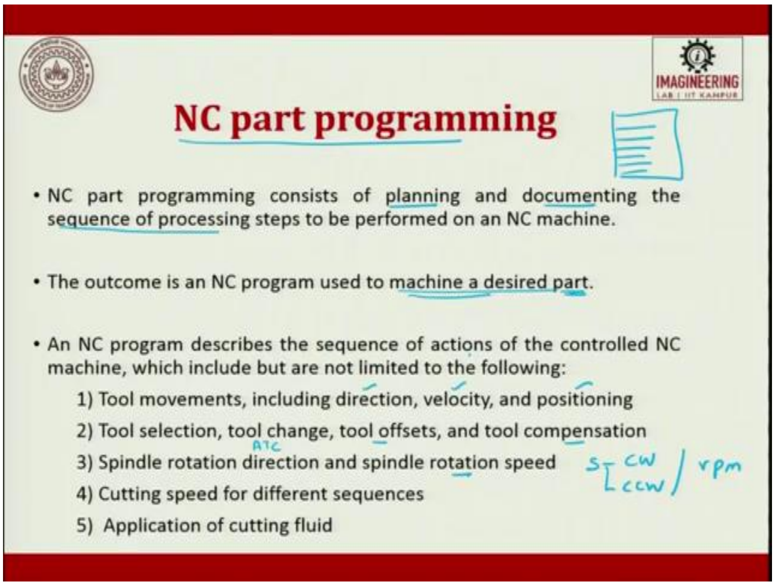
Computer Integrated Manufacturing
Professor. J. Ramkumar
Department of Mechanical Engineering and Design Program
Indian Institute of Technology, Kanpur
Lecture 20
CNC Part Programming - Part 2 of 4

(Refer Slide Time: 00:19)



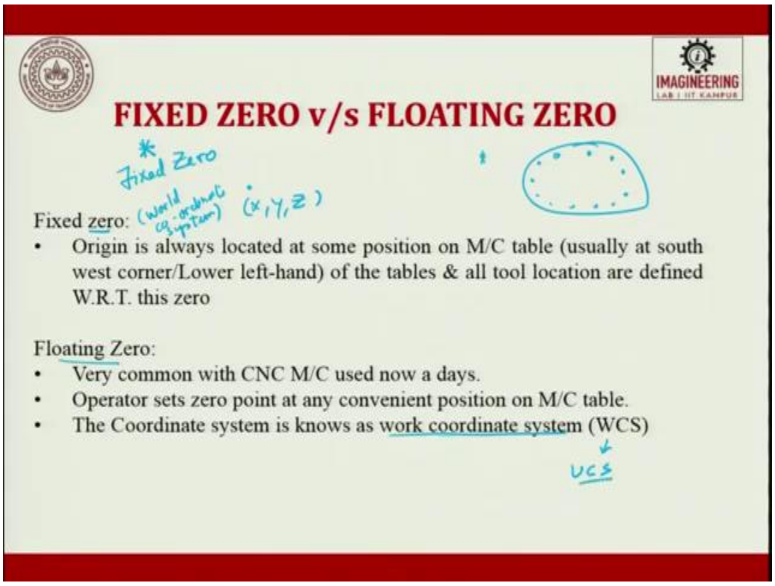
The next important topic in CNC is going to be, NC Part Programming and to how to write a program. This is a program, so now what are all parts of our program we are going to see. NC Part Programming consist of learning and documenting the sequence of processing steps, to be performed on NC machine. So, what we are trying to say is we are not, see in CAP what we studied? we studied sequence of processing. So here, we are trying to see a sequence in a process. For example, in turning, what are all the step by step, by step has to be followed to produce the part?

So, consist of planning and documenting the sequence of processing steps to be performed on a NC machine. The outcome is an NC program used to machine a desired part. So, that is why it is called as part programming. An NC program describes the sequence of action of the controlled NC machine, which include but are not limited to the following, tool movement including direction, velocity, and position.

So, till now we were talking about position and velocity. Now, we are also trying to say direction plus or minus. So, tool motion, tool selection, tool change, tool offset, tool change we saw ATC Automatic Tool Changer, tool offset and total compensation, these are there. What is tool offset? Suppose, you have a drill and of height 18 millimetre. It has got broken while operation, so you have another and stock which is 6 millimetre. Now, the height compensation has to be done, that is what it is. Then diameter compensation offset, so diameter change, so that will be an offset.

So, tools selection, tool change, tool offset and tool compensation will also be part of a program. Spindle rotation direction and spindle rotation speeds. So, whether it the spindle rotates in clockwise direction or rotates counter clockwise direction, while tapping we go front in clockwise and then we come back and counter clockwise, so clockwise, counter clockwise. And, we try to talk about rotation, RPM. Then, cutting speed for different sequence, then application of cutting fluid while machining. These are all sequence of actions of a controlled CNC machine, which has many more, but we have stopped with 5.

(Refer Slide Time: 02:47)



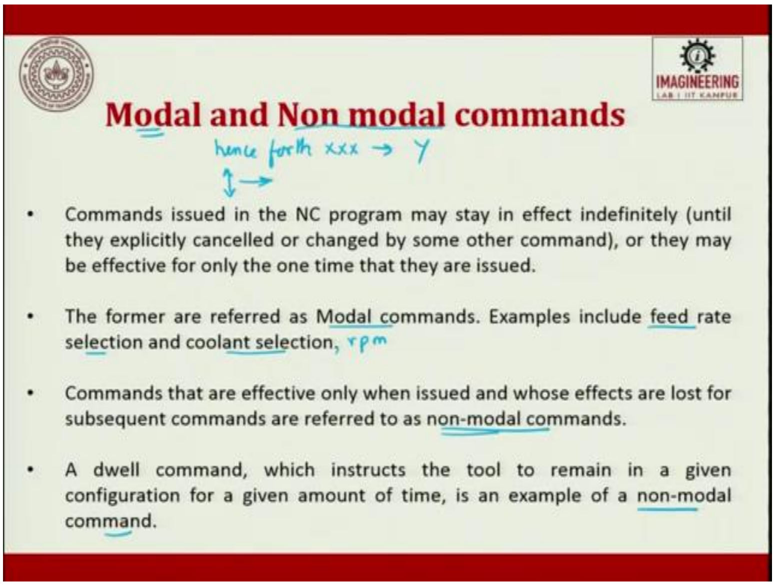
So, the first thing is we have to understand 2 things, which is called as float fixed zero and floating zero. Fixed zero is origin is always located at some point on the machine table or inside a machine tool, machine table off the table and all tool locations are fixed with respect to the 0. For example, as soon as you switch on the machine, it will move here and there and go back to your home position. That home position, location is X, Y, Z data is always taken with reference to your single point, which is called as fixed zero.

This is machine fixed data, which we cannot change, that is what is called as fixed zero. The other one is called as, floating zero. Very common with CNC machines used a nowadays. So, what you do is you try to shift the fixed zero into anywhere in the working space and set that as 0, 0 for convenience of your programming. The operator sets 0 point at any convenient position on a machine table.

See earlier, in a turning machine it should, it will be inside the spindle somewhere. In a milling machine also, it is the same. So, the locations are fixed and when we talk about the tool magazine, we will have several tools. So, all these tools also will have your reference datum inside it.

So that also, in turn will be calibrated with respect to this fixed zero. The coordinate system is known as work coordinate system. So, this is world coordinate system, which cannot be changed at all, world coordinate system, this is called as work coordinate system, user coordinate system. Some books call it as work coordinate, some book calls it as the user coordinate system.

(Refer Slide Time: 04:51)



Commands used in NC program may stay in effect indefinitely, until they explicitly cancel or changed by some other command or they may be effective for only the 1 time that they are issued. For example, if you are trying to say hence forth, in legal documents there. Hence forth, some XXX will be called as Y. So, wherever there is, Y you have to interpret it as XXX. That is throughout the document. So, that is called as modal commands.

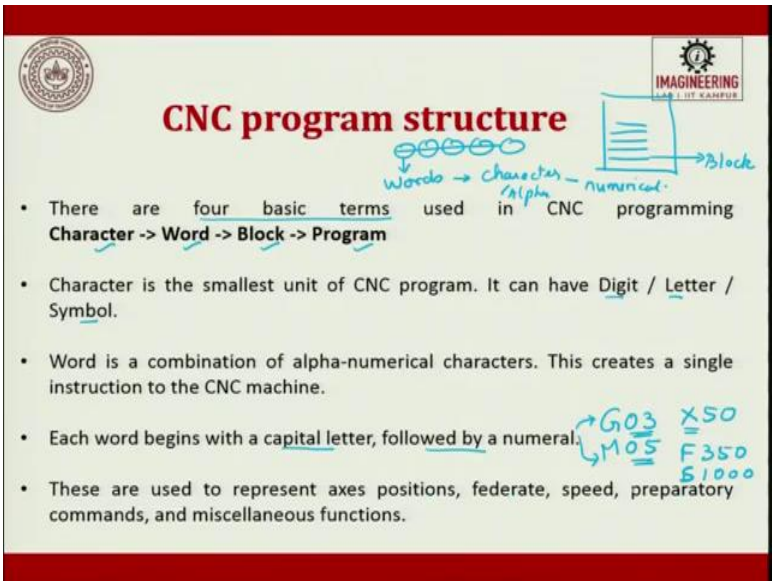
The non model command says, you try to say a small segment of the program will follow this command. So then, it is called us non modal commands. Modal commands once activated you have to go back and say deactivate, then only it gets deactivated. Non model commands it works for that particular part of the program, sub-routine of a program and then it gets deactivated off its own. We will see more details when we see the program. The former are referred as modal command.

Example, feed rates, selection and coolant selection, RPM selection, all those things are fixed once you write at the beginning of the program throughout the program it is the same. The commands that are effective only when issued and whose affects are lost for subsequent commands are referred as nonmodal commands.

So, you say that only in this particular segment you activate, so that is what is non modal command. A dwell command, which instructs the tool to remain in a given configuration for a given amount of time, is an example for non modal command. So, when we saw CNC machines wherein which there were measuring arms attached with the spindle itself, below the spindle there was an arm.

So that is used to check and calibrate the tool, dimensions or tool response. So, in the same way you can also use some gauging devices to check the features made, whether it is perfect or not. So here what we do, we write a program, we call that gauging device, and we say hold the machine for 20 seconds, until the gauging system gets activated, it gauges the feature generated and it is say go no go, that means it says okay further go machining or not. So, those commends are called us non modal commands. So, important thing is modal command and non modal commands.

(Refer Slide Time: 07:55)

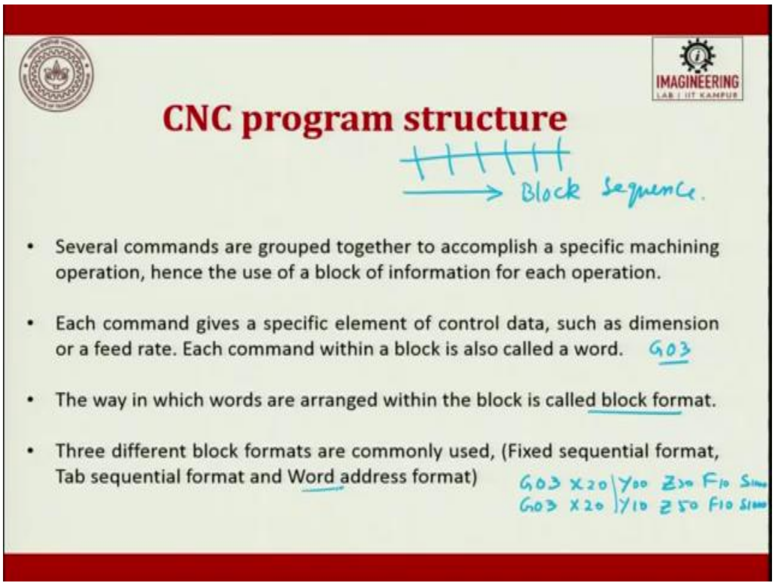


When we look into a CNC programming structure, there are 4 basic terms used in a CNC program, a character, a word, a block, and a program. So, this is a program, every line is called as a block. Every block has, these are words, every word has character. This character can be alpha, the character can be numerical. There are 4 basic terms used in CNC program; character, word, block, and a program block, program, block, then you have a word, then you have a character.

Character is the smallest unit of a CNC program. It can be a digit, a letter or a symbol, I forgot to say a symbol also can be there. A word is a combination of alpha numerical character, this creates single instruction to the CNC machine. Each word begins with a capital letter followed by a numeric. For example, G03, M05, X50. So, X is the coordinate, G and M codes are 2 different codes which are used. So, it starts with the capital letter followed by a numeric, starts with a capital followed by numeric. So, this is what it is.

So, these are, then for example, you can say, feed 350, RPM, S, you say 1000, all these things. These are used to represent axis positions, feed rates, spindle preparatory commands and miscellaneous functions.

(Refer Slide Time: 10:00)

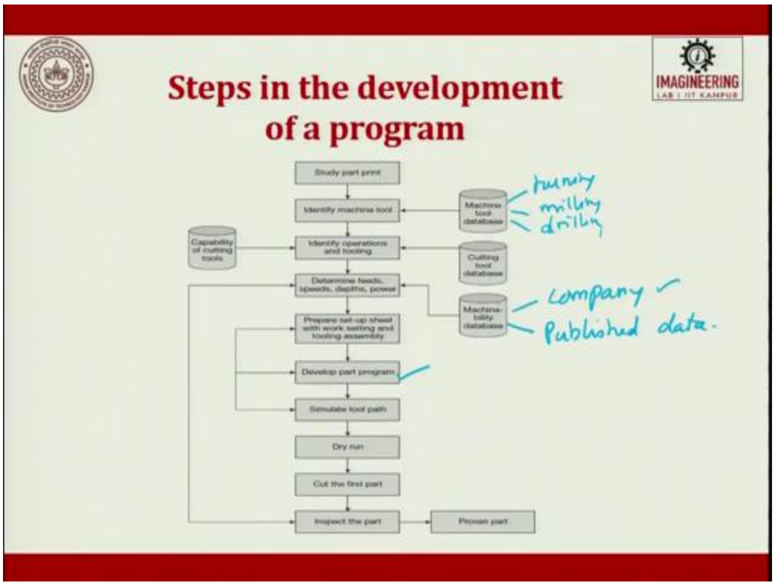


Several commands are grouped together to accomplish a specific machining operation, hence the use of a block. As I said, a line of information for each operation. Each command gives a specific element of control data, such as dimension or feed rate. Each command within your block is called a word. So now, G03 is a word. The way in which words are arranged within the block is called as a, block format. So, I said, a line. So, what has to start and how it has to end.

So, this is a block sequence, that is also a block format. 3 different blog formats are commonly used, fixed sequential format, tab sequential format and word address format. So, what is fixed it, fixed it is, you will be say for example, G03 X, Y, Z, then F, S, for every line you have to start giving this data and this is what it is. So, a sequence is fixed, all you have to do is fit in numbers. For example, here you say 20 here u says 00, 30, and then you say 10, and then you said 1000.

So here it is a fixed set sequence. So, the next line also, suppose it says D03, X, there is no change in X. Still you have to write 20 and then you have to say 10, 50, F10 and S1000. So this is, fixed to sequence. You can not change the sequence and also the data will be repeated if that is not change in the coordinate values. Tab system is you go here, put a tab, it goes to the next. The last one word address format, which is latest use, so you do not have to use the same sequence. For example, if there is no change in X, you just write the next line what is the change, and Y value alone. So, word address formats.

(Refer Slide Time: 12:02)



So, the steps involved in developing a program. First, study the part, identify the machine tool, so the machine tool database which is turning, milling whatever, drilling you will see that. So, machine tool identification. Then, identification of the operations and tools, so tools come from here, operations come from here. So, capability of cutting tool and cutting tool database comes from here. After this is done, then you try to look at the feed speed, the depth of cut power required.

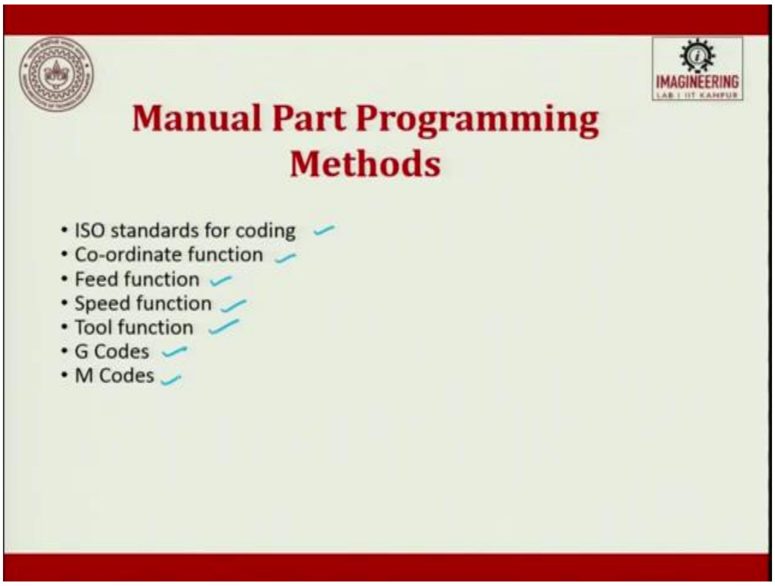
So, that comes from the machinability data. So, this can be company based, or this can be published database. Many a times company based will be used because machines are getting aged. So, every company has their own machinability data index. So, we take the machinability data index and put determine the feed, speed depth of cut. Once that is done, then what we do is, we prepare your setup sheet with work setting and tool assembly.

So, we talk about that, how to fix the part, how to do it. So, then what we do is, we develop a program. If the part is not fixed properly or the tool data is not obtained properly, part programming will not be executed properly. So next we go for a part programming. Then once we develop a part programming, we simulate and see how does the tool move with respect to the work piece development of part, simulation of part.

So here, it can go back and forth like within these 3. And then what we do is, we try to do a dry run. Without the work piece we allow the CNC machine tool to move, the tool to move, machine will be stationed the tool to move and then we see what is happening, whether it is touching anything, that is dry run.

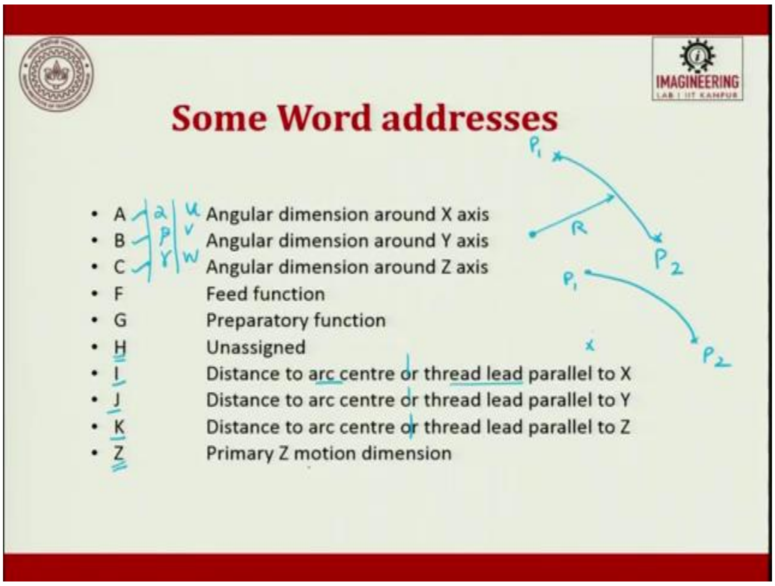
Then we do, a cut the first apart very consciously we cut the first part and we measure the dimension and see whether it is okay, so we inspected. Once it is done, then only it is said done. So, you run several parts. So, these are the steps which are involved. Currently what we are going to see is only this, rest all are assumed that as and when you get to the field you will start understanding. But so much of steps are involved, so many data basis has to support or you should have the data seats available with you to write a program.

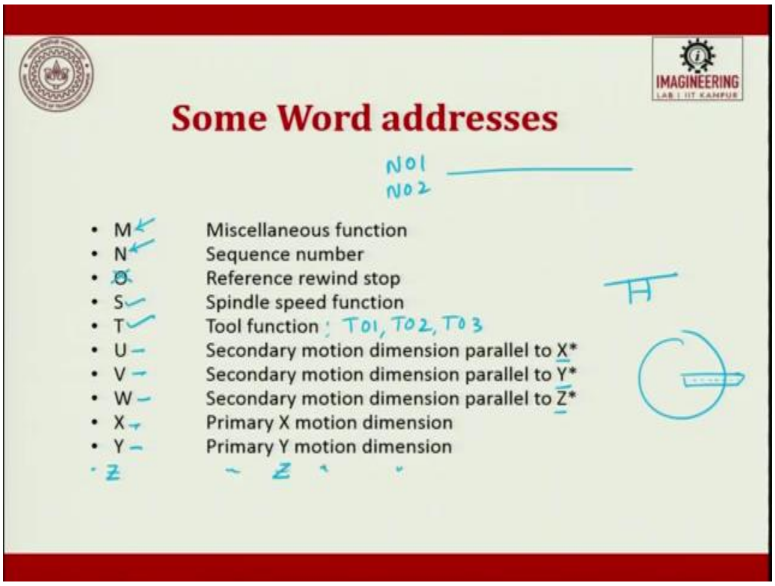
(Refer Slide Time: 14:37)



So, when we talk about manual part programming methods There is ISO standard codes available, the coordinate functions. Then you will have feed functions, speed function, tool function, G code, and M code. These are the things which you should understand when you write a manual part programming.

(Refer Slide Time: 14:58)





So, some of the words which are used A, so it is a rotation about X is A, B, C. Please understand the logic only here, and A, B, C can be changed from controller to controller. It can be alpha, beta, gamma, it can be u, v, w. So, look at the controller and then see what is it.

So here we are using A, B, C for our lecture discussion. So, it is rotation about X axis is A, rotation about Y axis is B, rotation about Z axis is C. Feed functions, preparatory codes, these are the words which you should remember. H is unassigned, you can use it for anything. I, J, K, are used to for arc distance or thread lead parallel to X axis. So suppose, you have an arc. So, this is the P1 point, this is P2 point. So, you can define the arc by P1 to P2 maintain this radius, this is 1 way of doing. The other way of doing it is you say P1, P2. What is the offset in X axis? What is the offset in Y axis? So, then what happens, I try to use I as an offset in X, J as an offset in Y, and K as an offset in Z.

This is one way of using I, J, K. The other thing is, what is the lead it gives to parallel to X axis can be I, parallel to Y axis can be J, and parallel to Z axis can be K. So you again, you have to look into the controller. The Z axis is nothing but X, Y, Z or say Z is a primary motion axis.

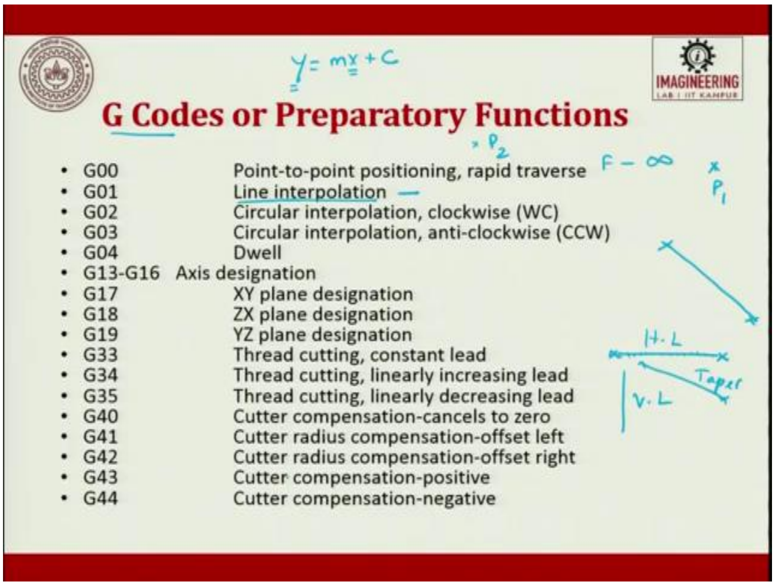
M stands for miscellaneous, N stands for the block number, N01, N02, that is N, sequence number. O is reference for rewind stop, this is nowadays getting removed. And why are we still talking about this tapes and the punch cards is, even today in very big industries where they bought the machines 40 years back, they still use the old techniques for pulling in the part programming from the stored memory.

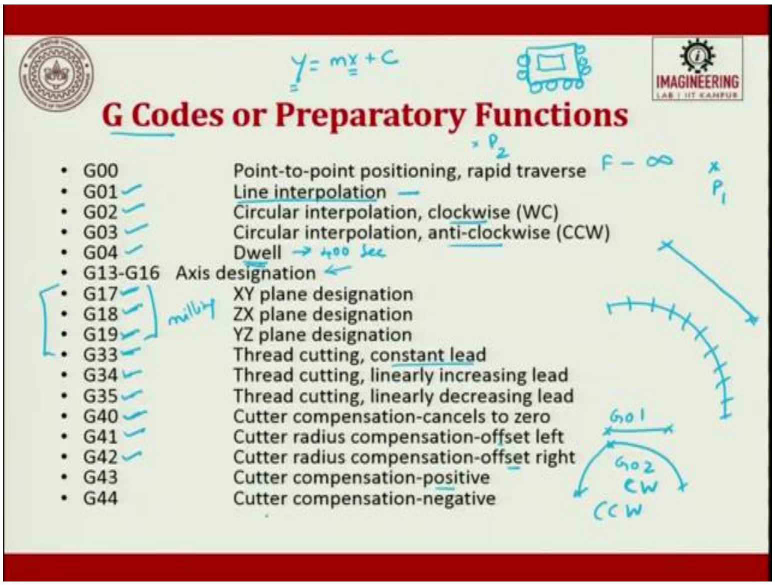
So, we have cassettes, we have punch tapes, even today, we have sheets, punch cards are even today available. But the advancements have happened, and why still these machines are existence, the machines are very capable, the accuracies are very high, and it is still able to produce the required quality part. Companies are not interested to scrap, and these readers are gone out of business.

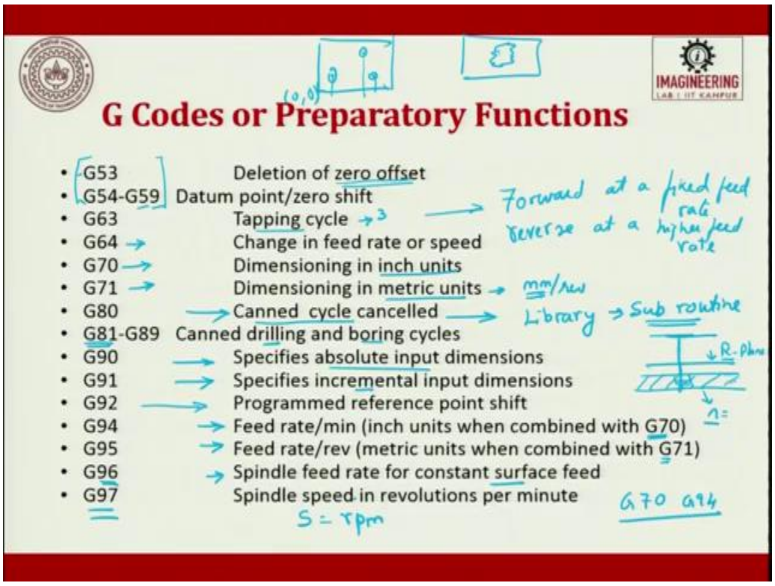
So, they are still holding the legacy of those machines for their use. So, O command it is better you know, but in recent CNC machines O commands are not used. Then spindle speed is defined as S, T is which tool you are using tool numbers, T is a tool function, T01, T 02. So, if you are having 3 tools to be used in a program, so each tool will be given a number and then you will call them. So, T will be used.

So next is U, V, W which are secondary motion dimensions parallel to X, Y and Z. This is X axis, in X axis you want to have some motion, secondary motion. So that is U, V, W. Where really complex jobs are made, we used to have secondary motions. For example, here is a disc and inside a disc you have one more thing to be controlled, so we use secondary motions, X, Y, and Z. The last one, are the 2 primary axes, X axis and Y axis. So, Z will be the other primary, Z motion dimensions.

(Refer Slide Time: 19:31)







Now let us see the, preparatory cords or G codes. These are the standard codes which ISO has given and people have to follow it. G00 is point to point positioning and it is rapid traverse. For example, in here I am moving the tool from one position to the other position. Here what I do is, the feed becomes infinity that means to say, the maximum possible speed. The only control is starting point P1 and P2, so it is point to point control. We are least bothered about the path it takes and we do not do any machining, so because of that the feeds are running at infinite speeds.

That means to say, it is going at its maximum feed rates. Next is, linear interpolation. Wherever you want to generate a straight line, either horizontal line or vertical line or a taper line, it is a line. So, we have a start point, we have an end point, and in between points are interpolated. In engineering we encourage people to do interpolation.

So what does that, I have defined an equation, y equals to mx plus c, and then I tried to give x values, and I try to find out the y values and I do interpolation, find out the discrete points and so the cutter or the tool will move along the straight line. The straight line can be horizontal, straight line can be vertical, the straight line can be tapered, depending upon your requirements. Whenever there is a straight feature to be machined, we use linear interpolation. And, so, if I know to draw an arc, then all objects can be split into lines. For example, I have a circle. If I know to split it into straight lines, facets, then I can draw this arc. So, in engineering we will, we have to know to make a line and we have to know to make an arc. So, line is G01 and arc is G02. See in line it goes along the start point, end point, end point, start point it comes. So here, there is no variation, if you start from here, go here or you start from here, come here it is single command G01 only.

But whereas, in an arc you have 2 options. One is clockwise, the other one is counter clockwise, clockwise and counter clockwise. So, you can come from here to here or if you go around, so you will have to move in the counter clockwise direction. So, if I want to move from here to here, I will go from here to here. So, we need to have 2 options. One is circular interpolation clockwise, circular interpolation counter clockwise. So we have, G02, G03, G01 is along a straight line. Today, there are controllers where they have removed G02, G03 and set G01 if I defined with a radius, then it does the function of G02 and G03.

So please check the controller, what you are using and then use these codes. So generally, it is expected G01 will be a straight line, G02 with respect to start point, end point, you have to choose whether to go in clockwise direction or to go in counter clockwise direction. Next one is dwell, which we saw in non-modal example, between 2 lines you wanted to have a break, so then we use a dwell cycle. So, dwell and then you say 400 seconds.

So, then it will stop for 400 seconds and then do it. Now, 17, 18, 19 are nothing but, in a circular component shaft, you do not have to specify the planes but when you do prismatic, you will have to define the planes. Whether it is the XY plane you are going to write a program, YZ is the plane you are going to write a program, or XZ is a plane you are going to write a program. Because if you take a cube you will have sides. So, it is XY, YZ or XZ. So, you have to first define the plane.

So that is why we use the commands, G17, G18 and G19 for milling operations only. For lathe, we do not use G17, G18 and G19. So, G13 to G16 are axis designations. If you want to have turn or mill and you want to give some assignment to the axes, so you can use this G13 to G16 which is open, you can define anything you want. Now, G33, G34, G35 these are all used for thread cutting. So, thread cutting constant lead, thread cutting linearly increasing lead, the lead can be changed, linearly decreasing lead, the lead can be reduced.

So, if you want to do a threading cycle with a normal lead, we try to use G33. If we want to use with increasing lead, we go for 34 and 35. G40, G41, G42 are cutter compensation, wherein which, if you want to do a radius compensation offset along left, you try to do G41. For example, if you have your competent like this, you are trying to cut. So here you have to understand, whether your workpiece is towards your left or right. So, this peripherally I am cutting it.

Now let us assume, I am doing a pocket inside. So, towards my right or towards my left is the competent. Depending upon that, we try to use cutter radius compensation offset left hand side or cutter compensation on the right hand side. So, this is what it is. And then, if you wanted to cancel that command so you go for G40. G43 is cutter compensation positive. G44 is cutter compensation negative.

So, when we talk about from G53 to G59 these are all codes which are used for shifting or deleting the offset, which is given to the tool. It can be to the tool or it can be to the workpiece, predominantly it is given to the workpiece. Next, when we talk about G63 it talks about the tapping cycle. So, when you talk about tapping cycle, when it moves forward, it will move at a fixed feed rates and when it returns back, reverses, it will travel at a high speed, at a higher feed rate.

This is to reduce the production time or increase the productivity. So, tapping cycle it is not that one shot like drilling you do it, you go forward few millimetres and then you reverse back. Then you go forward few millimetres, then you reverse back and finally you get it. And in taps also, you have 3 different taps for making one particular thread cycle. Next, we have change in feed rate or speed if you want to give in the command in the block you would write to put G64, which talks about changing feed rate or speeds. Then G70, G71 talks about inches unit or millimetres per unit.

So, wherever we use millimetre per revolution, we try to use G71 or whenever we want to use only millimetres, we will go write it as G71. Then we have Canned cycles, Canned cycles are nothing but library functions, which are supposed to do a fixed job or I will say it is a sub routine and this sub routine used to do one particular task. So for example, it starts from here, goes down, drills, and then it reverses back.

So, when it starts from here from infinite space and then it gets very close to the work piece, if this is the work piece, it goes very close to the work piece and from there, slightly above from this, it starts moving down in the specified feed rates. So now, this plane is called as the R plane. So now, when you move the tool, from the extreme end to the R plane, it will just come in very high feed rates and then from here it will get into the feed rate, what you specify.

Then from here, it will start going inside and drilling the work piece. Suppose, aspect ratio is very high, then you will have to give n number of steps with which the drill has to be made. So then, it will follow that and then come back. So, the complete cycle will be written in one sub routine or a library function. And all you have to do is, define the plane, define the number of n, and then you try to give the feed, and then you try to give the cutting speed. So, then what will happen is, it will follow several cycles and then it completely does one operation.

This is only for drilling cycle, like this you can have for boring cycle. You can have for many cycles like that, when you have a turning also, you can have threading cycle like that. So a set of functions, which is to be repeated and you put it in a bundle and give one particular name, that is the Canned cycle. So Canned cycle, we give you the command G81 to 89 for varying Canned cycles.

This is for drilling cycle, boring cycle, grinding cycle, you have tapping cycle, you have threading cycle, all these things you have. So, if you want to call those commands, it is G81 two G89 and if you want to cancel it you put G90 and then close it. So, when I said here, you can have the dimensions in inches and metric, the same way you also have G90 for absolute and G91 for incremental inputs. If you go back and look at this holes, which we have made. If you take this as (0, 0) you drill a point here, and then drill a point here and here. So, if I try to say absolute all the datum will be taken with respect to this (0, 0).

So, if I have to do such programming then I will call it as, absolute input dimension programming. When I have to take with respect to the previous point, then it becomes incremental. Whenever you have a simple geometry, it is absolute (())(31:35). When you have a complicated geometry, we go for incremental. You can also have in a single program, mixed mode of both absolute and incremental. For example, in a work piece, you have certain features. This feature making can be an incremental and the rest of machining can be in absolute form. So, we use G90 and G91 commands.

So, G90 is model command, G92 is programmed reference point shift. So, we have word coordinate system, user coordinate system, you will have work piece coordinate system, you will have in a work piece again a user coordinate system. So, that is programmed reference point shifts. Then we will have G94, G95 is feed rate in millimetre, so feed rate per minute in terms of inches unit. It is feed rate per revolution in terms of metric unit, when combined with G70 and G71.

So that means to say, G70 and then you use G94 something like that combined. Then G96 you will use it for spindle feed rate for a constant surface feed. And then G97, you will use for spindle speed in revolutions per minute, so that is S which is written in RPM. So, this is G97, G96 is when you start turning a component the cutting velocity keeps on changing. In order to maintain the constant cutting surface speeds, so constant surface speeds, what we do is, we use this, commands. So, you can use this commands, G96 spindle feed rate for constants surface speeds, and then you can start machining it. So, these were all G codes. Thank you.