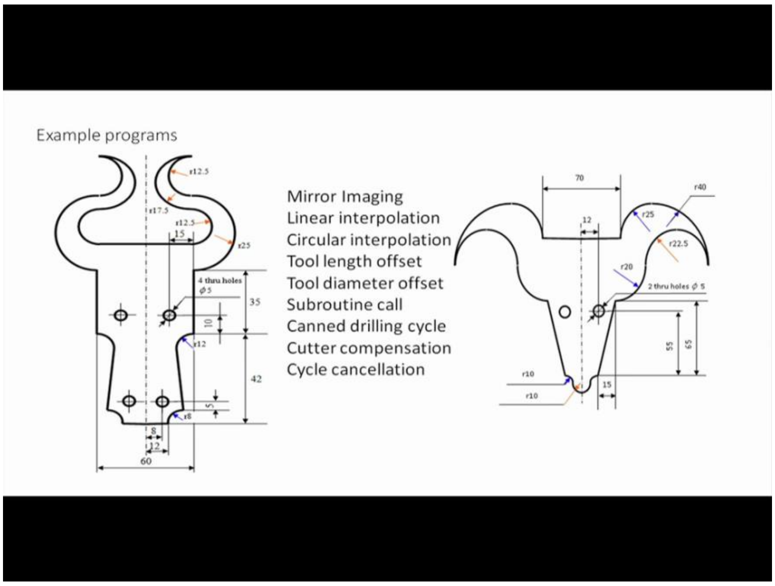
Computer Numerical Control of Machine Tools and Processes
Professor A Roy Choudhury
Department of Mechanical Engineering
Indian Institute of Technology Kharagpur
Lecture 11
Programming Practice 2

Welcome to the 11^{th} lecture of our open online course "Computer numerical control of machine tools and processes". In this lecture, we will be discussing about Programming Practice involving G and M codes and we have already completed programming on point-to-point control on a drilling exercise we have carried out. Now we will be carrying out Programming on milling, which means continuous control and of course some drilling exercise will be included in. After that, we will concentrate upon a turning exercise CNC turning that means basically lathe work. So to start with, let us go right into the particular programming practice.

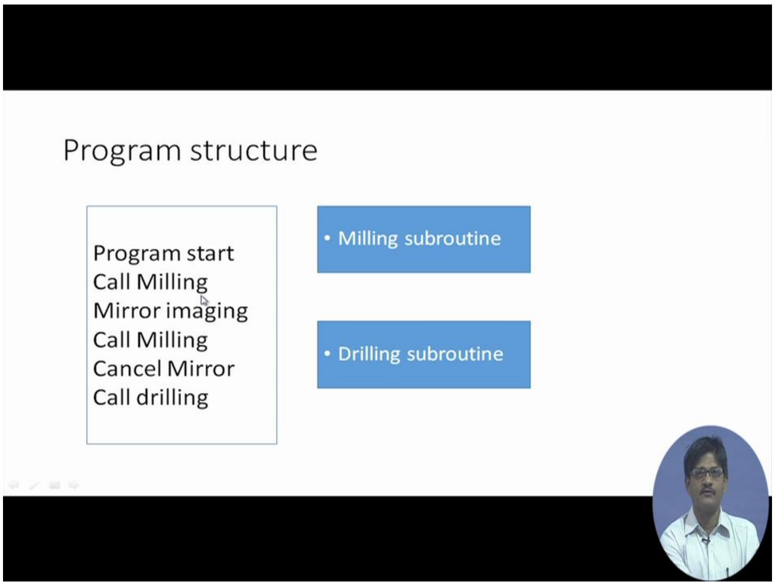
(Refer Slide Time: 01:30)



We are taking shapes of this type and this might seem to be very much quite funny and not at all connected with engineering practice, but actually they provide us very good opportunity to test out and practice certain programming features which are available in G and M code CNC programming. First of all, these jobs are bilaterally symmetrical therefore we can test out mirror imaging on them, mirror imaging is a facility by the help of which we can program for one side and after that call the same program once again with mirror image so that the other side of the particular mirror axis is done is carried out without rigorously writing down all those program lines.

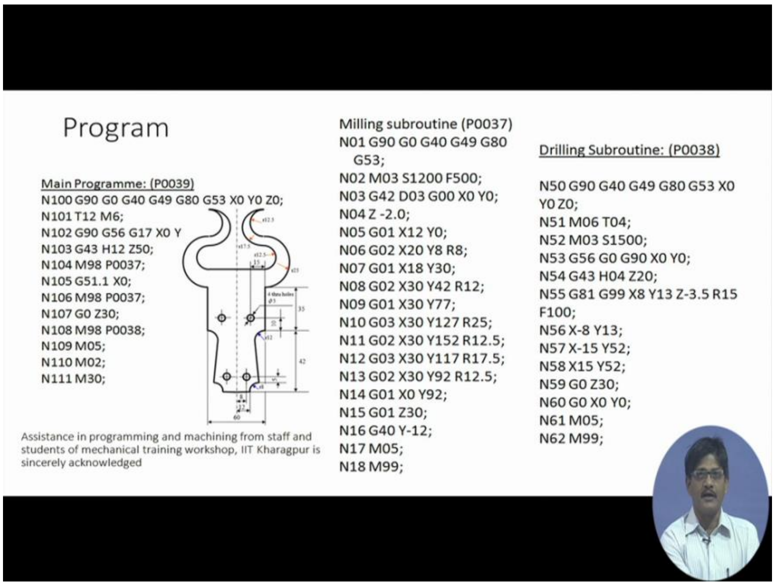
We will also learn about linear interpolation, there are linear interpolation applications here, circular interpolation and then tool length offset, tool diameter offset, subroutine call, Canned recycle drilling, Cutter compensation and cycle cancellation and many other ordinary features are also there. So we will be taking this particular part and applying CNC programming on it so that it can be a program can be made which will cut it out on a CNC machining centre.

(Refer Slide Time: 03:21)



The program structure will be like this, there will be a main program in which the program will start, so this is the main program so starting commands will be executed, after that there will be a call to the milling subroutine and one side of the part will be milled that means the outer profile will be cut by milling operation, then it will come back to the main program. After that mirror imaging will be declared and after that the milling program subroutine will be called again so that the other side of the path will now be carried out by milling. Then it will return to the main program once again, cancel mirror command and give a call to the drilling subroutine, drilling will be carried out and it will return to the main program and there the programming will end, so this is the basic structure of the program.

(Refer Slide Time: 04:25)



Um I hope it is visible very clearly to everybody, let us start with the program. 1st of all in the main program, it is residing in line number 0039 and this program has a starting line which is consisting of G90 absolute programming, G00 fast positioning, then G40 cancellation of tool cancellation of cycles which are active when the program is started, cancellation of tool length compensation sorry, G40 was cancellation of tool diameter compensation, G49 is cancellation of tool length compensation, G80 is cancellation of cycles which may be active and then G53 deals with the program the coordinate systems, the machine coordinate system.

And connected with G00 command we are moving to the point X0 Y 0 Z0. After that, we go I mean for Tool selection, we give a command M06, which is concerned with tool change, which is the tool that we are selecting? We are selecting tool number 12, mind you when we are loading the tool number 12, we at this moment we are not loading any tool offsets. Tool offsets mean whenever tool is loaded onto the machine spindle, its length and the job height I mean the length of the tool protruding from the spindle and the height of the job above the table surface, these things are not known to the machine unless they are declared by something called tool offset, length offset and tool damage and is also not known to the machine, that also has to be declared through specific command.

Coming to the line number 102, we have G90 G56 is the work coordinate system which is being declared, G17, the plane of work is the X-Y plane and in this work coordinate system, we move to the point X0 Y0 I think at this moment something might have been blocked off by this figure, it was Y - 10, please make this correction it is Y - 10, so after that line number 103 we have G43, G43 declares tool length compensation and the actual length and tool protrusion and work piece height, they are together stored in this particular address location, this register H12 will be storing it and therefore, the machine control will go and access that particular value stored here and use it in order to activate tool length compensation.

Once tool length compensation is activated, the machine would be having work piece surface as Z = 0, the spindle axis generally has the Z axis and in this case we are were dealing with a machine in which there is a vertical spindle axis and that is the Z axis and the work piece surface becomes Z = 0 ordinarily, that table surface is Z = 0. After this, Z50, so it moves to Z50, 50 millimetres above the job surface. Next we have M98 P0037, 0037 is the address location of the milling subroutine and M98 gives a call to the milling subroutine that means the machine control will send the program control to this particular location 0037. The 1^{st} line once again is the cancellation of all possible active commands program machine coordinate system.

And after that we have the declaration of spindle speed and the spindle starts moving due to the command M03, this is the spindle RPM and this is the feed value of 500 millimetres per minute. Next, G42 is Cutter compensation right or Cutter diameter compensation right. It means that if you are intending to move your Cutter all along the periphery of the part the Cutter is not infinitesimally small, it is not a point object but it has a finite diameter okay, it is a circular piece with a finite diameter so that if you simply ask the Cutter to move along this particular part, if centre point will be on this particular part and therefore it will machine on this side as well as that side and your job will be spoiled.

So a circular cutter with its centre on this part moves along and spoils the job, so you would like the Cutter to move on the right side of the moving line, is not it this way? But in that case you have to do so much calculation involving coordinate geometry, et cetera an offset line has to be drawn, offset circles have to be withdrawn, so all these things can be avoided by just telling the computer see I want to move my circular cutter of this particular radius always on the right side of the moving line and that that command is represented simply by a single command as G42. Where does the Cutter get, where does the machine get the information about the diameter of the job that is stored in D03 address location memory location, okay?

So it is understood by the machine now that Cutter right compensation that means looking along the moving line, the Cutter has to be shifted by its by its radius to the right side so that the inner part is preserved in the outer part is anyway waste. So after this we are simply going to move along the moving line and these are the commands, let us quickly go through them like this one this one line N04 gives the depth, it goes down in depth to 2 millimetres and then starts machining. How would it machine? It would 1st move straight and then along a circular path and then along an inclined path and then along a circular path, so all these movements will give in G01 X12 Y0 okay.

And G2 X20 Y80 R8 these things we have already covered previously that means circular interpolation okay, let us try out one here for example, this one is G02 then we will be giving the target point location, target point location here is 20 comma 8 that is why it is written here and the radius about which the circular interpolation is going to take place is 8 millimetres as per drawing and the feed value is not mentioned here, so that the feed which has been declared previously here as 500 millimetres per minute that will be adapted here. So, after that circular interpolation it has finished up this part of the job and come to this particular location.

Next we will have a straight linear movement write up to this point where this line is tangent to the circle and therefore it moves to G01 X18 Y30. Here, it is interesting to notice that though this part seems quite complex, it is absolutely extremely simple because all the circular paths they have been chosen intentionally to be either half circles or quarter circles. If they are half circles or quarter circles, finding out their coordinate locations and in the end coordinate locations starting from the 1st coordinate location it is extremely simple. Moreover, there are not only half or quarter circles, they are basically you know, the segments have been taken in line with the coordinate axis.

Have they been inclined, they would have created little bit more of a problematic situation for us. So, I am not going to painstakingly go through all these but basically we will be doing linear interpolations, circular interpolation, straight-line interpolation, circle, linear and then circle what you call it, counterclockwise then circle clockwise and then circle once again counterclockwise and then circle clockwise and we will come here after that linear interposition, this will be our mode of movement, so after this we will reach this point and let us see where it is, it is this point. Only X movement has been carried out so that it is back at the access point, all these points are absolute coordinates.

That means they refer to the actual coordinates as per the drawing, so X0 Y92, it has come to this particular point and this side of the job has been done with compensation, radius compensation. So once this side has been done, what we are supposed we are supposed to do is, we are simply going to rise up the tool to a height so that it does not interfere with work piece and take it back to the initial point. That is good, so in line number 15 we simply write G01 Z30, since Z = 0 is the work piece surface and as you go upwards you have more and more positive values, Z = 30 makes the tool move to 30 millimetres above the job surface.

After that, we go right down with Cutter radius compensation cancel G40, once we do not need it; we are cancelling it because if it is active it does not permit some of the commands to get executed some of the other commands. So Y - 12 which has come, it just cross this line and come right to this side, so this particular side has been completed and after that we reach to this particular part, G51 0.1 X0, this is mirror image okay programmable mirror image. That means no axis is getting physically shifted, the job is physically not shifted anywhere but simply in the coordinate axis X is becoming - X that is all.

So if that is being done, so in by this particular command G51 0.1 mirror image, the axis about which the rotation is to be performed, this is mentioned here X = 0, so once this is declared whatever commands follow in all those subsequent commands, X will be taken as - X and after this give a call to the subroutine once again. So if this particular call is given, we will again go to this particular program, execute all these commands and all the X values will be solved, all these locations would be solved for the movement of the tool interpolator and at the last stage, the sign of X will be change to the opposite sign.

If this is done, all the opposite side geometries will be defined and the tool will be moving through all of them with compensation, that means we will still be using this program, so still G42 will be declared, but when G42 will be declared at the last moment the coordinate values are changed, so ultimately automatically it will be shifted to the left of the work piece now. The tool is now shifted to the left of the work piece when it is doing the mirror image sign, but we do not change any commands we keep it G42, we give the same command and automatically it shifts.

So after this particular command is executed, the other side is cut automatically and after the execution it comes back to the main program, in the main program what do we have? We have the Z value to be 30, so the tool is raised from the work piece is free from the work piece and there is a call to the drilling subroutine now, what does the drilling subroutine look like? It is residing in memory location 0038 and the command goes to this particular position. What happens here? First of all again those cancellation commands are given in the very beginning, the coordinate I mean the coordinate system and other things I will not repeat it.

So in the 2nd line we have M06 T04 that means a separate tool naturally the drilling tool is now selected and the drilling tool is selected by the command M06 okay, a tool change takes place, the previous N milling cutter flat ended N milling cutter goes back and the drilling tool is selected. So, after the drilling tool is selected it is set into motion that means M03 sets it at the motion of 1500 rpm and next it goes to this particular coordinate this particular coordinate system is selected, which takes care of the discrepancy between work piece coordinate system and machine coordinate system because the draughtsman is drawing his figure with a particular coordinate system, this might well not match with the machine coordinate system when the blank is put on the table.

So the distance between machine origin and the origin of the work piece given here, this is 00 point, this is taken care of by the declaration of a separate coordinate system. Next, line number 54 once again length offset for this particular tool and work piece pair is declared and this value is residing in address location H04 and after that we execute, we are inside the subroutine now we are executing the drilling cycle, drilling cycle is given by G81, we have done this previously in another type of programming that was programming with Siemens system, this is Phanup System.

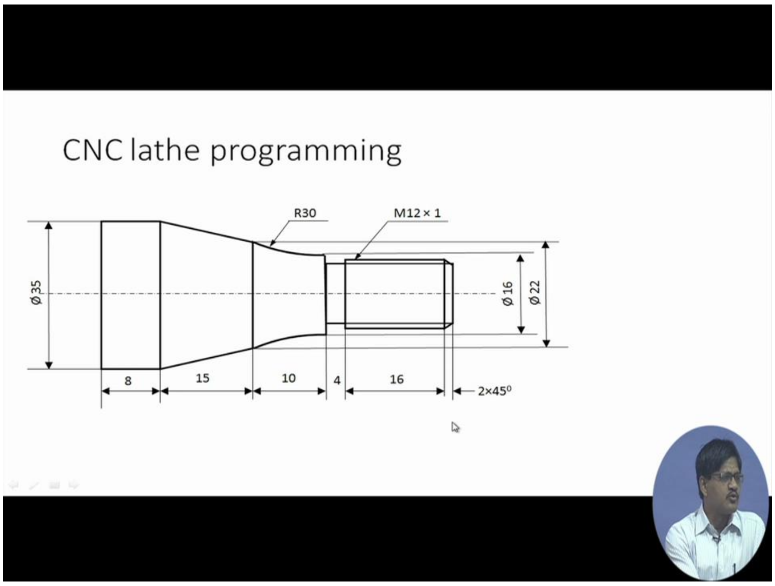
So G81 G99 feed in expressed in millimetres per revolution, previously the feed was in millimetres per revolution and what you call it, X8 Y13 Z - 3 R15 F100. So 1st of all we have, I would like to point out one thing, this F100 is a bit too much, if the feed is taken in millimetres per revolution, it will be much less, so please keep a note on this, this value should be very much lower maybe 0.3 to kindly excuse me for the moment. So after that X8 Y13, so what do these things mean? X8 Y13 gives the location of the 1st hole that is going to be drilled it is good and Z gives the depth of hole and R15 is the level at which high-speed a high-feed value along Z axis of the drill towards the work piece will be changed from high-speed to machine feed okay.

So 1st is moving down by rapid motion and then it changes over to feed motion at 15 millimetres above the job. This 15 millimetre is also quite high, it can well be 2 millimetres and then F if it is programmed in millimetres per revolution, it will be much less 0.3 like that. After that in the next line we have simply the location of the next position of the whole centre mentioned in the XY plane. So once this G81 is declared and once it is active as we go on giving different coordinate locations on the X-Y space, we will be having holes drilled at those locations with these specifications, depth equal to - 3.5 millimeters.

That means depth equal to 3.5 millimetres from the surface of the job and the changeover from rapid to machine feed I mean for a machining feed at 15 millimetres above the job surface and feed please read this as 0.3 millimetres per revolution. So one these points have been mentioned, all these 4 drills will be done by these particular commands. Next we have the tool moving up to 30 millimetres and then the tool going to the point X0 Y0 and spindle becoming off and going this one going back to the main program and that is it.

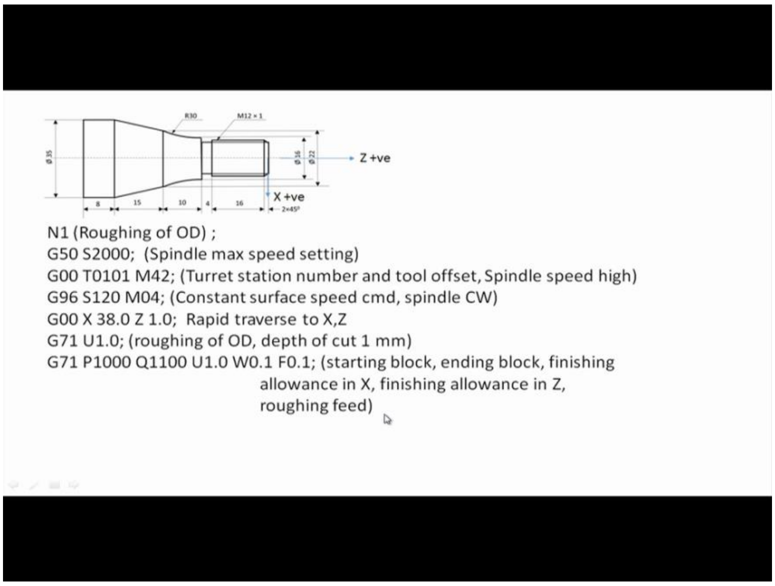
The spindle goes off, the program is put off put to a stop and then ultimately M30 stops the program okay. And assistance from staff and students of mechanical training workshop IIT Kharagpur is sincerely acknowledged for making this program and executing it. Now let us take a Lathe programming exercise.

(Refer Slide Time: 24:22)



In the lathe programming exercise what we have here? We have a straight turned portion and we have a taper turned portion, we have a circular interpolation carried out here. Next, we have a groove and last of all we have threading, we have chamfering and we have and we have facing done here. So we will start with the assumption that facing has already been done here and let us see the different diameters are given, this is a metric thread with 1 millimetre pitch and the diameter of the thread is 12 millimetres and this is having 30 millimetres diameter is circular interpolation part and these dimensions are also shown in the figure, let us move right away into the program.

(Refer Slide Time: 25:12)



First of all, here we will be having roughing of the outer diameter values okay, what do we mean by the roughing? That means before taking the finishing cut we need not bother much about the feed that we are employing and we need not bother about the depth that we are employing. If you employ high feed, you are going to have coarse surfaces finish. If you employ high depth of cut, you are going to incur very large deviations of the part because the part will either bend as a cantilever or as a simply supported beam if high forces high thrust forces are applied. So if you have high depth of cut, high-speed, these problems are going to occur but during the course turning we do not bother much about it.

So the spindle speed is set to a maximum value of 2000 by this command G50 2000 and next we have G00 T0101 M42, what does this mean? This mean, this particular tool, tool number 01 with offset given by the command 01 is loaded and M42 means high spindle speed range is selected, which actually executes a gear change operation inside the CNC turning centre okay M42 will lead to an automatic gear change operation, so there is a speed range and a low speed range.

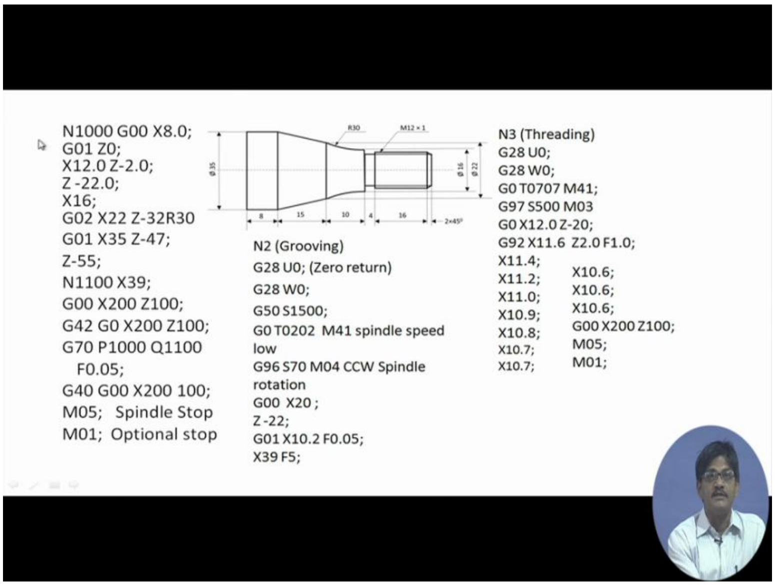
Next, G96, speed mentioned and spindle speed to be this spindle speed actually it is written clockwise, but M04 stands for counterclockwise, so this one is this one may please be read as counterclockwise or M03 can be used which is clockwise okay M03 clockwise. So coming to the next line we have a fast movement, where are we going? X sorry this one is X negative this one is X negative please make this change, X negative and that side is X positive, so that means this particular space is Z positive and X positive okay. The spindle, rotating spindle always contains the Z axis.

Next, if we start the program now it moves rapidly to a particular position 38 Z1, so it comes somewhere here, the tool is on this side okay it is on this side and it approaches the job and cuts it this way, recedes, goes back. In conventional machines, the tool is generally on that side okay, this is because the operator is here and he has to approach the machine and operate the tool, so it is useful for him for the tool to be on this side. And the tool is on that side because it helps, in automatic machine it helps in work piece setup and further the tool can be inverted so that all the chips which are coming, they will be falling down and the spindle will be rotating in the opposite direction.

So let us start, we have the tool coming to this particular position and from here it starts a machining cycle the rough roughing of the outer diameter, how is this done? This particular G code is executed G71 and line numbers are mentioned here, so from this line number to this line number, it will go on executing this roughing cycle that means one cut after the other will be taken so that all the material mentioned here from the outer diameter to the final diameter, it will be going on machining. What is the depth of cut used? It is 1 millimetre, so what are the other values expressed here?

Like you finishing allowance that means some material will be left behind that is 1 millimetre along X that means along the diameter and 0.1 millimetre along the Z axis that is left behind and the feed is 0.1 millimetres per revolution okay. So next we come to the actual movements, what are these movements?

(Refer Slide Time: 30:31)



Starting from line number 1000 to 1100, we will have these commands repeated, so if we are having these commands repeated, let us see. We start from here and mind you, spindle is rotating and we start from here actually and we start moving this way and the material is getting cut, so let us quickly follow these movements. G00 8 millimeters why? Because this is chamfered and this has 45 degree slant, so we consider this diameter to a to be 12 millimetres from there to here to here this comes to 8 millimeters, so the cutter 1st comes to this particular point okay 8 millimetres, generally diameters are programmed.

So 8 millimetre diameter, it starts from here, it goes by 2 millimetres sorry, it goes to the Z = 0, so it right on the surface of the job. Next, it moves by linear interpolation, linear interpolation it will be moving from here as it will cut this slant this chamfered portion and then move by linear interpolation up to this particular point okay. Here these movements are shown for example, G01 Z0 it comes and touches the job surface. G01 X12 Z - 2 okay, it comes to this particular point, it has gone 2 millimetres inside, cut the chamfered portion, then Z - 22, it moves right from here end comes to this particular portion 22, 16 okay + 2 millimetres, it is 18 + 4 = 22, so with cuts right up to this portion.

After cutting right up to this portion, it moves out 60 millimetres and then it cuts this circular interpolation part, G02 target location and then the radius, this is mentioned here. And next it will cut up to 35 millimetres and Z47 taper cut okay by linear interpolation. Next, again by linear interpolation this turned portion Z - 55. Once all these commands are executed, written down and executed by the rough turning, it will go on taking cut after cut with 1 millimetre depth of cut until this particular part is finished. Once this is done, it goes to G42 that means now cutter offsets are mentioned because it is going to take a final cut, previously it was a rough cut so we did not mention any offsets.

So now offsets are mentioned in, again it has repeats it with a lower feed. If you have lower feed, you have better surface finish, if you have lower depth, we will have better if you have lower depth you will have less of thrust forces and less of deviation, okay. So after this one is finished, next we go for G40 cancel of compensation, spindle stops and it goes to 00 position for a tool change, why do we have a tool change? Because now we are going to do the groove, so 1st we have rough turned the periphery, left some machine allowance and we are doing the grooving.

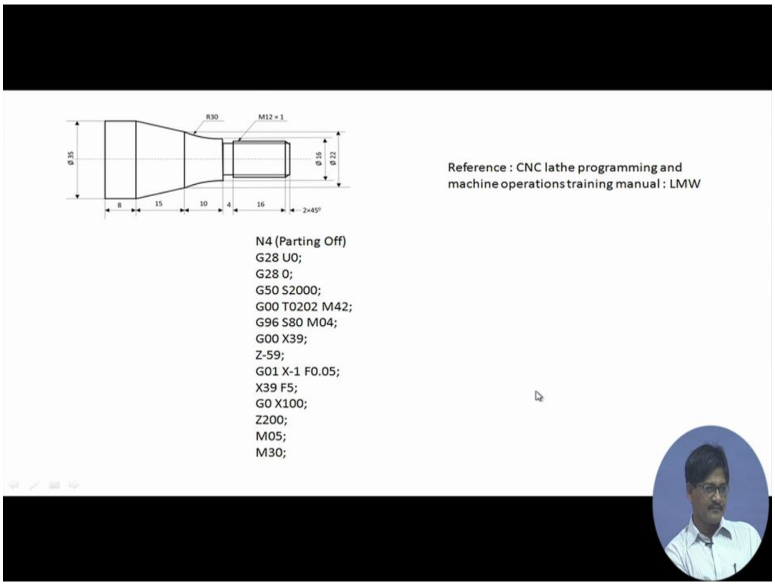
How do we do the groove? We simply change the tool okay, T0202 go for low spindle speed option that is the gear change and after that with constant speed option, we go for 70 millimetres speed and M04 that means counterclockwise spindle rotation. Now, what is G96? G96 means constant surface speed that is even if the diameter is changing just like in rough turning also the diameter was changing as we took one after the other cut. What is happening is, we are simply going for constant surface speed option so that there is no speed loss. If we do not cut at the recommended speed, we will lose time so we are going for no speed loss here.

So the moment of the parting tool is simply down, down to the particular depth X = 10.2 millimetres that means very close to the root diameter of this thread, so we cut up to 10.2 millimetres so that when this thread is cut, the tool will come out and get some relief, it will not be in touch with this particular surface, all right. So now the threading part, once again tool tool change will naturally be required, so tool is changed and then we will cut from the side outwards to that side. In that case, the spindle will be rotating the spindle will be rotating counterclockwise okay.

Sorry spindle will be rotating in such a way that the tool will be moving outwards okay the tool will be moving outwards and 500 spindle speed is taken and G97 means constant RPM is chosen, this time we are not choosing constant surface speed because if it is chosen, then it will interfere with the particular pitch that is being cut. So this way we 1st take it to Z = - 20 here and after that it is moving outwards okay, moving outwards and the spindle is rotating clockwise. The spindle rotation is always determined by looking from the head stock towards the tail stock okay and deciding whether it is clockwise or counterclockwise.

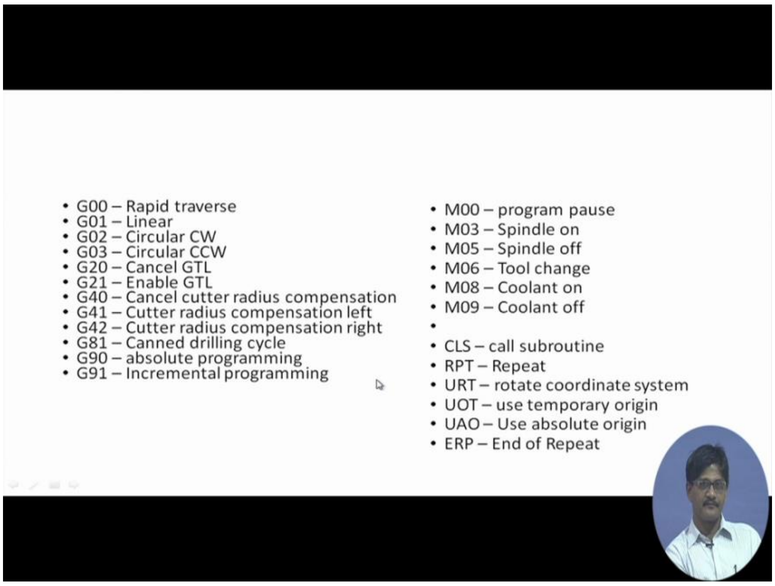
So this way, this is the thread cutting command we give here the final points where it is moving, it is moving to Z = 2 out from the work piece 2 millimetres and 11.6 millimeters is the X movement first and it goes on repeating this command by taking smaller and smaller depths of cut till it reaches the ultimate diameter okay. Why do we do this? Because the thread profile has to be very accurate and if you give a large depth of cut, there will be deviation or deflection and that will interfere with the 1st of all with the diameter of the thread and also to some extent with the geometry of the job.

(Refer Slide Time: 38:01)



Here some detail about the parting operation is provided that is after the job is done, we will simply part it off here. Now that you have come across all the other commands previously, this will not be very difficult to follow so I leave this part with you people and this for this we have generally taken the help of CNC lathe programming and machining operations training manual of LMW Lakshmi Machine Works, thank you very much.

(Refer Slide Time: 38:34)



Thank you.