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

Lecture - 33
Lab - 10

(Video Starts: 00:23)
Hello and welcome to the course on manufacturing technology - theory and practice. We are in the simulator room. So, here we have the simulators which are from the EMCO company from where we have purchased the machine and as I told you that after you make the program manually in a piece of paper, that program has to be tested. To make sure that there is no flaw in the program so that program can be put in through the run, through the simulator.

And simulator will show you whether the program is right and the relative movement between the tool and the workpiece goes on without any conflict after you get the result; then only the program has to be brought to the machine and the machine will then fabricates the parts. So, this is the simulator and now we will show you how each of the lines in the program can be made here in the simulator and how that can be executed.

Now you can see the screen where we have to bring that page where we will start typing. So, that new page will be now open; so this is a new page, fresh page and you can see that the program name is given as 01111, that is the name of the program. So, here you are writing that 10101 that is the N5 here you have the T 0101 that is for turning. So, N is the sequence number that can be taken as here it is N5.

So, it can be taken as 10, 15, 20 and so on T 0101 is calling the turning tool; it means that we need a turning tool and that is being called for in the N5 first and after that the semicolon is the block end that means here we are calling the turning tool and that ends the block. Now the next is the N 10 this is the next series that will have the G, G90, G95, S1000, F0.1 and M has to be 4.

So, what does it mean? First of all, that M series is the number 10 sequence 10, so first one was 5 then we are using the G code and particularly the G71, this stands for that we are

working in the matrix system. So, we are telling the machine that we will be working in the matrix system and whatever dimensions will be given that will be the matrix system.

Next to that is the G90 this is the absolute programming I told you already about the absolute programming that when the all the dimensions are from one base either it is from one side one axis or from another axis and all the dimensions will be from there only after that it is G95 which is the feed rate and G95 make the machine to understand that we will have the feed that would be in the rate in revolution that is some feed value you have to take.

Now next to that is the S1000 S is the RPM 1000 is a speed and F0.1 is the feed rate in millimeter per revolution next to that is the M04, M04 is the main spindle on the counter-clockwise meaning we are defining whether the spindle will be rotating clockwise or counter-clockwise. Next is N let us say 15 we will have the G00 and X26 after that will be Z1 so in N15 we have given G00 X26 and Z1.

And again, every time we can see that at the end of the block there will be a semicolon. So that will mean that this is the end of the block. Now G00 is the positioning the rapid traverse, that is, we are positioning the tool by the rapid traverse in the tool meaning that from the 0 position we are coming to a position rapidly since the tool is not removing any material.

And that way we are indicating that G00 is the program which tells that this is for positioning the tool and that traverse will be rapid next to that is X26 Z1 that means the first tool calling point from home that is the reference point we are setting the reference point by giving that X26 Z1 that is the coordinate and after that we have the N20 which is G01 code we are using the linear interpolation which is how the tool has to move and that has the Z - 0.2.

So, this is the value that we are showing in the Z axis is Z axes is a particular direction that it takes and I have already told you those axes the directions in the turning it is one, in milling it is different. N25 is the next and so on. So, the entire program is made. I will call the entire program and you can see that in the N25 next to that is the X - 1 that means this is given as a facing cycle.

X means it is in the direction perpendicular to the axes of the workpiece. So, at this moment we are calling the entire program which is already there. So, you can see that in N25 here this

is the G01 and X - 1 so G01 is the linear interpolation this is remaining. In fact, if you do not give the G01 it will take the same from the N20 and here it is given X - 1.

So, you are facing, facing means that the tool is moving perpendicular to the axes of the workpiece, this is the turning, in case of turning now N30 it is given G00 X26 and Z2. So, this G00 we have already told, this is the positioning and that coordinate given as the X26 and the Z2 and that is the end of the block; next is N35 this is the G73 U1 and the R0.5 so G73 stands for the counter turning cycle.

Now in the lectures through the power point, I have already told you that there are codes and the table will be with you, meaning that in the table all these codes, what each code stands for will be given to you. So, the G73 here means that counter turning cycle if you remember we had the main spindle counter-clockwise was the M4 earlier.

So, here we are saying that the counter turning cycle and I will show you how it goes when we will simulate it then the U1; U means the depth of cut and U stands for the incremental this will be 1 incremental. So, R is 0.5 so U is the incremental and 1 is the value that incrementally you are giving the depth of cut and the value is 0.5 R is the retract height after depth of cut finishing.

After that you have the N40, N40 is the G73, P45, Q75, U0.2 and the W0.2. So, G73 is the block number the first block the G73 we have already told you that is a counter turning cycle and then you have the P45. So, the first block for the program that is the block number and you are going to the 45. So, this is bringing the code that we have written later, that can also be done.

So, here it is written P45 meaning that you are bringing a block which is by number 45 the number of 45 will be coming later but that can be done here. Now after that it is a Q75, Q is the block number of the last block for the program 75 is here. So that means it will take this line G01 and Z - 43 this is from the N75 and then we have the W0.2, W stands for the finishing offset.

And that has to be in the Z direction and this is the 0.2 after that we have in N45 is the G01 X12 and the Z0 so this is 12.159 and the Z0. So, first point coordinate for turning now the

G01 we have already told you that this is the linear interpolation and the coordinates are given as the X12 and the Z0. Now the next N50 this is G01 that is remaining that is the linear interpolation and the coordinates given as X14 Z - 0.2.

So, these are the coordinates which dictate the tool how to move from the previous position for example in previous position it was N45 and the coordinate was 12.159X. So, now it is changed to 14 in the N50 that coordinate in the X direction, X is perpendicular movement of the tool, perpendicular to the workpiece axis and the Z is - 2. So, it has a direction plus or minus depending on whether the tool is moving to the right side or to the left side.

And Z is the movement along the direction of the axis of the workpiece in case of turning. Now next is the N55 is given same linear interpolation and the coordinate is that Z is remaining, Z is going to only minus 17 next is N60 and N60 block. So, there it is X G01 that is remaining as linear interpolation X20 and the Z - 32. So, these are the coordinates given X and Z as we have seen earlier also like for example in N50 it was X14 and Z - 2 in N50.

So, here it is X20 and a Z - 32 so this coordinate is changed next is N65 again the linear interpolation remains and the Z - 37 this is the coordinate along the Z axis and Z axis once again, it is along the axis of the workpiece next is N70, N70 is the G02, G02 we have changed now from linear interpolation we are going to the 02 is a circular interpolation and whether it is counter-clockwise or it is clockwise that is actually given by G02 or 03.

And then the coordinates are given X24 Z-40 and the radius is 5 since it is the linear interpolation and these are clockwise so it will be moving clockwise direction with a radius of 5 and the 5 unit it can be millimeters we are using matrix system and the coordinate will be X24 and the Z - 40. So, meaning that within this coordinate what is dictated is that the tool has to move along the X direction and the Z direction simultaneously.

But in the X direction, it will move only 24 units whereas in the Z direction it will move 40 in the negative direction, that is minus. Next is N75, again we are coming to the linear interpolation and that coordinate is given as the Z - 43. So, in the opposite direction minus direction the Z axis along the Z axis that is along the axis of the workpiece it has to move by 43 units. Then it is N90 G72, G72 is a program and it is the P45 and the Q75.

So, we are going to tell you accordingly how this program is made and I have already described in the class through the power point presentation. So, similarly, you can find out from the table the meaning of each of these codes, for example what is T 0606? So, what is a tool? That is a tool G95 what is stands for G77, what it stands for G28, for example in N90 all those things can be found out from the table.

So, when you are making the program, you have the part and it is not so difficult because you have the tables from where you can find out the G code M code and the values, coordinates you can find out from the dimension given in the drawing; so now the entire program is made and as you understand that this programming itself is not a difficult task because you have all the codes in the table.

And you can figure out which code stands for what and which code is required for that particular program because you have the drawing with you. So, you know how you have to give the command, how the tool has to move from one point to another point depending on the dimension given in the program or in the code that you have generated. Now this can be simulated so for simulation the entire program is given as an input, the various factors are selected in the simulation program.

And here what you can see is the blank already simulated, the blank shows that the blank is mounted in the 3-jaw chuck and you can see the dimensions where the diameter of the blank is 26 mm, 70 mm is the length and so on. So, now the machining will start you can see that this is in the 3 dimensional, so, the tool is positioned, just touching the workpiece this is all according to the program.

So, the program is made in such a way that the tool has to move from that point to the next point and you can follow that according to the program. So, these are the different passes the tool is making and it is turning, making the cylindrical surface as per the drawing and as per the program given. So, you can see that the chamfering is made, the cylindrical surfaces made, the taper surface is made.

And after the taper surface there is a flat made and finishing so there will be few passes of rough turning and the final pass will be of the finished turning and the entire part will be

made. So, after that we have the cylindrical surface, conical tapper surface and so on. So, next is the operation according to this drawing you have to have the groove.

So, the tool is fed towards the axis of the workpiece, perpendicular to the axis of the workpiece and each step of the program is being executed; this way and you can see that this is made so this means that the program is right there is no flaw in the program and then the thread is being cut; so, there are few passes the final depth of the thread is obtained by few passes of doing the rough and finished turning finally.

So that is the whole program is being executed and you can see the entire part is being made according to the drawing and this is the one that we have actually planned so this is how the drawing is made. Now we are going to demonstrate you how this milling in the CNC the program can be simulated. So, we have made a program let us say this is the program according to the drawing and I have shown the drawing to you.

According to the drawing we have to make the flat surfaces. We have to make the round surfaces and so on. So, let us see what we have written so this has to be made by us according to the drawings. So, first line is the N5 that says G54 is a block number N and the G54 is the 0 offset, 0 offset means that you are bringing the tool to the 0 position and from where you are starting from where all the coordinates will be taken.

N10 says T01, M06, G43 and H1 so T01 is a tool number like it was in the case of turning tool number 1, 01 which is in the turret we will show you what is the turret, turret has all the tools located there and each tool is numbered. So, from the turret the tool number 1 has been selected M06 M is the tool change and from the turret. So, whatever tool was mounted earlier that has to be changed.

And the tool number 1 has to be called from the turret and it has to be mounted. So, then the G43, G43 stands for to length compensation tool length compensation in this case is important because here we are using the milling cutter because it is a milling operation; in milling cutter the coordinates are given with respect to the center and different milling cutters may have different diameters.

Therefore, that compensation is required so that the coordinates can be considered from the center of the milling cutter and not from the peripheral point next is H 1, H1 is the offset value for tool length compensation for tool number 1. So, what is the value that has to be given afterwards N15 is the G94, S1200, F100 and M03. So, N is the number that number is the 15 now so G94 is the feed per minute.

So, we are defining that this is feed per minute in case of turning it was revolution per minute RPM it was the N and the feed was given in millimeter per revolution here this is the milling cutter. So, here the feed has to be taken in feed per minute so this is G94 and S1200 this stands for the RPM of the tool and the tool is 12 mm diameter we have selected for the aluminum.

And that RPM will be 1200 revolution per minute; next is N20 and N20 is the G00 that is the rapid positioning X coordinate is minus 20 Y - 20 and Z + 20 is a coordinates X, Y, Z. So, G00 rapidly it is positioning from the 0 position to these coordinates and all these X, Y and Z in all these 3 coordinates values according to the values given here minus 20 minus 20 and plus 20 that tool has to move simultaneously.

So, initially I told you that difference between the NC and the numerically controlled or computer numerically controlled and the conventional machine is that in the CNC and the NC machines we have the individual prime mover of the motor along the axis, that is the X, Y and Z individually can be given motion, movement. Therefore we can have different kinds of relative movements between the tool and the workpiece to generate various surfaces that is the basic difference.

Here you can see that rapidly it has been brought to this coordinate and then there are different other programs, for example in N21 it is G43. What does G43 means that you can find out from the table that has been given what is that H stands for and the 11 is the value given, so what I mean to say is that you know that G codes and what each of them stands for.

And then knowing what the other codes stand for you can always write a program like that on your own depending on the drawing that you have and that can be executed. So, after that we have to have the simulation like we have done it in case of the turning this program will be

simulated to see that the program does not have any flaw and the relative movement between the tool.

And the workpiece is right which is being executed according to the program which is given here. All the parameters will be set like it was done in case of turning. Here is the drawing which shows that the blank out of which you will make the workpiece is clamped on the table of the milling machine, this is the workpiece these are the clamps and all these heights are given here.

So, right now it is in the 00, now it will be made. The X, Y, Z coordinate is changing you can figure out from the program that we have written we have given you can find out how much X how much Y how much Z the tool is moving and should move and see that movement of the tool in the red it is shown and the white it is the workpiece, the workpiece material removed.

And it is exactly moved to make the workpiece that is given in the drawing and you can find out that it is according to the drawing the tool is moving, so, you can be ensured that the program is right and if the program is put in the machine where the tool will be moving the same way as in the case of the simulation. So, now just an example the same program that you have seen for making the part in the CNC milling.

So, the same way it can be made by a program; instead of making that program manually we can generate that program using the software which is called the ‘fusion’ as I was telling you and in the ‘fusion’ the first thing that you have to do is you have to make the drawing. So, the drawing has to be made according to the requirement that is given in the drawing that is all the dimensions all the tolerances everything.

And then you can see we are running this video and you can see here step by step how the process is made how from the very beginning the drawing has been generated and then you will see that from that drawing how the codes are generated. So, once the code is made that is the entire program that I have shown you that we can make manually, the same program will be made here for the CNC milling of this kind of a part.

That can be interfaced with the actual machine and in the machine then the relative movement between the tool and the workpiece will be executed according to the code which is generated here by the fusion program and the part finally will be manufactured. So, now we are running that video it will be very quickly made and you can see the steps. So, you run that these are the steps that how the part is being generated you can see that in the video.

This is the part which is made from a block this is the 3-D picture and now the code has to be generated. So, you go to that 'manufacturing', this is the 'workplace' so according to that you find out what are the coordinates, select the coordinates in the manufacturing area of this program. So, you put the system of axis at a certain point which will be the 0 point.

Selecting the tool for manufacturing you have to select all those parameters 'tools'. So, for that there is a bank of data and from where you are selecting the right kind of tool which is required for your choice. So, depending on your choice your tool will be selected and the command that you have given; so, this is the geometry which is selected that has to be fabricated; these are different values which are being selected.

So, this is the simulation that is, how the part is to be generated for holding the tool. Now you can see how the tool will be generating the surfaces; this is the movement of the tool with respect to the workpiece step by step and the final surfaces are being made this is the central hole which is being made so this is generated. So, depending on the surfaces that have been generated now the NC code will be made by this program.

That can be executed in the machine. So, we will save that program in a folder, in a file and that can be called and you can see this program, these codes are made this is the program that otherwise you would have made manually and we have shown you how manually we have made this kind of a program. So, you can see that if we have the drawing and if we have the fusion program, then the fusion program itself will generate the entire code and that can be interfaced with the machine and the interfacing will be demonstrated to you now.

So far what we have shown you is how to make the program for the computer numerically controlled machines, how to make the program manually and how to make the program using the software. Now those programs have to be put into the machine so that the machine can operate according to the program that we have made. Now that program also, I would like to

remind you we have shown you how those programs to be tested before we put it in the machine.

Because if there is any fault in the program, if there is any contradiction in the program in that case there may be an accident because there may be some clash between the tool post and the spindle and they are rotating at a high speed. Therefore, we always test it, simulate it and then when we find out that the program is flawless then that program has to be put into the machine. And then the machine will automatically make that part.

So, here is that CNC turning lathe as you can see, here is the spindle it is like in normal conventional machine the spindle as it is there. It has a 3-jaw chuck which holds the cylindrical workpiece, here is the turret head. And as I was telling you that turret head will hold many tools. So, depending on the tool that you require, either it is a turning tool or it is a facing tool or it is a parting tool or is a forming tool.

So, you have to call that tool in the program. If you remember we have said T01. So that is one tool, T02 and so on. So, each of these tools in the turret head is numbered and according to the number given in the program, that particular tool will be coming in the position where it will be ready for machining. So, this is the turret head and that turret head can move we will show you the movement here, we have the center.

So, when there is a bigger or lengthier cylindrical job then that job can be mounted between the centers like in the case of the conventional machine that we have shown you. Now here we have that provision for coolant, in case the coolant is required then this will be switched on but everything will be done according to the program; in the program you have to write whether the coolant has to be on or off, only then it will be done.

Because no intervention from the human being will be required to operate the machine; once the program is input here in the machine then the machine will automatically work without the intervention of human being whatsoever. So here you can see the panel, this is the control panel. Now if we have to input the program manually in that case, we use the control panel and using the control panel line by line we can put this program in the computer.

So that computer can finally execute. Now Mr. Rakesh will show you how this program can be put in here and after that we will show how the program can be interfaced, also the one that we have automatically generated, the code if you remember and in that case the fusion software was used. So that program can also be interfaced with this; in another machine we will show you how it can be interfaced.

And that program can be used without writing the program manually or without inputting the program manually in the machine itself. So, this is the machine control keys and if you see that in the panel there are different kinds of knobs and those knobs are indicative of a certain function which the machine has to perform. So, for example one is the skip, skip blocks will not be executed.

If you put the skip so that a particular block which is in the program will not be executed. Now the next is the dry run, dry run this is the test run of the program; if you have to do that, optional stop program stop and the program you know that code is given as M01 M is the miscellaneous code. Next is the reset then the single block machining then the program stop program start that is the key to start and stop the program.

And then there is a set of 3 into 3 keys which are for the manual axis movement. So, plus X plus Q you can see minus Z plus Z and so on, so this is for the manual axis movement then there is approaching the reference point in all axes at this is the key then the feed stop, feed start and other keys there. Then we have the spindle override lower 100% higher so that is done by the knob and the knob is there.

Now there is another one which indicates the spindle stop spindle start and spindle can start in the jog and NC1 to 1000 mode. Now what is jog is that we want to slowly move that spindle because we do not want the rapid movement to prevent any accident. So, in that case the jog mode is switched on; it can have the clockwise or it can have the counter-clockwise; now open close the door he has seen that we have closed and open the door this is knob.

So, what I mean to say is that all the knobs for all the functions are in the control panel machine control keys and this is a control panel which commands the machine what to do. So, we have to make that in the program by using this and then we will keep it for the

machine to execute the entire program which has been input to the machine. Then there is a mode selector, in the mode selector you can see that it can be referenced.

Or it can be MEM, MEM is the auto mode then there is EDIT mode so there you can edit the program and so on. So, there are different modes and all those modes you have to put it in that position so that a particular function can be executed. What you see in this display is the entire program which is already given. So not to put it one by one here, this program has been made using these keys.

And this is the same program which we have already shown you, that we have generated and then that program has been given. Here all the codes are given; that is, what kind of G code to be used, what kind of M code to be used, what kind of coordinates to be given in the X and the Z and as I will repeat once again that the X in the turning is in the vertical direction, to and fro.

And the Z is along the axis of the workpiece; so these are the 2 axes which are used then that feed value has to be given, then if there is a radius then the radius has to be given in terms of R, you can see that in the program it is given. So, how this program can be made according to the part that we have to fabricate we have shown you in the simulator. Now, with this program, one by one the program will be executed.

And finally, you can make the part we can show that part, so, finally you can expect that part has to be made. This is the final product which we can expect from this program that we have made. You can see that in that part we have the flat, the conical surface, the threaded portion, groove and so on. That you have seen earlier.

Now, according to the program line by line we will execute. For that the first step is that we have to call the appropriate tool which is required, which in this case is tool number 1 and that tool number 1 has to come to the 0 position. So, it will touch the exact 0 position of the tool which you have determined and for that we will be using the ‘jog’ mode as I told you already that the ‘jog’ mode is when the tool has to move slowly, we will not put it very rapidly.

That means to prevent any kind of accident in case it may happen. So, we will put it in the jog mode and the tool will be called, that is tool number 1 and it will be brought to the 0 position on the workpiece. The first step is, first of all we have to close them because otherwise it can be potentially dangerous. Now we are putting the program and we are putting that in the reference mode first.

You can see that this is the adjustment of the axis then the rotation of the spindle command is given and you can see the spindle has started rotating. Which direction the spindle has to be rotated, that is either clockwise or anti clockwise that has to be also programmed. And this is the clockwise, earlier it was the anti clockwise just to give a demo to you that it can be programmed whether the jog will be rotating clockwise or anti clockwise.

Now you can see the program in the display that is the tool number, tool number 1 that we have selected and when you execute that then the tool number 1 will be selected; you can see that the turret head is rotating and it is adjusting to the tool number 1 which will be in the position ready for removing material from the workpiece. Now it has to be brought to the 0 position; workpiece started rotating.

And in the jog mode you can see in the display in the jog mode we are bringing the tool to the 0 position so that it can start from there start the machining process from that point this is the X direction prior to that it was it was moved to the Z direction. So, those coordinates are already given there in the program according to that program it will be moved. Now we are simply showing it by doing it manually.

Because we want to give a demonstration to you otherwise it can be also done automatically by executing the entire program in the machine. So, the X and Z coordinate in the display and those coordinates are given so that the tool can be moved accordingly. Again, we are coming back to the reference point. So, this is just a demo for to show you that how the tool can be moved along the X and along the Z axis.

So that the tool can be brought to the initial position that is the 0 position, after that the program has to be executed; according to the program now it will be working so it is coming now it is not manually done anymore. Now the material has started being removed you can

see that deliberately we have chosen plastic materials so that the machining can be shown more effectively with the less force required.

So, this is the turning of the cylindrical surface and as you can see that there is no intervention, so the program is given and according to the program the tool will move and it will machine the surface, so you can see that, the taper is being now machined after that. This is still the cylindrical surface and you can see in the display that the X and Z coordinates how they are moving and how much they are moving.

So, we have given a particular coordinate for the X and Z it will move up to that and that is why there will be no clash between the tool and the 3-jaw chuck. Because according to the program it will be moving, it will go up to that point as per the program and then it will come back. So, there is roughing in the beginning, rough operation then there is the final finish operation; this is how the final diameter has to be achieved.

And it has been done in different passes you can see that all the surfaces are being produced according to the drawing that includes the cylindrical surface, tapered surface then the thread will be cut, then grooving will be done and so on. And then it is going to the initial position, then another tool is coming, this is for the grooving, so there is a grooving tool and the program has been given in such a way that the grooving tool has to come there.

And without the intervention of any human being, according to the program the grooving tool is selected, it will come to the position where the grooving has to be made and then you will be given feed as per the program, as per the value which is given in the program and it is done in various passes; then it will again go back to the initial position and then another tool is selected which is the threading tool.

So, you can see the threading now is done and the speed is decreased; in case of the threading operation and in several passes with incremental depth, the thread depth has been achieved; this is how the threading is done with the threading tool. So, we have deliberately switched off the coolant because in this case the coolant is not necessary since it is a soft material and we have switched it off so that it does not cloud the machining zone.

So that you could see the machining zone clearly; so, in various passes the thread is cut and then again the tool will go back to the turret, will go back to the initial position. Now you can see the part is made, it is ready. So, it has to be removed manually and that will be exactly according to those dimensions which you have given in the drawing and the dimensions can be measured or in these cases we normally do not because, we are sure.

That these dimensions will be exactly according to the dimensions which had been given here; we can see this is the milling machine CNC milling machine. Here the control panel is absolutely the same as in case of the turning that we have demonstrated to you in the machine you can see that this is the spindle in which the tool is mounted. Now, here in this case the tool has to be taken from the turret head which we are going to show you.

And the tool will be taken by the automatic tool changer, there is an automatic tool changer which will remove this tool and the required tool will be coming from the turret head according to the program. So, here also you see that there is coolant provision then the vise here is like in the case of conventional milling machine, we have shown this is the table and on the table with the help of the T slots we are fixing the vise.

And in this machine the X, Y, Z coordinates are different than the coordinates that we have shown it to you in the in the training. Here the X is this movement along the X axis, Y axis is the cross movement and the Z axis is the lowering and lifting up the tool meaning that this is the actual Z axis. So, here is the Z axis, here is the X axis, and here is the Y axis; this is the axis system for the CNC milling machine, all milling machines will have the same axis.

Now we are going to show you how this tool has to be changed. And the tool that we require is given in the program and as per the program that exact tool will be coming to the spindle and it will be attached to the spindle, you can see the turret head here and each of these positions are numbered. For example, you can see that this is number 20, this is number 11, this is number 8, 7, 6 and so on.

So, according to this number, the particular tool can be taken to the spindle by the automatic tool changer. And this program, I mean the program that we have made we have to indicate that which tool is required. So, any of these numbers any of these 20 tools, starting from 1 to 20 and any of these 20 tools can be selected that we require for machining.

And that has to be taken by the automatic tool changer and it will be put in the amount in the spindle we will show you one of the tools and you can see that how that tool can be changed how this automatic tool changer can come select the tool pick it up from here and mounted and the spindle. So, you can show that in the program and you can see this now. So, this program is given in the panel now we are showing it manually.

But when this will be executed this execution will be done automatically; we can see that changer it has been kept here and then it is a particular tool has been selected. Now the tool changer has come and it has mounted the tool in the spindle. The tool you can see there which is mounted, that is the required tool for the current fabrication and the command will be given to bring the tool from the reference point to the 0 point of the workpiece.

So, you give them the coordinates and it will be in the jog mode, you can see the jog mode, from the reference point it will go to the Z and the X and Y coordinates as per the program given there. I will come to the 0 point from where it is just touching the workpiece. So, from this 0 point all the dimensions which are given in the program are in the in the drawing which will be executed.

That is why it has to be brought to the 0 position and those are the coordinates in the panel you can see that X, Y and Z at this coordinate this is tool is positioned at this moment. So, from the 0 position the tool will take the dimensions as per the program and when it will be put in the automatic mode you can see that automatically the tool will come to that position and then it will move.

According to the program given there, along with the groove that it has to make the circular surface. Now we are bringing back the tool, this is just manually you have seen how the tool can be brought to the 0 position of the workpiece; now it is the initial position that is tool has gone to the reference point and then the program will be brought.

So, the program which is being shown to you has been made with the help of this machine. So not to waste time here, this program is being demonstrated to you that this is the program which is made and now it has to be executed. That is the program, you can see that program, I have already demonstrated to you and we have also simulated this. So, we know that the

movement of the tool and the relative movement of the workpiece and the tool will be perfect.

So, there is no flaw in the program so now we can run the program so this is in the MM mode and it is being executed tool has come now there is no manual intervention. So, tool is rotating it will come to the 0 position and then it will take the dimensions as per the drawing; these are the coordinates also it is showing that it is in the absolute and those are the program steps that are being executed.

It is moving in the X direction and the feed that is taken, those values are given in the display you can see that the tool number is given T. So, this program is being automatically executed; now there is no human intervention and manually we are not doing anything there everything is going as per the program; step by step all this values, the feed, the rotation RPM all are taken automatically and all these values are given in the program.

So, overall, the program has to be very accurate, if there is any flaw in the program so in that case you will not get the perfect fabrications or the dimensions and the workpiece will be faulty. You can see that the program has been already tested and we found out that there is no flaw in the program, it has been simulated and therefore, according to that program we can expect that the part will be as per the drawing and the dimensions will be as per our requirements.

After that it will again go back, tool will stop rotating and will go back to the reference point and all those commands are given in the steps that I have already explained to you earlier. Now after the machining, you can see that this is the part that we have obtained and if you remember that in the drawing also, we have given this the cut here this is the round shape this is the half round and this is the corner.

And all these dimensions are given in the drawing and as per drawing this part will be obtained and you can make sure that all these dimensions are according to the drawing. Thank you for your attention.

(Video Ends: 01:02:15)