Manufacturing Systems Technology

Prof. Shantanu Bhattacharya

Department of Mechanical Engineering and

Department of Industrial and Production Engineering

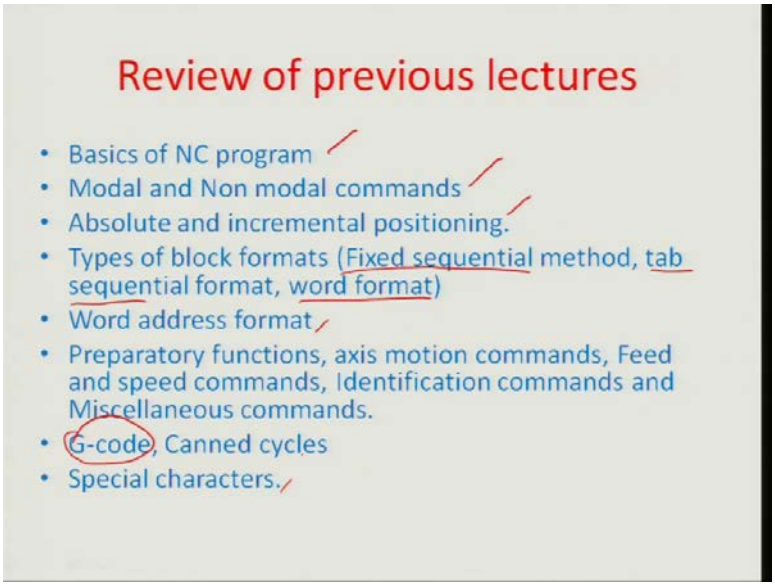
Indian Institute of Technology, Kanpur

Module – 07

Lecture -41

Hello and welcome to this module 41 of Manufacturing Systems Technology.

(Refer Slide Time: 00:19)

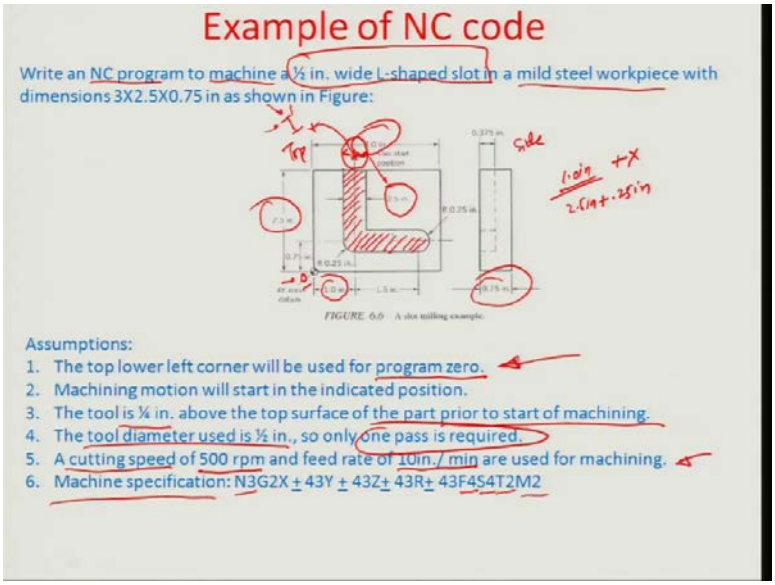


Quick recap of what we have done so far in the NCC, NC area, we talked about the basics of NC programming including a discussion on the modal and non modal commands, the absolute and incremental positioning aspects. We also talked about the various types of block formats, which are used fixed sequential for example, tab sequential, word format, so on and so forth.

We also discussed word address formats in great details including all the different command classes which are there, the preparatory commands, the identification commands, the axis motion control commands, the miscellaneous commands, so on feed speed command, so on and so forth. And also we discussed then individual aspects of

particularly the G code and what it represents geometrically and discuss finally, the special character in the particular NCC, NC set up.

(Refer Slide Time: 01:12)



So, today we would like to do some actual programming for a certain machining process, which is illustrated in this particular figure here. So, let us first explain or let me first explain to you, what exactly is the machining requirement for which, you have to design the NC code. And for that, we want to write a program for machining it a half inch wide slot and this is in l shaped it is slot as you can see here right about in this particular area, I will just represent this slot by this hatched you know lines, so this is a slot.

In other wise continuous block piece of metal, which is probably rectangular in shape and this is a mild steel work piece in which, this slot has to be sort of mailed and for that a program has to be developed. So, various things are given about the block dimensions etcetera. So, the initial block dimension is about 3 inches in length and 2.5 inches in the breadth and then, the thickness of the block is closed to about 3/4th of an inch, so this is shown clearly in this particular figure. And various assumptions are needed to be made here for starting the programming; obviously, you have to do a 0's and etcetera on the NC program.

So, the first assumption that we are having here is that we want to start machining from the top lower left corner of the block, which is indicated here. So, why top? Because; obviously, the block has a thickness and we are talking about the top surface of the block

and we are recording this 0 setting of the tool. So, it is on the top surface of the block and also towards the left corner of the left bottom corner of the block as can be seen in this top view, this actually is the top view side view.

So, top view really represents the top left lower corner of the block, so using this is the program 0, therefore the tool has to be positioned here. So, typically the position of the tool is done by jogging the tool at that particular place, you know trying to align manually and seeing, whether it reaches that point. So, there are many methods for doing the tool positioning in a NC system otherwise, which can be done by the operator himself.

So, the machining motion starts in this position indicated right about here, this particular position and at that particular time, when the machining has started the tool is about of fourth inch or quarter for an inch above the top surface of the block. Meaning thereby, that if I am talking about square or rectangular section like this at the block, the tool is really not touching the block in the left lower corner, but it is act a certain distance away from the block and that distance is about the quarterfinal.

So, the tool is actually not in any direct contact with the block and that is prior to the start to the machining process. And further we using a tool diameter of half inch, which is thereby it implies that only one pass is needed for machining result slots, so the tool can be somewhere around here with the same diameter as the dimension of the slot here. So, the diameter and the dimension of the slot are both 0.5 inches in the tool, there is a cutting condition, which has been provided cutting speed of 500 rpm.

For example, of feed rate of 10 inches per minute, which are used for the machining operation and, what has also been clearly mentioned is the machines specification, which is part of the controller, which comes to the machine. So, you can go up to from 000 to 999 about thousand steps in this particular NC system you can also you know go up to probably G code starting from G 00 to G 9900 the X, Y and Z capabilities are indicating about 10 million dimensions each 0000.000 to 9999.999. Please repeat all this again and again, because this is the way to sort of learn, what is the value that you are programming, so that, the controller understands your language properly.

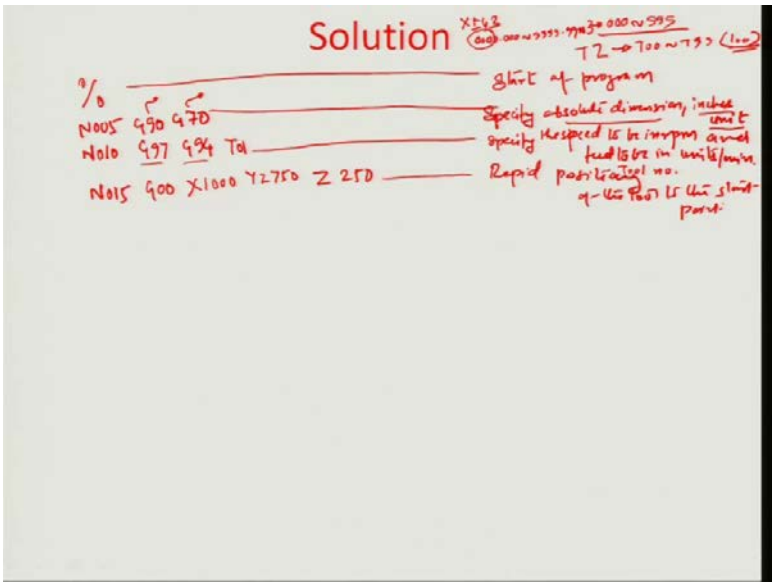
Then; obviously, the radius also is programmed in a similar manner particularly in case of polar coordinates you need to have a radius in place and then; obviously, the feed, the

speed, the number of tools in the miscellaneous commands given to be various values, so given this machine specification, now we start doing the programming.

So, the first job really for the machines would be to take this tool first of all initialize the tool, initialize the access of motion probability define things related to whether it is an incremental process or whether it is an absolute dimension that you have to consider. Also decide about the units of the inches, units of the access motion commands with that they are in inches or millimeters, decide about the units of feed and speed commands. So, these things are initialization schemes for letting the actual motion event happen within the tool.

So, we want to step by step first arrive at a condition, where we have more or less defined everything for the tool, motion, the rate of motion etcetera at this the units we have defined. So, that now, the controller is able take a decision as to what point of time we need to start and what point of time we needs to stop in terms of relative motion between the tool and the work piece.

(Refer Slide Time: 07:23)



So, the first step that we write in this particular program probably indicates the start of the program. So, we called this the percentage sign as I think I had illustrated it earlier, it is a special character used for starting the program. So, once we have that in place, because the n value as you can see here ((Refer Time: 07:41)) is actually representing 3.

So, therefore the way that this N would have to be defined is, you know the N varies between 000 to 999 about 1000 N values can be program.

So, let us say we do the programming steps of 5, so that leaving there by four values in between four any modification correction later on if the operator, so wishes to have, where the program or wishes to have. So, the first line starts from N 005 meaning there by this is the fifth line of the program and let us first actually tries to indicate the absolute dimension. So, we specify the absolute dimension will do this programming in the absolute mode and let us also try to indicate the length scale here in inches units that is all we indicate.

So, we actually write here N 005 G 90 and G 70; obviously, 90 indicates absolute dimension as can be seen here and this G 70 indicates inches units, which is actually giving you a basis of, what units to be followed while moving the access commands. And you also need to define the simultaneously the speed on the feed rate input this particular case if I just go back to the last problem statement you see the feed and speed rate have been defined here as in the cutting speed is about 500 rpm.

So, the unit is rpm and the feed rate is 10 inches per minute unit being inches per minute, so these two have to be somehow define using the correct G code for that and. So, in the next line we can probably mention this line to be numbered 010 meaning there by we are just giving way four steps in between for correction etcetera and we write G 97 and G 94 meaning there by that we are specifying the speed to be in rpm and the feed to be in inches per minute or units per minute units have already been defined earlier as inches, so inches unit, so let us just write units per minute for the sake of clarification here, so its units per minute.

So; obviously, we also need to do some kind of tool selection I just want to indicate what all these description mean. So, the tool selection here can be made at this particular step and, let us say we want to use the tool 0,1 the tool is numbered between again the specification says t 2 meaning there by it goes between t 0,0 all the way to t 99 about 100 tools are capable to be programmed us for us the controller goes you know on the machine. So, we select the first tool for example, let say the milling cutter that we are using is the first tool in this particular case.

So, you do the selection of the speed to be in rpm G 97 and the feed unit in units per minute G 94 and then, also the tool number and these are the things you have selected using this particular line or block of the NC command. So, more or less now you are capable of moving ahead, because you have defined the units of the various value whether its feed speed whether it is access motion commands whether its absolute incremental these are important for now, the controller to start understanding, what you're doing in terms of machine motion a programming the machine motion and the system.

So, let us start that motion now, so let us say the first step that we make here is the 15th line of the program N 015 and the initial part of the program would be associated with really the tool moving from this particular position write about here all the way to the start position of the tool, which you can see indicated here. So, the center of the tool moves to this position, let us call this position you know some t dash, where the tool has to be moved. So, basically the point here is that the tool should move between the point o that it was initially in this point o all the way to the point t dash.

So, you have to move this in a rapid manner because; obviously, the tool is away as far as possible by quarter of an inch above for the surface and it is not touching the work piece as such to the tool can directly move from the point o to the point t dash in a rapid manner with you know, what rapid positioning means earlier G 0,0 code has been very well defined in terms of the full capability of the stepper motors in both axis to move at full speed. So, that the time of positioning becomes minuscule in that process.

So, we try to rapid position by mentioning that G 0,0, so that the control controller takes up the rapid positioning. And then, specifying the with respect to the origin, which is again point o remember we had set this to be the machine 0. So, this is the origin, so with respect to that, because it is an absolute dimension we are now, reading may early the coordinates with respect to this origin. And, so we have to move to a point right about here which is probably about.

So, the values of the X and Y coordinate has to be commensurate with the tool position at this particular position as I earlier illustrated t dash, which is actually if you look at how much it is from with respect to the origin it is about one inches in the positive X direction, which is indicated by the value shown here from the machine 0 this 1 inches

and about 2.5 inches plus the cutter radius, which is actually 0.25 inches. In this case you have to understand one thing here is that the tool was centrally located at this particular corner and it has to go all the way to clear off the machine machined portion or machining zone.

So, therefore the cutter needs to be all the way to this point t dash here, so this goes in the plus Y direction of the reference frame. Obviously, now, one thing you have to understand is that how the milling is behaving in audit to a certain the plus X and plus Y direction, what I had earlier told is that if I had a sort of a slot milling operation or a you know some kind of a face milling operation, where there is a vertical arbor, which is rotating and there is a column tool column, which is in place.

So; obviously, the position that we are indicating here is about 2.75 inches in the plus Y direction we have to remember that we are looking towards the, so when the Z the negative Z motion basically means the tool comes back to the work piece. So, if the positive Z axis is in the direction of the tool going upwards from the work piece, then towards the right is the positive Y the X axis and towards the column is the positive Y axis that is all the optimal coordinate would be define.

So, in this case we are going towards the column of the tool milling tool and also towards the towards the right you know as we are pointing to the column or as we are seeing into the column. So, we are maintaining the directions in terms of signs etcetera sign conventions etcetera as for as possible. So, we now write that with G 0,0 you have to go to a position, which is X 1,0,0,0 mind you the X capability shown as plus minus 4,3 meaning there by its starts from 0000.000 all the way to 9999.999.

So, you do not have to really record the as for as possible the capability of by making this extra 0's. So, you can just write 1.000 inches as the X value. And then, 2.75 inches can be recorded as Y 2750 inches. And; obviously, now you have to move the Z away from the work piece a little bit, because your earlier resting at a place, which was the corner. So, before going there you have to really assume that the tool also gets away from the work piece and the distance that it moves away is quarter for an inch that is what the example sides actually.

So, you say that Z 250 meaning there by Z is 0.250 inches. So, this is the position that you have to reach quarter for an inch over the upper plate or the upper portion of the

block and the coordinate 1.000 and 2.750. And basically the rapid positioning can be done because its G 0,0 there for you do not have any feed rate at, which you have to really move etcetera.

So, you can either say f zero or that is unnecessary in this particular case and, so this actually indicates the rapid positioning of the tool to the start point of the machining. So, I think I will in the interest of time close on this module and start the next module, where we will describe more about the remaining steps of this program which are associated.

Thank you.