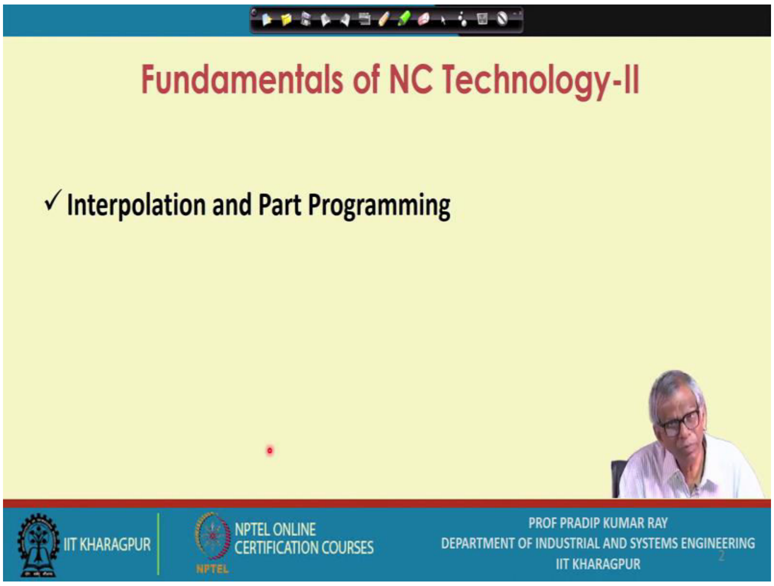
Automation in Production Systems and Management
Prof. Pradip Kumar Ray
Vinod Gupta School of Management
Indian Institute of Technology, Kharagpur

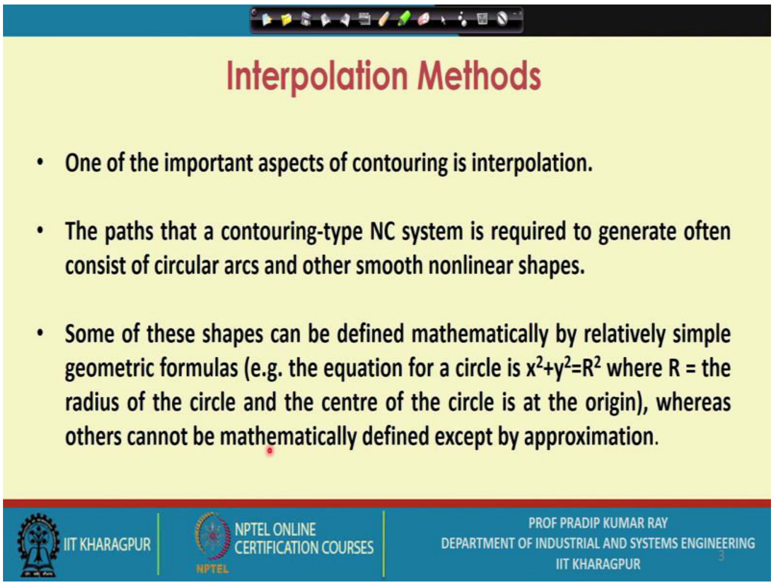
Fundamentals of NC Technology - II
Lecture - 24
Interpolation and Part Programming

During this week we are discussing several important issues related to NC technology and we are referring to fundamentals of NC technology.

(Refer Slide Time: 00:33)

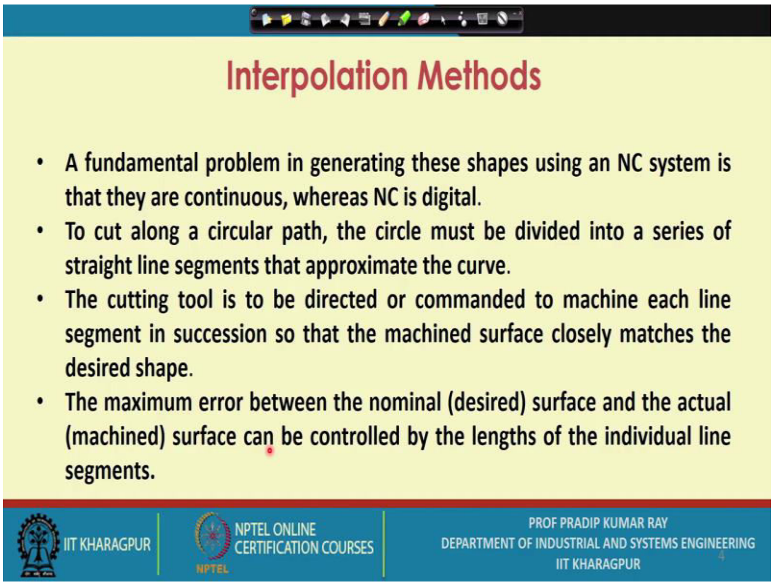


(Refer Slide Time: 04:27)



One of the important aspects of contouring is interpolation. The paths that a contouring-type NC system is required to generate often consist of circular arcs and other smooth nonlinear shapes. Some of these shapes can be defined mathematically by relatively simple geometric formulas (e.g. the equation for a circle is $x^2+y^2=R^2$ where R = the radius of the circle and the center of the circle is at the origin., whereas others cannot be mathematically defined except by approximation.

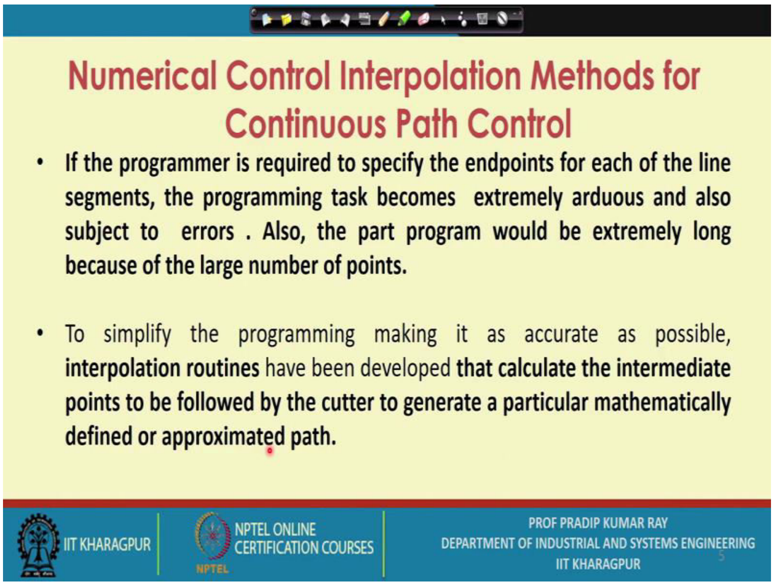
(Refer Slide Time: 09:01)



A fundamental problem in generating these shapes using NC equipment is that they are continuous, whereas NC is digital. To cut along a circular path, the circle must be divided into a series of straight line segments that approximate the curve. The tool is commanded to machine each line segment in succession so that the machined surface closely matches the desired shape. The maximum error between the nominal (desired) surface and the actual (machined) surface can be controlled by the lengths of the individual line segments.

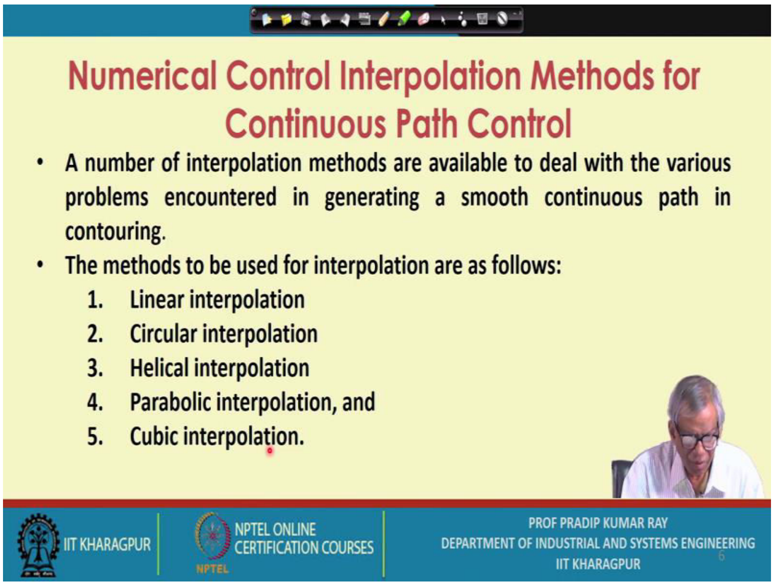
.

(Refer Slide Time: 13:25)



If the programmer were required to specify the endpoints for each of the line segments, the programming task would be extremely arduous and fraught with errors. Also, the part program would be extremely long because of the large number of points. To ease the burden, interpolation routines have been developed that calculate the intermediate points to be followed by the cutter to generate a particular mathematically defined or approximated path.

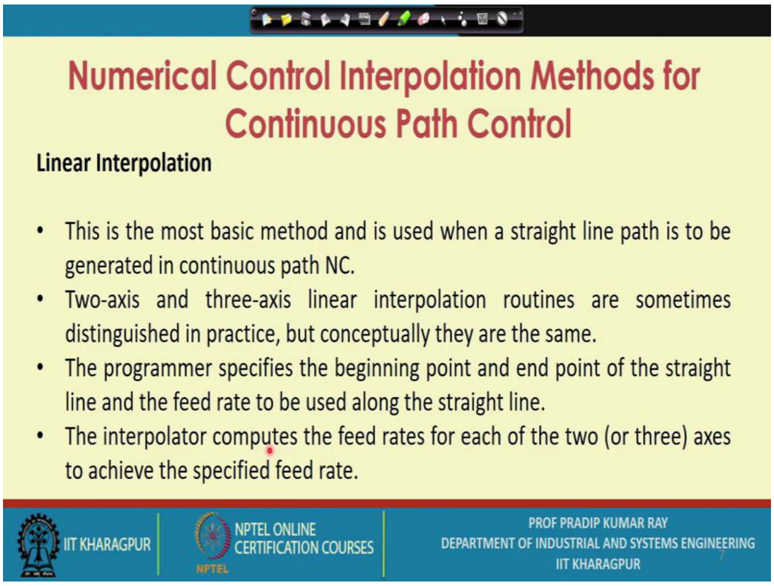
(Refer Slide Time: 16:53)



The number of interpolation methods are available to deal with the various problems encountered in generating a smooth continuous path in contouring. Basically, for contouring you have to use these interpretation methods. The number of the methods you can use define the continuous path or depending on the type of the continuous path. So, this continuous path could be linear, circular, helical, parabolic, cubic.

The cubic is most the complicated one. Parabolic interpolation is also required and you need 5 axis machine, 6 axis machines NC machines. So, what are the different types of methods for interpolation.

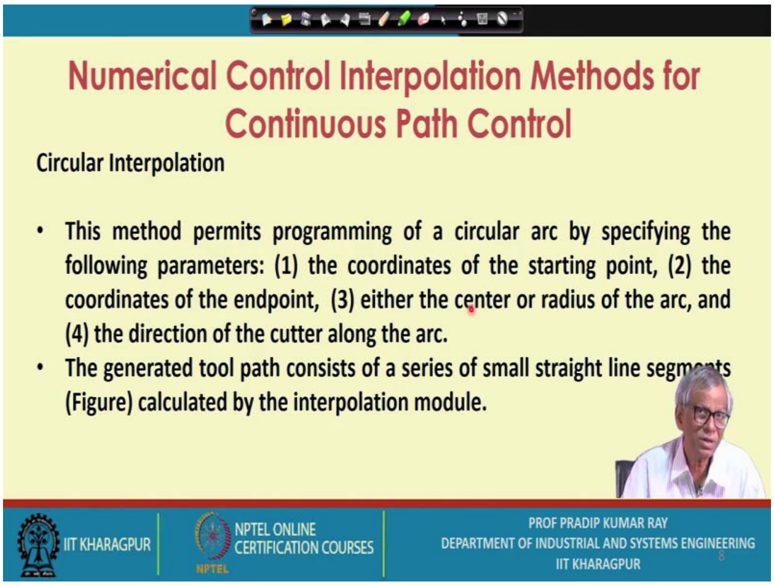
(Refer Slide Time: 18:31)



First one is the linear interpolation. This is the most basic method and is used when a straight line path is to be generated in continuous path NC. Two-axis and three-axis linear interpolation routines are sometimes distinguished in practice, but conceptually they are the same. The programmer specifies the beginning point and end point of the straight line and the feed rate to be used along the straight line. The interpolator computes the feed rates for each of the two (or three) axes to achieve the specified feed rate.

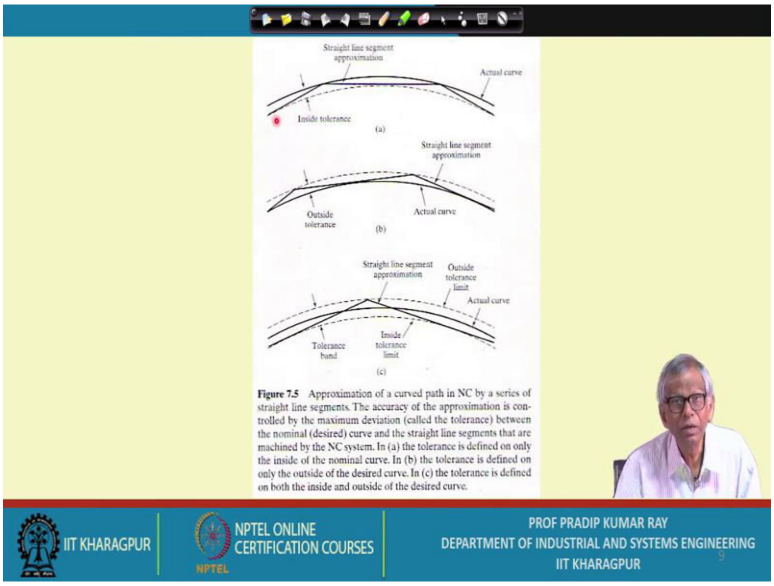
.

(Refer Slide Time: 21:29)



Next one is circular interpolation This method permits programming of a circular arc by specifying the following parameters: (1) the coordinates of the starting point, (2) the coordinates of the endpoint, (3) either the center or radius of the arc, and (4) the direction of the cutter along the arc. The generated tool path consists of a series of small straight line segments (see Figure 7.5) calculated by the interpolation module.

(Refer Slide Time: 23:27)



In the first figure here what you find that this is basically your desired path. This is your desired path, as you are using digital technology and the tool can move only in a straight line. This path is basically to be defined in terms of a number of the straight lines. So, here in this case you say yes, one straight line, the second straight line and the third straight line, you produce this shape obviously, this shape is not matching with the actual shape which you desire. There will be some error. Straight line segment approximation. These are the approximate straight line is an approx approximate. you define inside tolerance because here the tolerances are defined on only on the inside of the nominal curve not outside.

That is why it is basically called the inside tolerance and you determine these number of straight lines in such a way that this inside tolerance is minimum. It cannot be 0 because you have been using the straight-line segments, this way you could define that is figure a.

In the figure b you have an alternative you say I will define not inside tolerance. I will define outside tolerance. The reason is that instead of placing the straight lines within the curve, you are placing outside of the curve. This is one straight line, this is the second straight line and this is the third straight line; that means, all the straight lines are existing outside of the curve, not inside.

These will not be any inside tolerance there will be outside tolerance, there will be outside tolerance, you define the outside tolerance like this. So, as you might have noticed that as you increase the number of straight lines what do you find that the inside tolerance or the outside tolerance will be very less. And, you go on increasing the number of straight lines.

There is a research cell, they have developed those empirical formulations, and while get a software, they have carried out all this research.

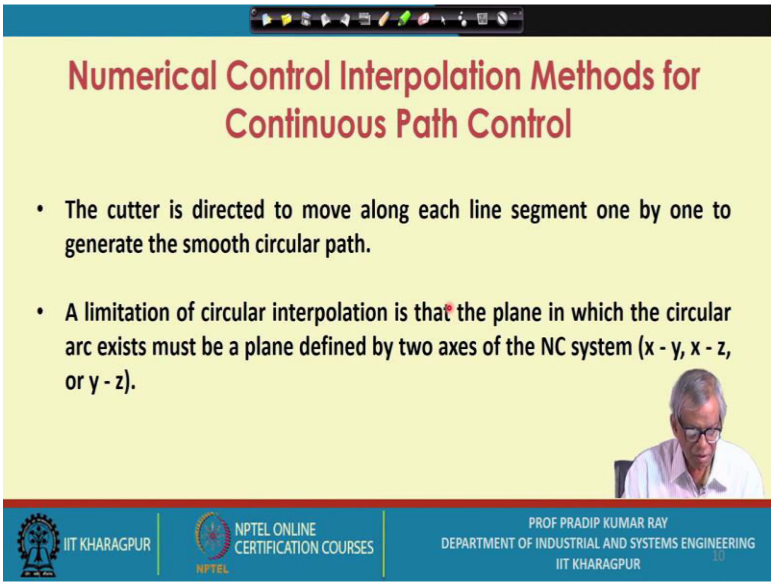
You check whether your, the NC machine tool is having this kind of software or not whether these are already loaded or not. There will be machine control unit or say basically the computer, just check whether this software is either loaded or not.

Third alternative is to define this path not only just keeping all the straight lines within or outside.

In the third figure, figure c if you notice you will find that we defined the tolerance band. Basically your actual curve or the nominal curve the desired curve and these are the straight line segments; these are the straight line segments. So, certain portion of the straight line is within and certain portion is outside.

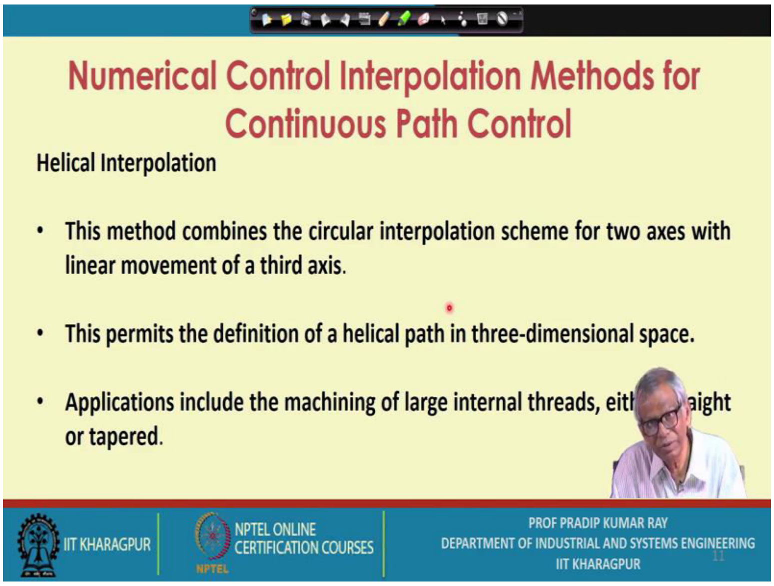
You define this tolerance band; you have the inside tolerance limit as well as the outside tolerance limit. You define and you determine the number of straight lines in such a way, through research so that this total by the tolerance band is minimum. This is the error basically the error is defined with respect to the tolerance band.

(Refer Slide Time: 29:41)



The cutter is directed to move along each line segment one by one to generate the smooth circular path. A limitation of circular interpolation is that the plane in which the circular arc exists must be a plane defined by two axes of the NC system (x - y, x - z, or y - z).

(Refer Slide Time: 30:25)

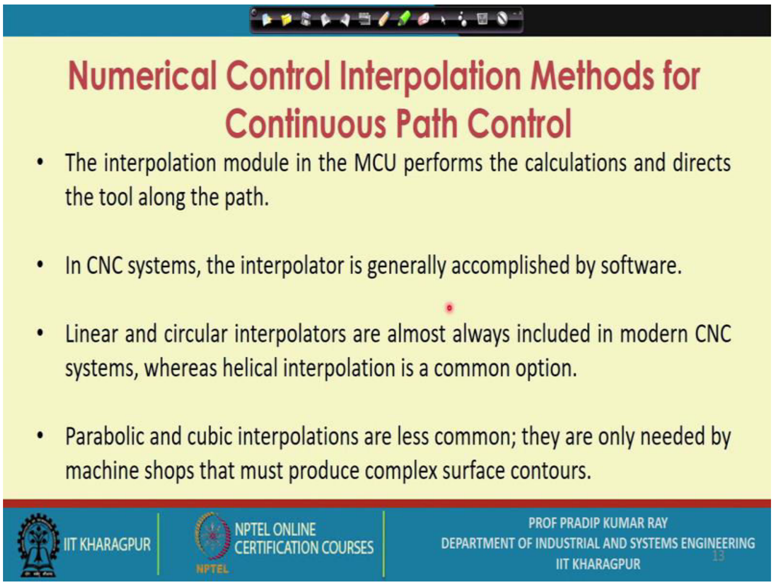


Next one is helical interpolation. This method combines the circular interpolation scheme for two axes with linear movement of a third axis. This permits the definition of a helical path in three-dimensional space. Applications include the machining of large internal threads, either straight or tapered.

(Refer Slide Time: 31:53)

Next one is Parabolic and cubic Interpolation. These routines provide approximations of free-form curves using higher order equations. They generally require considerable computational power and are not as common as linear and circular interpolation. Most applications are in the aerospace and automotive industries for freeform designs that cannot accurately and conveniently be approximated by combining linear and circular interpolations.

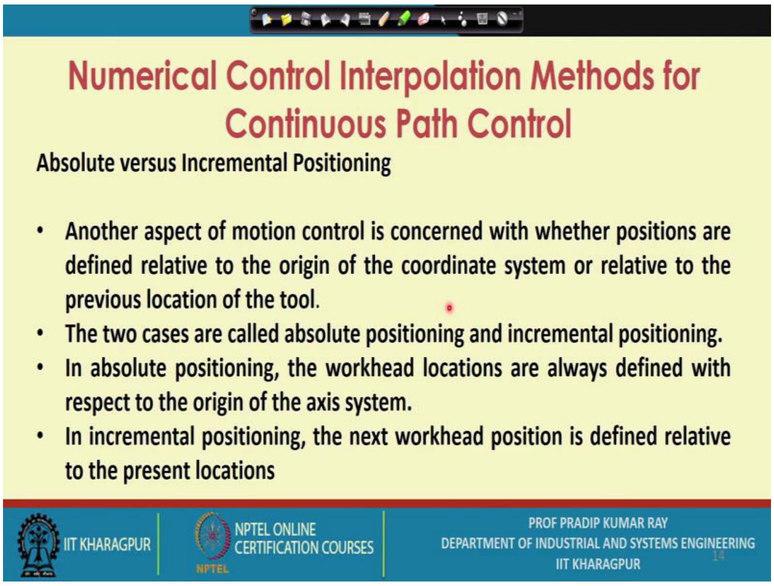
(Refer Slide Time: 33:17)



The interpolation module in the MCU machine control unit performs the calculations and directs the tool along the path. So, the tool path movement you have to do. In CNC systems the interpolation is generally accomplished by software. Linear and circular interpolators are almost always included in the modern CNC systems whereas, helical interpolation is a common option.

Parabolic and cubic interpolations are less common, they are only needed by machine shops that must produce complex surface contours.

(Refer Slide Time: 33:59)



Now, one particular issue is absolute and incremental positioning. Another aspect of motion control is concerned with whether positions are defined relative to the origin of the coordinate system or relative to the previous location of the tool. The two cases are called absolute positioning and incremental positioning. In absolute positioning, the workhead locations are always defined with respect to the origin of the axis system. In incremental positioning, the next workhead position is defined relative to the present locations

(Refer Slide Time: 34:59)