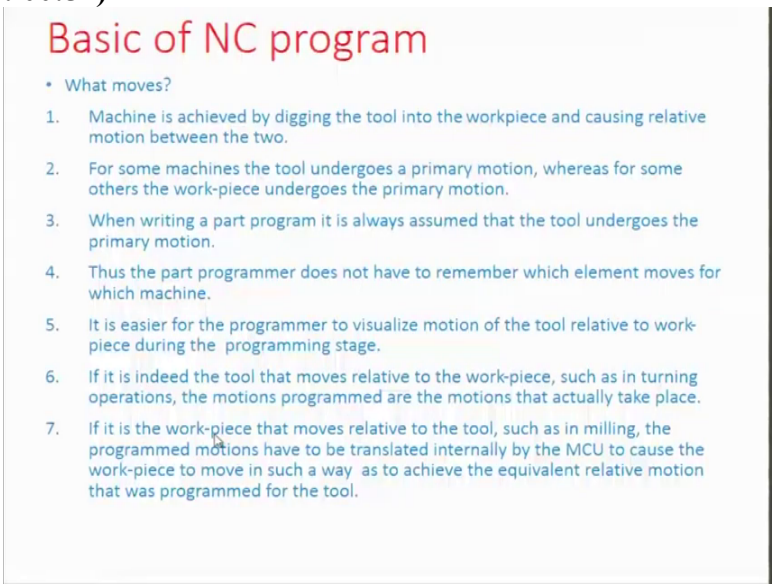
Design Practice - 2
Prof. Shantanu Bhattacharya
Department of Mechanical Engineering
Indian Institute of Technology-Kanpur

Lecture - 27
(CNC Tooling/ Rapid Tooling/ Rapid Prototyping)

Hello and welcome to this design practice to module 27 we were talking about computer control of machinery. In context of that we learned about how you have different orientations of control systems how you can change the relatively the positions in between such a retrofitted and CNC machine. We were about to go into the next step of how do you talk to such controllers develop an organized language to talk to controller. So, what is it that a basic NC programmer must know about the program.

(Refer Slide Time: 00:51)



So, the first aspect of any such programming knowledge that a person should have is about what moves. So, as we know that in metal removing machines or metal cutting machines typically the tool is dug within the workpiece and then there is a relative motion in this orientation with the tool digging into the workpiece and so because of that relative motion there is going to be a metal chip which comes out and it can be a continuous shape or a discontinuous chip depending on the relative the speed of the relative motion and also the depth of card and a variety of other things.

So, the first knowledge that NC programmer must have is what moves in the machine whether it is the tool or whether it is a workpiece. So, there may be different kind of machines some in

which the tool undergoes the primary motion there can be other machines where it is the workpiece which may undergo the primary motion whatever it may be one must assume is a part programmer that it is the tool which would undergo the primary motion.

So, in case it is an optimization problem in case the controller realizes that you know the program is written for principally incorporating tool motion. But it may mean a lot more tool path then automatically in some cases the workpiece motions are executed. So, that is how the controller has been set up and built and there are some algorithms which would give you an option of deciding whether it is going to be the tool of the workpiece which would move.

But that is of no significance or no concern to the programmer who always assumes that he is a part of the tool and moves along with the tool. So, this is a very important aspect somehow this strategy has been done to simplify controllers which are available internationally and in any business which is being followed which have known as standards which kind of promotes interchangeability and other aspects.

And so therefore you can think of that there is an international convention that any part programmer should consider that is the tool path that he is programming. So, I think one of the main reasons why that is so is that the programmer is easy; it is easy to visualize the motion of a tool for a programmer relative to the work then the other way round. So, if it is indeed the tool that moves relative to the workpiece suggest.

Let us say the turning operation the motions programmed are motions that actually take place if is the workpiece that moves sometimes relative to the tool. So, just let us say the milling operation the program motions have to be translated internally by the MCU it is none of the programmers business because the workpiece to rotate. So, this can achieve the equivalent relative motion that was programmed for assuming that the programmer was the part of the tool.

So, that is one very basic aspect of NC programming so the other question that needs to be addressed is where is the tool? Now the NC part program includes among other things commands that move the tool to various locations relative to the workpiece. And if it is you know if you are able to specify the required tool positions obviously these specifications need to be with reference to some points which are known as the origin.

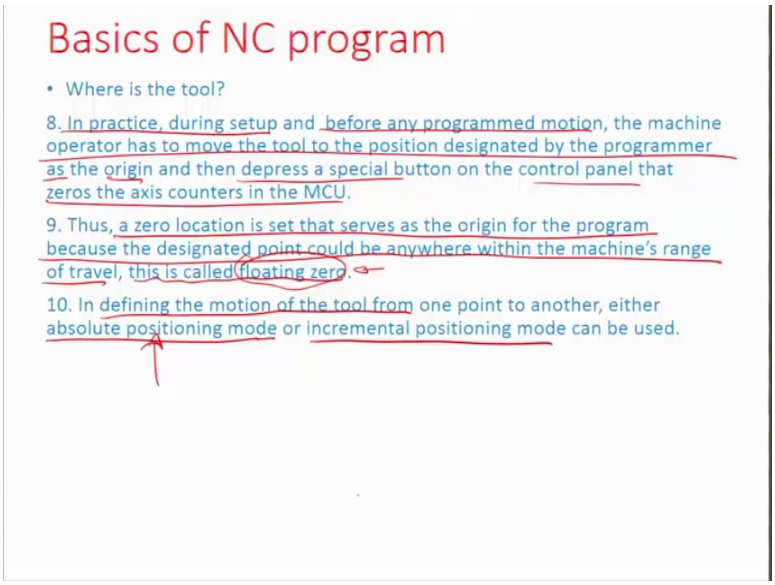
All the geometric entities which are defined in terms of the tool path okay and the motion of the tool is a progression in time would be with reference to that particular origin and so there has to be some fixed point in space which is considered by the machine control unit to be origin and all of the motions are executed relatively of the tool with reference to this origin. Now for single tooth cutting tools, let us say for example an insert; we just used to turn, let us say a certain circular workpiece cylindrical workpiece.

It is probably not a very big issue to define the point which talks for the engagement between the tool tip and the workpiece and it is not very difficult to then track how that point moves in space as a function of time. However there are many instances within particularly CNC machines because of high-speed operations which utilize multiple cutters or tool with multiple cutters multiple teeth tools.

For example look at for example different drills or milling cutters in fact some of the inserts are also present like a circular battery of tools and so therefore in such cases the point of defining the tool is actually the center of such a cutter and the cutter has a finite radius. So, the engagement which is taking place at the edge between the one end of the tool and the workpiece is that a certain radius from the center and that has to be somehow compensated in the job of a programmer.

So, there are certain machines which would typically have a command for tool compensation but vast majority of the cases the tool the programmer may need to actually account for that radius and he may consider himself to be sitting in the center of such a multiple teeth cutting tool where he is trying to map his own movements with respect to the work piece as machining carries out. So, that is one another of these important aspects important for a programmer to know.

(Refer Slide Time: 07:12)

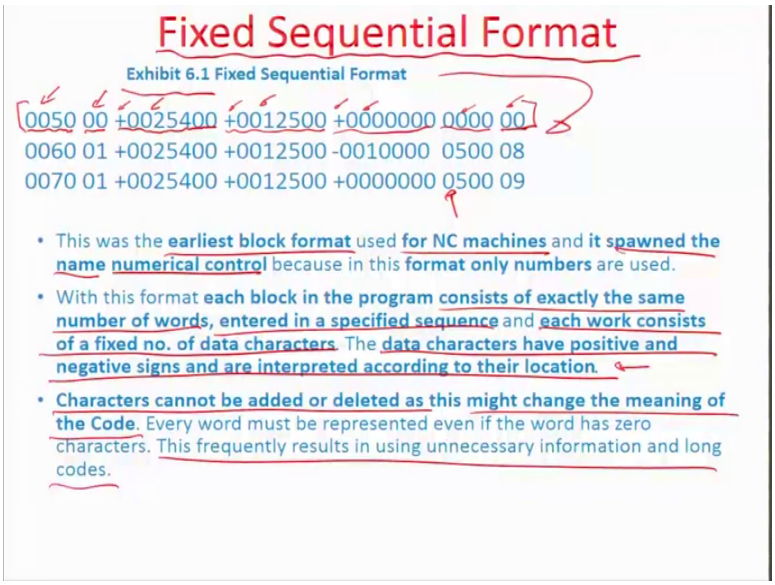


So, in real practice during setup or before any program motion the machine operator has to move the tool to the position designated by the programmer as origin and this can be also done by just depressing a special button on the control panel that zeroes the axis counters in the MCU. Therefore a zero location is set that serves as the origin for the program because the designated point could be anywhere within the machines range of travel.

So, in case you are setting such a zero with the controller okay where the existing point is the origin without any movements back to the real origin which started the whole operation it is otherwise known as a floating zero. And if you want to obsess absolutely move back every time to the place where it was initially docked it is called a fixed zero. So, based on this concept you know in terms of defining the motion of the tool from one point to another you can either use so you have two broad modes which you could use one is called the absolute positioning mode which is in reference to a fixed zero.

And the other is actually an incremental positioning mode which can always be in reference to the last position which could then be treated as a floating zero for the next operation to take place. So, these are some basics which are needed for any NC programmer to start understanding and now let us look in the history of programming how numerous numerix were used in order to define or control through various commands some applications or some specific applications related to controlling of the tools.

(Refer Slide Time: 09:08)



So, when we look back in history we find out that there are three different formats which are by and large emerged for giving some giving a set of commands to the machines one of them is called the fixed sequential format which is given in this exhibit right here you can see a set of numerical values here which talks about this format. So, incidentally this was one of the earliest block formats which was used for NC machines and this is from which actually the name numerical control spawned because it was all numerix as can be seen in this exhibit right here.

Format formats only made up of numbers so one can see that there are a group of numbers for example there is the first group of number here there is this is another group of numbers this plus 0025400 could be another group of numbers so on so forth. So, there are actually about in this one line about 7 different groups and each of them must be having a certain purpose we should like to know ok you can even think of this as a big table with different columns and rows where the columns are corresponding to certain aspects of control related to the machine command.

And the rows are again and the sequence of some of these aspects which would carry out a certain machining operation. So, one can see some very peculiar features of this exhibit one of them being that each block which is actually one line okay in the fixed sequential format contains the exact similar number of words which are these groups okay just as in a sentence what is constructed you have different words which come in a certain sequence to start making sense to a person you can think of a language being developed in identical manner through blocks and number groups.

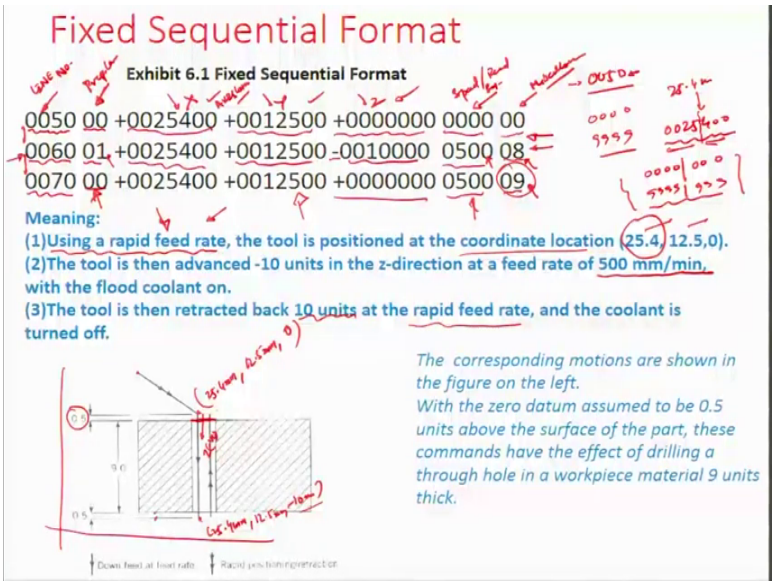
Where block kind of represents the sentence and the number groups represent the different words which go into the sentence. So, they are entered in a specified sequence just as words are interred within sentences and each work of course consists of a fixed number of data characters and as you can see that there are positive and negative signs related to this data which kind of is indicative of the directions in which controller would be able to move the tool with respect to the word piece you have already done the positive negatives that x and y directions respectively.

So, obviously you know the plus and minuses would indicate to those directions. So, the data characters have positive and negative signs interpreted along their locations if we are talking about recording everything with respect to a fixed zero or an origin we use the coordinate frame centered about the origin where there are negative coordinates and positive coordinates and so we can give where the tool moves from the positive to the negative quadrant as moving from the positive coordinate to the negative coordinate in a certain direction characters cannot of course be added or deleted this might change the meaning of the code.

So, every word must be represented even if the word is 0 or what does not have any character to represent so you can omit. And this frequently results in unnecessary information very, very long codes after they start running into a few pages. It is probably very difficult to refer to the beginning of the column and what was the purpose of the particular column. And for programmers who are not so trained and knew it may be horrendous activity every time keep filling row-wise manner different columns, we do not have any headings.

Program spell run as high as thousand pages in some applications so there are certain aspects which are probably not so suitable to the fixed sequential format.

(Refer Slide Time: 13:47)



For which there was a so there are the format's which have generated which would try to take care of some of these issues. So, let us actually look at the meaning what this group of numbers would mean. So, the first number here in this particular format signifies the line number and you can see that there are some spaces in between left over at least 10 space between the first in the second number. And one of the reasons why that is so is that people can modify people can add as many lines as possible without really jeopardizing the whole code.

Because when the controller reads it reads increasing sequence of numbers and so that is one very big advantage of the system. The other issue is that if I looked at you know what the capability of the controller would be every controller has a certain capability aspect and that is in terms of numbers. So, if I look at the number 0050 what it basically means is that the controller has in this particular column the option of going from 0000 to 9999 exactly 10000 different lines.

So, these numbers and the way that they have been laid out do signify an aspect of capability for example the second column here so it refers to the preparatory command. And similarly there are these numbers which talk about the individual coordinates so these are probably corresponding to the different axis which are involved in their coordinates and while representing this we can see that is represented in a very peculiar manner 0025400 okay.

So, there are so many numbers to denote so basically in this particular case it is giving a controller capability in terms of how many places would be there before the decimal and after the decimal. So, if I want to represent 25.4 millimeters as is going to be the position coordinate in this drilling operation by 0025400 it basically means that the controller has a capability of

accommodating right from 00008.00029999.999 so many different numbers can be accommodated by the controller.

So, you can have variety of different coordinates of XY and Z as can be represented in these three subsequent columns okay from the preparatory command. So, you have a line number you have a proprietary command, now you have axis commands or coordinates you know so you can call it access commands. And then of course you have something related to some basic operation which is indirectly related to the machining or metal removal but not directly related.

For example it could be the spindle on or off, the coolant on or off or something like that and so this can be or may be something to do with the feed rate rpm etcetera. So, this could actually be the; would be recorded as could be recorded as the feed of the speed command. And these are some other operations which are for example related to the coolant or any other operation which are important yet not the main operation okay and these are known as the miscellaneous commands okay.

So, these are the you can say that these commands are corresponding to the speed and feed setting aspects and these commands are corresponding to miscellaneous aspects of machining so there is a certain way a fixed way in which numerous numbers are in a line by line basis able to give a sequence of operation this code right here is a sequence which is defined for a simple drilling process. And if I look at what is going to be the tool path like following this code so in the first line it says using a rapid feed rate which I am going to define later quite well.

So, there is going to be a linear and a rapid Positioning System which we will talk details about the tool is positioned at the coordinate location 25.4 and 12.5 mm and 0 mm respectively. So, what it basically means is this particular point which with respect to the origin corresponds to an x coordinate value of 25.4 mm y coordinate value of 12.5 mm and a Z of 0 mm okay with reference to the origin again is where the tool has to be positioned.

So, let us say the two lows position somewhere here corresponding to some other address. So, it would definitely read the overall X Y Z motion and come to this particular position at a rapid rate. One must remember that when the tool is not being engaged into the workpiece you could use a very rapid speed, rapid feed kind of a strategy. But if it is engaged then obviously the rate of material removal will define its speed of feed and so you cannot just rapidly position as you

may have luxury to do at the very beginning when the tool is still not started in interaction with the workpiece.

In the next command as you know a set of commands as you know in this particular line it is referring for a linear feed okay. So, because there is a linear feed the feed rate has to be defined so is not defined as 500 mm per minute and of course there is a coolant on option which is already there we assume the drill to be moving along its spindle or rotating along its spindle already there is no separate spindle on command which needs to be used in this particular case. So, from this coordinate 0000.000 it is going exactly 10 units in the negative Z direction as you know that the Z direction corresponds to decrease between the tool and the workpiece.

So, this particular gap which was there from this position to the surface of the tool here is now decremented. So, this is a negative so you go exactly 10 units downwards and while going 10 units downwards you are crossing 0.5 unit of clearance that the tool had earlier and again going all the way to 0.5 away from the lower surface of the work piece so that you ensure this is a through-hole okay.

And so this logic is being provided in the second step right here so the block size may be about 9 mm and you need a 10 mm movement for providing a through-hole on the 9 mm block. And then in the third line you are retracting the tool back to the 0 position. So, basically whatever -10 position the tool had gone which is somewhere here keeping the 24. 25.4 and the 12.5 X and Y positions intact okay so you go to -10 mm here okay.

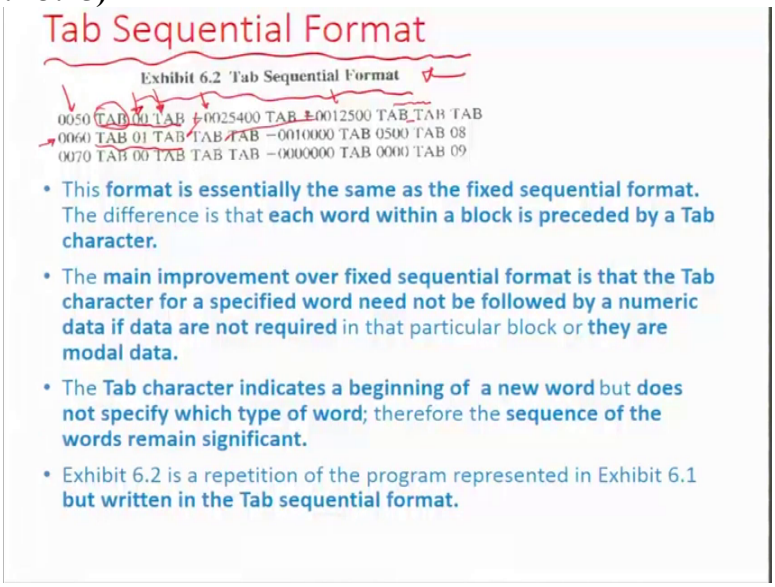
And then go back again so while going back just because this is -10 mm and you probably consider at origin 0 to be this point, so you will have to go back to the origin okay. So, you can go back from -10 to +0000.000 so this is actually the absolute frame of reading wherever it has been the origin you are reading everything in terms of coordinate map from that particular origin the origin was at 0 and it has been you know from 0 you have gone in the negative Z directions 10-unit.

And then again come back 20 meaning thereby that you have moved the tool from the deep interiors of the workpiece back to the origin position so, you are reading everything with a fixed frame in mind and not changing incrementally as a function of where is the last motion or where

is the last position of the particular tool. So, obviously the rapid feed rate can be utilized here so therefore you have already changed the preparatory motion to rapid rates.

And you can also use again 500 millimeter per minute feed because training has already been performed with maybe once you reach the zero you can actually use the coolant off strategy to save the coolant etc. So, that is how you can actually look at this fixed sequential format and numerically be able to program a very simple drilling process. So, I think it is kind of clear to you what we are trying to do or trying to achieve using this kind of numeric program.

(Refer Slide Time: 23:25)



And there are many shortcomings of numeric programs for example reputation rules or on the major short comings because of which this new exhibit 2 here right about here which is the tab sequential format card into existence. So, in this particular format the only difference it makes is that whenever there is a change between the previous line and the next line you need to enter the change.

For example there is a change in the first number from 005020060 so you have entered the change and in order to ensure that this column is no longer the active column you use the tab key on your keyboard so that the cursor can go to the next column right here okay. So, in the next column you fit or you put the preparatory function for the Machine axis to start preparing to move ahead.

Again press the tab key on the liver the keyboard which takes you to the next level that is the axis commands. So, you have positioned yourself at +00 25.400 + 012.500 and you know

wherever there is no requirement like for example either the origin is origin is there a zero is there or for example you do not need to change the entry you just simply press two tabs so that it skips or bypasses the particular column and goes to the next step.

So, now you do not have to really start writing again and again if you have two tabs in sequence it automatically takes the previous value. For example in this case the x axis motion and so the y axis motion also the z axis mission has been initiated by double tabs which takes the last value and you really not worry about re-entering into another line and the program format becomes a lot simpler in comparison to the sequential format.

So, that is one step advantage that the tab sequential format would have I am going to close this particular module in the interest of time and the third module we will advance some more and try to talk about actually trying to use some of the earlier formats to frame what we call the word address format which is the latest in the run and try to program using that thank you very much till then.