Manufacturing Systems Technology

Prof. Shantanu Bhattacharya

Department of Mechanical Engineering and

Department of Industrial and Production Engineering

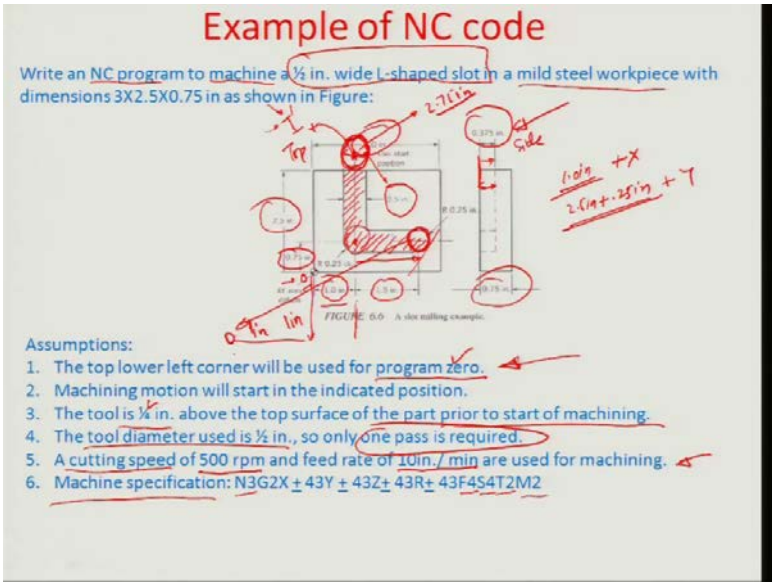
Indian Institute of Technology, Kanpur

Module – 07

Lecture - 42

Hello and welcome to this Manufacturing Systems Technology module 42.

(Refer Slide Time: 00:17)

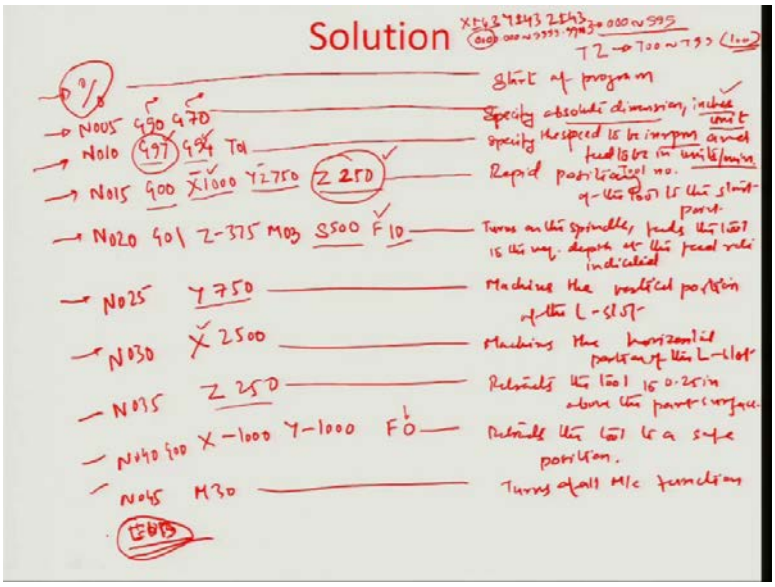


Quick recap of what you were doing, you are trying to write NC code for the particular situation shown in this question, where there is a slot milling operation carried out and then, l slot needs to be developed in a block, which has been given as 3 cross 2.5 cross 0.75 inches. And also it was there are certain assumptions that have been made including, where is the location of the program 0, which is at the lower left top corner of the block surface.

It is assumed that, when the tool starts moving it is about one forth of an inch or a quarter of an inch above the top surface prior to start of machining. So, you have to position the tool from probably the, this corner all the way to somewhere here, where the start of the machining process would happen, but you would ensure that the tool is about quarter for

an inch above that particular processes. And only one pass is needed, so therefore, it assumes that the cutter sizes with same dia as the slot size, which is about half inches. We also have cutting speeds, feeds and machine specification given.

(Refer Slide Time: 01:30)



So, we started writing this program and we had covered up, you know issues related to the start of the program depicted by a percentage, which is the special character. Then we started writing three different lines, one where specification of or one which specifies the absolute dimensions to be followed; that means, the reading would be only with speed reference to the origin every time. We also talked about the G command, which is needed for the particular system of measurements in progress of it is inches. So, a prior system is used here, so inches units for describing the length units.

We also were able to describe the speed in rpm and the feed in units per millimeter per minute using the commands G 97 and G 94 and selected the tool as tool number 1. And then finally, rapidly positioned the tool to the start point using the command G 00 and I already described, how we obtained these coordinates Z plus 250 means 0.25 inches above the surface. The specification mind you is about X Y Z plus minus 4 3. So, in that ambit this represents actually the decimal is represented to the third place and then, there are four 0's, which are the material even if you do not write them.

So, that is why you are describing Z as 250 and positive, because it is moving away from the tool. So, you are having the tool going up from the surface quarter for an inch and

that is why the ((Refer Time: 03:08)) position of the tool is as represented in the question. So, once this happens, now we are ready to start the machining, so we write to the next line of the program N 0 2 0 here, which describes the linear positioning.

So, obviously, we use the G code G 0 1 now, because now; obviously, the milling the slot milling operation would start to happen and it has to be happening at a slow manner with a certain speed rate, which has been specified in the particular machining conditions.

So, you start with that and then, first thing is you have to go is you do not really need to change the X and the Y positions. So, even if you do not write them in these particular steps, it is obvious it is taken as the previous X and Y positions, because their mode will combined unless you actually change them, they are not going to change, get changed by themselves. So, basically all the change that happens here is only in the Z direction, because you were placed quarter for an inch above the surface at the machining center and now you going to go down and you are going to down from the surface up to an extent of about 0.375 inches as has been seen here in this specification here.

So, that milling slot depth is given here in the side view as 0.375 inches. So; obviously, now, you have to make a choice, because it is an absolute dimensioning system and you are assuming that this whole plain here on the surface of the particular block is treated to be the 0 plane or at least the plane, which is containing the origin. Therefore, we are more concerned with, how the tool would go below the 0 plane.

So, below this is the 0 plain therefore, you know the top side of the block and you are going in the negative Z direction and you are traversing 0.375 inches, that is how the tool has to travel. So, assuming the tool thickness to be very, very negligible in comparison to the dimension that has been traverses although there is in practically there is a thickness of the, you know the slot miller or the slot cutter, which has to be accommodated here.

But we just neglecting at this particular point of time, so we just write Z minus 375, so minus 375; obviously, indicates that the tool now has to go from position quarter of an inch that is 0.25 inches to the position minus 0.375 inches. Mind you again, it is an absolute system of dimensions. We are recording the coordinates only from the origin and the earlier coordinate quarter for an inch was with respect to the origin and this coordinate minus 0.375 inches also is with respect to the origin.

So, this is something that mostly the programmers make a mistake. So, you have to be very, very careful you are writing, because it is an absolute system. Obviously, you can use the miscellaneous command M 03 to illustrate that you know it switches on the spindle and M 03 probably indicates the spindle starts moving in the clockwise direction. You can also give a speed at this particular point to the rotating tool, the speed was given to be 500 rpm; obviously, the rpm has been described earlier here as you can see by the code G 97.

So, we are left with S 500, which is the speed of the tool and now you also can give a feed rate; obviously, there was no feed rate when you are talking about the rapid positioning, because the feed rate would be defined by the shortest possible time path that the tool would take as I described in the great details earlier. So, here this feed rate is given again in units per minute and suppose 10 inches per minute.

So, I just write F 10 here as my, you know the whole command block and, so this command block basically indicates, what all it turns on the spindle. It feeds the tool to the required depth at the feed rate indicated and then; obviously, it turns on the spindle at a certain speed and actually, let us it go up to the 500 rpm speed. So, now, it is all set to go into the block, because now it is correctly positioned in terms of depth etcetera, it is still cleared off the block and it has to not traverse, so that the l slot that has been given here, can be now fully machine.

So, the next position that we want really this tool to go with respect to the origin is about in the Y direction. So; obviously, the tool was somewhere here, which was corresponding to the 2.75 inches and now, this tool position has to come all the way to about, you know this particular position, which is at a distance of 0.75 inches from the origin. So, we can say that this comes, the Y changes from 2.75 absolute coordinates to 0.75. The remaining other things remaining same, because it is at the same X axis of 1 inch are represented earlier by X plus 1000 on the first step.

So, we now write the next step simply as N 025 to be equal to Y 750, this is only the axis motion command that we are using, which means that merely the controller moves the tool from the Y step of 2750 all the way to Y 570. So, this is already defined at the predefined, you know we have already predefined the feed rate etcetera earlier. So; obviously, whatever the controller does in terms of motion between the point Y equal to

2750 and Y equal to 750, that motion is actually at the feed rate that has been specified, which is this 10 units per minute or inches per minute given earlier.

So, this becomes equal to, then effectively cutting the vertical portion of the l slot, so I can write machines, the vertical position or portion of the l slot. So, once this machining is done then; obviously, the tool now is at the new position located somewhere here and it is actually centered about this 0.75. So, the other thing that we need to certain is that the tool goes in this direction from the current location 1.000 inches to 1.500 inches and, so by traversing by another 1.500 inches. So, because it is a absolute dimension the coordinate here really the X coordinate here is 2.5 inches.

So, basically the other step that is involved here now is that asking the tool N 030 identifier number asking the tool to now, traverse from the position, which is recently acquired that is Y 750, X 1000 to X value 2500. So, automatically the access motion combined indicates that the two has to feed forward in the X direction by about 1.5 inches whose earlier you have defined the position as 1.000 inches and now your defining as X 2.500 inches.

So, therefore, the relative difference is only 1.5 inches, which is suppose to move. So, therefore, this machine this ensure that the tool machines the horizontal portion of the l slot and the only other thing that is remaining now, is to sort off you know a line in a manner, so that you can retract the tool away from the position that it has finished in the last step. So, the tool finishes somewhere here and still rotate in the rotating condition and you have to somehow now, retrieve this away from the block or l slot. So, that tool does not do any further damage to the system.

So, the next step therefore, can be that first of all whatever depth had been provided to the tool in term of going below the origin plane or reference plane by 0.375 needs to be retracted back and you know in order to keep it safe the best idea is to sort off take it again quadrant over to the surface. So, that it does not, now touch with the surface and you can take it easily to a direction or location coordinate, where it won't foul or interfere with any of the working block etcetera.

So, therefore, we give a machine command here N 035 that is the identifier number corresponding to the 35th line, which we say that, now since all the machining is done the z needs to go back to 250. So, Z actually was earlier 250 if you actually remember

the tool was stationary at the start point above the work piece surface by quadrant for inch. And then, it came down to the plane and through the plain into minus 0.375 inches, which is on the other side of the you know the plain containing the origin.

And now, we are talking about again retrack it from the after the cutting of the whole l slot has been over retracting it back quarter of an inch above the machine surface that is why Z 250, so this actually retracts the tool to 0.25 inches above the part surface. And then, finally the tool needs to now, get retracted to a point, where it does not interfere any of these work piece features. So, we can go back from the origin to, let us say some position somewhere here ((Refer Time: 14:15)) corresponding to about you know 1 inch in the negative Y direction and again 1 inch in the negative X direction.

Then; obviously, the tool is in the safe zone, so somewhere here the tool can be located. So, tool needs to get translated from here to here and we can say that the 40th line of this program can give that command saying that take the X back to minus 1.000 and Y back to minus 1.000 and this can be rapidly positioned again. So, we can actually, because feed command is something that needs to be executed at the you know at the outside.

Otherwise the feed would be still the same 10 and then, you do not have rapid positioning access back again. So, you can actually write G 0 0 rapid position and X minus 1000 and Y minus 1000 and then, you have to set the counter of the feed to 0, so that the feed can now, can be taken by the G 0 0 commands. So, this retracts the tool to a safe zone the tool to a safe position and then; obviously, you have to turn off the machine.

So, you actually write another small instruction here saying M 30, which sort of turns of all machine functions and you can probably again insert a special character EOB here signifying end of the block. So, this whole block write about containing write about 9 steps gets started from the percentage you know line and then, ends up all the way to the end of block. So, you saw now that how we can actually do the really ANSI programming, so there are many process is in which you know validate what code you have written is really going to have a non interfering tool path and an continuous tool path.

So, with dell cam etcetera there are certain packages, which are there you can actually develop auto codes and also can try to stimulate the auto codes on a platform, where you

can really estimate, what is the tool path like and, what is tool behavior like. So, once there is a validation of this code and you can feed in two NC system and start to do machining on work pieces. So, with this I would like to end this module in probably next module will start a little bit of you know issue related to lean manufacturing and Japanese production system, till them have a good day from us here or NPTEL.

Thank you.