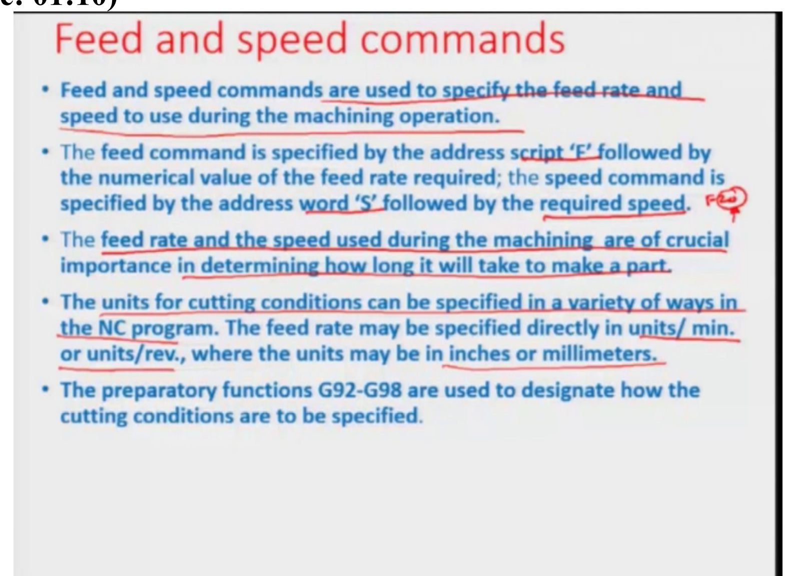
Design Practice - 2
Prof. Shantanu Bhattacharya
Department of Mechanical Engineering
Indian Institute of Technology-Kanpur

Lecture - 29
(CNC Tooling/ Rapid Tooling/ Rapid Prototyping)

Hello and welcome to this course design practice to module 29 we were talking about CNC tooling rapid tooling rapid prototyping these different areas and topics and also we were discussing about how to do CNC programming where we had learned about the different formats namely the fixed sequential format, the tap format and the word address format. In context of that we had also looked at the American National Standards Institute code of how you define controller capabilities with different words to talk about the different aspects of the CNC program.

As you may recall a program contains of statements or blocks and each block contains certain commands. So, it is pretty much a language that you are developing with a certain controller.

(Refer Slide Time: 01:10)

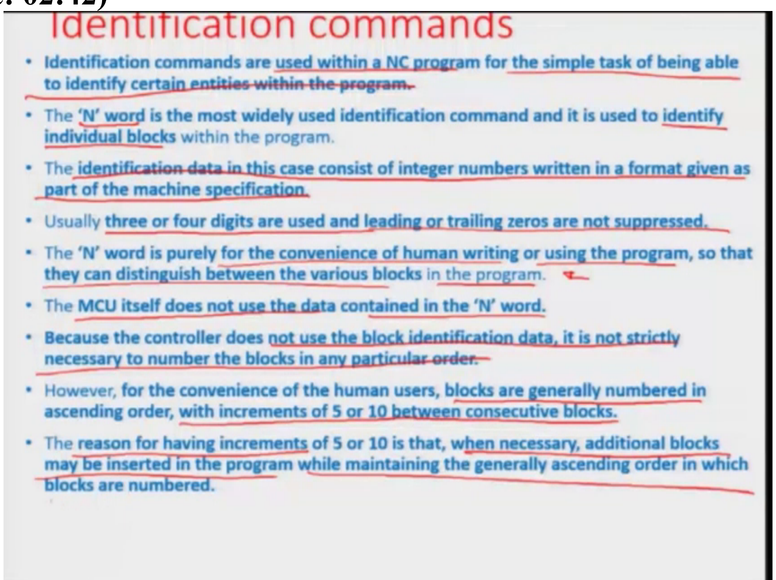


So, we will continue on that and try to look at some of the other aspects of that word address format and the one of the important aspects that such format has is feed and speed commands. These commands are used to specify the feed rate and the speed to be used during the machining operation from the feed command is generally given by the script F and the speed command again given by the word S and it is followed by the required speed which can be mentioned as numerals.

And if I say S20 or F20 it basically means that this 20 is mentioning the ability each measuring the number which corresponds to the initial value of the coordinate system which has been put as G 90 or G 91 where it could be inches units or metric units respectively. So, the feed rate and the speed used during the machining are very crucial in determining how long it would take to make a part units of cutting conditions can be specified in a variety of ways in the NC program.

Feed rate may be specified directly in units per minute or units per revolution where units may be in inches or millimeters. Preparatory functions 92 to 98 are used to designate how the cutting conditions are to be specified so this kind of gives you an idea of how you prepare the axis before starting the machining operations.

(Refer Slide Time: 02:42)



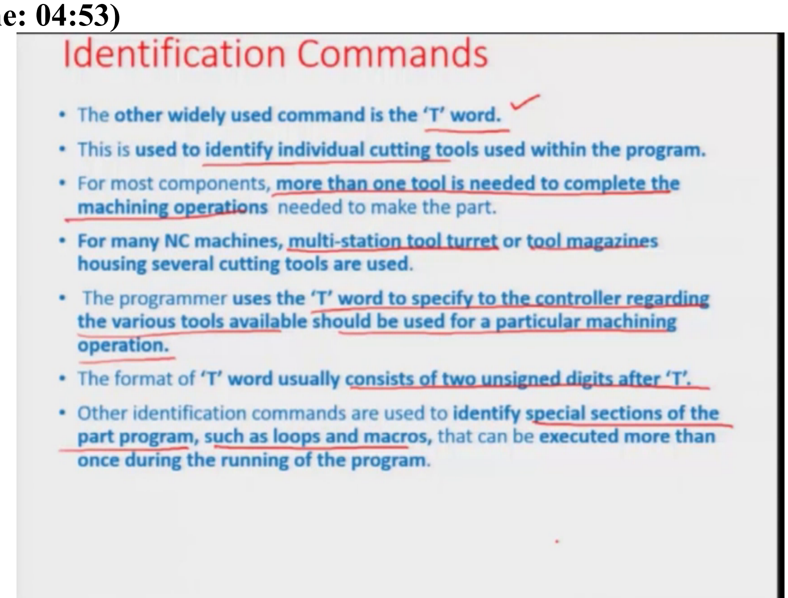
Next in the line comes identification commands these are used within NC program for simple task of being able to identify certain entities within the program. The most common identification command is the N-word which is used to identify the individual block or the line along the whole descriptive which is called a program. So, you typically have values of N which leaves gaps in between so that it could insert in a increasing sequence as many modifications as possible to a certain set code which has been drawn out once the program is freezed.

So, the identification data in this case consists of integer numbers written in a format given as a part of the machine specification usually 3 or 4 digits are used and the leading or trailing zeros are not suppressed the N-word is purely for the convenience of human writing or using the program so that they can distinguish between various blocks in the program in fact when the NC controller numerical you know executes the numeric control they will just go on reading out in the organized sequence of how they are outlaid in the program okay.

So, they really do not need the command line number but for a programmer who is there on board one needs to understand what is corresponding to a command line and between which to command lines you want to specify some improvement or some modification that you want to make to the code and for that this N word is a very important aspect in the in the block itself. So, the Machine control unit itself does not use the data contained in there. It is only for the operator and the programming sake okay that this N word is being used.

So, because the controller does not use the block identification data it is not strictly necessary to number the blocks in any particular order however for convenience of human users blocks are generally numbered in ascending order with increments of 5 or 10 between consecutive blocks and the reason as we all are aware of having increments of 5 or 10 is that when necessary additional blocks may be inserted in the program while maintaining the generally ascending order in which the blocks are numbered.

(Refer Slide Time: 04:53)

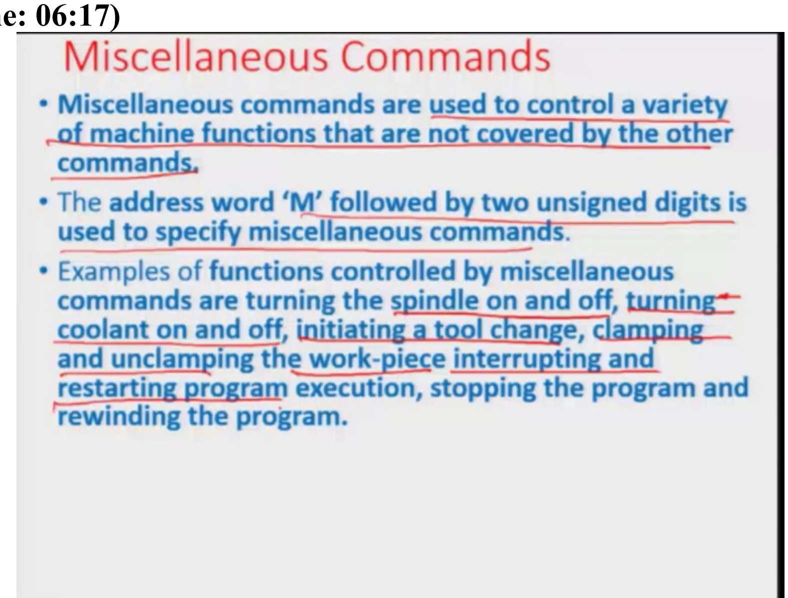


So, this is all about identification commands there are some other commands which also hour of the identification type for example the T word represents the number of tools. So, basically you could have a certain tool number let us say T01 or T02 in a magazine and you may be able to choose one over the other using this particular word here in the program so you get to implement that a certain set of blocks which are there in a sequence manner is using what particular tool or what particular tool number ok in a magazine of tools which is otherwise not distinguishable unless there is some kind of identification code there.

So, you used to identify individual cutting tools ok within the program more than one tool is needed to complete the machining operation in most of the components and therefore it is important that we have a magazine of tools ok and so therefore identification number serves as that particular number which will identify one tool over the other. Obviously these are used widely for multi station tool turrets or tool magazines.

So, the programmer uses that you were to specify to the controller regarding the various tools available should be used for a particular machining operation in the format of T words usually consists of two unsigned digits after T. Other identification commands are used to identify special sections of the part program such as loops and macros that can be executed more than once during the running of the program.

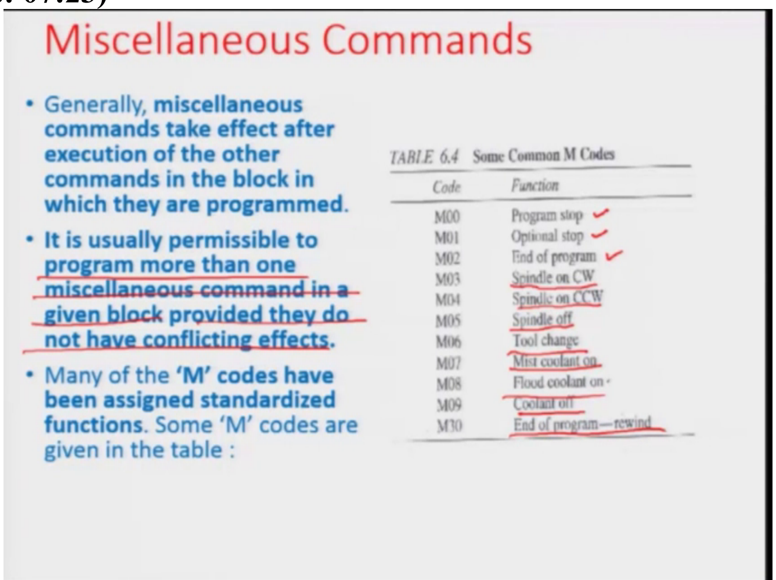
(Refer Slide Time: 06:17)



We also now talk about miscellaneous commands miscellaneous commands are majorly used to control a variety of machine functions that are not covered by the other commands. The address of M followed by two unsigned digits is used to specify the miscellaneous command and examples could be many for example there could be a spindle on and off there could be turning coolant on and off which are directly or indirectly related to the machining operation of the process initiating a tool change.

For example clamping and clamping a workpiece so these are commands which are of some importance winds once it comes to the whole machining setup although they are not directly involved in the machining process okay. And so therefore you can you can say that these are essentials for a program to be executed although they may not directly participate in other case could be interrupting and restarting the program execution for example or stopping the program altogether, rewinding the program to a certain level okay. So, all these things come under the purview of miscellaneous commands.

(Refer Slide Time: 07:23)

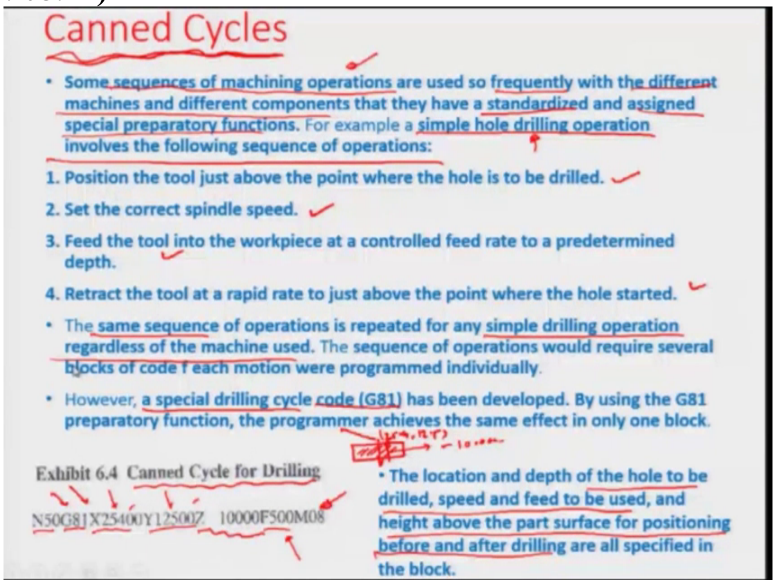


In fact if I look at what are the common M codes which are there normally for the standardized programming format M 00 suggests program stop M 01 is optional stop 02 is end up program. You can also start this spindle counterclockwise or clockwise so clockwise becomes M03 M04 becomes counter clockwise you can also do spindle off that means stop rotating the spindle perform a tool change using M06 mist coolant on or a flood coolant on mode you know are generally a coolant off mode as M 07, 08, 09 so on so forth.

And then end of program rewind is M 30 so these are some different M commands miscellaneous codes which have been characterized. So, it is usually permissible to program more than one miscellaneous command in a given block provided that there are no conflicting effects. For example in a certain block you can say coolant mist coolant on and then you it is not a better idea or it is not a good idea to do the flood coolant all as well because they will have conflicting interests and there may be an error in reading.

So, unless those kind of you know Co factors are arrived at you can have more than one miscellaneous command in a single block.

(Refer Slide Time: 08:44)



I would also like to now come to a certain different aspect which is about canned cycles. Now one must understand that the whole drilling cycle what we did in the last you know few slides can be represented as one unit or one command okay. So, every time there is a drilling operation you need not go to the different aspects of okay take the drill at a certain position you know dive into the workpiece by having a Z negative value up to a certain extent and then come out of the workpiece okay.

So, all these things can be avoided by using some kind of a canned Constitution where all these things are put together under one command you know or one let us say command line which will execute the whole drilling process. So, the canned cycles are particularly used to define sequences of machining operations and particularly those machining operations which are very frequently used okay which is quite unnecessary to every time start writing a subroutine of how a particular machining operation will work rather it is better to define it as a G code.

So, that the whole G code would be instrumental and the line there and would be instrumental of giving the whole idea of how the machining would happen in that particular operation is to be executed okay. So, it is sequencing the machining operations which are frequently used with different machines different components. And you are kind of standardizing some of these machining operations okay and assigning some special preparatory functions.

So, that every time you use the function and the command block it signifies okay. Let us look at for example I was just talking a little bit earlier about a simple hole drilling operation you saw in the fixed sequential format how the hole drilling operation was carried out on a particular block, just in the last lecture. So, it involves the following sequence of operation here. Let us say the first sequence is that the positioning of the tool is made just above the point where the hole is to be drilled.

Then you set up the currents correct spindle speed okay and then feed the tool into the workpiece at a control feed rate to a predetermined depth and retract the tool at a rapid rate to just about the point where the hole started. So, these four commands are implemented in sequence when we talk about a drilling process. So, position the tool set the speed that means rotate the spindle start feeding the spindle inside for the drilling action to happen then retract the tool okay just so that you can you know you can keep the coordinates in a manner so that the whole can either be a through-hole or just you know blind hole depending on what is the thickness of the block and thickness of the proceed of the drill.

So, all these things can be executed exactly at the same sequence okay for any simple drilling operation whatever it may be regardless of the machine used by just setting up the numerical values every time like the coordinate can be different where the tool has been docked or the proceed can be different by how much is the tool has to be proceeded into the workpiece and remaining everything being the same.

So, the sequence would all operate if you design a special drilling cycle okay which is of course G81 and let us say in this particular cycle you mentioned that at a certain command block number the G81 code exists corresponding to the initial X and Y position and I am taking the same values as the fixed sequential format was describing. Let us say 25.4 mm and 12.5 mm of X and Y was the first point over which the drilling was being carried out.

If you remember there was a block in this case and we were trying to drill from a certain point which is somewhere here which reads 25.4 and 12.5 XY and then going into this whole thing and coming out back okay and the tool was placed somewhere here from which you had to come to this particular position. So, automatically the tool from its talking position comes to this and then goes to an extent of $Z = -10$ millimeters and comes back okay.

So, the feed at which it should go is also defined and you know that is M 500 and M 08 would correspond to the you know the flood coolant on case where you know when the spindle is on one has to ensure that there is proper heat transfer etcetera. So, that there is no fractured tool fracture which takes place. So, this in a way is an example of a canned cycle okay. So, the whole sequence is now programmed under one line the location and depth of the hole to be drilled the speed and feed everything is in that same line.

The height above the part surface for positioning before and after the drilling all specified in this particular block one block. And automatically the code is designed in the manner so that it will understand what are the XY is and what is the Z and what is the sequence at which G81 needs to be executed corresponding to a special drilling cycle.

(Refer Slide Time: 13:50)

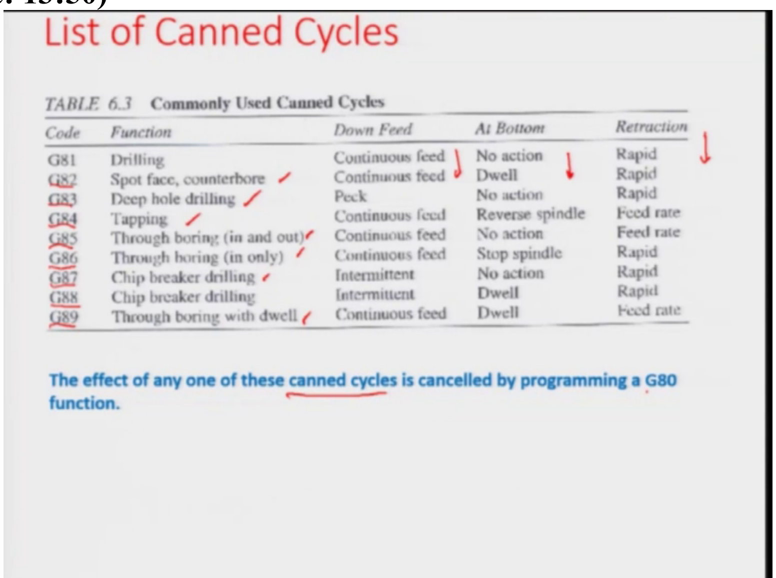

List of Canned Cycles

TABLE 6.3 Commonly Used Canned Cycles

Code	Function	Down Feed	At Bottom	Retraction
G81	Drilling	Continuous feed	No action	Rapid
G82	Spot face, counterbore	Continuous feed	Dwell	Rapid
G83	Deep hole drilling	Peck	No action	Rapid
G84	Tapping	Continuous feed	Reverse spindle	Feed rate
G85	Through boring (in and out)	Continuous feed	No action	Feed rate
G86	Through boring (in only)	Continuous feed	Stop spindle	Rapid
G87	Chip breaker drilling	Intermittent	No action	Rapid
G88	Chip breaker drilling	Intermittent	Dwell	Rapid
G89	Through boring with dwell	Continuous feed	Dwell	Feed rate

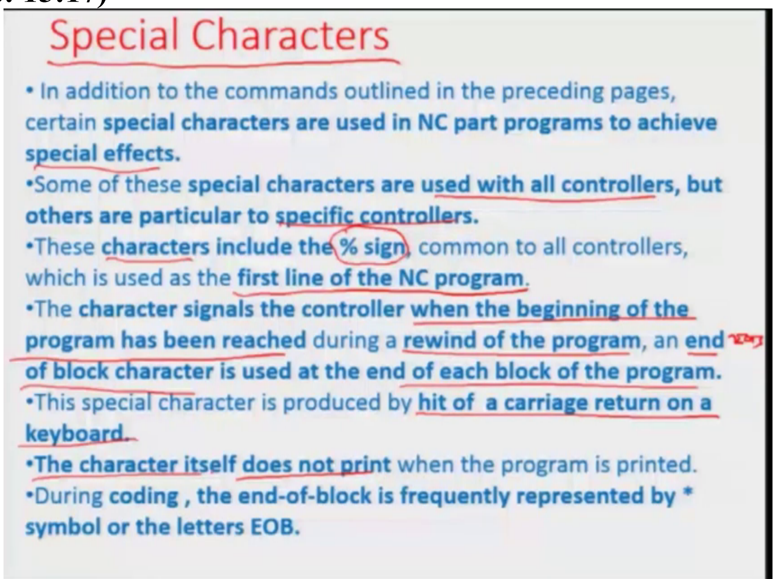
The effect of any one of these canned cycles is cancelled by programming a G80 function.

There are many other commonly used canned cycles it could be used for spot phase counter boring, deep hole drilling, tapping, through boring both in and out as well as in chip breaking or chip breaker drilling, through boring with dwell okay. So, you have different down feeds in all these certain action points which are what happens to the final depths where the tool reaches. And then some points related to how the tool has to be retracted okay.

And then these codes are accordingly designed in a manner so that every time this challenge is faced you know for designing a CNC code you just simply put the command and it executes a sequence of events just like the drilling okay where you know only this value specification of specified value in a block would suffice for the whole operation to take place. The effect of any one of these can cycles obviously would be canceled by another G code so that the G80 function that is normally used for cancelling any of the canned cycle.

So, please make sure that whenever there is a question of separate you know parts of the blocks which would be executing canned cycles you must cancel it to get back into the normal process of you know the machining okay.

(Refer Slide Time: 15:17)



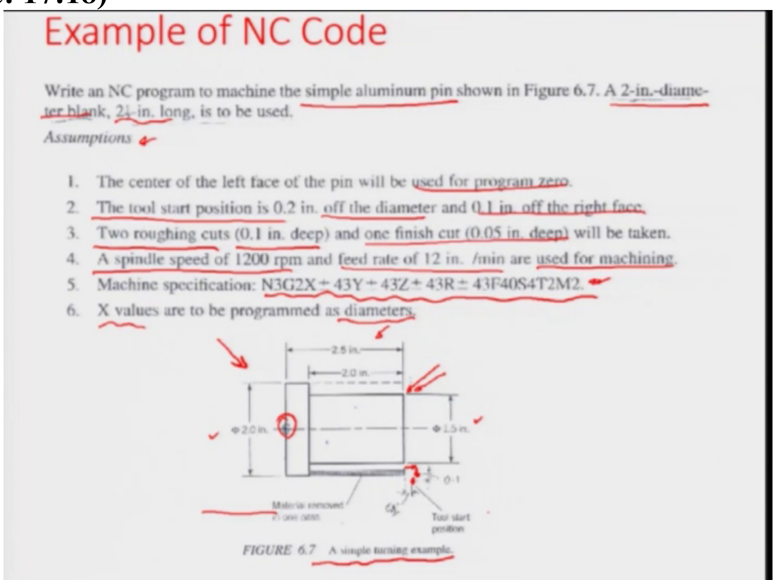
So, last but not the least there are special characters which are also a part of the code these are in addition to the commands outlined. And special characters normally have special effects some of these characters for example are used with all controllers but other can be with specific controllers. There is a character called percentage sign which would be common to all the controllers and it is generally used as the first line of the NC program to suggest that it is a program start ok.

So, the character signals the controller whenever the beginning of the program has been reached during even rewind process of the program and an EOB or end of block as we commonly call is to signify that you know at the end of each block of the program there should be such an EOB. So, that it can go to the next block okay so the special character is produced by a hit of a carriage return on a keyboard so whenever you are entering the different blocks.

So, after the block is finished typing you generally enter to go to the next line and so that enter automatically gives you an end of block command ok. So, the controller then reads that line up to the EOB command and then starts the next line character itself does not print on the screen but it has to be there because of the tabbing that you are doing ok when the program is printed.

And during coding the end of block is very, very frequently represented by star symbol ok which says that it is as well the star the end of the program. So, what I am going to do is to actually help you to write certain codes and then move ahead probably in the next section for the other very important part on rapid prototyping and rapid tooling.

(Refer Slide Time: 17:18)



So, let us look at this one example problem here which describes the machining of a simple aluminum pin with a certain head as you can see in this figure right here. it is a simple turning example and here some specifications are given as to what would be the final dimension of the pin which we want to program and a 2 inches diameter blank is carried out or used for formulating the pin which has a head dia of 2 inches.

But then there is a tilde which is 1.5 inch also a 2 1/2 inch long block needs to be used in this particular case for you know the turning operation. And there are certain assumptions which we follow for writing the code the first assumption being that the center of the left face of the pin that is this face right here would be used as a program zero. So, every time there is a machining operation to be done it has to be with respect to the program zero and all the dimensions are here and defined with respect to the program zero.

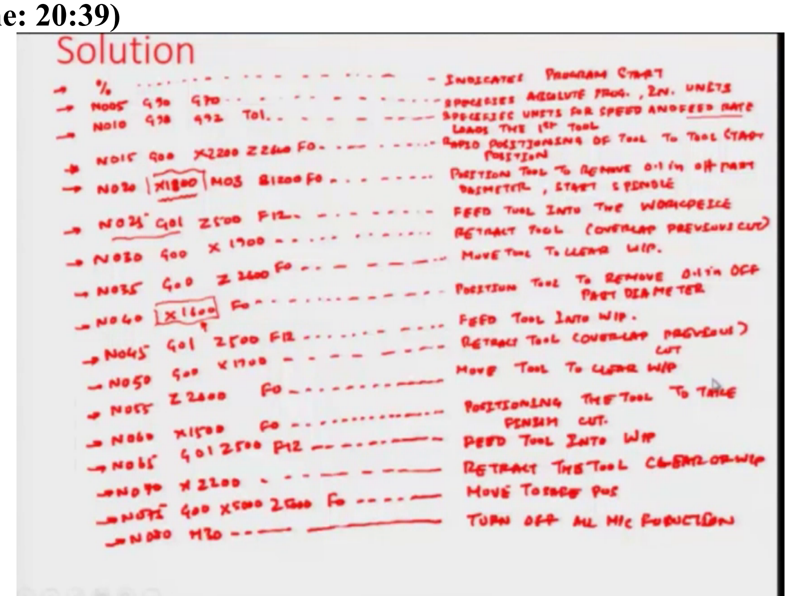
You know that the Z would be positive as the tool moves away from the origin in this particular case in an axial manner parallel to the axis of the workpiece. This right here is the workpiece okay from which you know it is a cylindrical simple you know aluminum blank from which you have to do the machining. So, the stool tool start position further is given to be .2 inches off the diameter and 0.1 inches of the right face this is the right face right here at the end of 2.5 inches from the origin.

And so we go again .1 inches to the right and then because diametrically at this .2 inches radially it should be 0.1 inch. So, that is where the tool position should be somewhere here okay. So, 1.1 inch toward the right and .1 inch away from the radius of the particular pin. So, there is where the start position is it is represented by this particular dot and we also are wanting to define through this machining process a certain sequence of operation.

It is not a direct cut affair you are having to roughing cuts of .1 inches deep and one finished cut off 0.05 inches deep okay. So, they will be taken simultaneously and then there is a spindle speed of about 1200 rpm a feed rate of about 12 inches per minute which are used for machining and then finally we know that the controller specification that is being defined here is N3G 2X + - 43 43Y +- 43Z + -43R +- 43 F40 S4 T2 M2.

So, this is how the controller is specified and we want to find the x values as or we want to program the X values mostly as diameters. So, whatever we will be mentioning would be automatically 2 folds okay. So, we start our programming now.

(Refer Slide Time: 20:39)



So, let us look at how we do the solutions so let us say we start the program with a % sign which indicates the start of the program so let me just write this down here start of the program start.

So, then we want to set up the particular system of reference including whether it is going to be inches units or whether it is going to be absolute positioning. So, in this case we use assumed absolute positioning we will write the first line of command starting at the identification number 9005.

And say G90 and G70 okay so this corresponds to; this specifies absolute programming in inches units. The next option could be for speed feed rate units and you know which tool number so that we can identify that it is a turning tool cutting edge which has to be aligned you know in this particular case the spindle can make the workpiece rotate. So, I will write the next line which is N010 which is the identification number followed by G98 and G92 okay.

And this corresponds to tool number T01, so this so I let us just demarcate the different descriptives okay. So, the next descriptive is about rapid so specifies units for speed and feed rate loads the first tool. So, then we go into N015 the next particular line and then we try to rapid position the tool to the start position. So, we have G00 and then we have to write what is the position aspect. So, in this particular case if I go back to the drawing here the start position is somewhere here in this particular case at a distance of you know 2.2 inches in the Y in the X direction okay.

So, as you remember that in this particular case the Z axis is along this direction so this is the positive z where the cutting edge of the tool which is probably starting or somewhere here okay it has to start somewhere here or start from the you know because it is a single point cutting tool obviously. So, it will have to go parallel to the axis after it has got in certain depth okay into the; so, the depth can be in this direction and parallely to the axis is in the -Z direction minus that corresponds to is the tool approaches the workpiece.

So, sitting in this condition if I look at my right here so this corresponds to the positive X. So, this was what the sign conventions said at the very beginning ok and X and Y do not matter in this scale because they are pretty much similar to each other. So, we will be actually starting only with the X and the Z coordinates. And in this particular case the a coordinate happens to be 2.2 inches remember we are programming in diameters.

So, if we are talking about 0.1 inch radius it corresponds to 2.2 inches diameters. So, that is what the whole idea is that X values that we programmed as diameters. So, you have 2.2 inches and then you have you know 1 inch off the 2.5 inches here from the origin everything is with respect to reference to this point right here which is the origin. So, it is 2.6 inches ok along the Z direction positive Z direction.

We are going to write this here as X 2200 Z 2600 F0 ok and this corresponds to the rapid positioning of tool to tool start position. In the next line we would like to put another command N020 and right you know exactly in the first instance when we will like to shave off point 1 inches what is going to be the basic extent up to which the tool needs to be going. So, if it is point 1 inches on 2 inches diameter it means that radially 0.1 means corresponds to the final diameter being 1 point 8 inches.

So, I will say that for the after the rapid positioning of the tool to the tool start position which is X22 you know corresponding to X2200 and Z2600 so we want to make now obviously something to position the tool to remove or shave off the first cut which is point 1 inches part diameter okay. So, the first thing you have to do is to start the spindle that means the workpiece should start rotating and so there has to be a certain rpm you know which is going to be given to the speed okay.

So, in this particular case we will say that the tool now needs to reach X let us say Z equal to or X = 1800 okay corresponding to again the point of time when the spindle starts spinning in the clockwise direction that is M03 all the way when the speed of the spindle has been defined by 1200 rpm which has been given in the question itself okay. And in this particular condition once the positioning of the tool goes to 1800 what it means is that from this position when it is still off the face of the workpiece it is going in this vertical direction and standing here as the spindle has started moving and the workpiece has started rotating.

And this length right here is a diameter which corresponds to 1.8 I am sorry somewhere here corresponds to 1.8 diameter or .9 inches radius okay. So, the tool in its current location goes to 1800 or 1.8 inches so corresponding to the X. So, because we are reading in absolute manner please note that we are merely referring to the coordinates written here and coordinates written here without considering the plus or minus because everything written on the right side here and the downside here are all positives okay.

So, in this position there is no feed given to the tool so the feed let the feed be 0, I will just write this down as position tool to remove 0.1 inch of part diameter and start the spindle and so with everything else is ready and the workpiece is rotating you can possibly give a feed and in this particular case we will write this command as N025 the feed a linear feed. So, we will change the G00 option to G01 and we have to go to a certain Z value.

So, in this particular case if you look at the Z value which it needs to go to is actually corresponding to 0.5 inches okay right about here which is the size of the head of the pin as given in the diameter. So, I will say this corresponds to a position Z 500 up to which the tool must go and the tool must do so with the feed which is given to be 12, you know the units are basically inches per minute as has been defined earlier for the speed and speed rates in line 3 of this particular program.

And so you can say that this corresponds to the feed tool in to the workpiece for the first case that is the .1 inches depth case. So, in the next step we would like to retract the tool to overlap the previous cut because obviously in the next cut we will again have to move from the start position you cannot just you have to complete you know the whole cycle and then start so we will retract it so that could be obviously rapidly being able to get done.

And so we will do G00 and write the next command of 1900 which again clears off the tool from the workpiece remember we had gone up to 1.8 dia and we are now going to 1.9 which is actually towards the right of 1.8 okay and so it is still clearing off. So, we will say this is a retract tool option and overlap previous cut. In the next sequence we will again go to the clearing off mode okay. So, once we go out to that particular domain 1900 X 1900 in this particular manner okay.

We can go to a sort of a safe zone maybe so we can say that we again rapidly position this all the way to Z2600 okay to make it clear again at a certain feed of 0. So, let us say this corresponds to move tool to clear workpiece you may or may not decide to go for this step you could actually directly proceed from the last step but because the start position it at that particular point it may be a better idea of route time to take the tool to the start position.

And then you give the next feed of 0.1 inches which is defined as a X motion of 1600 at feed 0 just that we had given 1800 in this particular step we are giving now from 18 to 16 so another .1 along the radius or .2 along the diameter remember the tool values have to be programmed in the or the position has to be programmed as diameters. So, this is position tool to remove again .1 inches off. So, position tool to remove 0.1 inches of part diameter.

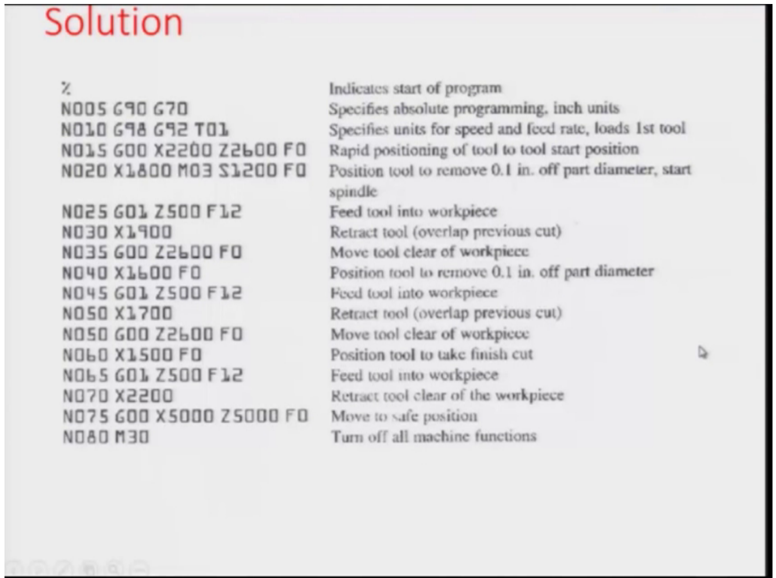
We will go like this again just as we did in this step linear position all the way up to Z500 so that we can shave off at a certain rate that is 12 inches per minute rate. So, this corresponds to feed tool into the workpiece, so your 2 cuts are already done once this is done we can probably again do the same process of retraction. So, we will now clear off the part diameter first so we will go G00 and clear off to X 1700 maybe okay.

So retract tool so this is to overlap the previous cut go back again into the start position okay. If so that you can clear off along the Z-axis also so 055 is written as G00 you do not even need to derive G00 here okay just you know Z 2600 okay G00 has already been programmed in the last step give a zero feed in this particular case. And this corresponds to again position so move tool to clear workpiece.

In the next step again we will have the finished cut so here we need to go to X = 1500 okay corresponding to a zero feed again 1500 makes sure that we have a finished machining of .05 inches deep that means .05 inches radius or 0.1 inches diameterly okay from the 1600 position that we had hit upon in the last step okay. So, essentially you are giving that finish cut stepped in this particular you know instance and so basically here you are positioning the tool to take finish cut go deep into the workpiece.

So, now you do G01, Z 500, F12 in the same manner so by virtue of doing it you are basically feeding tool into the workpiece again do the same step of retracting back come to X2200 and you know you could say that you are able to retract the tool in this particular case clear of the workpiece. Finally again move clear to a safe position and turn off all machine functions. So, you could see that how diameterly we have programmed and obtained a situation where we have to roughing cuts and one finished cut okay together.

(Refer Slide Time: 39:06)



So, this is the whole program you can find the program details here in a printed manner okay this is how the print comes out to be when you want to extract it from the from machine. I am going to close this particular module okay because I think it is already gone overboard by so many different minutes. So, in the next module we will probably try and look into other aspects some informational aspects related to rapid tooling or even CNC tooling and how those are important and those are fitted.

You have more or less now had a very good inkling about how to do CNC programming what are the basic aspects of interacting you know with the controller and so therefore you need to understand in what capacity the tuning may be designed or made. So, that now you can mix various things together and design and develop different parts so with that I would like to close this module, thank you very much.