Manufacturing System Technology
Prof. Shantanu Bhattacharya
Department of Mechanical Engineering and
Department of Industrial and Production Engineering
Indian Institute of Technology, Kanpur

Module- 07
Lecture- 37

Hello and welcome to this Manufacturing Systems Technology module 37. Quick recap what we did in the last lecture. We were talking about the different formats in numerical control systems. Particularly started with the 5th sequential format where there was a sequence of numbers, which were able to give you these steps you know, of reading and drawing and trying interpret a machine as that particular drawing on a surface. We also modified the format little bit and use to the tab button. So, that there could be lesser amount of Fortan the part of a programmer in terms of writing etcetera and there would be essentially a repeatability from step 1 to step 2. And some of the values which are more like of the model values would be retained. In comparison to the other format earlier it was much simpler that way and much easier for the programmer to not keep on repeating the values almost always.

So, then I said that in case you are reading a long file, several commands, a several blocks and instructions. It is really hard weather it is fixed or tab sequential format to interpret about for each and every row, unless you either memories the rows and you know exactly what is means etcetera. And supposing there is a tab it additionally creates complication because now you have gaps in between were you do not, you just simply repeating. So, if the tab continues for the last about 30 steps. So, you have to go back in 30 steps and see that what was that particular value that you operated on, you know in order to writing the next step or the next program line. So, there for this evolved new line of thinking and the latest format incidentally which is also known as the word format, a word sequence, or the word format of the NC systems. Where instead of writing any particularly only numbers you would also associate some English alphabets, and these alphabets would mean one or the other command and then it will be easier at anywhere along the program to read that line and be able to interpret about what it means actually by the programmer.

(Refer Slide Time: 02:35)

So, this was where we left with. And now I would like to just described this word adders format in a little more detail, as you can see here. You know if I talk about the American National Standards Institute (ANSI), they have established a standard method of specifying the word address data for any controller and this actually comes has more or less with the controller. So, any controller available new on the market would have to send this kind of a one lines associated with that controller would demonstrate the capability I mean just in a minute going to tell you about what really these numbers and these words mean about the capability of any controller.

So, this was adopted mostly by all the manufactures this ANSI standard and who ever supplies a controller in the market this days also give this specification, relating to the capability or the process capability of the range of motion etcetera of the controller itself. So, with in the specification a letter identifies a specific type of word as in the table, I will just show you about this the different word actually and they will give you also a small table a letter on. The A plus minus symbol this sorry, the plus minus symbol as you can see here after the letter indicates that sign is significant. So, if it is probably mentioning some kind of a direction. So, if you look at the letters which are there in this particular, are this N which can be indicate a of the step number or the step line or the line number or the block number.

So, something related to date and then you are mentioning a number 4, meaning there by that probably N4 might mean that the index number or the identification number for the particular line type that we are showing. So, this is also the step number of the block

number and the numeral N4 merely represents that the capability of this N in terms of 0000 line all the way to 9999 lines. So, you can write 1000 line of the program for this particular controller. That is what it means maximized number of lines that you can entry in the controller at single go, is about 1000. So, than there is an option G here, which represents probably the why that the axis would try to behave and here you refer to these kind of commands as go G commands. And you had earlier look at the illustration in the fixed sequential format for the drilling system, where we pointed out that there is a number just succeeding the index number which talks about how the positioning of the tool with respect to the work piece would be done. So, for example, there was a rapid position system that was illustrated there or a let us say linear position system which was illustrated there.

So, in this case all these are grouped together as set off commands called the G commands, and mostly the number G2 identifies that this will start probably from G00. And it can go all the way up to G99. So, there about 100 such G commands which actually exist in the CNC system, and the controller is capable. This line show the controller is capable of handling all those 100 commends. You also talk about the x system here. So, the x actually probably means the x axis and then you have x plus minus 43 as an indicator. So, this represents the letter x and then also the plus minus indicates what is the direction in which the x is read weather it is away from the 0 in the positive x direction or whether it is away from the 0 in the negative x direction. And this number 4 3 should mean something in terms of the again the controller capability to handle. So, it is merely number of places.

So, this actually means 4 places in front of the decimal and 3 places behind the decimal. So, basically the controller now is capability to read x value is starting from 0000.000, all the way to 9999.999. So, it is about close to 10 million values that the X can is capable of representing, as well as the controller capability goes. Similarly you have the same kind of capability illustrated in the Y and Z, meaning there by the Z is capable of reading either side in the positive as well as negative Z axis direction and it can read about 10 to the power of 7 about 1 million value starting from as I told you 0.00., 0000.000 to 9999.999. It is merely indicating the number of places up to which is the controller has a capability to read. So, mind you this has to be develop as a part of the hardware, which the controller would have and that is why it is important to the manufacture it should mention. This one line of the specification which tells all about

how much the code is capable to get execute or how much of the code can be executed through this controller, given the capability of the controller etcetera.

So, there is also is you can see another line or another what which mentions R plus minus 4 3, which talks about probably the radius had some point. We also having a word F plus minus 40. So, all these have some of the other meaning, and for that what I would like to do its short of share with you a small you know, list of short of words which probably can tell us about the details behind the use of such a word adders a format.

(Refer Slide Time: 08:42)

So, I would like just to sort of go back to this one table here, to mention about the various things related what you saw in those words. So, Fmeans the feed rate command, the G means the preparatory function or the as you can commonly call G command basically, which is about all these rapid positioning linear positioning so on so forth. The I, J and K this signify circular interpolation and I will mention in details about what really interpolation is meaning. So, there is X axis of set, there is a Y axis of set, and there is Z axis of set for which you need 3 independent coordinates or 3 independent values of I J K. To really determine the shaper the nature of the curve you know, round the centric center position; this is more like spherical coordinates we are talking about. So, I will illustrate this in great details just little bit later.

Then M basically means miscellaneous commands where all these coolant on off, these kind of commands are basically recorded under the miscellaneous commands. I will again go back when we complete or this ANSI programming process, I will again go

back to some of these words and illustrate the details of how many miscellaneous commands can be available, how many G codes can be available, what are the interpretations of those once and so forth. Obviously, the sequence number is given by N. So, this is the principle identification number or the block identification number that is being recorded, so the sequence number. R records the arc radius. So, supposing there is circular interpolation you are wanting to execute, then you have to go along the radius of certain arc given a central position and given an offset value that radius sometimes comes automatically because of the dimension of the I J K. So, you need not a specifically mention the arc in some places, but in other cases when have a point about which you have a describe an arc, you better prefer the other method that you give the coordinate of that point and then mention an arc radius. And then somehow you have to also know whether the arc is in positive or the negative direction depending on the direction of the axis system.

So, I think I had recorded earlier with you that if supposing the axis, Z axis is in particular direction positive and if you use the right hundred grip rule you get the exact value of the arc rotation direction. So, basically the fingers show the positive arc rotation direction given this is the positive axis that we are talking about.

Then S stands for the spindle speed, T is tool number; obviously, there are going to be many tools in the CNC system and there is also attendance of the CNC system to pick and choose certain tools and preside one operation followed by another operation etcetera. And then; obviously, the XYZ data actually represents the coordinates which are red on a drawing where the tool has to access or from one place to another the tool has traverse on the particular drawing. So, this is illustrated by XY and Z such convent formats. So, given this and given the now the understanding that you have from this particular, you know sequence of word addresses and specifications.

The controller is totally now capable of sort of giving you an idea of what is the range across which it can operate. T2 merely illustrates. So, the whole controller specification is sort of defined by this one particular line. So, T00 to t99, means 100 to links can be accommodated with that particular machine. M2 again indicates M00 to M99 meaning there by about 100 miscellaneous commands can exist an; obviously, this IJK that I will probably detail little bit latter is about how much would be the circular interpolation offset value in comparison to the arcs radial center. So, I would talk about this and great details little bit later. So, that is about the word address format.

And so, basically now we want to looking to the first step in writing an NC program. Once this controller specification has been given it comes with the machine by the by. So, you have to have prior knowledge of the controller specification before being in able to start programming on the NC machine or NC system. So, the first stepping writing and NC part program is to determine and organize the data that will be use to within the program to reflect the exact machining operation that you want to finally, want the machine to continue. And that is really defined by the part drawing or the dimensions of the part that needs to that would be machine in the by the NC system.

So, the fully coated NC part program, basically consist again of high broad categories or classes of the commands. So, as you already saw that you know, one of the identification commands, the N command which is starting any block or any line of the program. The other one is probably a preparatory function which allows the machines tools to position with respected to work piece and N number of ways or manners. You have access motion command which talks about exactly the relative amount of axial motion along the X Y and Z direction and if it is in absolute systems it just keeps on reading the coordinates, with respect to the origin. And it is the job of the controller to now do the path planning from going from one coordinate position to another coordinate position. The program is not in able to program is not required to give; you know the exact path etcetera. The optimization is carried out by the controller itself. So, all you need to bother is an absolute in absolute program as programmer is only the absolute coordinates with respect to the origin.

So, you also want to give the feeds and speed comments like for example, you already saw that feeds of so many mm per revolution or speed of so much RPM of a particular tool has to be executed. So, there are different command FNS for that particular thing. And that finally, the miscellaneous commands which are about the associated systems like the tool start stop or the coolants start stop those kinds of activities. So, these are the five principle categories and which an NC program can be split up or it can be divided. So, I am going to now sort of go into a dull a little bit later a into each of these categories an share with you a few commands which exist for example, the preparatory function there N number of a function which are there. As you saw that the controller is an able to handle from G00 to G99. So, what those individual commands mean, to some extent I will be able to give you an idea and at least the most widely used I will probably discuss in this particular session.