Module

5

CNC Machines

Lesson
23

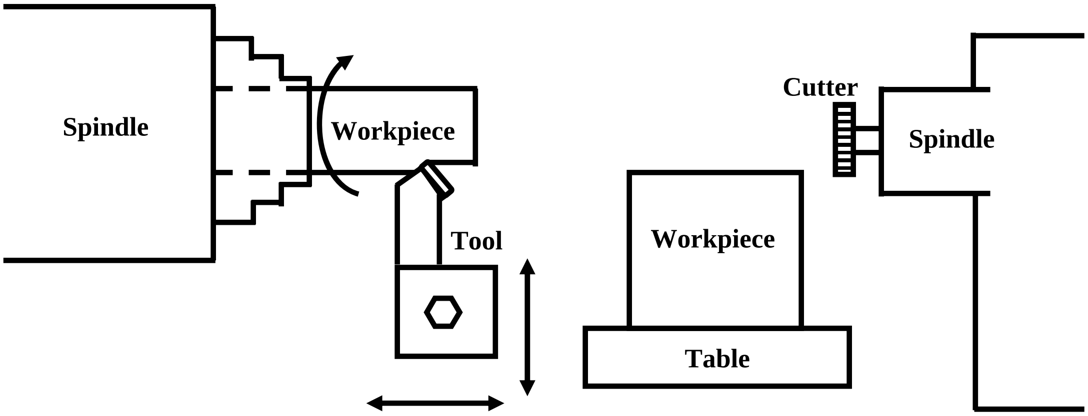
Introduction to Computer Numerically Controlled (CNC) Machines

Instructional Objectives

After learning the lesson students should be able to

A. Define Numerical Control and describe its advantages and disadvantages
B. Name and describe the major components of a CNC system
C. Explain the coordinate systems adopted for CNC programming
D. Describe the major types of motion control strategies
E. Describe the major classifications of CNC machines

Introduction

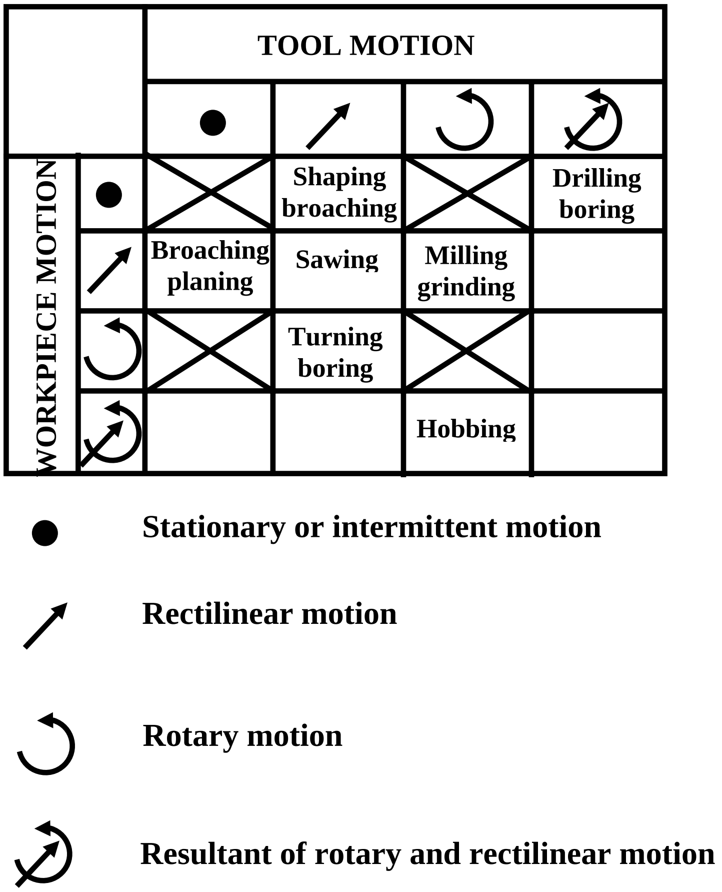


(a) Turning **(b) Milling**

Fig. 23.1 Drive in a metal cutting

Introductory Concepts of Machining

Machining is basically removal of material, most often metal, from the workpiece, using one or more cutting tools to achieve the desired dimensions. There are different machining processes, such as, turning, milling, boring etc. In all these cases metal is removed by a shearing process, which occurs due to the relative motion between the workpiece and the tool. Generally, one of the two rotates at designated and generally high speed, causing the shearing of material (known as chips), from the workpiece. The other moves relatively slowly to effect removal of metal throughout the workpiece. For example, as seen above in a turning operation of lathes, the “job” or the workpiece rotates in a chuck, while the tool moves in two dimensions translationally. On the other hand, in milling, it is the cutter which rotates on a spindle, while the workpiece, which is fastened to a table, moves in X-Y dimensions. While, a precise and high speed rotational motion is needed for good finish of the machined surface, for dimensional accuracy, precise position and velocity control of the table drive are essential.



		TOOL MOTION			
		●	↗	↺	↺↗
WORKPIECE MOTION	●	╳	Shaping broaching	╳	Drilling boring
	↗	Broaching planing	Sawing	Milling grinding	
	↺	╳	Turning boring	╳	
	↺↗			Hobbing	

●	**Stationary or intermittent motion**
↗	**Rectilinear motion**
↺	**Rotary motion**
↺↗	**Resultant of rotary and rectilinear motion**

Fig. 23.2 Nature of motion of the Job and the Tool for various Metal Cutting Processes

For all metal-cutting processes, the cutting speed, feed, and depth of cut are important parameters. The figure below shows the important geometry for the turning process. The cutting speed, which is a measure of the part cut surface speed relative to the tool. Speed is a velocity unit for the translational motion, which is may be stated in or meters/min. The depth of cut, DOC is the depth that the tool is plunged into the surface. Feed defines the relative lateral movement between the cutting tool and the workpiece. Thus, together with depth of cut, feed decides the cross section of the material removed for every rotation of the job or the tool, as the case may be. Feed is the amount of material removed for each revolution or per pass of the tool over the workpiece and is measured in units of length/revolution, length/pass or other appropriate unit for the particular process.

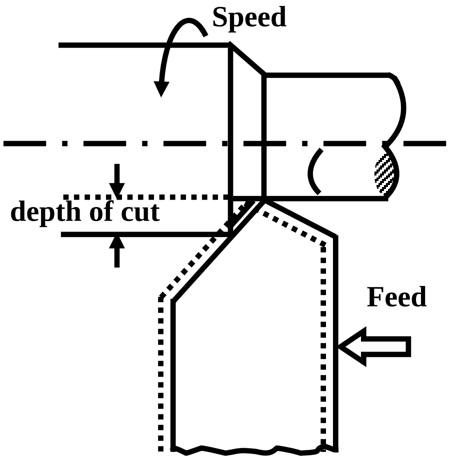
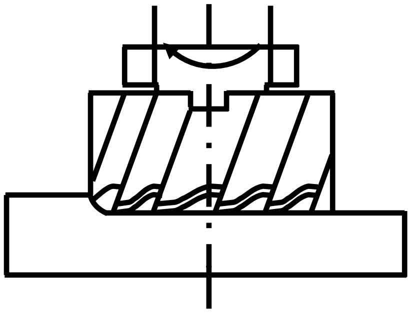


Fig. 23.3 Speed, feed and depth of cut for turning operation

Point to Ponder: 1

A. *Consider the metal cutting configuration for milling shown. Identify the motion directions corresponding to speed, feed and depth of cut*



B. *"Metal cutting is essentially a shearing phenomenon", justify the statement with respect to the entries in Table 23.1.*

What is Computer Numerical Control?

Modern precision manufacturing demands extreme dimensional accuracy and surface finish. Such performance is very difficult to achieve manually, if not impossible, even with expert operators. In cases where it is possible, it takes much higher time due to the need for frequent dimensional measurement to prevent overcutting. It is thus obvious that automated motion control would replace manual "handwheel" control in modern manufacturing. Development of computer numerically controlled (CNC) machines has also made possible the automation of the machining processes with flexibility to handle production of small to medium batch of parts.

In the 1940s when the U.S. Air Force perceived the need to manufacture complex parts for high-speed aircraft. This led to the development of computer-based automatic machine tool controls also known as the Numerical Control (NC) systems. Commercial production of NC machine tools started around the fifties and sixties around the world. Note that at this time the microprocessor has not yet been invented.

Initially, the CNC technology was applied on lathes, milling machines, etc. which could perform a single type of metal cutting operation. Later, attempt was made to handle a variety of workpieces that may require several different types machining operations and to finish them in a single set-up. Thus CNC machining Centres capable of performing multiple operations were developed. To start with, CNC machining centres were developed for machining prismatic components combining operations like milling, drilling, boring and tapping. Gradually machines for manufacturing cylindrical components, called turning centers were developed.

Numerical Control

Automatically controlling a machine tool based on a set of pre-programmed machining and movement instructions is known as numerical control, or NC.

In a typical NC system the motion and machining instructions and the related numerical data, together called a *part program*, used to be written on a punched tape. The part program is arranged in the form of blocks of information, each related to a particular operation in a sequence of operations needed for producing a mechanical component. The punched tape used to be read one block at a time. Each block contained, in a particular syntax, information needed for processing a particular machining instruction such as, the segment length, its cutting speed, feed, etc. These pieces of information were related to the final dimensions of the workpiece (length, width, and radii of circles) and the contour forms (linear, circular, or other) as per the drawing. Based on these dimensions, motion commands were given separately for each axis of motion. Other instructions and related machining parameters, such as cutting speed, feed rate, as well as auxiliary functions related to coolant flow, spindle speed, part clamping, are also provided in part programs depending on manufacturing specifications such as tolerance and surface finish. Punched tapes are mostly obsolete now, being replaced by magnetic disks and optical disks.

NC equipment has been defined by the Electronic Industries Association (E1A) as “A system in which actions are controlled by the direct insertion of numerical data at some point. The system must automatically interpret at least a portion of this data.”
This is an old definition as is apparent from the terminology used in the definition

Computer Numerically Controlled (CNC) machine tools, the modern versions of NC machines have an embedded system involving several microprocessors and related electronics as the Machine Control Unit (MCU). Initially, these were developed in the seventies in the US and Japan. However, they became much more popular in Japan than in the US. In CNC systems multiple microprocessors and programmable logic controllers work in parallel for simultaneous servo position and velocity control of several axes of a machine for contour cutting as well as monitoring of the cutting process and the machine tool. Thus, milling and boring machines can be fused into versatile machining centers. Similarly, turning centers can realize a fusion of various types of lathes. Over a period of time, several additional features were introduced, leading to increased machine utilisation and reduced operator intervention. Some of these are:
(a) Tool/work monitoring: For enhanced quality, avoidance of breakdowns.
(b) Automated tool magazine and palette management: For increased versatility and reduced operator intervention over long hours of operation
(c) Direct numerical control (DNC): Uses a computer interface to upload and download part programs in to the machine automatically.

Advantages of a CNC Machine

CNC machines offer the following advantages in manufacturing.

- *Higher flexibility*: This is essentially because of programmability, programmed control and facilities for multiple operations in one machining centre,
- *Increased productivity*: Due to low cycle time achieved through higher material removal rates and low set up times achieved by faster tool positioning, changing, automated material handling etc.
- *Improved quality*: Due to accurate part dimensions and excellent surface finish that can be achieved due to precision motion control and improved thermal control by automatic control of coolant flow.
- *Reduced scrap rate*: Use of Part programs that are developed using optimization procedures
- *Reliable and Safe operation*: Advanced engineering practices for design and manufacturing, automated monitoring, improved maintenance and low human interaction
- *Smaller footprint*: Due to the fact that several machines are fused into one.

On the other hand, the main disadvantages of NC systems are

- Relatively higher cost compared to manual versions
- More complicated maintenance due to the complex nature of the technologies
- Need for skilled part programmers.

The above disadvantages indicate that CNC machines can be gainfully deployed only when the required product quality and average volume of production demand it.

Classification of NC Systems

CNC machine tool systems can be classified in various ways such as :

1. Point-to-point or contouring : depending on whether the machine cuts metal while the workpiece moves relative to the tool
2. Incremental or absolute : depending on the type of coordinate system adopted to parameterise the motion commands
3. Open-loop or closed-loop : depending on the control system adopted for axis motion control

Point-to-point systems

Point-to-point (PTP) systems are the ones where, either the work piece or the cutting tool is moved with respect to the other as stationary until it arrives at the desired position and then the cutting tool performs the required task with the motion axes stationary. Such systems are used, typically, to perform hole operations such as drilling, boring, reaming, tapping and punching. In a PTP system, the path of the cutting tool and its feed rate while traveling from one point to the next are not significant, since, the tool is not cutting while there is motion. Therefore, such systems require only control of only the final position of the tool. The path from the starting point to the final position need not be controlled.

Contouring systems

In contouring systems, the tool is cutting while the axes of motion are moving, such as in a milling machine. All axes of motion might move simultaneously, each at a different velocity. When a nonlinear path is required, the axial velocity changes, even within the segment. For example, cutting a circular contour requires sinusoidal rates of change in both axes. The motion controller is therefore required to synchronize the axes of motion to generate a predetermined path, generally a line or a circular arc. A contouring system needs capability of controlling its drive motors independently at various speeds as the tool moves towards the specified position. This involves simultaneous motion control of two or more axes, which requires separate position and velocity loops. It also requires an interpolator program that generates the position and velocity setpoints for the two drive axes, continuously along the contour.

In modern machines there is capability for programming machine axes, either as point-to-point or as continuous (that is contouring)

Before the next type of classification is introduced, it is necessary to present the basic coordinate system conventions in a machine tool.

Point to Ponder: 2

A. *Comment on the sensing requirements for PTP and Contouring axes*

B. *Do you think the overall cutting time can be optimized for PTP and Contouring systems? Are there any constraints to that?*

Coordinate Systems

The coordinate system is defined by the definition of the translational and rotational motion coordinates. Each translational axis *of motion* defines a direction in which the cutting tool moves relative to the work piece. The main three axes of motion are referred to as the X, *Y*. and Z axes. The Z axis is perpendicular to both X and *Y* in order to create a right-hand coordinate system, such as shown in Fig. 23.5. A positive motion in the Z direction moves the cutting tool away from the workpiece. The location of the origin is generally adjustible. Figure 23.4 shows the coordinate system for turning as in a lathe while Fig. 23.5 shows the system for drilling and milling.

For a lathe, the infeed/radial axis is the x-axis, the carriage/length axis is the z-axis. There is no need for a y-axis because the tool moves in a plane through the rotational center of the work. Coordinates on the work piece shown below are relative to the work.

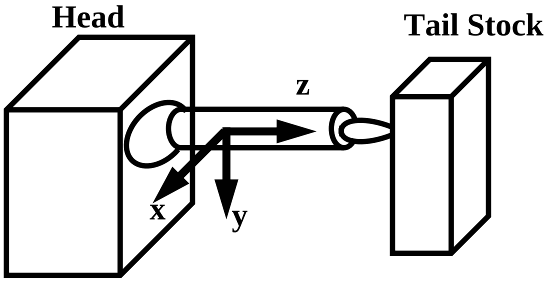


Fig. 23.4 Co-ordinate system for turning

In drilling and milling machines the X and *Y* axes are horizontal. For example, a positive motion command in the drill moves the X axis from left to right, the *Y* axis from front to back, and the Z axis toward the top. In the lathe only two axes are required to command the motions of the tool. Since the spindle is horizontal, the Z axis is horizontal as well. The cross axis is denoted by X. A positive position command moves the Z axis from left to right and the X axis from back to front in order to create the right-hand coordinate system.

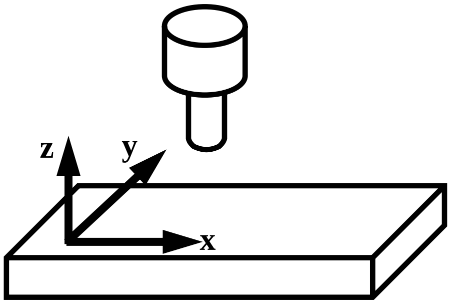


Fig. 23.5 Co-ordinate system for drilling and milling

For a tool with a horizontal spindle the x-axis is across the table, the y-axis is down, and the z-axis is out.

In addition to the translational motion, rotary motions around the axes parallel to X, Y, and Z can also be defined. Similarly, in addition to the primary motion coordinates, secondary coordinates can also exist.

Incremental Systems

In an incremental system the movements in each Part program block are expressed as the displacements along each coordinate axes with reference to the final position achieved at the end of executing the previous program block.

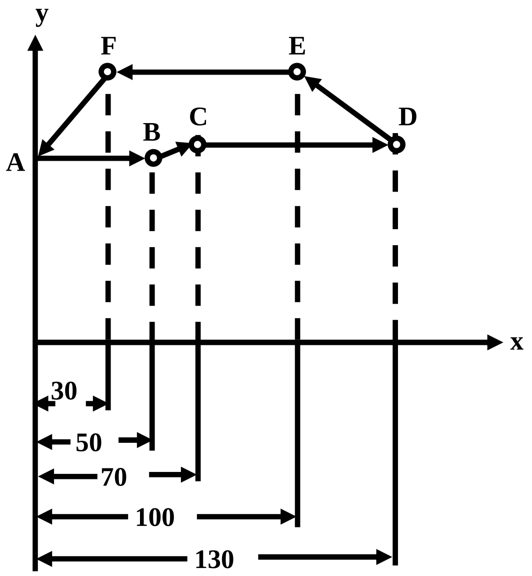


Fig. 23.6 A trajectory for drilling

Consider, for example, the trajectory of rectilinear motions shown in Fig. 23.6 for a PTP system. In an incremental system, the motion parameters, along the X-axis, for the segments, A-B, B-C, C-D, D-E, E-F and F-A, would be given as, 50, 20, 60, -30, -70 and –30, respectively.

Absolute System

An absolute NC system is one in which all position coordinates are referred to one fixed origin called the zero point. The zero point may be defined at any suitable point within the limits of the machine tool table and can be redefined from time to time. Any particular definition of the zero point remains valid till another definition is made. In the Fig. 23.6, considering the X-coordinate for point A as zero, the X-coordinate for points B and C would be 50 and 70, respectively, in an absolute coordinate system.

Most modem CNC systems permit application of both incremental and absolute programming methods. Even within a specific part program the method can be changed These CNC systems provide the user with the combined advantages of both methods

Unit of Displacement

Displacements are expressed in part programs by integers. Each unit corresponds to the position resolution of the axes of motion and will be referred to as the *basic length-unit (BLU).* The BLU is the smallest length of motion that can be repeatably sensed in the machine. It therefore determines the accuracy of machining possible with a given machine. For example, if a shaft angle encoder having a sensitivity of 500 pulses per revolution is mounted on a lead screw having a pitch of 5 mm, the BLU is 0.01 mm. In modern CNC machines the programmer can use floating-point dimensional data, which would be converted in to BLU's by the interpreter.

Point to Ponder: 3

A. *Can you think of one advantage and one disadvantage of the incremental coordinate system compared to the absolute one?*

B. *Is there any connection between the choice of coordinate system and the position sensing machine used for the machine tool?*

C. *How would you decide on the BLU for systems with position sensors such as LVDTs and resolvers?*

D. *Is the BLU affected by the motor or the drive system also?*

Part Programming

As mention earlier, a part program is a set of instructions often referred to as blocks, each of which refers to a segment of the machining operation performed by the machine tool. Each block may contain several code words in sequence. These provide:

1. Coordinate values (X, Y, Z, etc.) to specify the desired motion of a tool relative to a work piece. The coordinate values are specified within motion codeword and related interpolation parameters to indicate the type of motion required (e.g. point-to-point, or continuous straight or continuous circular) between the start and end coordinates. The CNC system computes the instantaneous motion command signals from these code words and applies them to drive units of the machine.
2. Machining parameters such as, feed rate, spindle speed, tool number, tool offset compensation parameters etc.
3. Codes for initiating machine tool functions like starting and stopping of the spindle, on/off control of coolant flow and optional stop. In addition to these coded functions, spindle speeds, feeds and the required tool numbers to perform machining in a desired sequence are also given.
4. Program execution control codes, such as block skip or end of block codes, block number etc.
5. Statements for configuring the subsystems on the machine tool such as programming the axes, configuring the data acquisition system etc.

A typical block of a Part program is shown below in Fig. 23.7. Note that the block contains a variety of code words such G codes, M codes etc. Each of these code words configure a particular aspect of the machine, to be used during the machining of the particular segment that the block programmes.

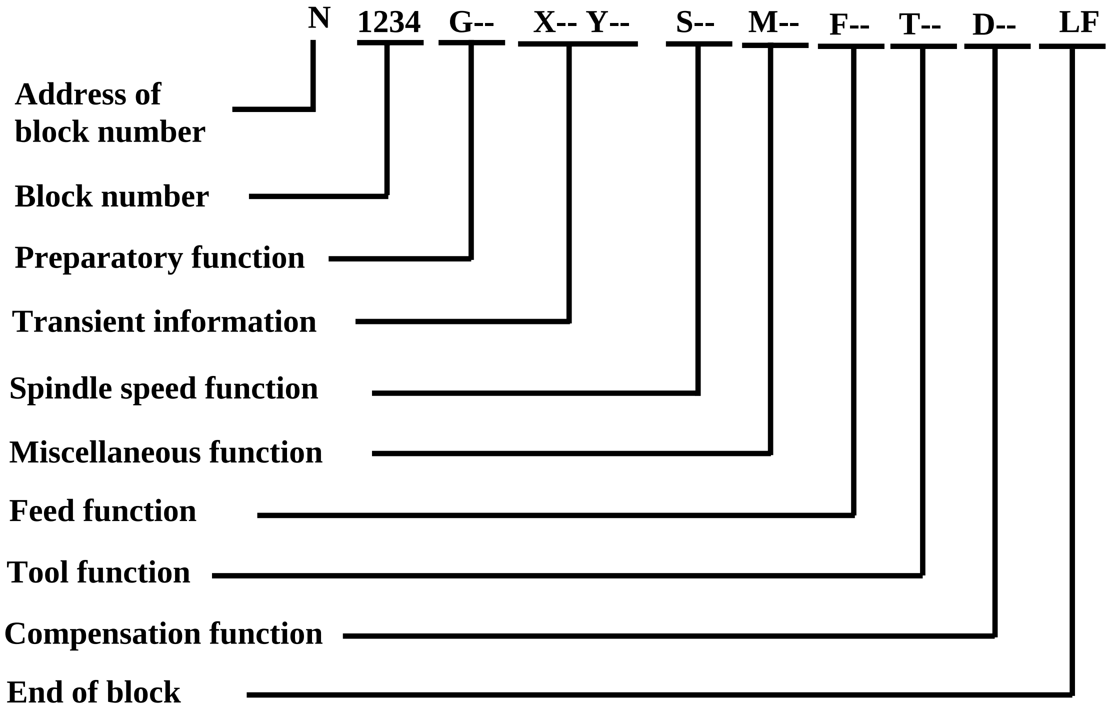


Fig. 23.7 Structure of a block in a part program

Appendix-1 provides some details of these codes. A typical sequence of operations in a part program would be,

A. Introductory functions such as units, coordinate definitions, coordinate conventions, such as, absolute or relative etc.

B. Feeds, speeds, etc.

C. Coolants, doors, etc.

D. Cutting tool movements and tool changes

E. Shutdown

Point to Ponder: 4

A. Consider the part program segment given below.

```
N0010G90;
N0011G01X1Y2;
N0012G01X2Y2;
N0013G91;
N0014G01X1;
N0015G92X2Y2;
N0016G01X1Y1;
```

Identify the meaning of the codes

B. Draw the trajectory of table motion that this program seeks to create.

C. Consider the part program segment given below for cutting a circular arc.

N10G01XY1;
N11G03X2Y512J1;

Determine the parameters of the circle.

D. *Is there any other way of operating machine tools other than by Part programs?*

Before we discuss the next categorization of CNC systems, it is important to understand how the instructions in a part program are converted to the operations needed for machining the parts. The control system software, which controls the axis motion, is called the axis manager. The axis manager controls the movement of the axes on the machine tool. This control may be divided into two distinct activities, namely,

- Axes interpolation
- Axes servo control

These two activities are executed by two specific routines, namely the interpolation and servo control routines, which communicate by means of a buffer for the exchange of data.

The axis manager is processed by one or more dedicated CPUs. In a multi-processor architecture, the interpolation and the servo control can be split between the various CPUs according to different combinations, such as,

- interpolation of all the axes on one CPU and servo control of all the axes on another CPU
- interpolation of all the axes on one CPU and servo control of part of the axes on the same CPU and servo control of the remaining axes on another CPU.

Interpolation

Interpolation consists in the calculation of the coordinated movement of several axes using the programmed parameters, in order to obtain a resulting trajectory, which can be of various types, such as:

- Straight line
- Circular
- Helicoidal

The interpolation module computes instant by instant position commands for the servo module, which in turn, drives the motors. There are two types of interpolators, namely:

- Process interpolator (for continuous axes)
- Point-to-point interpolator (for point-to-point axes)

Servo Control

Servo control consists of all the activities which allow several axes to effectively maintain the trajectory calculated by the interpolator. Continuous axes are continuously controlled by the system both for “speed” and for “position” so as to guarantee that the calculated trajectory is maintained. In contrast, for point-to-point axes there is no guarantee that the trajectory will be maintained. The only guarantee is that the final point will be reached.

Types of servo control for motion axes

The axes controlled by the axis manager may be divided into various types according to the specific function they perform on the machine tool. Some of these types are described below.

Coordinated Axis

This is a working axis, which may be interpolated along with other axes of the same type. This is necessary for generating specific 2D or 3D contours. The movement of one of the axes can be taken as the master and the other axes slaved to it. The mechanical and electrical features of the slave axis must be identical to those of the master. A coordinated axis can also be rotary and programmed in degrees. Note that for rotary axes, it may or may not be needed to map angular displacements to a $(0\text{-}2\pi)$ interval.

Point-to-point Axis

This axis is not required to be interpolated with others, since it is used for only for positioning from one point to another. Such an axis may be viewed as an independent mechanical component fitted with a positioning transducer.

Spindle Axis

There are two types of spindle axes. For some, only the speed of the axis need to be controlled and not the position by the spindle servo control system. Such an axis essentially realizes a “motorized” tool. For the second type, the speed of this spindle axis, as well as its angular position can be controlled. This has application in controlling threading processes. It is also possible to drive the spindle in coordinated motion, interpolated with the other axes. This uses the spindle transducer value as the set point for the other axes. A typical example is the C axis in lathes. One can command a controlled acceleration ramp for the spindle rotation command. However, for improved angular positioning, this must be eliminated. It is also possible to have spindle drives without servo control, generally for spindles driven with ac motors. The only control needed in such a case is for reversal of spindle rotation.

For control of tool and workpiece motion in the various ways described above, one of two kinds of control systems is employed.

Open Loop Systems

The term open-loop means that there is no feedback, and in open loop systems the motion controller produces outputs depending only on its set points, without feedback information about the effect that the output produces on the motion axes. We have already seen that the effects of controller outputs on the plant may not be the same always, since it depends on factors such as loads, parameter variations in the plant etc. In open loop systems, the set points are computed from the instructions in the Part program and fed to the controller, which may reside in a different microprocessor, through an interface. These motion commands may be in the form of electrical pulses (typical for step motor drives) or analog or digital signals, and converted to speed or current set points by the controller. These setpoints, in turn, are sent to the power electronic drive system that applies the necessary voltage/current to the motors.

The primary drawback of open-loop system is that there is no feedback system to check whether the commanded position and velocity has been achieved. If the system performance is affected by load, temperature or friction then the actual output could deviate from the desired output.

For these reasons, the open-loop system is generally used in point-to-point systems where the accuracy requirements are not critical. Contouring systems do not use open-loop control.

Closed Loop systems

Closed-loop control, as described in the module on controllers, continuously senses the actual position and velocity of the axis, using digital sensors such as encoders or analog sensors such resolvers and tachogenerators and compares them with the setpoints. The difference between the actual value of the variable and its setpoint is the error. The control law takes the error as the input and drives the actuator, in this case the servo motor and its drive system, to achieve motion variables that are close to the set points. As we know, closed loop systems can achieve much closer tracking of set points even with disturbances and parameter variations in the system with, say, with temperature. Closed-looped systems, on the other hand, require more complex control as well as feedback devices and circuitry in order for them to implement both position and velocity control. Most modern closed-loop CNC systems are able to provide very close resolution of 0.0001 of an inch.

Point to Ponder: 5

A. *Why is closed loop control required for continuous axes and not for PTP axes?*

B. *What sort of motors, drives, controllers and sensors would you recommend for continuous axis control?*

C. *Comment on the comparative requirements of computing speed and memory for PTP and continuous axis control*

Lesson Summary

In this lesson, the following topics related to CNC machines have been discussed.

A. Fundamentals of machining and its parameters

B. Definition and advantages

C. Introduction to coordinate systems

D. Part programming

E. Basic axis control strategies

F. Classification

Appendix-1

In this appendix we provide a list of G and M-codes for the reader to have an idea of the kind of functionality that can be realized using these codes. These codes were originally designed to be read from paper tapes and are designed to direct tool motion with simple commands.

A basic list of ‘G’ operation codes is given below. These direct motion of the tool.

G00 - Rapid move (not cutting)
G01 - Linear move
G02 - Clockwise circular motion
G03 - Counterclockwise circular motion
G04 - Dwell
G05 - Pause (for operator intervention)
G08 - Acceleration
G09 - Deceleration
G17 - x-y plane for circular interpolation
G18 - z-x plane for circular interpolation
G19 - y-z plane for circular interpolation
G20 - turning cycle or inch data specification
G21 - thread cutting cycle or metric data specification
G24 - face turning cycle
G25 - wait for input to go low
G26 - wait for input to go high
G28 - return to reference point
G29 - return from reference point
G31 - Stop on input
G33-35 - thread cutting functions
G35 - wait for input to go low
G36 - wait for input to go high
G40 - cutter compensation cancel
G41 - cutter compensation to the left
G42 - cutter compensation to the right
G43 - tool length compensation, positive
G44 - tool length compensation, negative
G50 - Preset position
G70 - set inch based units or finishing cycle
G71 - set metric units or stock removal
G72 - indicate finishing cycle
G72 - 3D circular interpolation clockwise
G73 - turning cycle contour
G73 - 3D circular interpolation counter clockwise
G74 - facing cycle contour
G74.1 - disable 360 deg arcs
G75 - pattern repeating
G75.1 - enable 360 degree arcs
G76 - deep hole drilling, cut cycle in z-axis
G77 - cut-in cycle in x-axis

G78 - multiple threading cycle
G80 - fixed cycle cancel
G81-89 - fixed cycles specified by machine tool manufacturers
G81 - drilling cycle
G82 - straight drilling cycle with dwell
G83 - drilling cycle
G83 - peck drilling cycle
G84 - taping cycle
G85 - reaming cycle
G85 - boring cycle
G86 - boring with spindle off and dwell cycle
G89 - boring cycle with dwell
G90 - absolute dimension program
G91 - incremental dimensions
G92 - Spindle speed limit
G93 - Coordinate system setting
G94 - Feed rate in ipm
G95 - Feed rate in ipr
G96 - Surface cutting speed
G97 - Rotational speed rpm
G98 - withdraw the tool to the starting point or feed per minute
G99 - withdraw the tool to a safe plane or feed per revolution
G101 - Spline interpolation

M-Codes control machine functions.

M00 - program stop
M01 - optional stop using stop button
M02 - end of program
M03 - spindle on CW
M04 - spindle on CCW
M05 - spindle off
M06 - tool change
M07 - flood with coolant
M08 - mist with coolant
M08 - turn on accessory (e.g. AC power outlet)
M09 - coolant off
M09 - turn off accessory
M10 - turn on accessory
M11 - turn off accessory or tool change
M17 - subroutine end
M20 - tailstock back
M20 - Chain to next program
M21 - tailstock forward
M22 - Write current position to data file
M25 - open chuck
M25 - set output #1 off
M26 - close chuck
M26 - set output #1 on

M30 - end of tape (rewind)
M35 - set output #2 off
M36 - set output #2 on
M38 - put stepper motors on low power standby
M47 - restart a program continuously, or a fixed number of times
M71 - puff blowing on
M72 - puff blowing off
M96 - compensate for rounded external curves
M97 - compensate for sharp external curves
M98 - subprogram call
M99 - return from subprogram, jump instruction
M101 - move x-axis home
M102 - move y-axis home
M103 - move z-axis home

Some other typical codes and keywords used in part programs are given below.

Annn - an orientation, or second x-axis spline control point
Bnnn - an orientation, or second y-axis spline control point
Cnnn - an orientation, or second z-axis spline control point, or chamfer
Fnnn - a feed value (in ipm or m/s, not ipr), or thread pitch
Innn - x-axis center for circular interpolation, or first x-axis spline control point
Jnnn - y-axis center for circular interpolation, or first y-axis spline control point
Knnn - z-axis center for circular interpolation, or first z-axis spline control point
Lnnn - arc angle, loop counter and program cycle counter
Nnnn - a sequence/line number
Onnn - subprogram block number
Pnnn - subprogram reference number
Rnnn - a clearance plane for tool movement, or arc radius, or taper value
Qnnn - peck depth for pecking cycle
Snnn - cutting speed (rpm), spindle speed
Tnnn - a tool number
Unnn - relative motion in x
Vnnn - relative motion in y
Wnnn - relative motion in z
Xnnn - an x-axis value
Ynnn - a y-axis value
Znnn - a z-axis value
; - starts a comment , or end of block

Appendix-2

Typical Specifications of a CNC System

1. Number of controlled axes	Two/Four/Eight, etc.
2. Interpolation	Linear/circular/parabolic or cubic/cylindrical
3. Resolution	Input resolution (feedback)
	Programming resolution
4. Feed rate	Feed/min
	Feed/revolution
5. Rapid traverse rate	Feed rate override
	Feed/min
6. Operating modes	Manual/Automatic/MDI(editing)/Input/Output/ Machine data set-up/Incremental, etc.
7. Type of feedback	Digital (rotary encoders with train of pulses)
	Analog (transducers, etc.)
	Both
8. Part program handling	Number of characters which can be stored
	Part program input devices
	Output devices
	Editing of part program
9. Part programming	Through MDI
	Graphic simulation
	Blue print programming
	Background editing
	Menu driven programming
	Conversational programming
10. Compensations	Backlash
	Lead screw pitch error
	Temperature
	Cutter radius compensation
	Tool length compensation
11. Programmable logic controller	Built-in (integrated)/External
	Type of communication with NC
	Number of inputs, outputs, timers, counters and flags
	User memory
	Program organization
	Programming Languages
12. Thread cutting/Tapping	Types of threads that can be cut
13. Spindle control	Analog/Digital control
	Spindle orientation
	Spindle speed overrides
	RPM/min; constant surface speed
14. Other features	Inch/metric switchover
	Polar coordinate inputs
	Mirror imaging
	Scaling

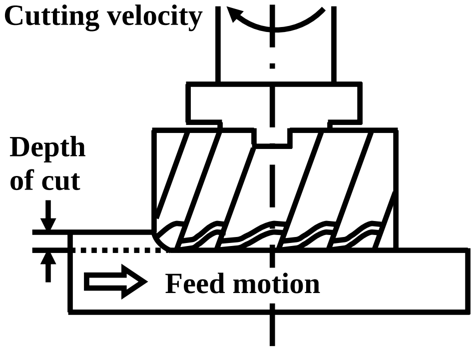
: Coordinate rotation system
: Custom macros
: Built-in fixed cycles
: Background communication
: Safe zone programming
: Built-in diagnostics, safety function, etc.
: Number of universal interfaces
: Number of active serial interfaces
: Direct numerical control interface
: Network interface capability

Answers, Remarks and Hints to Points to Ponder

Point to Ponder: 1

A. *Consider the metal cutting configurations shown. Identify the motion directions corresponding to speed, feed and depth of cut*

Ans: The directions are shown below.



B. *"Metal cutting is essentially a shearing phenomenon", justify the statement with respect to the entries in Table 23.1.*

Ans: All entries in the table would show that the motion of the tool and the motion of the work piece create a shearing.

Point to Ponder: 2

A. *Comment on the sensing requirements for PTP and Contouring axes*

Answer: PTP systems require only feedback of position. Contouring axes require feedback of position and velocity both. In PTP systems velocity is varied in open loop to achieve rapid traversal of the table with preprogrammed acceleration, deceleration patterns. However, often the hardware capabilities of the axis conforms to contouring.

B. *Do you think the overall machining time can be optimized for PTP and Contouring systems? Are there any constraints to that?*

Ans: For PTP systems the rapid traverse feature saves overall machining time. This means that the table movements from point to point are carried out as fast as possible, with acceleration and deceleration features. Deceleration is needed, so that the final position is reached accurately and quickly. The extent to which time can be minimized depends on the motoring and braking torque levels of the table drive.

For contouring systems the instantaneous ratios of velocities along the motion axes are important to maintain the contours. Therefore over all time has to be minimized maintaining them. Note that whenever the contours take sharp turns, the velocity ratios also do so. Therefore it is very difficult to change the ratios sharply, if the velocities themselves are high. Thus contour cutting around sharp edges have to be done at low speeds. This constrains the minimization of overall machining time to maintain dimensional accuracy.

Point to Ponder: 3

A. *Can you think of one advantage and one disadvantage of the incremental coordinate system compared to the absolute one?*

Ans: An advantage of the absolute system is that a change in any one a position coordinate in an instruction does not affect the rest of the part program. In the case of the incremental system, any such change would require corresponding changes to be made in all subsequent instructions in the part program.
On the other hand, for incremental systems programming for parts with mirror image symmetry is easier, since it involves only changes in signs for the position commands with respect to the symmetrical points. Similarly, verification of program dimensions with drawing is easier with incremental system.

B. *Is there any connection between the choice of coordinate system and the position sensor used for the machine tool?*

Ans: Among the position sensors used, the most common ones are the linear optical scales or inductosyns which directly measure movement of the table slide with respect to the fixed parts of the machine. The other kinds are the rotary sensors such as the shaft angle encoders and resolvers. The translational motion of the slide is deduced from these using the ball screw pitch constant. Among these, the linear optical scale and the rotary encoder can either give absolute or incremental displacement, depending on the encoding on the grating. However, in fact incremental output is more common, since better resolution can be achieved. Thus, in an absolute coordinate system, this output must be integrated to generate absolute positions by electronic means. The resolver or the inductosyn, on the other hand naturally generate absolute positions with respect to a fixed origin. This means that these signals have to be differenced from the current origin of the incremental system, to generate incremental coordinate positions.

C. *How would you decide on the BLU for systems with continuous position sensors such as inductosyns and resolvers?*

Ans:

D. *Is the BLU affected by the motor or the drive system also?*

Point to Ponder: 4

A. *Explain the part program segment given below.*

```
N0010G90;
N0011G01X1Y2;
N0012G01X2Y2;
N0013G91;
N0014G01X1;
N0015G92X2Y2;
N0016G01X1Y1;
```

Ans:

```
N0010G90; PUT IN ABSOLUTE MODE
N0011G01X1Y2; MOVE TO (1, 2)
N0012G01X2Y2; MOVE TO (2, 2)
N0013G91; PUT IN INCREMENTAL MODE
N0014G01X1; MOVE TO (3, 2)
N0015G92X2Y2; SET NEW ORIGIN
N0016G01X1Y1; MOVE TO (3, 3) ABSOLUTE
N0017G92X0Y0Z0; RESET THE ZERO
```

B. Draw the trajectory of table motion that this program seeks to create.

Ans: