Manufacturing Systems Technology

Prof. Shantanu Bhattacharya

Department of Mechanical Engineering and

Department of Industrial and Production Engineering

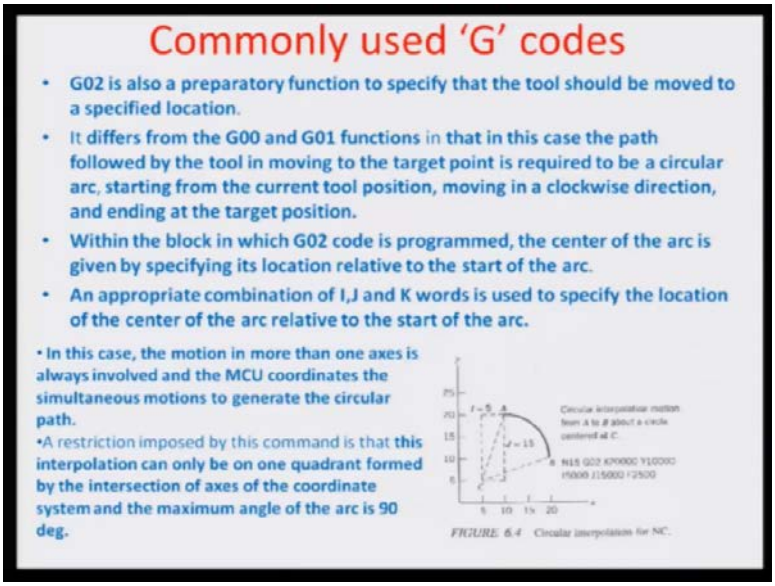
Indian Institute of Technology, Kanpur

Module- 07

Lecture- 39

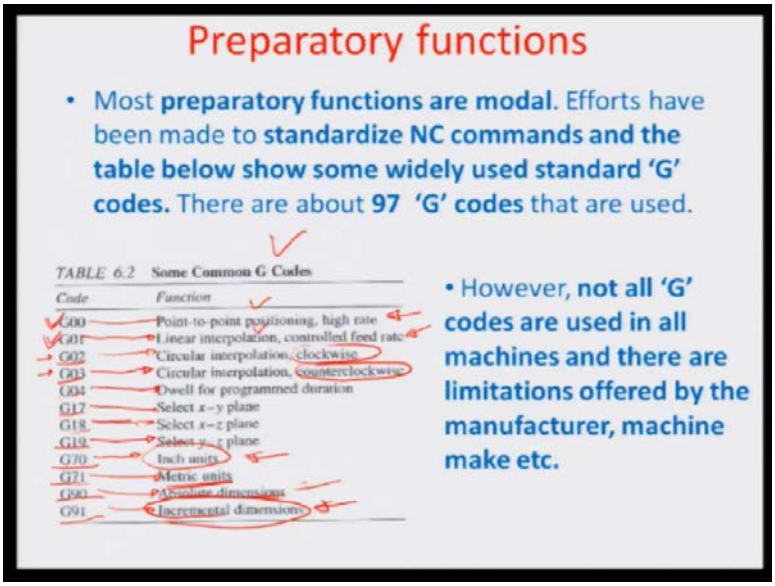
Hello and welcome to this Manufacturing Systems Technology module 39. We were talking about rapid and linear positioning in last module.

(Refer Slide Time: 00:26)



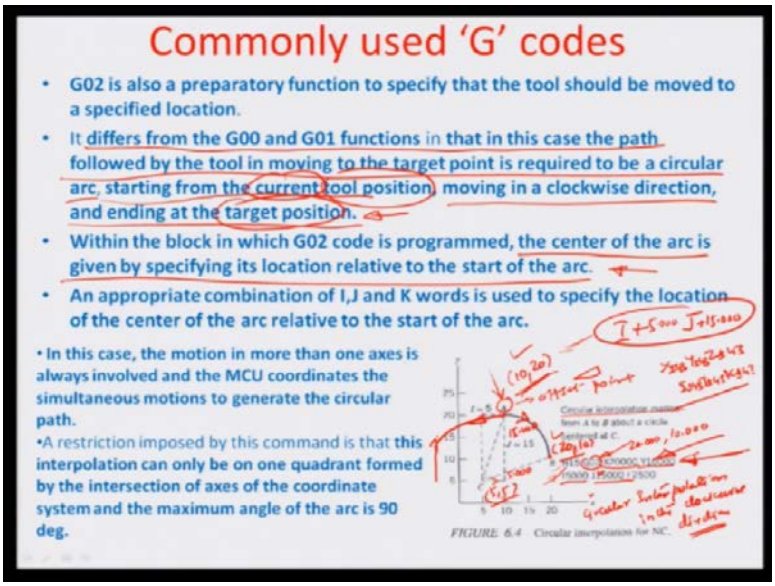
Let us look at a few more issues are related to the circular interpolation, which is actually a command given by you know G02 as can been seen in this particular table illustrated here.

(Refer Slide Time: 00:36)



So, this motion command is definitely going to illustrate that the circular interpolation would take place in the clock wise direction. The question is from where to where that has to be somehow specified? So, what you need is actually a degree offset in terms of a central position. And from that particular off set point you have to go to another offset point. So, you start utilizing coordinates I J and K, to represent what are those offset points from central points when you talk about defining a certain arc. Obviously, the direction of arc are been clock are counter clock wise is represented by this G02 and G03 command.

(Refer Slide Time: 01:23)



So, if you look at that the particular preparatory function when we are executing it, it

differ from the G00 and G01function in that in this particular case the path followed by the tool is moving to the target point, is required to be a circular arc; starting from the current tool position moving in the clock wise direction and ending at the target position. So, what we really now need to mention is the current tool position and the target position somehow. So, that now the arc can be defined in term of a central you know central radius etcetera.

So, for this the center of arc is given by specifying its location relative to the start of the arc. In fact, what we refer doing is to sort of you know represent certain point. Let say in this particular point this value A here, I would urge all of you to look at this particular figure. So, this value A here is at the position 20, 10; oh sorry 10, 20. If you look at the common X Y coordinate system that there are using its position 10, 20 that we are talking about.

So, here in this illustration this position 10, 20 is the start position for the particular arc and the final position that, this arc is going to go up to XY, is basically this position right here which is about 20, 10. So, you have to go from 10, 20 coordinate to 20, 10 coordinate to describe so called arc. And further you have been given a sought of you know position coordinate 5, 5 at which the tool is probably placed at some point of time. So, that tool has to know start from this particular C, and go all the way to this point A. The definition of A is not yet provided in a proper manner and; obviously, only the coordinate is something that we know of A. And then, from there the circular interpolation starts and you remember that G02 if we mention, mention circular interpolation in the clock wise direction.

So, it is basically mentions circular interpolation in the clockwise direction. So, this controller probably is unable to read everything in terms of you know X Y Z plus minus 43 and similarly I J K again plus minus 43. So, I have already illustrated in great details what these things mean, what four three means. Is merely indicating the position of the decimal point which is there you know from the start numeral all the way to the end of the numeral.

So, here now the question that is asked is that if supposing the final coordinate of I would urge all of you to look at this particular line, this particular block of statement that has been given here. So, the block says N15; obviously, it is first in principle identification number that is probably its 15th line of the program that is what it

represents. And the G02 obviously, indicates a circular interpolation that it should execute, and the position coordinates X20.000 and Y10.000 has been provided. Meaning there by this 20.000 and 10.000 are actually the final positions, at the point B where the tool has to reach by doing a circular interpolation. And obviously, what is now mentioned this that let the tool go in the positive X direction by 5000, and in the positive Y direction by 15000; meaning there by that from the 0.55 we are moving 5.000 directions, 5.000 you know in the positive X direction to define the offset. So, this is called the offset point and merely the I J K are defining the offsets that would be there with respect to this start point or the start position of the tool. So, the start position is at C and you are defining the offset point by mentioning that go to the positive X direction by an amount 5.000, which is mention here is I 5000 and go to the positive Y direction by 15.000 which is mentioned here as J15.000. So, that is how you basically are giving the offset. So, I will just like to sort of indicate this once more for your convenience. So, that you are able to see the, the difference that this offset creates actually.

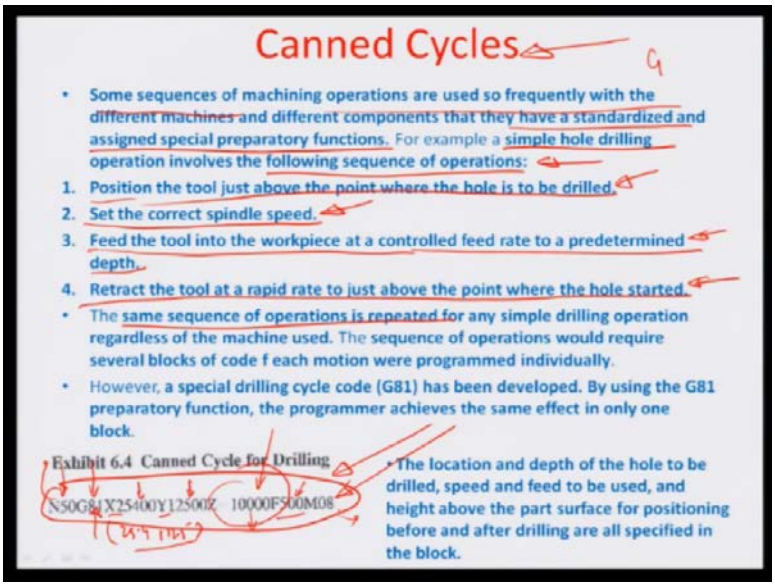
So, you have been enable by giving a position C, and you have been said that the C is at 5 5; and you are asked to move for offsetting to start the arcing process in the clock wise manner to a point A by saying that go in the offset direction corresponding to x axis positive 5.000 and go to the offset direction J in the positive Y axis by 15.000. So, that is exactly what you mean by this term here I 5000, J 15000. So, from this start position you have reach the offset point and now you are trying to execute the arc by going from this position to this position; obviously, it will center around the C, because you have mention the offset. So, offset means offset from the center.

So, the controller interprets as if how much offset is from the central, so automatically by defining C at 5 5. You do not see the definition of C at this particular line, probably this has been defined in an earlier line, when you talking about the tool position. So, probably the last step had defined the 0.55 for this particular you know position C here from which the machining would start to take place. So, it is probably the origin of the tool position or something like that.

So, having said that all you need to do is how much you have offset in the X and Y to start the arc about the point C. And it is moving all the way to the 20s 10 coordinate which is the point B actually, which you see in this particular figure here; the 20 10. And that is how you actually do the definition for the circular interpolation; obviously, if the circular interpolation is G03, the same thing would happen in the counter clock wise

manner meaning there by now the B would be somewhere here. Rather than going in this direction clock wise manner it would go in the counter clock wise manner and the B would be somewhere here. So, that is how circular interpolation is being done. So, you have to be very careful about the direction; obviously, by defining the position C from an earlier step you have the center for the arc, and you all need to do is to sort the program the offset value from the center and the final position up to which the arc would be executed for really executing the arc. That is how you do the circular interpolation.

(Refer Slide Time: 10:14)

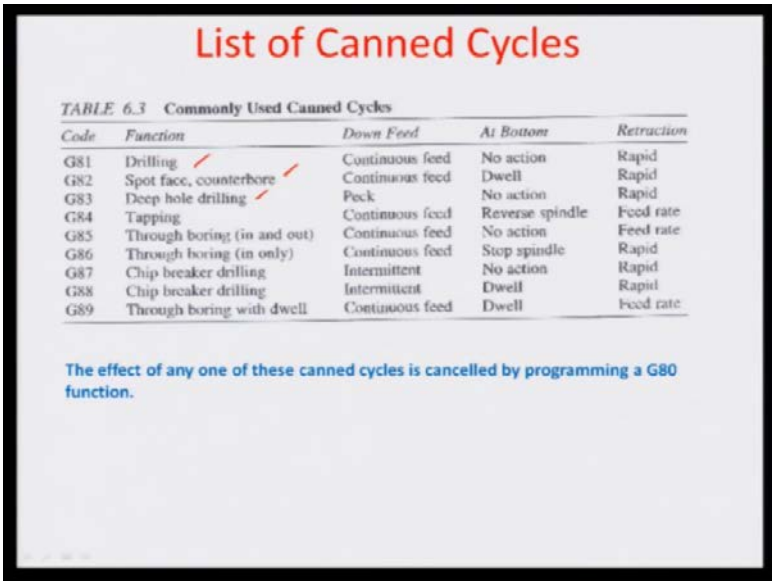


So, I will now go to the, another aspects of the preparatory functions, the G functions; we talks about Canned Cycles. And basically the Canned Cycles came into existence because there was certain operation which needed to be really repeated more than some other operations; like for example, when we are talking about the engineering part, wherever there is a fastener which is kept within the parts somewhere there is the requirement of the drilling and a taping process. Similarly if supposing there is the let say even a reverting that you are doing at some particular place to join a temporarily two or more members of a engineering assembly you have to again do the drilling and taping. So, such operations are very very common place in engineering designs has compare to some other very specific operation for example, may be turning of the shaft by certain you know value or making it lets say some kind of a step shaft.

So, that can be only a one time operation. So, we have to have some is pumped into this whole CNC system. So, that a programmer does not have to repeat the small program that would be there for let say a very widely used process like drilling, every time he

wants to do the program. So, what he can do is, he can make certain cans; with the can would kind of record all the different aspects of a single process let us a drilling process every time, you have to do drilling you just merely repeat that can. So, that is the logic which has been utilized for giving this basis of Canned Cycles. So, some sequences again you know machining operations as I told you are use very frequently and that are sort of standardized on special preparatory functions now.

(Refer Slide Time: 12:12)



So obviously, this Canned Cycles would be represented by now certain G codes. For example there is a list of such G codes which exists. The drilling can be G81 for example, G82 is spot face counter bore, G83 is deep whole drilling, G84 tapping, G85 is through boring and now through boring is N only, G86 chip breaker drilling. So, these are all the sort of you know a functions or machining operations which have very common place engineering assembly and they have basically been converted into different cans. Now every time we do G81, you only needs to specifics some aspects of the drilling into that single line of G81, are proceeding the G81. So that you can actually let the controller know that you do the whole drilling cycle in accordance to the canned G81 now. So, that is the advantage. So, basically a bridging or summarizing the whole drilling from a three step processing into single step processing, that is all what you are doing by a Canned Cycles.

So, let say we just see how this abridgment is done here, so in a simple drill hole whole drilling operation. The following sequences of operations are used, you had seen this before. So, position the tool just above the point where the hole is to be drilled and then

you set the correct spindle speed and start the spindle make it on. So, that it start you know doing the material removal and then feed the tool into the work piece at a control rate so; obviously, because now it is rotating and it is doing machining, so it has to go inside the control rate, feed rate to a predetermined depth. And then you retract the tool at a rapid rate to just above the point where the hole started. So, these are the four steps which are utilized now for doing the drilling operation.

So, how will you do it how will you mention in one block? The same sequences of operation can be again and again repeated once this has been carried out. So, here is for example, a Canned Cycles which has been illustrated for this drilling. So, here you can see, the moment it goes into the can mode. Let say, this 50th step of the program says that there is the canned mode G81 meaning that there is the drilling process that we are executing here. So, the X and Y coordinate corresponds to the initial position of the drill or the start position of the drilling process as the process continues. So, here for example, it is saying 25.4, 12.5 mm meaning there by that these are the coordinates this which are where the position of the tool would need to be before the or just before the start of the machining process or drilling process. And then they are saying Z minus 10.000,meaning there by now at this position you have to give a depth of about 10 unit in the negative Z direction; obviously, negative Z direction again means you know the traversal of the tool towards the work piece.

That is why the negative term here, as I discuss in my earlier classes or earlier lecturer on CNC. And you can go all the way to 10 units along the Z direction, and then you subsequently give at what feed rate. So, we probably giving at about 500 mm per minute, because obviously, probably somewhere before this line there may have been line which might have utilized the command for the matrix unit. So, you can do everything in mm or may be it can also be inches depending on some assuming here that 500 Millimeter per minute. So, the feed rate 500 Millimeter per minute and then; obviously, when the drilling happens the coolant has to be on conditions, so M08.

So, this whole line is the one integrated line, which talks about the whole machining process the whole CNC machining process at one go. So, the advantage now is that whenever you want to use drilling process, simply give the G81 figure out all these values: what is the start point, what is the depth, what is the coolant on off condition, what is the feed forget about the whole process. So every time it will be executing the three steps automatically. You do not need to change this step really again and again.

So, this in fact is an abridgment over the one which you have seen earlier in the tap sequential and the fix sequential formats, where the same one line was replaced by three lines. So, you are basically having three lines of code putting together in the single line, so that is about Canned Cycles. So, I think we are towards the end of the all the preparatory functions now, and we can move ahead with some of the other functions like excess motion controls or even the miscellaneous command or what kind of command exists before starting the programming. So, we will do that in the next module.

Thank you.