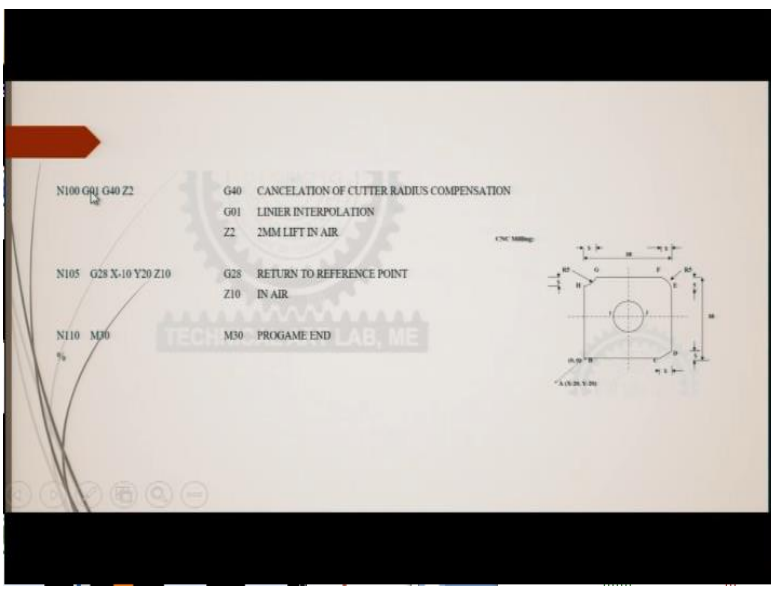


Then the rapid positioning it is going to the I position, which is not shown here it is in the Z direction. That is why all right and then it is the depth of cut linear interpolation and so on. So, what is shown here in the next command is that from here, the tool has to go to this point. Tool has to get a depth of cut and the tool has to make a movement like this. Then from J it has to go to the counter-clockwise.

Here it is clockwise, the clockwise and then it will be a hole will be drilled at the center. That is why it is given that given that the linear integration will be done with Z depth of cut, it will go from here to here linearly then there will be a circular interpolation clockwise. And the tool position will be somewhere in J, because it is coming from I to J and then it is again going and the circular interpolation clockwise from J to I.

And again the position will be I with a coordinate X19 and Y9. So, these coordinates will be X19 and Y9 you can find out from the diagram or it has to be given in the drawing, R10 is that tool with a radius of 10 millimeters. This is a profile tool which is used to make this round shape or the drill hole.

(Refer Slide Time: 59:41)



Then we have the rapid movement. So, before that we have the 100 is a G01 linear interpolation G40. Now we are canceling the radius compensation which is done by G40 and 2 millimeter we are lifting the tool on the air. Next is the return to the reference point and Z10 will be in the air. So, it is gone and the parking meaning that it has to go from here, after drilling the hole it will be lifted up and it will be taken to the safe point.

So that the tool can be parked. Overall, you can see that this has been made already with this program, if you make it manually and then this manual program will be given to the machine and you will see that according to the program that is made the tool will be moving along these points and this entire profile is made. So, you can use an actual tool, an actual workpiece with a certain material.

And we will show you in the machine how such kind of programs can be written, can be run and can be utilized to fabricate the part, how this profile is made. I will discuss in more details in our next class. Thank you for your attention.