Automation in Production Systems and Management
Prof. Pradip Kumar Ray
Vinod Gupta School of Management
Indian Institute of Technology, Kharagpur

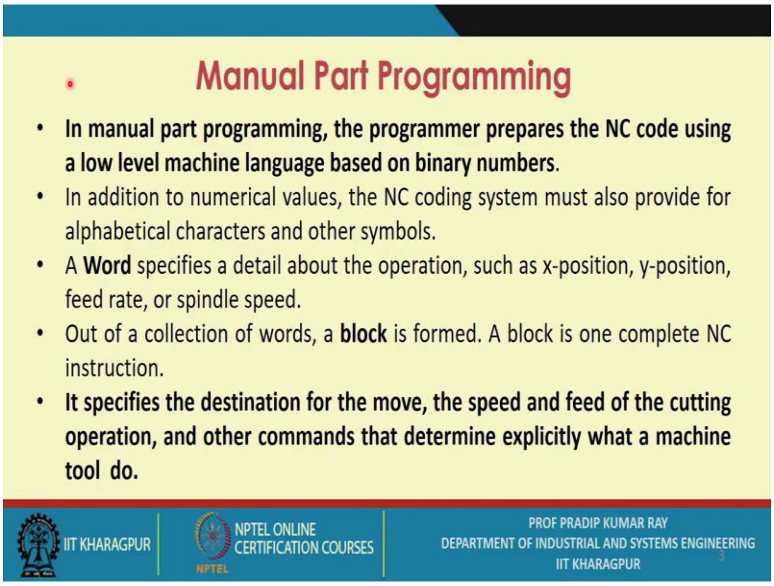
Fundamentals of NC Technology - II
Lecture - 25
Numerical Examples

During this lecture sessions we have the discussed-on NC technology. And, as you are aware that for application of NC technology in machining processes or manufacturing systems or in a production system, there is a systematic the procedures we have to follow and one important activity we should carry out that is NC part programming.

(Refer Slide Time: 02:34)

Now, the core issue is when we refer to the applications of NC technology that how do you perform as a part programmer.

(Refer Slide Time: 02:50)



In manual part programming, the programmer prepares the NC code using a low level machine language based on binary numbers.

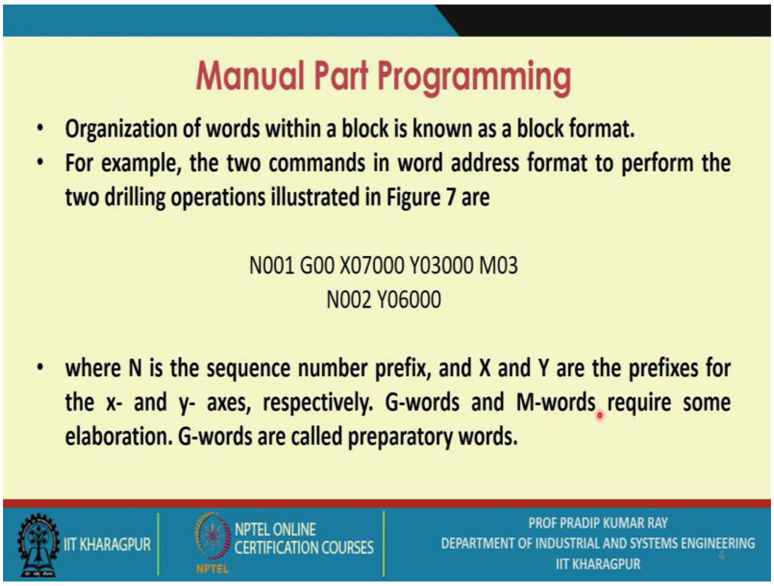
In addition to numerical values, the NC coding system must also provide for alphabetical characters and other symbols.

A Word specifies a detail about the operation, such as x-position, y-position, feed rate, or spindle speed.

Out of a collection of words, a block is formed. A block is one complete NC instruction.

It specifies the destination for the move, the speed and feed of the cutting operation, and other commands that determine explicitly what the machine tool will do.

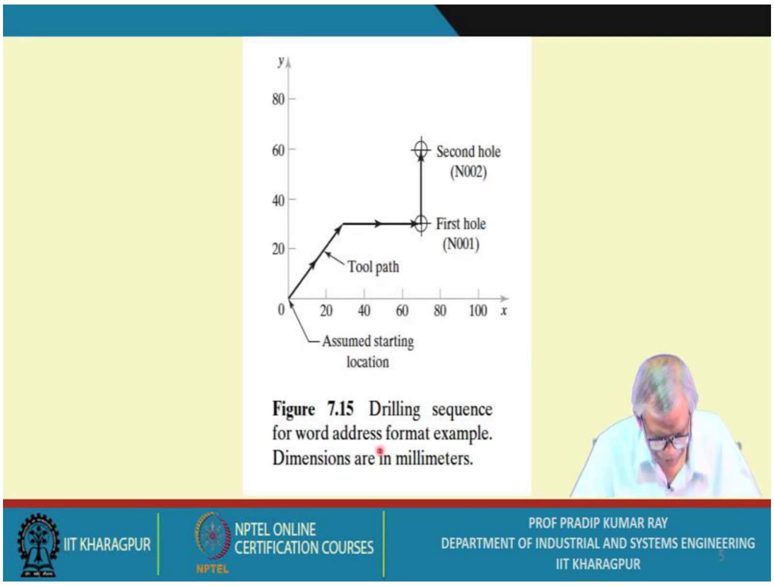
(Refer Slide Time: 07:15)



Organization of words within a block known as block format, you organize these words as per the sequence of operations, as per the location, as per the route it should be followed, that is one important aspect to be considered while you form a block. For example, the two commands in word address format to perform the two drilling operations illustrated in figure.

You will find in figure that is N001 these are the very common terms, we use G00, X07000, Y03000, M03, N002, Y06000. where N is the sequence number prefix, and X and Y are the prefixes for the x- and y- axes, respectively. G-words and M-words require some elaboration. G-words are called preparatory words.

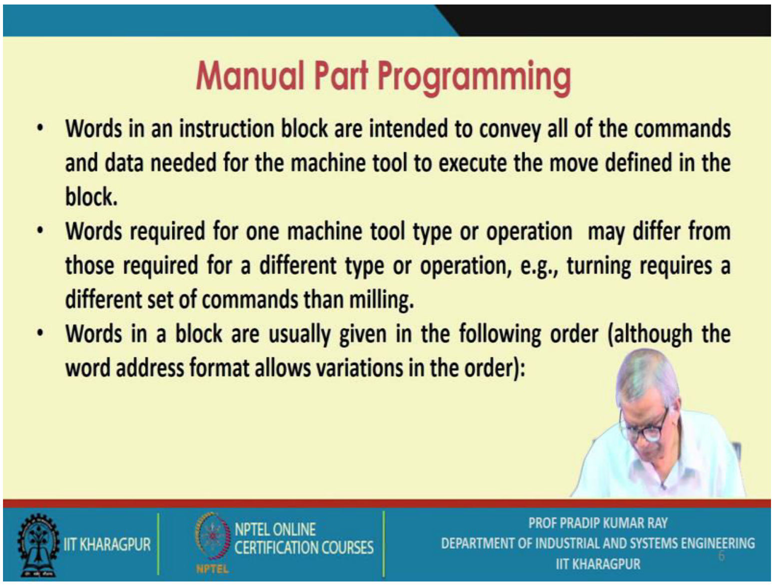
(Refer Slide Time: 09:37)



This is basically either drilling sequence, there are the two holes to be carried out. The first hole N001 that is the first sequence, first you drill this hole and then you have to drill the second hole and the second one is N002.

This tool path you have to define coding where there is a starting point and then you specify this path. This particular location you have to specify x, y coordinates and similarly this location, similarly the second hole location. You have to assume that starting location somewhere here, that is basically the 0 point, and then you start writing the program.

(Refer Slide Time: 10:54)

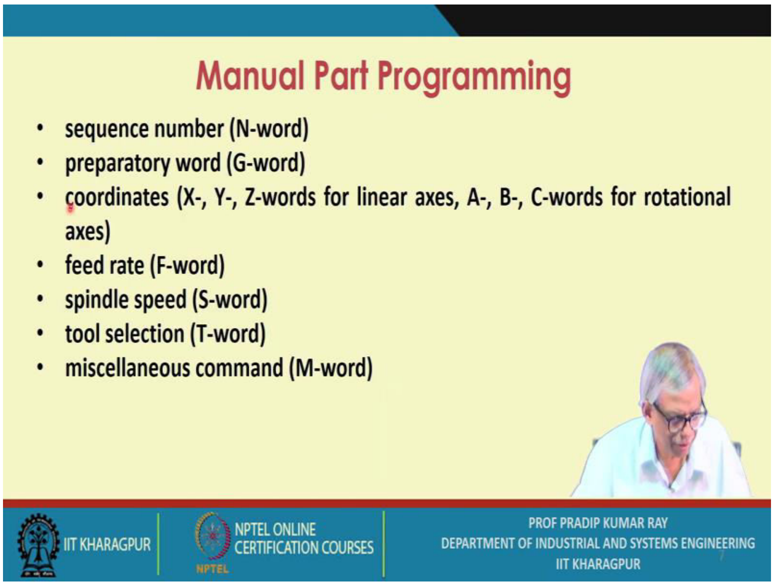


Words in an instruction block are intended to convey all of the commands and data needed for the machine tool to execute the move defined in the block.

The words required for one machine tool type may differ from those required for a different type; for example, turning requires a different set of commands than milling.

The words in a block are usually given in the following order (although the word address format allows variations in the order):

(Refer Slide Time: 13:47)



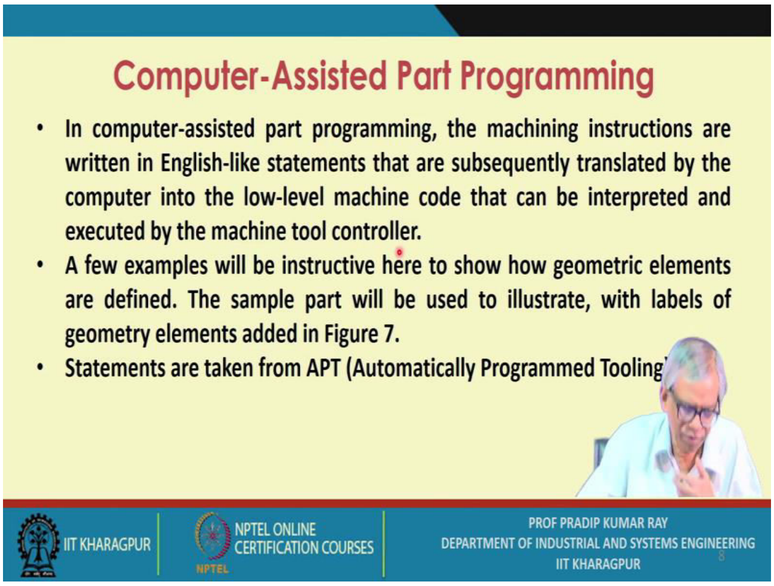
First is sequence number, referring to previous examples, it refers to the first hole to be drilled and then the second hole is specified as N002.

Then the G-words is basically a preparatory word. Coordinates X, Y and Z words for linear axis and there will be rotational axis also. A, B and C words for rotational axes. Already the NC coordinate systems we have discussed in detail, there could be the six possible axis X, Y, Z and A, B, C and in both directions, the feed rate that is F-word as soon as you find there is a capital F is written.

Spindle speed S-word these are very standard. The tool selection there is T-word. This could be automated tool changing like in typical FMS automated systems. Each tool is given a specific code and this is related to a particular code called the capital T.

Miscellaneous command normally we say it is M. So, the M-coding, miscellaneous command, the M-coding refers different kinds of commands.

(Refer Slide Time: 16:22)

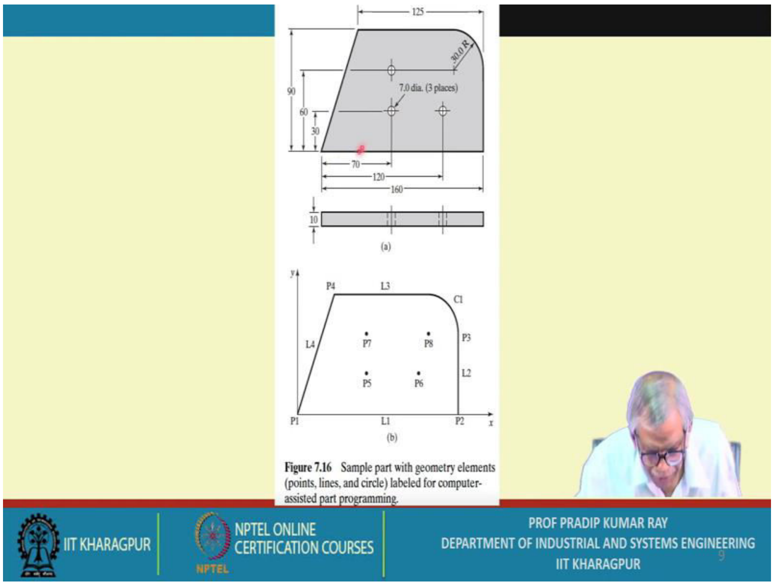


Normally the part programmer is well trained on all these coding systems and for different types of operations. In computer-assisted part programming, the machining instructions are written in English-like statements that are subsequently translated by the computer into the low-level machine code that can be interpreted and executed by the machine tool controller.

A few examples will be instructive here to show how geometric elements are defined. The sample part will be used to illustrate, with labels of geometry elements added in Figure 7.16(b).

The statements are taken from APT, which stands for automatically programmed tooling.

(Refer Slide Time: 18:17)



These are the two examples and this is basically the sample part with geometry elements. This is the line, this is also line, this is a circle, this is also a line, line 3, this is a point P refers to the point that you have to define, and again this is also a line.

Here these are the points; that means, could be the drilling location. The hole locations, like P7, P8, P6, P5, this way you have to specify you have points, you have lines and you have circles, you get the profile. Now you specify the x-axis and y-axis, plus the rotational axis also.

The first hole will be like this, this is 70 x-axis and this one will be 30 y-axis, that is the location and the diameter you specify right and three places 1 2 3. The location you specify. Now, this one is the line and its starting point is over here and ending over here; so, 160. So, that you have to specify, we get the drawing and you specify this one.

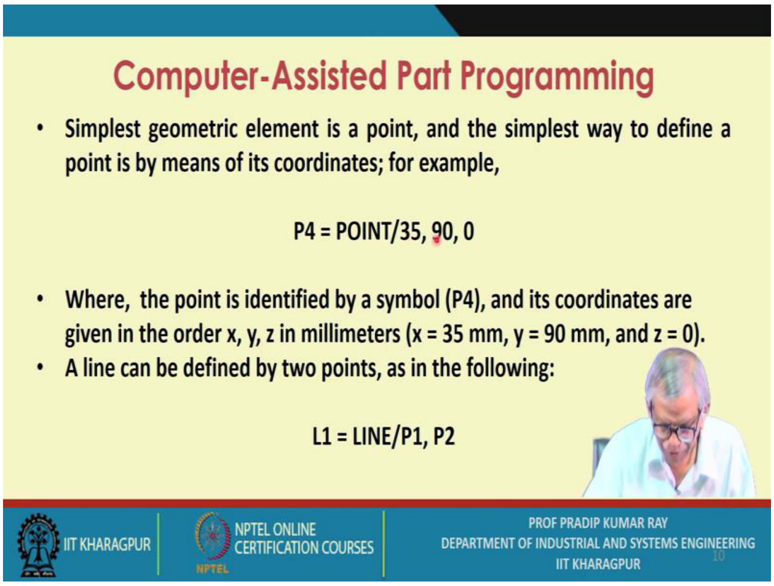
Then this point also you should specify; that means, this is the line and this distance is 60; from here it is 60 on y-axis and while you define this curve, you specify the radius.

This is 30 radius. This way we have to specify and now you say that here there should be a turn, what do you say that this point is to be specified? This is from here you go take a turn and then you move up to say this point; that means, the distance from here is 125.

Then the cutter moves along this right and then you get this particular shape. Again it comes back to this, now once you get this sort of part plus the thickness also you have to mention that is the 10 and these are the holes. There are three holes you specifying, fine.

This way you have to define the part, with all its feature and this is referred to as the geometric element, depending on the shape, size and other features of a particular part so, the number of geometric elements varying.

(Refer Slide Time: 21:38)



The simplest geometric element is a point, and the simplest way to define a point is by means of its coordinates; for example,

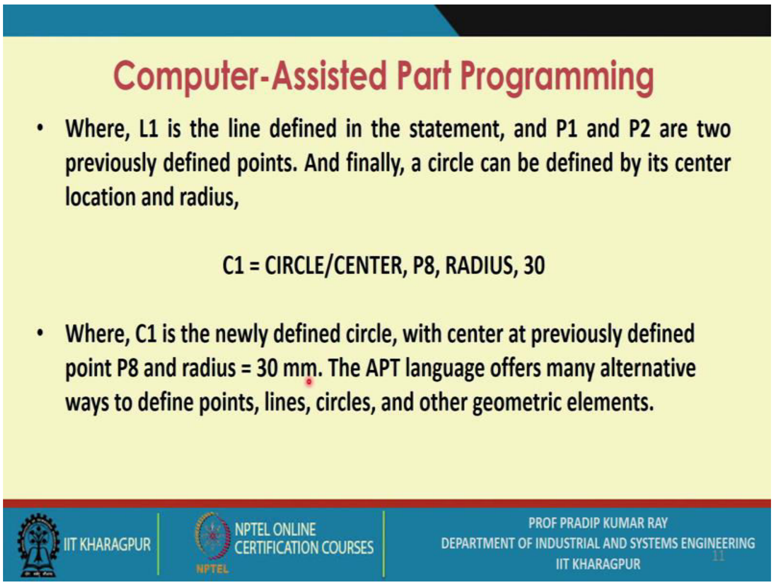
P4 = POINT/35, 90, 0

where the point is identified by a symbol (P4), and its coordinates are given in the order x, y, z in millimetres (x = 35 mm, y = 90 mm, and z = 0).

A line can be defined by two points, as in the following:

L1 = LINE/P1, P2

(Refer Slide Time: 24:08)

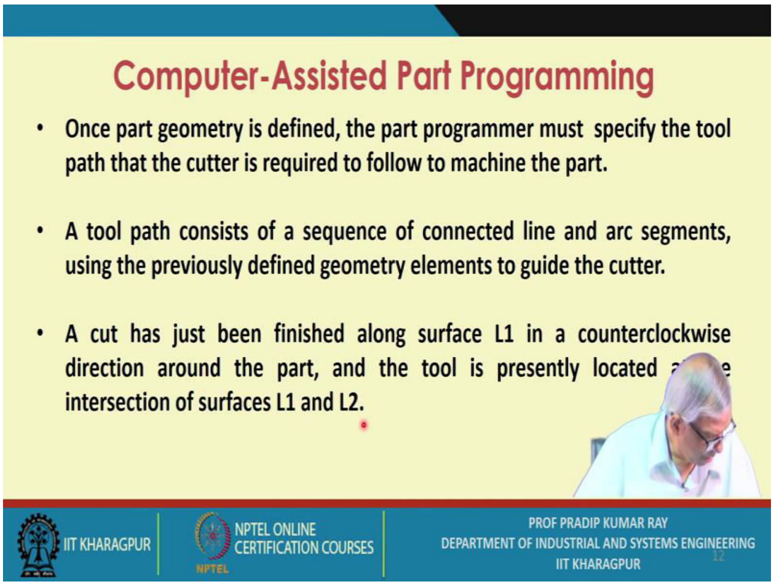


where L1 is the line defined in the statement, and P1 and P2 are two previously defined points. And finally, a circle can be defined by its center location and radius,

C1 = CIRCLE/CENTER, P8, RADIUS, 30

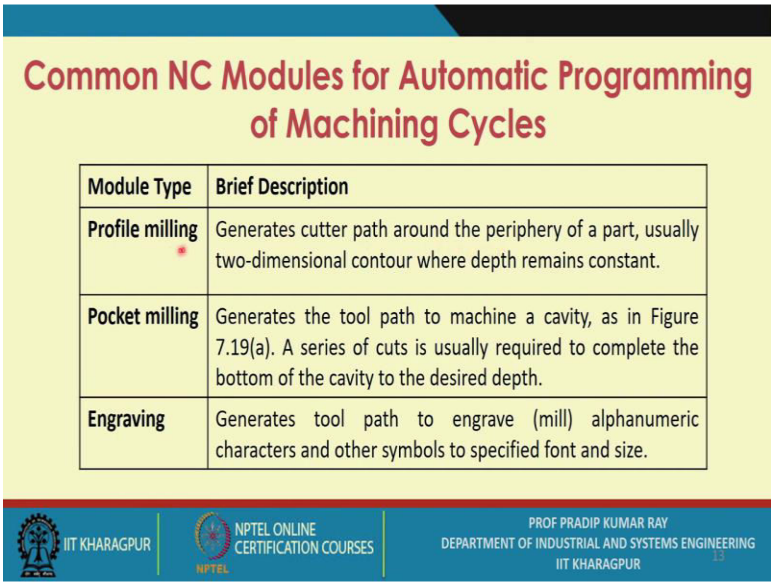
where C1 is the newly defined circle with center at previously defined point P8 and radius = 30 mm. The APT language offers many alternative ways to define points, lines, circles, and other geometric elements.

(Refer Slide Time: 25:28)



After the part geometry has been defined, the part programmer must next specify the tool path that the cutter will follow to machine the part. The tool path consists of a sequence of connected line and arc segments, using the previously defined geometry elements to guide the cutter. A cut has just been finished along surface L1 in a counterclockwise direction around the part, and the tool is presently located at the intersection of surfaces L1 and L2.

(Refer Slide Time: 26:50)

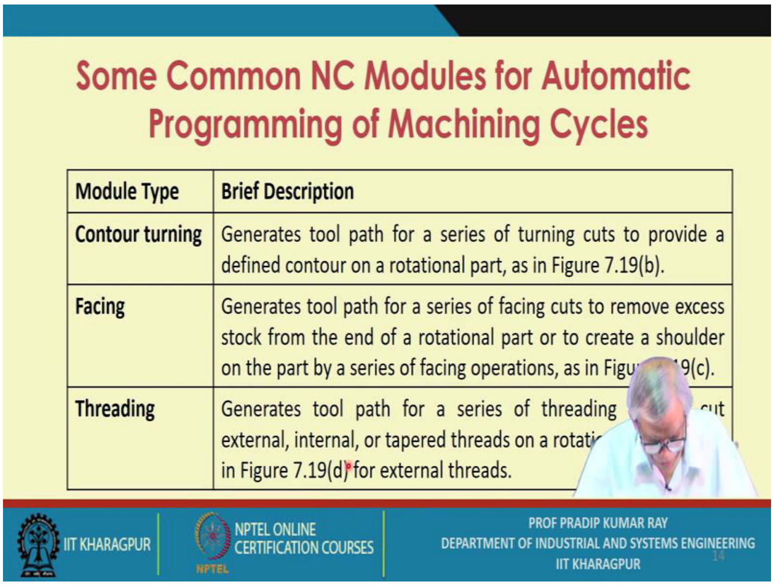


Let me highlight some of the important NC modules for automatic programming of machining cycles, one is profile milling, generates cutter path around the periphery of a part. Usually, two-dimensional contour where depth remains constant.

Then there is another the module called pocket milling. It generates the tool path to machine a cavity, as in Figure 7.19(a). A series of cuts is usually required to complete the bottom of the cavity to the desired depth.

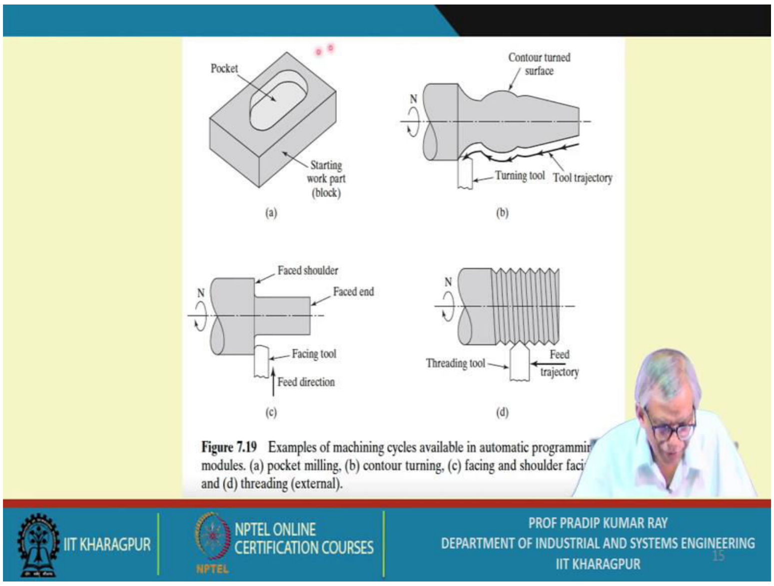
Then the next module is engraving. It generates a tool path to engrave alpha numeric characters and other symbols to specified font and size, this is very common operation.

(Refer Slide Time: 27:48)



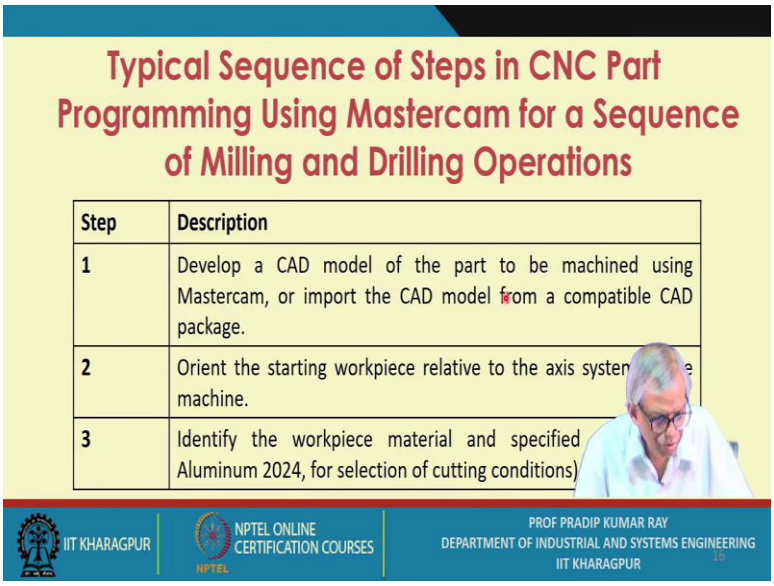
Then contour turning, generates tool path for a series of turning cuts. Then facing as an important machining operation, and threading.

(Refer Slide Time: 28:09)



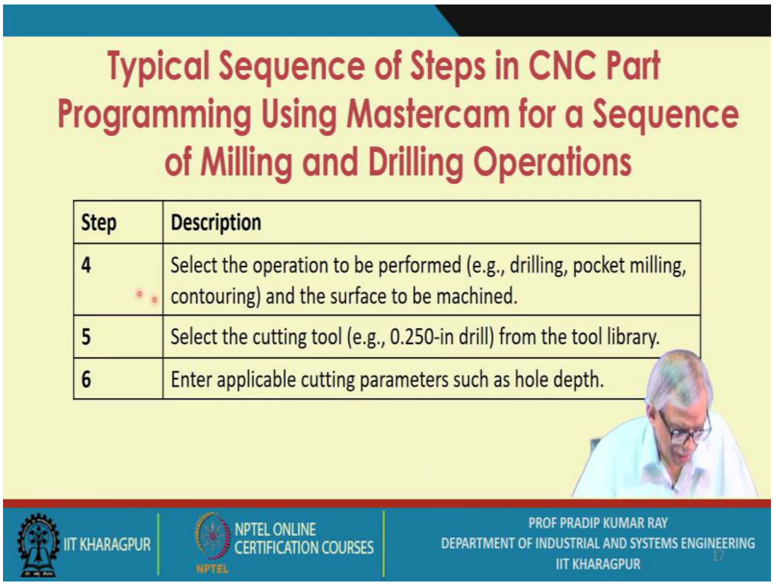
And, these are the examples of machining cycles. First one is the pocket milling. Second is contour turning, third one is facing operation and the last one that is given that is the threading. One should take up one particular operation and check what are the geometry elements and with respect to a particular machine one have to define NC coordinate systems and their values.

(Refer Slide Time: 29:19)



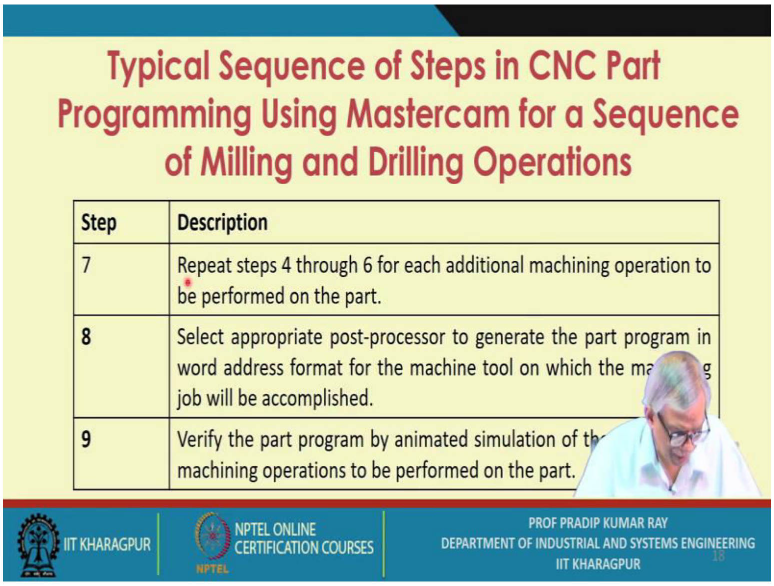
The typical sequence of steps in CNC part programming using Mastercam for a sequence of milling and drilling operations, develop a CAD model is the 1st part. 2nd part is orient the starting workpiece relative to the axis system of the machine. At the 3rd step identify the workpiece material and specified grade.

(Refer Slide Time: 29:56)



4th step is to select the operation to be performed and the 5th step is select the cutting tool which is very important and enter the applicable cutting pattern parameters such as hole depth is the 6th step.

(Refer Slide Time: 30:40)



Repeat steps 4 through 6 for each additional machining operation, once you change the machining operation, you have to select a particular tool. Select appropriate post processor to generate the part program, that is a part of computer assisted part programming.

And, next step is to verify the part program; that means, before you go for using it for mass production, first you have to check, there must be some trial run is refer to animated the simulation of the sequence.

(Refer Slide Time: 31:23)