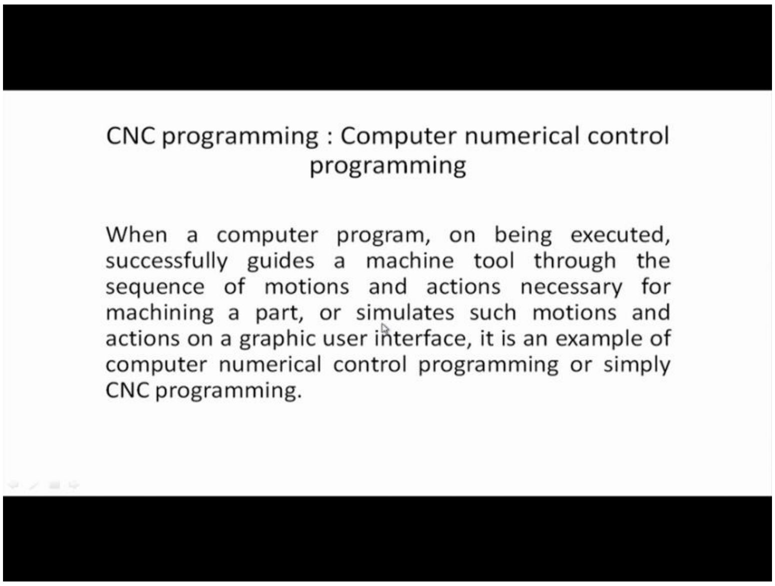
Computer Numerical Control of Machine Tools and Processes
Professor A Roy Choudhury
Department of Mechanical Engineering
Indian Institute of Technology Kharagpur
Lecture 09
Programming Practice

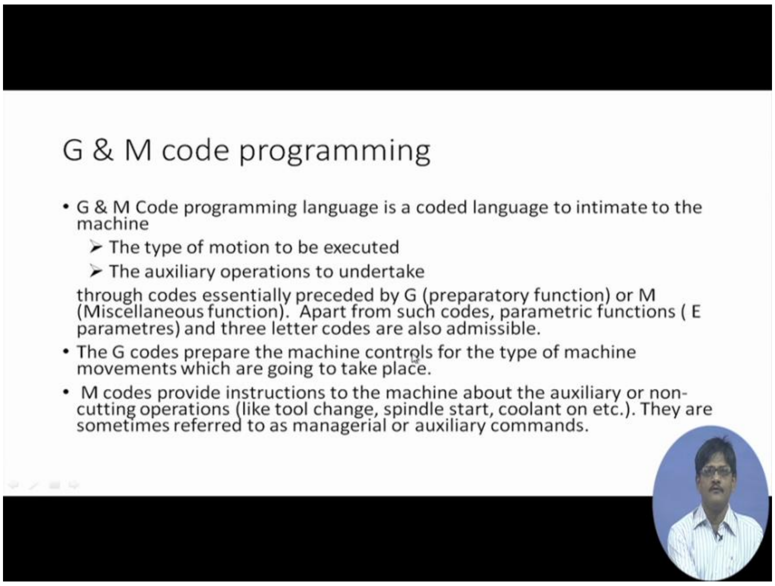
Welcome to the 9th lecture in the course just a moment, 9th lecture in the course "Computer numerical control of machine tools and processes". We will be talking about programming practice in this particular lecture, what is programming practice? Whenever we are giving instructions to the machine to carry out a definite number of machining operations, then there is a specific way in which we can give these instructions and this is generally called computer numerical CNC programming, okay it is going to be a sequence.

(Refer Slide Time: 01:11)



That is when computer program on being executed, successfully guides a machine tool through the sequence of motions and actions necessary for machining a part or simulates such motions and actions on a graphic user interface, it is an example of computer numerical control programming or simply CNC programming okay, so let us have quick look how CNC programming is carried out.

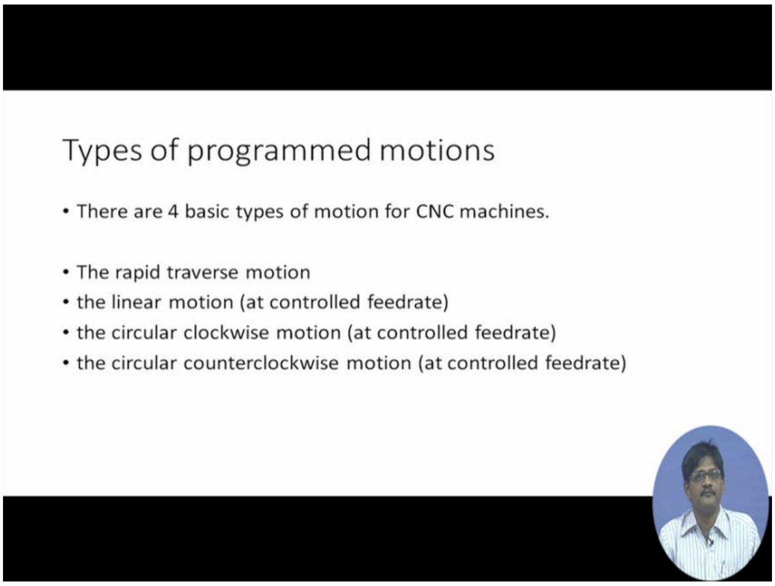
(Refer Slide Time: 01:39)



First of all, the basic simple way of communicating with the machine a control unit about the motions and operations to take place is the G and M code programming. Some basic codes are written down and the G and M code programming language is simply to get the motions that are to be executed and also the auxiliary operations that means non-cutting but essential operations. For example, say coolant ON coolant OFF, then tool change then declaration of tool length and diameter measurements, et cetera, et cetera. So, if the codes are preceded by the letter G, they are called G codes or preparatory functions and they are generally connected with motion.

And if the codes are preceded by M, they are called miscellaneous functions and they are generally concerned with the non-cutting auxiliary operations. Now, there might be other codes as well, there might be 3 letter codes, there might be parametric functions, et cetera in addition to G and M codes. They basically um help us to carry out the operations in a better fashion. So G codes will be preparing the machine controls for the type of motion and M codes for non-cutting operations in general, so M codes are sometimes also referred to as managerial or auxiliary commands. Let us look at the type of programmed motion that we are going to get from a general-purpose continuous control machine.

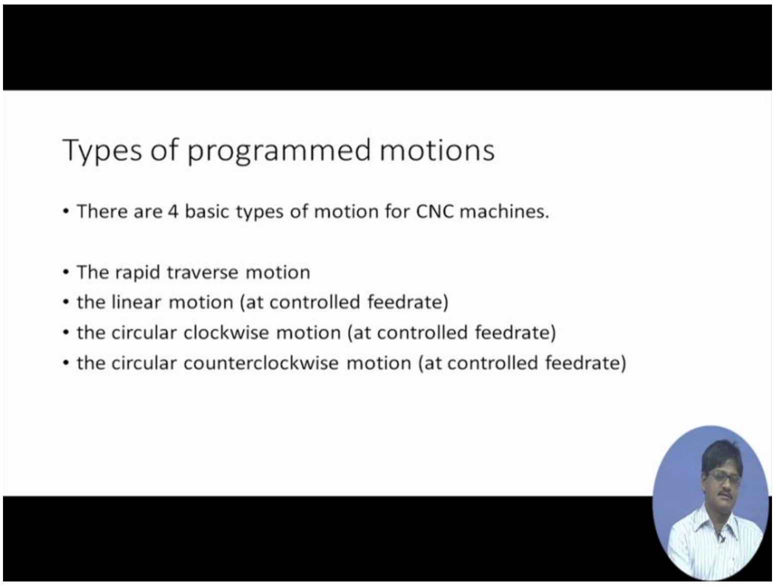
(Refer Slide Time: 03:38)



Continuous control machine can carry out all point-to-point operations, but point-to-point operation machine cannot carry out certain motions which are possible in case of continuous control, so there are 4 basic types of motion for continuous control CNC machines. First of all comes the rapid traverse motion, this motion is generally used to take the tool or the cutter to the point at which it is supposed to carry out the cutting action, so rapid traverse as the name implies rapid traverse motion generally takes place at the highest possible speed available for that particular axis unless there are other factors to modify that. The linear motion okay, linear motion is also possible at controlled feed rate, and what does it mean?

It simply means that you can move from one point to another along a straight line at a controlled velocity, control means predetermine; you can assign a particular velocity for that particular movement and that is generally referred to as feed velocity. The other 2 types of motion, they are the circular clockwise motion at controlled feed rate, so as as the statement implies, the motion is going to be in a clockwise direction naturally in a circular manner and the feed rate will also be controlled along this path. And in the same manner you can also have counterclockwise motion circular manner at controlled feed rate, these are the 4 basic motions which will always be available. Apart from these, if some machine has some other specialized motion available, that is a specific case.

(Refer Slide Time: 05:45)

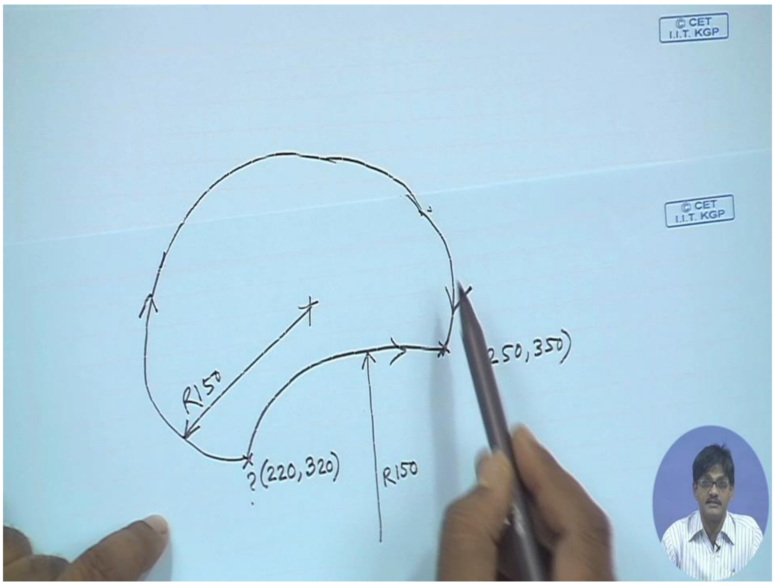


So let us have a look how they are depicted, actually. So G00 is rapid traverse and its format will be G00 followed by the syntax reads this way G00 code followed by the target point coordinates, the target point coordinates in this case they are X200 Y300 and Z400. Unless otherwise stated, we will understand we will take it this way that this refers to coordinates of a particular point on the machine coordinate system. By machine coordinate system say in case of a milling machine, we will understand it is a coordinate system fixed on the table as if by paper and glue you have stuck a coordinate system on a graph paper on the machine table and when the machine table moves, it means that other items are moving relatively in opposite directions in the coordinate system of the table.

So it is going to a particular point whose coordinates are given here as 200, 300, 400. G01 in a similar manner, it refers to linear motion as we have discussed in controlled feed rate and its syntax is G01 followed by target point coordinates and since it is taking place at a definite rate, the rate is mentioned as F200, which reads unless otherwise defined 200 millimetres per minute. We will understand this way in our discussion it reads 200 millimetres per minute feed rate. G02 circular clockwise motion can have different ways of expressing the data and there are I have given two such ways, there are other ways also.

For example, we might write G02 followed by target point coordinates X250 Y350 and Z is not mentioned here because generally these circular motions will be associated with a particular plane of action and in this case it is X-Y plane. The radius along which this rotation is supposed to take place, it is 150 millimetres and the feed is supposed to be 200 millimetres per minute, shall we have a quick look in a piece of paper how this will be coming out?

(Refer Slide Time: 08:30)



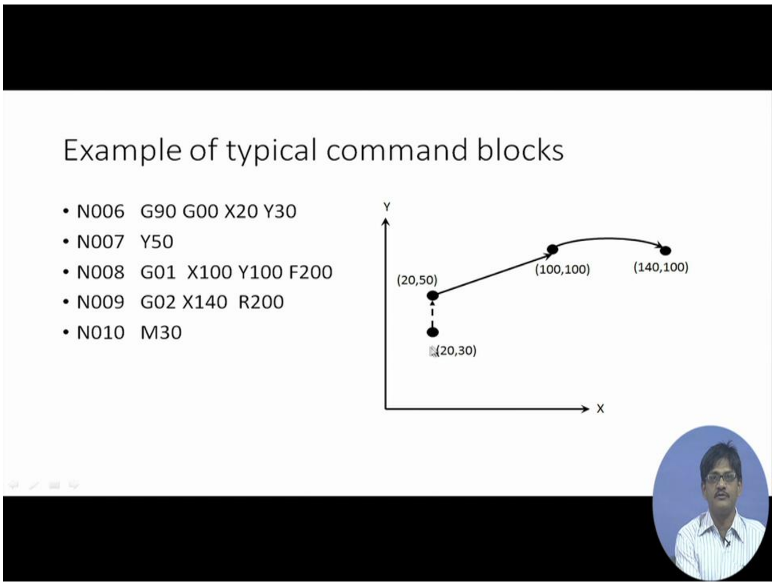
Let us see this one is the 1st point say, the initial point and this is the final point. So this point must be equal to 250, 350, you might ask what about this point? I do not know its coordinates. Well, this is the last point at which it was residing when this command is executed. Let us say this point is the previous G01 coordinates 220 just for the sake of having a definite concrete values, 320 that is fine so after that what is stated? It is stated R150 and G02, we have to move clockwise manner, you might say why not this one. Well, this will be accepted if this radius is R150. My drawing is not very good please understand that this is supposed to be part of the circle with radius 150 and the motion of the tool will start from here and it will end up here.

Is there any other path which will be satisfying all these condition that is this is the start, this is the end, this is essentially clockwise and this is R150, yes it is a possibility that it might start this way out of our space of consideration and end up coming here and end up here where this radius is also 150, what about that? So if there are 2 possible solutions for the motion that that has been asked before, then what path is the tool going to take up. The answer to this is that unless otherwise stated, it is always going to take the minor arc. If you specifically want this particular path, longer path major arc to be executed then something else has to be stated, what is that?

In general in machine if we come back to the this one again in general some machines permit that you can just write add a minus sign in front of this numerical value just after radius that means if you write R - 150 in some machines they will permit the cutting of the major arc otherwise if you do not state anything, the default case that means the thing that is going to happen yet nothing else is stated that is going to be the minor arc, the smaller arc okay. Now coming back to this one to other formats which allow, this one says, G02 XY of the target point followed by Centre coordinates 270 and Centre coordinates sorry, Centre coordinates X 270, Centre coordinates Y 370 and feed.

So this means that the ambiguity which was existing in the previous format that is done away by actually choosing the centre coordinates, but here you have to be very careful, you have to give the correct Centre coordinates, if you give some incorrect centre coordinates it will give you an error message and the program will not be executed. There are other ways also of giving command for circular interpolation where I and J are made use of, but we will not be discussing it at this moment. When we are discussing interpolators, we can take up that particular definition. Coming to circular counter clockwise motion, only difference here is the G03 term instead of G02 term, all other things I am sure you can understand by applying some common sense.

(Refer Slide Time: 12:46)

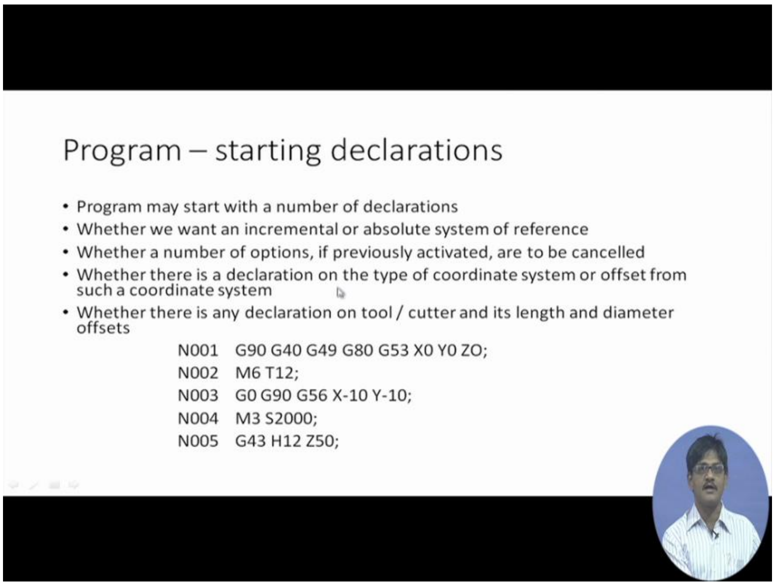


This is an actual example if we see is visually. G90 means, these are the different lines of the CNC program, each line may correspond to a declaration okay about some data or it might be corresponding to an actual movement, the 1st line corresponds to a movement. G90 means absolute system that means we are referring to the points by their absolute coordinate values, so there is a point which is being referred to as 20, 30 and it is this point. So, we are saying that go fast to this point 20, 30. So wherever the tool might be residing, from there it moves fast and reaches this particular location at highest possible speed. After that nothing else is mentioned so that it means that G00 is still active, so it is still going to move fast and where is it going to move? To Y50.

What about X? It does not change and therefore it is not mentioned and therefore it simply moves up only in Y so that it attains a position Y equal to 50 and stops there, so the coordinate locations of this point will be 20, 50. After this, we have a movement here which involves movement both along X and along Y, so it moves along this path until it reaches the point 100, 100 and the feed value for that motion since it is G01, this feed value is mentioned. What if you do not mention this feed value in the G01 command? If you have previously executed G01 or G02 command or mentioned some feed value in a declaration statement, then it will adopt that value.

But if you have not given that, generally machines will stop giving an error symbol that is insufficient data has been declared. So in order to avoid that, please be careful always to declare a feed value for G01, G02 commands. After this is a circular movement and we write G02 X40 and Y is not mentioned so Y remains the same and R200, so it moves from simply from this point to that point we have already discussed this, how this is done and it reaches this point by circular interpolation. After that, M30 is end of program.

(Refer Slide Time: 15:26)



Starting declarations, so when a program is starting it might be starting with a number of declarations, what sort of declaration are we talking about? It might be stating whether it is adopting the absolute system of reference to coordinate values or is it referring to incremental values of movements that means, if it says G91 X30 it simply means that incremental system is being adopted by G91 and X30 would be referring to a moment which is 30 millimetres along X, it has nothing to do with the absolute coordinate value of the particular position of the tool. It might have a number of options activated previously by say the previous command which are to be cancelled. For example, what are these options? What can be these options?

It can be that you are declaring that the tool is of such and such length and work piece of such and such height, the tool is having this kind of diameter, the tool is supposed to carry out drilling operations unless it is asked to do, asked to stop that okay, the tool might be asked to carry out say spot facing operations at different locations, wherever it goes it is supposed to spot face. You might have a command for cancelling that, so all sorts of cancellation and cancellation of options previously opted for, which might be active still, they might be cancelled in that 1st line.

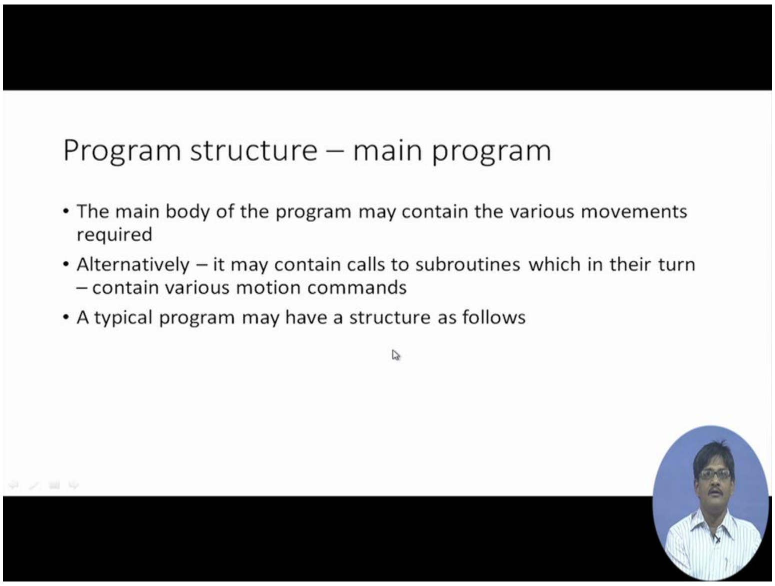
So there might be declaration on the type of coordinate system or offset from such a coordinate system or you might be having what you call it, the choice of a particular coordinate system or the rotation of a coordinate system, things like that might be declared initial lines. Or whether there is a declaration on a tool and cutter which we have already discussed. This is a typical line, starting line of program for example, you might be having declaration that G90 absolute system G40 cancel tool compensation okay, tool compensation means some declaration on the tool length , work piece height, tool diameter, et cetera sorry tool diameter declarations are being cancelled by G40.

Then comes, what is the importance of tool diameter? Whenever you are cutting along the Perry Ferry of a particular part, if you declare the tool diameter and the way in which it is connected with the part profile, then that machine can do some calculations which otherwise would have had to be done by you when we discuss programs, we will be definitely discussing this. So G40 cancellation of tool diameter or tool radius compensation, G49 cancellation of length compensation, G80 cancelling tool cycles which have been previously activated and now need to be cancelled.

G53 is a coordinate system option command that is you are asking for a definite coordinate system okay that means you might be opting for work coordinate system declaration. You might be opting for some coordinate system which is offset from the machine coordinate system, et cetera. X0 Y0 Z0 gives us the particular values connected with such a coordinate system, which you want to declare. Then you might be having M06, this is an M command this concerned with tool change. Suppose you want to be, you want a particular tool to be loaded on the spindle and that tool has a name "12" that is tool is designated a number, it might be associated with the very tool itself or it might be associated with a particular location on the tool's storage device and the tool happens to be residing there.

So the machining is being instructed go for a change of tool so that tool number 12 or tool residing in seat number 12 is loaded onto the spindle, this is the command. So after the execution of this command you will find that particular identified tool is mounted on the spindle of the machine or if it is some other kind of machine like turning centre, that will be in the location so that it can start a cut may be a turret is rotated so that that tool comes into the working position. So there might be other commands, we will be going through through all these commands in more detail fashion when we take up a particular program.

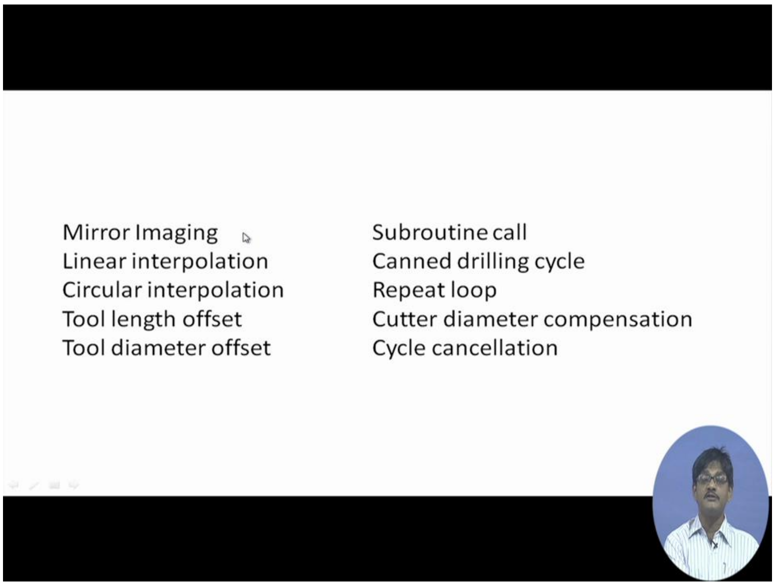
(Refer Slide Time: 20:42)



So there might be various movements, if it might have calls to different subroutines, what are subroutines? Subroutines will be a cluster of commands which are given a separate entity like a program and it might be called from a particular program in order to be executed sometimes a number of times. What is its utility? That means it can be called to be executed at any particular location, it is generally location independent okay.

So suppose you want a number of operations to be executed at different locations a different number of times, so you might be asking the subroutine to carry out those operations at those rotations and your work will be produced many folds, You do not have to write huge number of lines of commands, but you might activate the subroutine from time to time, we will have examples of that discuss in our discussion on programs. So, let us have a look at different options present inside a program.

(Refer Slide Time: 22:05)



We might be having mirror imaging option, as the name implies if you have a bilaterally symmetrical job that means it it is symmetrical on two sides of a central line, if you have a bilateral symmetrical job, in that case you can call mirror image program for one side only and use a repeat loop to do the other side automatically after calling for mirror image, this sufficiently reduces the line number of lines and possibilities of errors, et cetera. Linear interpolation is very simple we have already discussed it. Circular interpolation we have already discussed it.

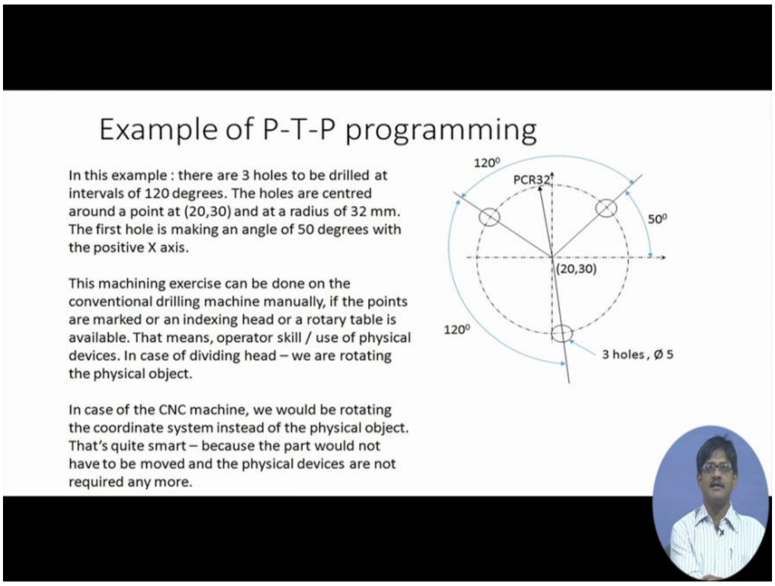
Tool length offset means that each and every tool which is used on the machine, they are not integral parts of the machine so the machine has absolutely no idea what is the length of the tool protruding outside the spindle after mounting it and what is the height of the job which is being done at present. The machine only knows about dimensions up to the spindle nose and the machine table, beyond that if something is loaded on the spindle or placed on the table, their dimensions the machine is completely oblivious about those. So we have to declare to the machine in some manner what is the length of the tool protruding from the spindle and what is the job height above the table surface.

So this way we also have tool diameter offset that means if the tool diameter is a certain value, we mention that to the machine so that if milling operation is to be carried out, the machine can take a decision where to place the tool so that correct geometry is machined out.

Subroutine call as we discussed when we are clustering a number of commands in a particular separate program, we can give a call to it may be several times in a program in order to execute. Canned drilling cycle, we sometimes put a number of items in a Can okay, number of say food items, sweetmeats you will be putting in a Can and that becomes a single unit.

In the same energy you can have all the operations required for drilling put in a single cluster and you can refer to that particular cluster of operations just by a single G code so that your effort at writing lines of program commands that is reduced. So this way we can have canned drilling cycle, canned boring cycle, canned so many cycles of operations are there. So repeat loop, repeat loop is simply do loop as used in ordinary programs. Cutter diameter compensation we have just now discussed, we will have a quick look when we are actually doing the program and cycle cancellation we have already discussed G40 G49 and G80. Let us have a look at an actual program.

(Refer Slide Time: 25:31)



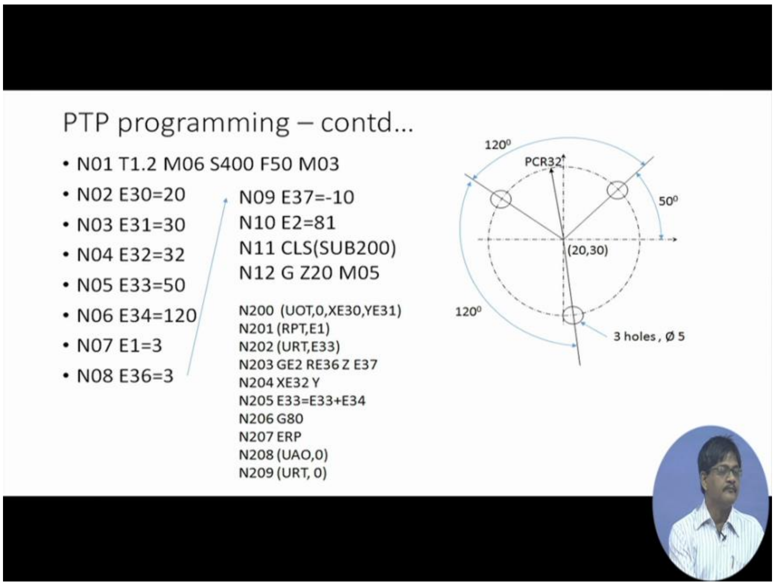
What is this program about? You can see that there are 3 holes to be made okay, 3 holes to be drilled at the circumference of a circle that is good. Only holes, no it might also this program will also solve the problem of 3 welled main to be made okay or 3 solders to be made or 3 what you call it enlargement of existing holes to be made as these 3 locations. Are the locations constant? No, if you use this program that we are going to discuss now, these values can be different, you can call a subroutine which is tailor made for handling this particular pattern of machining operation, machining or whatever manufacturing operation we are going to deal with. So this is understood, 3 operation needs to be done at 120 degree apart locations at a definite radius that is all, nothing else.

So let us have a look, what are we lot of material seems to have been written, let us quickly go through what it means. Okay, in this example there are 3 holes to be drilled at intervals of 120 degrees, the holes are centered around a point (20, 30) and at a radius of 32 millimetres, so the radius is 32 millimeters, pit circle radius is 32 millimetres, diameter of holes 5 millimetres holes and a centre of this pit circle is 20, 30 so it is not exactly at the origin might be. And the 1st hole is making an angle of 50 degrees with the positive direction of X-axis and this can be done in conventional, yes we can do it in conventional drilling, how will you do that?

Well, what we can do is, we can use an indexing head, place it with axis upright that means vertical on the machine table and use a vertical drilling device to drill these holes and the indexing head can rotate so that sequentially the whole positions will be just under the tool and drill will be drilling them. So indexing head can be used or you can even mark it out and then drill these holes or you can use a rotary table, et cetera that means either a skilled operator or you can use physical device, rotating physical devices can be used, indexing head can be used, rotary table can be used, et cetera, so physical rotation is being given. In order to give this physical rotation, the problem is we have to buy equipments.

Indexing head will be expensive and they will be prone to errors, so in CNC machine, as the philosophy is to get rid of physical devices to do away with skill of your operator, et cetera we are rotating something else, we are going to take the coordinate system. Virtually, the coordinate system rotates and it takes care of the rotation which was required in the physical world in the conventional machines. So we will simply rotate coordinate system and repeat the drilling action so it will be drilled automatically at the 3 locations, let us have a quick look how it is going to be done.

(Refer Slide Time: 29:03)



This is the basic program, we start in the 1st line okay, and in this program we are taking the help of E parameters okay and we are also taking the help of G codes M codes and 3 letter codes. So 1st of all, the 1st line reads that T1.2 that means a tool is being mentioned, to number 1 with offset written in the 2nd line of the offset file, now that is a bit complex to understand, let me express it once again. Tool number 1 is is being declared and the offset, that means its length and diameter offsets will be written in the 2nd line of file called offset file okay. We will take it in more detail, but at this moment we are declaring the tool number and we are declaring its offset that means its length details and diameter details.

And N06 asks the machine to load this tool on the spindle and it is declared that the spindle speed will be 400 rpm and the feed whenever required will be 50 millimetres per minute and M03 starts the spindle. So at the end of this command will find that the tool has been changed and the tool is rotating that means spindle is rotating with the tool at 400 rpm. So after that we are defining some E-parameters, what are E-parameters? Just like we have variables in algebra, E-parameters are simply variables which can be assigned some values. So, first of all the centre is at 20 30 and we are assigning those X and Y values respectively to two variables called E30 and E31.

So these are very strictly handled so that E30 can only be assigned such dimensions referring to say the coordinate values, so E30 is 20, E31 is 30, we have to designate these values in the initial programs. So in the main program we are first having designation of different E values, E32 is 32, what is that? The pit circle radius, it is sometimes referred to as grid radius. So E32 gets assigned this value, E33 gets assigned the starting angle, the angle which is made by the first hole meridional line with the positive X-axis okay. And E34 is the pitch angle that means the angle existing between two successive holes. After that, E1 reads the total number of operations to be executed which is 3 in our case.

And E36 is the small dispense which will exist between the drill position and the plane of the of the work piece top surface when the drill changes over from fast motion to feed motion. The drill when it is coming down to drill a hole, it moves fast but changes over to slow motion feed motion of 50 millimetres per minute only when it has come to a predetermined distance above the job surface and this is being designated as 3 millimetres. So the drill is being asked change over from rapid to feed motion at 3 millimetres above the job surface after that E37 gives the depth of machining, E37 = - 10.

E2 defines the particular Canned cycle fixed cycle that we are going to carry out, this fixed cycle 81 pertains to drilling. After that call subroutine is depicted by CLS three letter code and subroutine residing at line number 200 is being called and therefore we move onto the subroutine residing at line number 200 starts with UOT Use Origin Translation, so with respect to the origin depicted as 0 which is the machine coordinate system origin, we are supposed to shift the origin to the point XE30, YE31 that mean by 20 and 30 respectively because E30 = 20 and E31 = 30, so shift the origin with respect to the machine coordinate system which is given a designation 0 to the point offset to this machine coordinate system by 20, 30.

So we basically have simply an origin translation, after that repeat RPT stands for repeat loop do loop even number of times. That means we have 3 operations, those many number of times this loop should repeat. After that, URT goes for the coordinate rotation so the coordinate system as we discussed is going to be rotating above the Z axis by the angle 50 degree, E33 is 50. That is good, so it is rotated by that angle therefore the positive X-axis is now passing through, the positive X-axis first getting offset and then getting rotated, it is now passing through the 1st hole and the 1st hole is residing at the point 50, 0 in this rotated and offset coordinate system.

So after that, line number 203 we are having GE2 which means G81 which means Canned cycle starts RE36 and ZE37, so it is a declaration to the machine that get ready for the moment I declare a coordinate value, coordinate position carry out drilling with R means the changeover location from fast motion to slow motion RE36 that means at 3 millimetres above the job surface and Z value E37, go to a depth of - 10 millimetres below the job surface. I am always saying job surface because after you have declared 2 length offset, the job surface becomes Z = 0.

Next we declare the coordinate location, coordinate location is X = E32, what is E32? Rotated therefore, the location will first hold position is nothing but 32, 0 and a hole is made at that point you to this particular line 204. After this, the angle is be updated as E33 = 33 + E34, so whatever was the value of E33, we add E34 to it so 120 degrees is being handed and therefore, the angle depicted by E33 is now been changed to 50 + 120. So if it is 50 + 120, therefore G80 by the way G80 cancels the drilling cycle because we are not drilling after this and we go for end of repeat that means the Do loop ends, so this "Do loop" is going to be repeated 3 times.

Again we start the Do loop and it goes to EURT E33, E33 has changed now it is equal to 50 + 120 and therefore, coordinate system gets rotated so that the absolute angle of rotation is now 120 + 50. So the already rotated coordinate system undergoes a further rotation so that the rotated angle is 120 + 50 now, so it is now the second hole location the positive X axis. Again the hole will be drilled on the X axis at 32, 0 and the 2nd hole will be drilled now. So this way first hole, 2nd hole, 3rd hole gets drilled and after that it comes out and comes to line number 208, which reads use absolute origin 0 which means that offset is now cancelled.

The 20, 30 program shift coordinate origin shift is cancelled, the machine coordinate system is restored and URT cancels the rotation that means use rotation of 0 degree, so now the rotation rotated axis is now restored to its original position and comes back to the program at G00 Z20 M05. G00 means fast move up to the level 20 millimetres above the job so that the drill comes up to a safe position and M05 is M05 is spindle off okay, so at the end of this program we will have 3 holes drilled at the locations required carried out by the execution of a subroutine made use of what you call the E parameters and some G codes, thank you very much.