Manufacturing Systems Technology

Prof. Shantanu Bhattacharya

Department of Mechanical Engineering and

Department of Industrial and Production Engineering

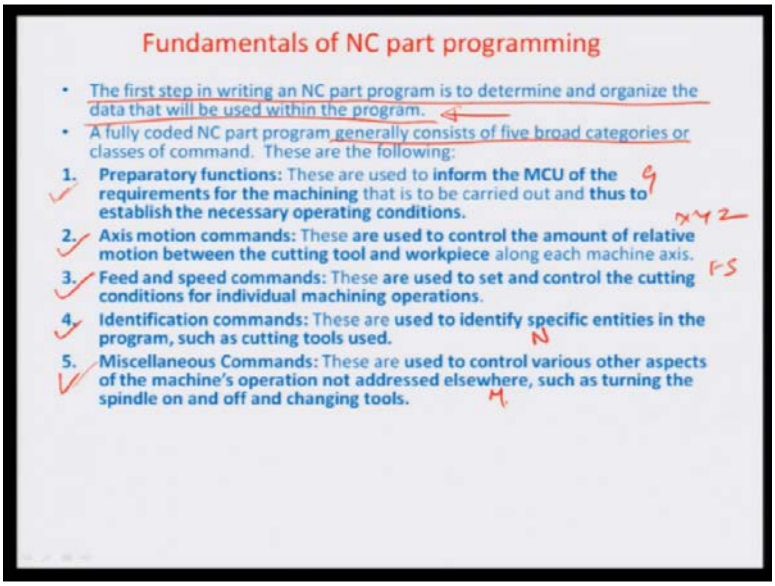
Indian Institute of Technology, Kanpur

Module – 07

Lecture – 38

Hello and welcome to this Manufacturing Systems Technology module 38.

(Refer Slide Time: 00:17)



Fundamentals of NC part programming

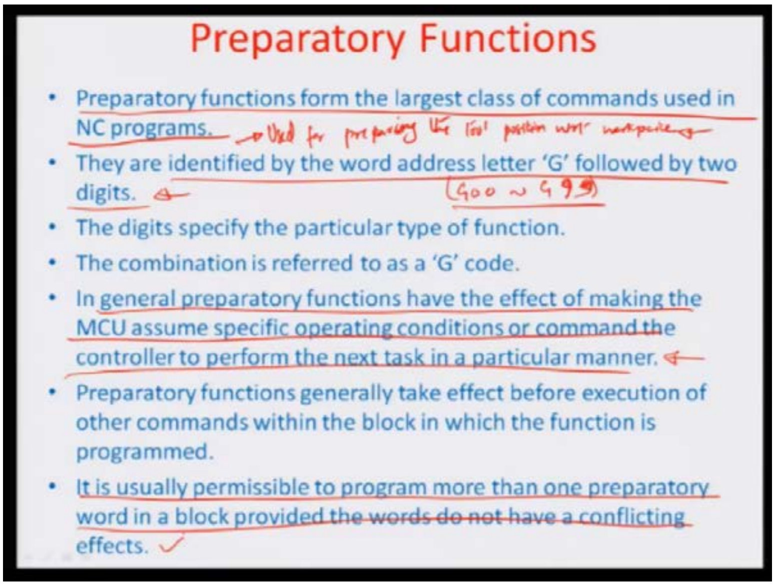
- The first step in writing an NC part program is to determine and organize the data that will be used within the program.
- A fully coded NC part program generally consists of five broad categories or classes of command. These are the following:

1. **Preparatory functions:** These are used to **inform the MCU of the requirements for the machining** that is to be carried out and **thus to establish the necessary operating conditions.** G
2. **Axis motion commands:** These **are used to control the amount of relative motion between the cutting tool and workpiece** along each machine axis. x y z
3. **Feed and speed commands:** These **are used to set and control the cutting conditions for individual machining operations.** F S
4. **Identification commands:** These are **used to identify specific entities in the program, such as cutting tools used.** N
5. **Miscellaneous Commands:** These are **used to control various other aspects of the machine's operation not addressed elsewhere, such as turning the spindle on and off and changing tools.** M.

We were talking about the various steps of NC program and we were generally concerned with what are the broad categories in which you can actually divide the different commands of an NC program. So, I think we had mentioned in the last lecture that there are preparatory functions for example, which are also known as the G functions, there is an axis motion commands which is the x, y, z kind of functions.

There is a feed speed which is F S kind of function and then there is an identification command, which is the N; obviously, the sequences as per the format of the controller that we had shown in the last module and then, there is a miscellaneous command which is defined by some M or something.

(Refer Slide Time: 00:55)



Preparatory Functions

- Preparatory functions form the largest class of commands used in NC programs. → Used for preparing the tool position wrt workpiece
- They are identified by the word address letter 'G' followed by two digits. (G00 ~ G99)
- The digits specify the particular type of function.
- The combination is referred to as a 'G' code.
- In general preparatory functions have the effect of making the MCU assume specific operating conditions or command the controller to perform the next task in a particular manner.
- Preparatory functions generally take effect before execution of other commands within the block in which the function is programmed.
- It is usually permissible to program more than one preparatory word in a block provided the words do not have a conflicting effects. ✓

So, now let us put together by first delving into the independent entities like, let say what are the different kind of miscellaneous commands or what are different kinds of preparatory functions or what are different kinds of let say even, you know the motion controls like x, y, z or i, j, k kind of motion controls. And once we have done that, then we are able to now and we understand the sequence in which we have to write the blocks independently, so that you can now do a construct of the program.

So, preparatory functions they are largest class of commands actually in the NC program. And in fact, you know these are the commands which are typically used for preparing the tool position with respect to work piece. So, this is how mostly these G commands or preparatory commands are used, you are preparing the tool position with respect to the work piece. So, they are identified by the word address G followed by two digits, two digits mean that you know there is a capability that the controller can handle about hundred such G commands.

Obviously, all that the requirements related to what is the coordinate axis in which the motion should happen or what is going to be the, you know units in which the motion is recorded, whether it is in millimeters or in inches, whether it is absolute units or incremental, all these angles need to come when we are preparing the tool motion with respect to the work piece.

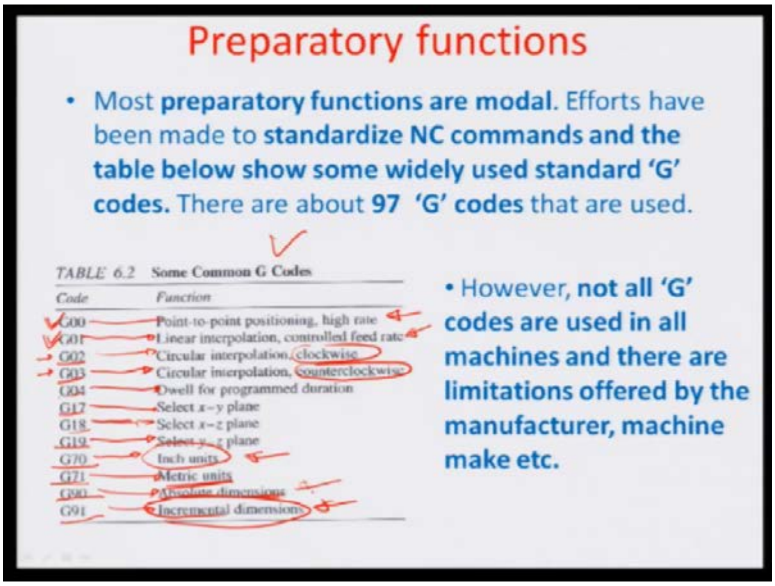
So, therefore, the G 00 to G 99 are in fact, independently may new word 100 such commands which some of them are very widely used, some of them are probably not very often used or even used only once, but they are all related to how the tool will need to get position with respect to the work piece. And this position has the definition of all the metric units or systems and whatever the measurements are in, whatever the relative motion is in, whether it is incremental or absolute, all those definition should be content typically within the preparatory functions.

So, in general the preparatory functions have the effect of making the MCU assume specific operating conditions or command the controller to perform the next task in a particular manner. So, they are basically one of the first set of commands which need to be there in a block for the block to get initiated or started in terms of it is execution. So, just following the identification number N there has to be the preparatory function enabling the system of motion, which is governed by all these stepper motor etcetera within the machine to start getting prepared to do it is job of the relative motion. So, that is why the name preparatory.

So, it is usually permissible to program more than one preparatory words in a block as long as the words do not have the conflicting effects. For example, you cannot use in one block that some G code says that the measurements is in inches and some says it is in meter that kind of sequence is not permissible; otherwise, if you want to use two different G commands, one giving the aspect of let say inches coordinates and another giving the aspect of the absolute motion or the incremental motion that kind of sequence of G in a same block is allowed.

So, you can use more than one such G commands to prepare the motion at different levels or for different aspects. But, one aspect cannot clash in a single block with respect to each other that we have to be very, very careful while using the preparatory commands or multiple preparatory commands within the single block.

(Refer Slide Time: 05:05)



Preparatory functions

- Most **preparatory functions are modal.** Efforts have been made to **standardize NC commands and the table below show some widely used standard 'G' codes.** There are about **97 'G' codes** that are used.

TABLE 6.2 **Some Common G Codes**

Code	*Function*
G00	Point-to-point positioning, high rate
G01	Linear interpolation, controlled feed rate
G02	Circular interpolation, clockwise
G03	Circular interpolation, counterclockwise
G04	Dwell for programmed duration
G17	Select x–y plane
G18	Select x–z plane
G19	Select y–z plane
G70	Inch units
G71	Metric units
G90	Absolute dimensions
G91	Incremental dimensions

- However, **not all 'G' codes are used in all machines and there are limitations offered by the manufacturer, machine make etc.**

So, let us look at some of the very, very widely used or commonly used G codes which are available. So, for example, G 00 actually signifies a point to point positioning at a high rate, G 01 similarly signifies a linear interpolation at a controlled feed. I am going to illustrate the difference between G 00 and G 01 in great details in the next slide, that what exactly these two are, G 02 is circular interpolation. So, whenever there is a need for a, you know an arc to be executed, symmetrical arc to be executed you go for this G 02 options.

So, you can have the circular interpolation in either the clockwise or the counter clockwise direction. So; obviously, this has to have two different preparatory modes. So, G 02 is the circular interpolation clockwise and the G 03 is circular interpolation counter clockwise. Then, you have a G 04 for a dwell for program duration option, that whenever you have this and you need that the two dwells in a certain place for a certain amount of time, use the command G 04.

So, there is also a G command for selecting the x, y plane. So, G 17 selects the x, y plane, G 18 selects the y, z plane, the G 19 selects the x, z plane particularly if you want to do something in a block and which has all these three different planes, it would be very easy to just often mention the preparatory function G 17 indicating, whatever has to be done is in the x, y plane something like that.

So, then you have G commands representing the basic dimensions for example, there is this G 70 for example, which represents inches units. So, now, once you have programmed G 70 every reading that the controller will be interpreting from the numbers that you have feeding on the system is a programmer would be taken in inches code and code. Similarly, G 71 which indicates metric units, where every reading that you was the programmer would be punching in the system would be taken to be in meter or millimeters actually, millimeters that is how metric system would be.

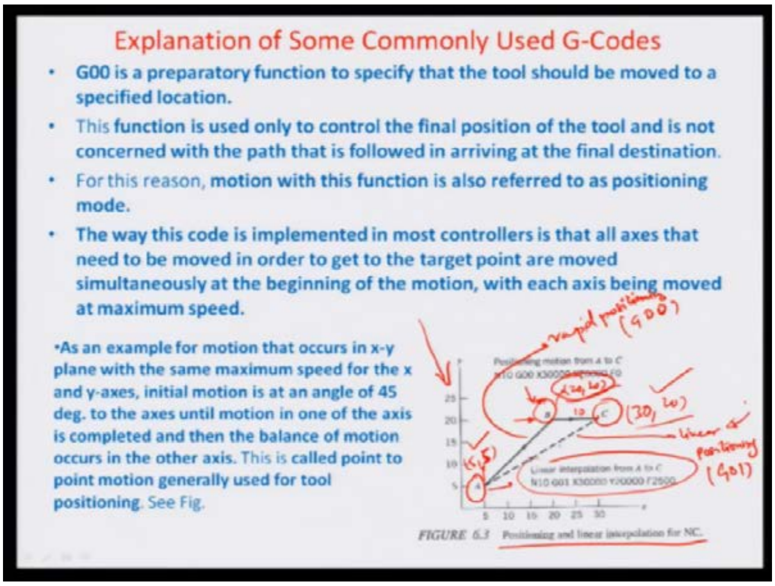
G 90 for example is absolute dimensions. So, here what it means is that I already told you before that either coordinates can be interpreted either by looking at the origin and from, you know from the origin how the coordinate is placed and it can also be looked in to as an increment or a difference between the previous position and the next position. So, in one mode it is called absolute and other mode called incremental programming.

Similarly, if the controller is capable of or if the controller is commanded to follow the absolute dimension strategy, then it is going to automatically read whatever you punching the x, y, z values as the coordinates and if it is actually in the incremental mode then; obviously, it will take the x, y, z value to be the next particular x, y, z where the tools has to be. And here, there may be a little problem, because if there are multiple tools and multiple operations happening, there is going to be a problem about cross interference between the tool paths.

So, typically all the NC controllers have an enabled simulation platform, which is associated with the machine, where wherever code you have been executing that will first get validated through this motion sequence, which otherwise gets simulated. And then once the tool paths is considered to be a good tool path, only then it is feed forwarded to the NC system or the NC machine. So, there are capabilities like y’s which are associated with all machines operating in the NC environment.

In fact, the programmer’s job becomes reasonably simple, because now he can program and see if his program is going to work. So, there is a validation which is provided by such a platform before actually finally, feeding the program in to a NC system and otherwise, if you had such interference the tool axis would we damaged. So, that has to be somehow prevented using this particular mechanism.

(Refer Slide Time: 09:16)



Explanation of Some Commonly Used G-Codes

- G00 is a preparatory function to specify that the tool should be moved to a specified location.
- This function is used only to control the final position of the tool and is not concerned with the path that is followed in arriving at the final destination.
- For this reason, motion with this function is also referred to as positioning mode.
- The way this code is implemented in most controllers is that all axes that need to be moved in order to get to the target point are moved simultaneously at the beginning of the motion, with each axis being moved at maximum speed.

•As an example for motion that occurs in x-y plane with the same maximum speed for the x and y-axes, initial motion is at an angle of 45 deg. to the axes until motion in one of the axis is completed and then the balance of motion occurs in the other axis. This is called point to point motion generally used for tool positioning. See Fig.

Positioning motion from A to C
N10 G00 X30000 Y20000 F0
rapid positioning (G00)
(20, 20)
(30, 20)
(5, 5)
A
B
C
10
linear or positioning (G01)
Linear interpolation from A to C
N10 G01 X30000 Y20000 F2500
y
x
5
10
15
20
25
30

FIGURE 6.3 Positioning and linear interpolation for NC.

So, that is what the preparatory functions are, there are also we try to just answer that question that I left earlier about the linear interpolation as opposed to the rapid point to point positioning or the rapid positioning. So, let us now look at the difference between G 00 and G 01, here is an illustration in this figure, which talks about such positioning and linear interpolation.

So, let us look at the fundamental of CNC or NC system. I had discussed that earlier that, basically an NC machine is nothing but, a retro fitted in normal machine. I had also illustrated in great details, that they are going to be principally two different control mechanisms, one is an open loop control mechanism, another is a closed loop and there I had illustrated that all the motions there in which are relative motions between the tool and the work piece are going to be there by virtue of lead screws and some rotation provided to those lead screws by motors, which can be either DC motors or stepper motors.

So, in case of closed loop control for example, it is mostly done with DC motors, because there is a resolver which needs to be used, then the motor is although height arch, you know it provides the height arch for machining etcetera. But, it is really not enabled to position accurately and there is always going to be an over shoot or under shoot, because of the inertial component of the motor itself.

Similarly, in case of stepper motor as I told it is normally use for open loop there is no such resolving sensor which would allow for the position etcetera and you already know for sure that if you given these many pulses this is exactly the precision with which the motion would take place and it will reach a point with certain precision and you forget about it, you do not really need the positioner data or position data again to sort of back track it back to the original position, because you already know that it is precisely position.

So, these are the two modalities with which all the NC systems operate I had illustrated in lot of details are earlier. Now, think of it that supposing you want to execute an x, y motion on such a tool system on a NC machine and there are for that purpose there are two leads screws which will take a table probably let say the work table and the work is moving relative to the tool in this particular orientation. So, the work table is moving some distance along the x direction in some distance along the y direction by virtue of two lead screws, one moving the x direction another moving are rotate in the y axis.

So, that there is lead screws based travel in both the x and y directions. Now, if these two motors are being operated at full speed, typically we can assume that if the lead screw pictures are exactly similar to each other and they are also both of them either a single starter or double start system in terms of lead screw and they are identical as such mechanically.

So, therefore, with equal rotations of motion per unit time the lead screws would travel linearly in equal dimensions or equal the distances. So, the x motion will be same to y motion and there you would always get the 45 degree angle across which the final table would move, because the x and y is now happening simultaneously in a particular direction. So, the table moves at 45 degrees angle in that case within the range; obviously, of the x and the y which the tool has been designed for and that is a very important aspect that I would like to consider here, when we talking about rapid position in particular.

So, rapid positioning is a system where it is typically used when you need to have motion, because you want to align the tool at the start of the machining or the start point were the machining should start to take place with the work piece. So, you want to take the tool there without much delays because; obviously, it is not machining while doing the positioning, it is merely a positioning that you are doing at that particular instant.

So, obviously, you want the tool to quickly as quickly as possible go to the start of the machining process and therefore, you need to give it rapid positioning aspects. So, having said that now with all this knowledge which I have thus discussed this over the last about 5 minutes, think of it that if you had situation where the x and y coordinates are not uniform in nature.

For example, let us say we talk about a motion between the point a and the point c on this x, y plane and there are several ways to go to c 1 is that you go from a to b and b to c; obviously, if you look at the coordinate of c, the coordinates of c are basically 30 units in the x direction and 20 units in the y direction. Similarly, the coordinates of b here or about 20 units in the x direction and 20 units in the y direction, coordinates of a being about 5 units and 5 units a and b direction respect to in the x and y direction respectively.

So, you are essentially going from the point 5, 5 to the point 30, 20 and the constraint you have is that you have to reach the position c with the rapid position; that means, utilizing the full capability of the NC system with both the motors rotating. So, that you can make this traverse from a to c within minimum possible time. So, whatever essentially doing is that till and until it permits till and until x is equal to y, it can go at full speeds on both the axis.

For example, now with a and b on both the a axis is rotating at the fullest speed and the b axis also is rotating at the fullest speed at the motor. Once it is has reached the b now, there is no more requirement to go in the y direction. So, only the full speed is operated along the x direction motion and the x lead screw goes for the remaining ten units in between. So, that it reaches the point 30, 20 from the point 20, 20 it is very important for you to gage this point that why I am calling it a rapid positioning.

So, essentially your splitting up the motion into a situation, where the x and y is would move to equal to each other at 45 degrees beyond which one of the axis is limited, because you have already reached the fullest of one of the axis and the other axis is still needing to go. So, now, you are having whatever balance you need to go on the axis with full speed of the motor along that particular axis.

So, that is how practically you are moving the x, y table from the position a to the position c at the minimum possible time although the path may seem to be as drawn here may seem to be more; obviously, two paths in this triangular figure or I mean you know length wise they would definitely sum up to be more and comparison to the dotted line here third side of the triangle, but the figure does not illustrate that factor here. The filler on the saying that what is the time advantage you will get in moving 45 degrees and moving the remaining balance by full speed operation of all these stepper motors, with the base a case where we have to position linearly from a to c.

So, the linear position really would be from a to c along the dotted line as you can see here. So, this is the linear positioning and this is the rapid positioning and there rapid positioning is basically G 00 and the linear positioning is basically G 01. So, it is very, very clear as to what our goal is here, when we talk about rapid in linear.

Obviously, linear positioning may be made in a case on the actual machining operation has began and you have to really feed it at a certain rate, because if you just simply are considerate of the over speeding factor of the motors alone for the time saving aspect, the material may not behave in a similar manner, every material has some kind of a residence time for the tool in certain places or at a certain rate only the material can be machined to for the machining to be optimum and the tool were to be the least.

So, therefore, the linear positioning is typically used in those controlled aspects, were the actual machining operation has started to take place. Otherwise, mostly for the positioning aspects etcetera the rapid positioning is formulated to be a very good tool. So, I think I am at the end of this particular module I am going to take of some few more such detailed commands, like particularly the circular interpolation commands before I proceed into actually the looking at the program.

Thank you.