NC, CNC & Robotics

By S K Mondal

What is NC/CNC?

- NC is an acronym for Numerical Control and CNC is an acronym for Computer Numerical Control.

What is the difference between NC and CNC ?

- The difference between NC and CNC is one of age and capability.
- The earliest NC machines performed limited functions and movements controlled by punched tape or punch cards.
- As the technology evolved, the machines were equiped with increasingly powerful microprocessors (computers) with the addition of these computers, NC machines become CNC machines.
- CNC machines have far more capability than their predecessor.

contd.....

What is the difference between NC and CNC ?

- Some of the enhancements that came along with CNC include: Canned Cycles, Sub Programming, Cutter Compensation, Work coordinates, Coordinate system rotation, automatic corner rounding, chamfering, and B-spline interpolation.

Where did CNC get started?

- 1940 Jhon Parson developed first machine able to drill holes at specific coordinates programmed on punch cards.
- 1951 MIT developed servo-mechanism
- 1952 MIT developed first NC machines for milling.
- 1970 First CNC machines came into picture

Now-a-day's modified 1970's machines are used.

IAS - 1996

Assertion (A): The temperature control of an electric iron is an example of servomechanism.

Reason (R): It is an automatic control system.

(a) Both A and R are individually true and R is the correct explanation of A

(b) Both A and R are individually true but R is **not** the correct explanation of A

(c) A is true but R is false

(d) A is false but R is true

Do all machines speak the same CNC language

- No, while there is fairly standard set of G and M codes, there is some variation in their application. For example a G0 or G00 command is universally regarded as the command for rapid travel. Some older machines do not have a G00 command. On these machines, rapid travel is commanded by using the F (feed) word address.

What is a “Conversational Control”

- CNC machine tool builders offer an option what is known as the conversational control. This control lets the operator/programmer use simple descriptive language to program the part. The control then displayed a graphical representation of the instructions so the operator/programmer can verify the tool path.

Are CNC machines faster than conventional machines?

- Yes, No, Sometimes. When it comes to making a single, simple part it is hard to beat a conventional mill or lathe. CNC machines move faster in rapid travel than conventional machines.

Are CNC machines more accurate than conventional machines?

- Yes, they can be. But like anything else it depends on who is running the machine, how well the machines has been maintained, quality of setup and so on.

GATE - 1994

CNC machines are more accurate than conventional machines because they have a high resolution encoder and digital read-outs for positioning.

True or false?

NC/CNC Machines-Advantages

- High Repeatability and Precision e.g. Aircraft parts
- Volume of production is very high
- Complex contours/surfaces need to be machined. E.g. Turbines
- Flexibility in job change, automatic tool settings, less scrap
- More safe, higher productivity, better quality
- Less paper work, faster prototype production, reduction in lead times

NC/CNC Machines-Disadvantages

- Costly setup, skilled operators
- Computers, programming knowledge required
- Maintenance is difficult

IES - 1999

Consider the following statements regarding numerically controlled machine tools:

1. They reduce non-productive time
2. They reduce fixturing
3. They reduce maintenance cost

Which of these statements are correct?

(a) 1, 2 and 3 (b) 1 and 2

(c) 2 and 3 (d) 1 and 3

IES - 1995

Consider the following characteristics of production jobs:

1. Processing of parts frequently in small lots
2. Need to accommodate design changes of products.
3. Low rate of metal removal
4. Need for holding close tolerances

The characteristics which favour the choice of numerically controlled machines would include

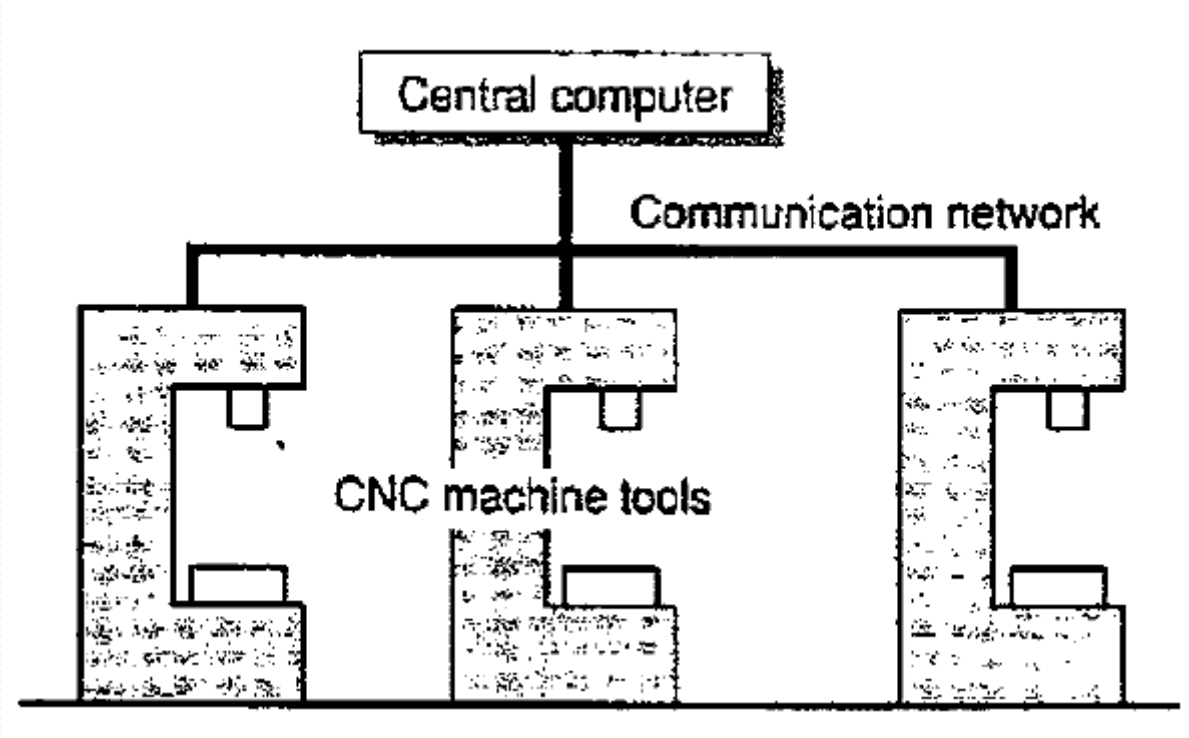
(a) 1, 2 and 3 (b) 2, 3 and 4

(c) 1, 3 and 4 (d) 1, 2 and 4

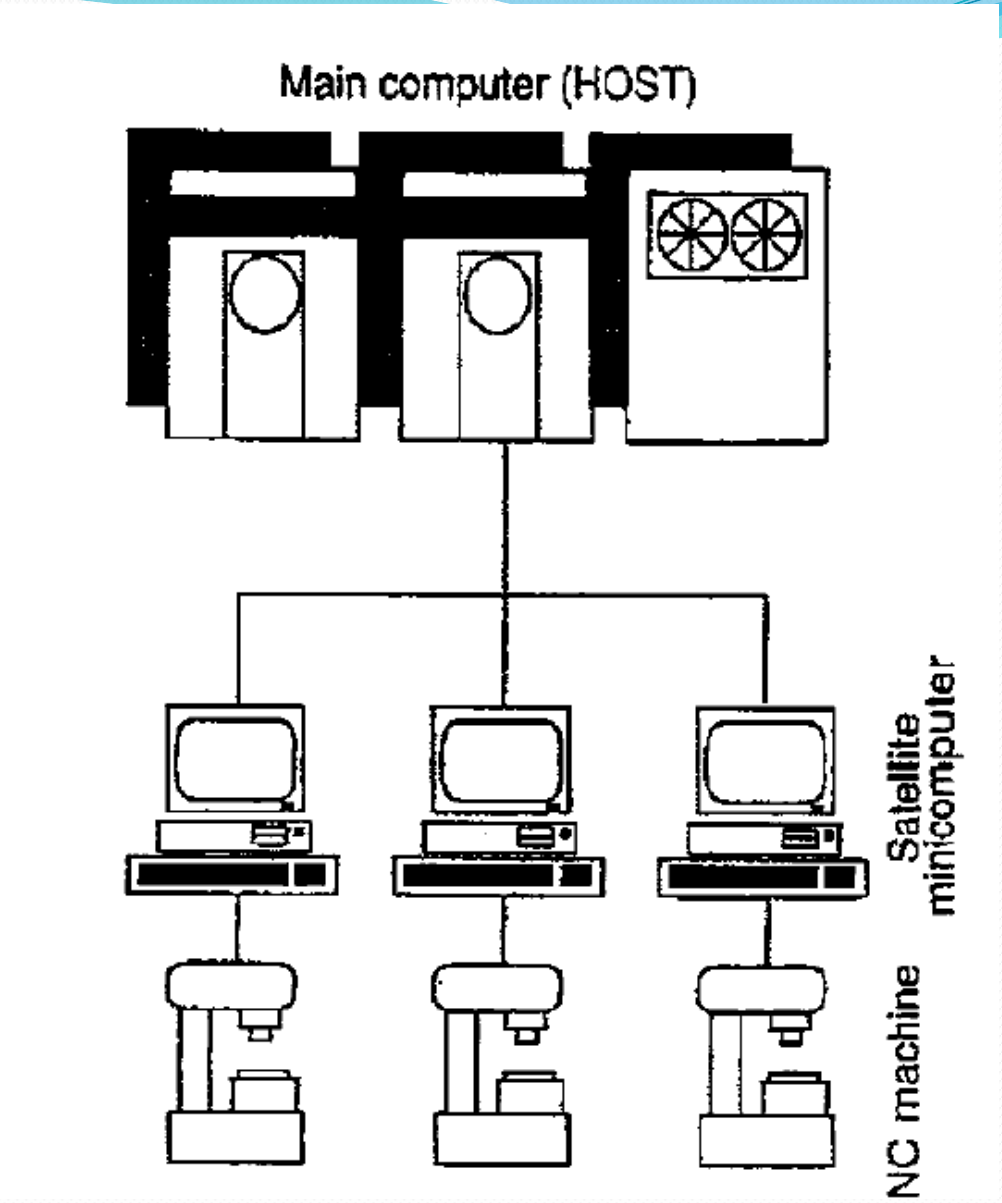
NC/CNC/DNC

- **Direct Numerical Control** is a system that uses a central computer to control several machines at the same time
- **Distributed Numerical Control (DNC):** the central computer downloads complete programs to the CNC machines, which can be workstations or PCs, and can get the information for the machine operations.
- The speed of the system is increased, large files can be handled and the number of machine tools used is expanded.

Direct numerical control



DNC



JWM 2010

Consider the following advantages of DNC systems :

1. Time-sharing

2. Greater computational capability

3. Remote computer location

Which of the above is/are correct ?

(a) 1 and 2 only **(b) 2 and 3 only**

(c) 2 only **(d) 1, 2 and 3**

IES – 2002 S-1

Match List I with List II and select the correct answer:

List I (NC machine tool systems)		**List II (Features)**
A. NC system	1.	It has an integrated automatic tool changing unit and a component indexing device
B. CNC system	2.	A number of machine tools are controlled by a computer. No tape reader, the part program is transmitted directly to the machine tool from the computer memory

IES – 2002 Contd….. From S-1

C.	DNC system	3.	The controller consists of soft-wired computer and hard- wired logic Graphic display of tool path is also possible
D.	Machining centre	4.	The instructions on tape is prepared in binary decimal form and operated by a series of coded instructions

Codes:	**A**	**B**	**C**	**D**		**A**	**B**	**C**	**D**
(a)	4	2	3	1	(b)	1	3	2	4
(c)	4	3	2	1	(d)	1	2	3	4

Stepper Motor

- The stepper motor is special type of synchronous motor which is designed to rotate through a specific angle (Called step) for each electrical pulse received from the control unit.

Basic CNC Principles

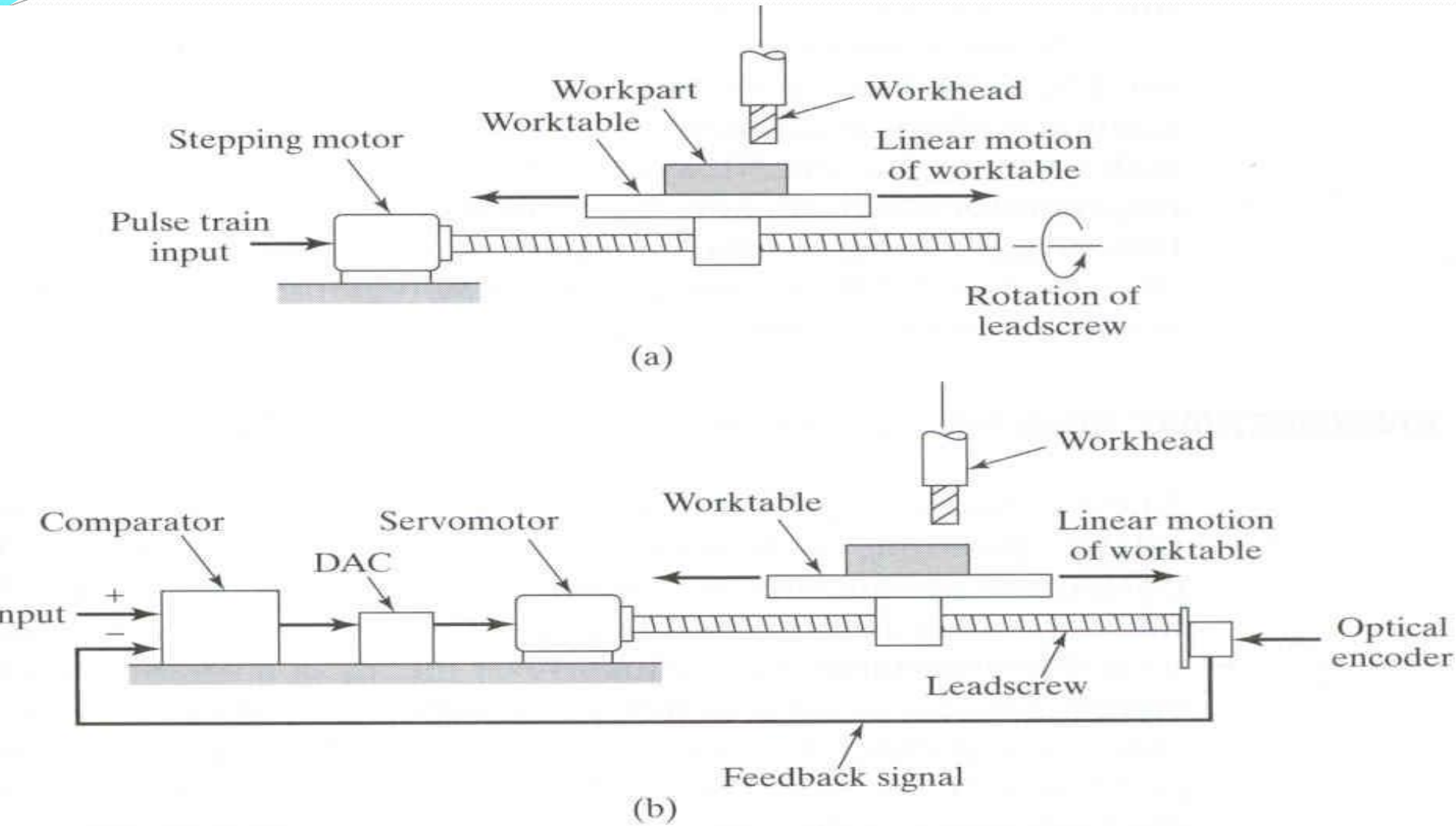


Figure 6.27 Two types of motion control in NC: (a) open loop and (b) closed loop.

Basic Length Unit (BLU)

- In NC machine, the displacement length per one pulse output from machine is defined as a Basic Length Unit (BLU).
- In the CNC computer each bit (binary digit) represents 1 BLU.

Bit = BLU

- Example: If one pulse makes a servo motor rotate by one degree and the servo motor moves the table by 0.0001 mm, one BLU will be 0.0001 mm.
- The lead of a ball screw is related to the displacement unit of the machine tool table.

Example

- **A DC servomotor is coupled directly to a leadscrew which drives the table of an NC machine tool. A digital encoder, which emits 500 pulses per revolution, is mounted on the other end of the leadscrew. If the leadscrew pitch is 5 mm and the motor rotates at 600 rpm, calculate**

  (a) The linear velocity of the table

  (b) The BLU of the NC system

  (c) The frequency of pulses transmitted by the encoder.

IES 2011 Conventional

- The table of a CNC machine is driven by a Lead screw which is rotated by a DC servomotor. A digital encoder which emits 1000 pulses per second is mounted on the lead screw as a feedback device. If the lead screw pitch is 6 mm and motor rotates at 500 rpm, find

  1. Basic length Units of the system
  2. Linear velocity of the table.
  3. Frequency of pulses generated by the feedback device.

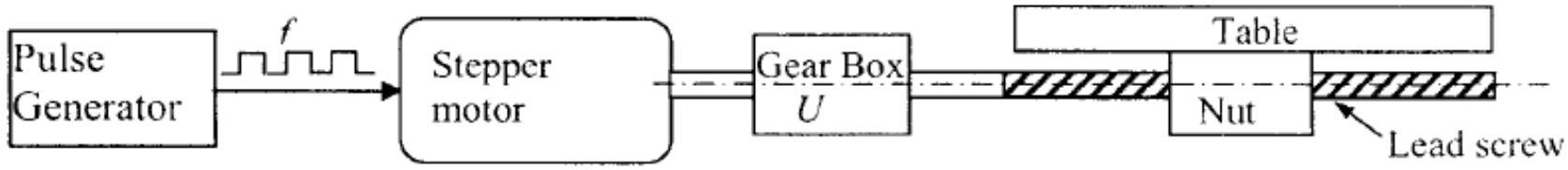
[5 Marks]

Statement for Linked Answers questions: S-1

In the feed drive of a Point-to-Point open loop CNC drive, a stepper motor rotating at 200 steps/rev drives a table through a gear box and lead screw-nut mechanism (pitch = 4 mm, number of starts = 1).

The gear ratio = $\frac{Output\ rotational\ speed}{Input\ rotational\ speed}$ is given by U = $\frac{1}{4}$

The stepper motor (driven by voltage pulses from a pulse generator) executes 1 step/pulse of the pulse generator. The frequency of the pulse train from the pulse generator is f = 10,000 pulses per minute.



GATE – 2008 Q-1 (Statement in S-1)

The Basic Length Unit (BLU), i.e., the table movement corresponding to 1 pulse of the pulse generator, is

(a) 0.5 microns (b) 5 microns

(c) 50 microns (d) 500 microns

GATE – 2008 Q-2 (Statement in S-1)

A customer insists on a modification to change the BLU of the CNC drive to 10 microns without changing the table speed. The modification can be accomplished by

(A) Changing U to $\frac{1}{2}$ and reducing f to $\frac{f}{2}$

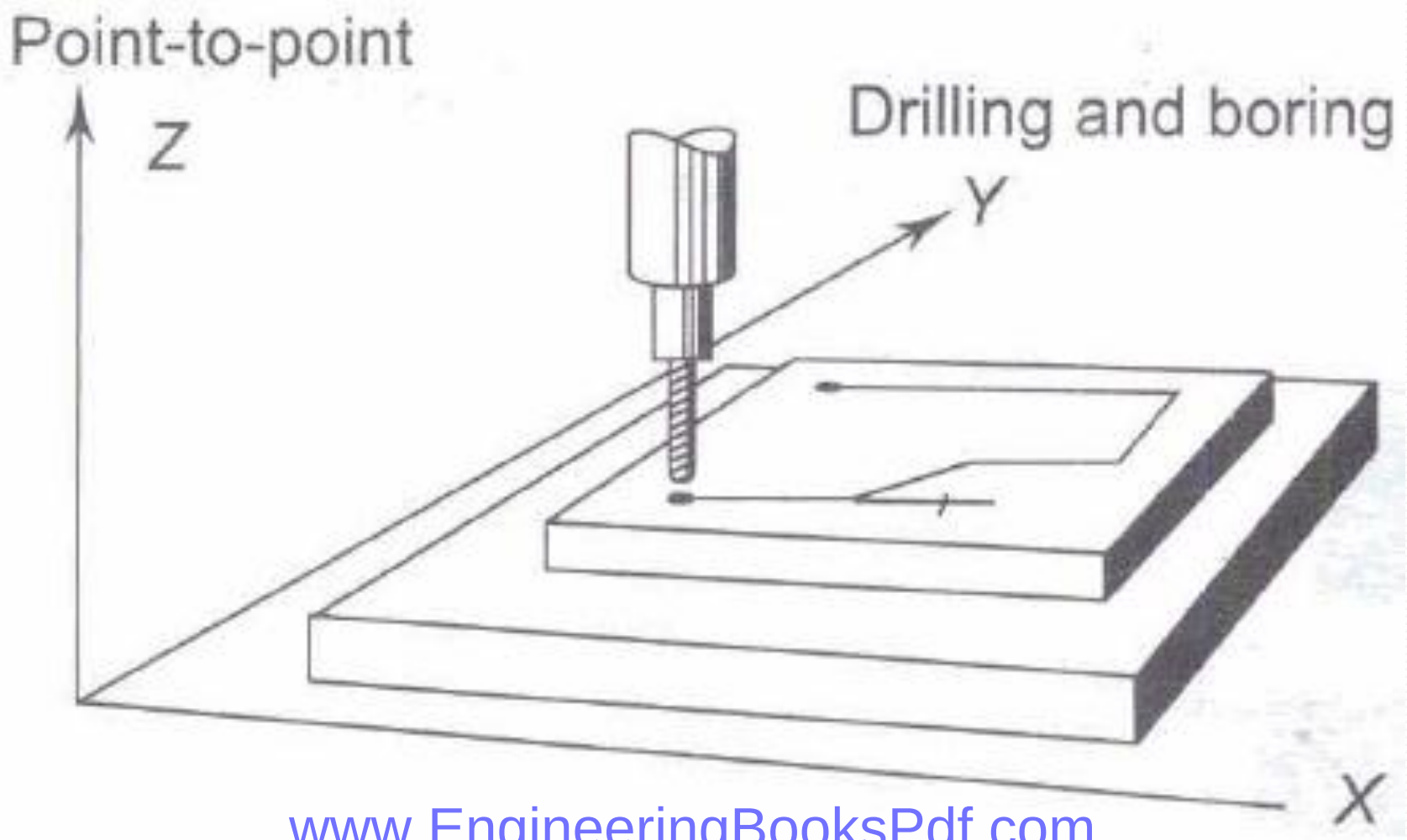
(B) Changing U to $\frac{1}{8}$ and increasing f to 2f

(C) Changing U to $\frac{1}{2}$ and keeping f unchanged

(D) Keeping U unchanged and increasing f to 2f

Control Systems possible in CNC Machine

- Point to point mode:



GATE - 1992

In a point-to-point type of NC system

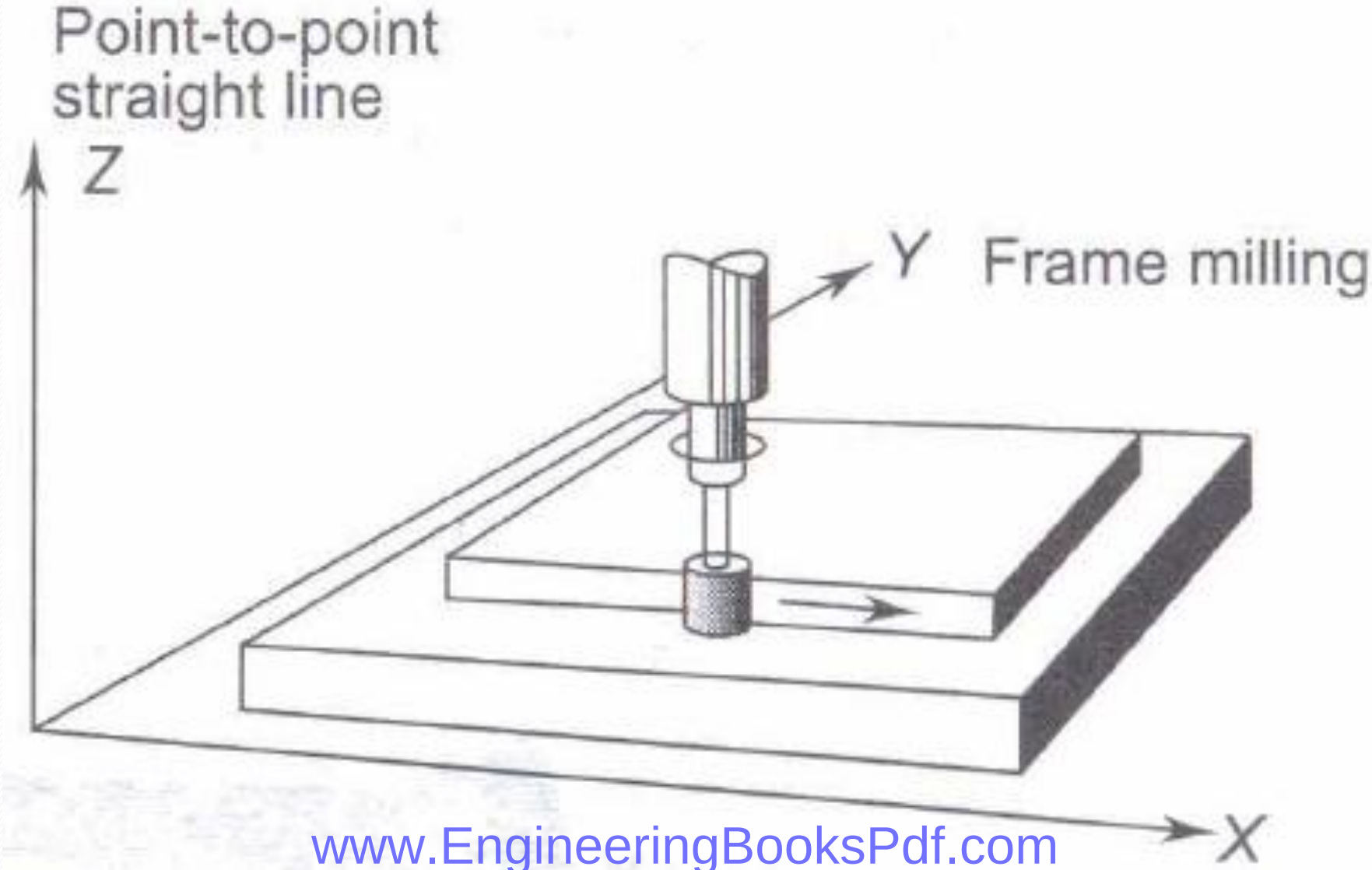
(a) Control of position and velocity of the tool is essential

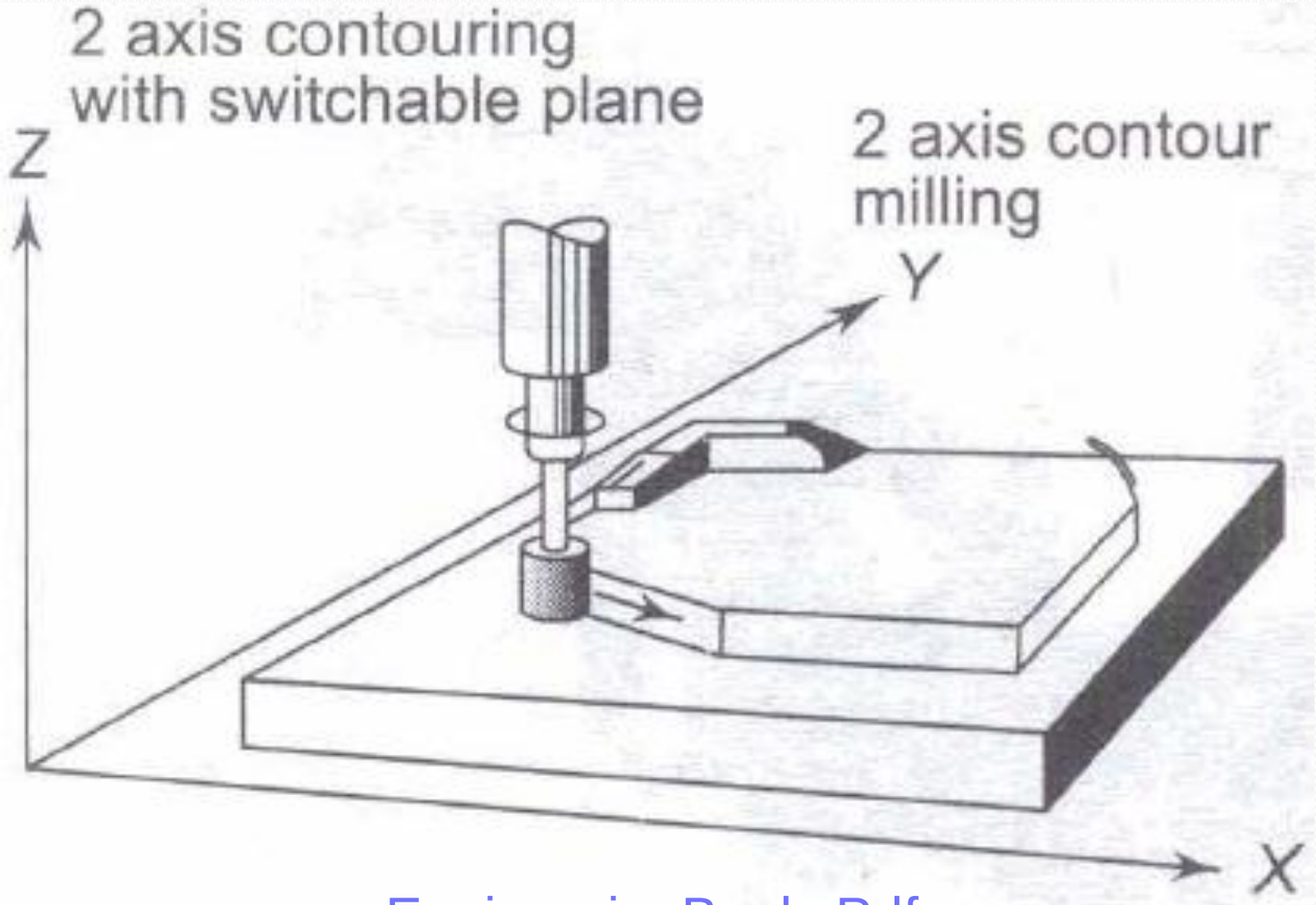
(b) Control of only position of the tool is sufficient

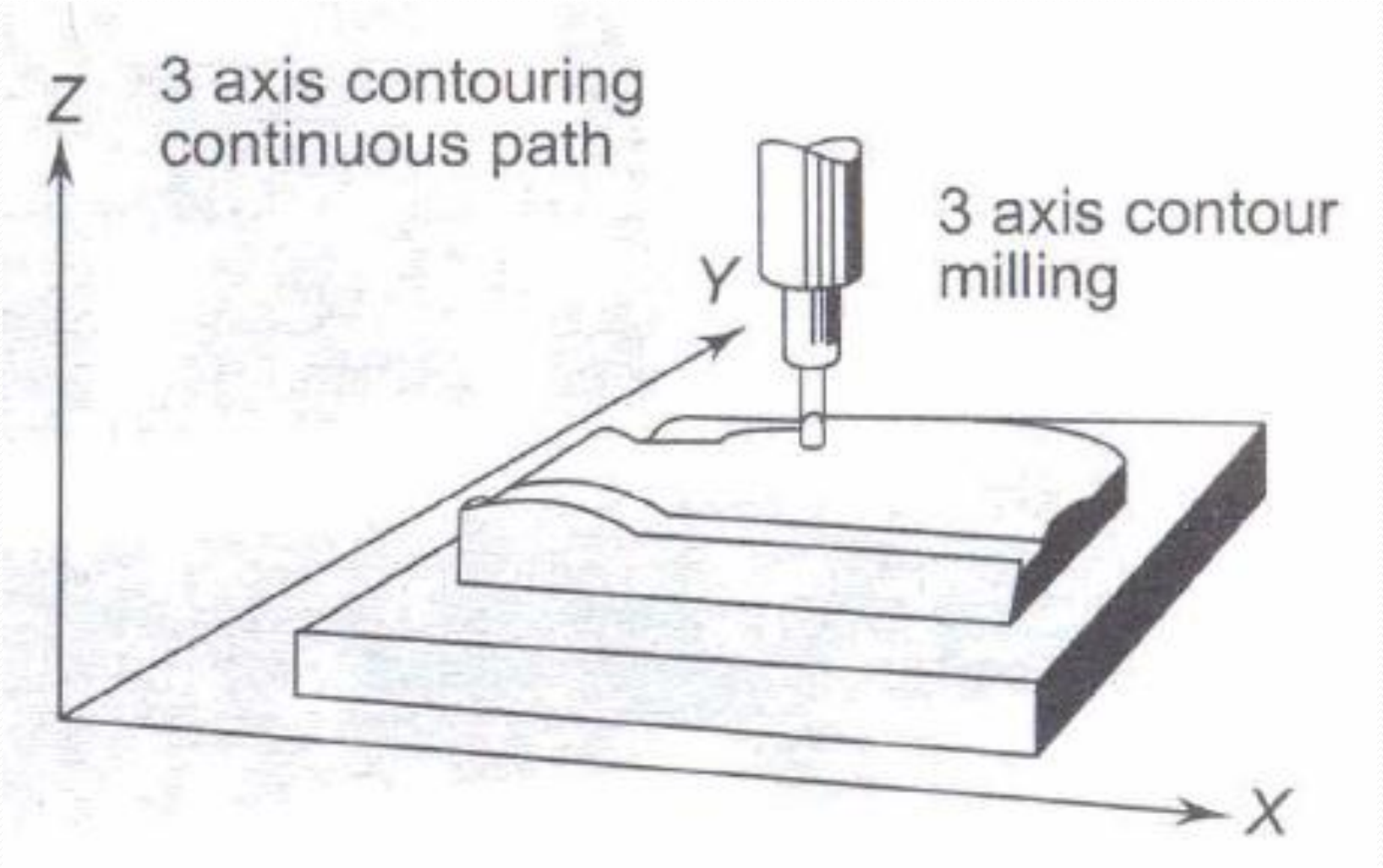
(c) Control of only velocity of the tool is sufficient

(d) Neither position nor velocity need be controlled

Point-to-point straight line mode







GATE-2005

Which among the NC operations given below are continuous path operations?

Arc Welding (AW)

Drilling (D)

Laser Cutting of Sheet Metal (LC)

Milling (M)

Punching is Sheet Metal (P)

Spot Welding (SW)

(a) AW, LC and M

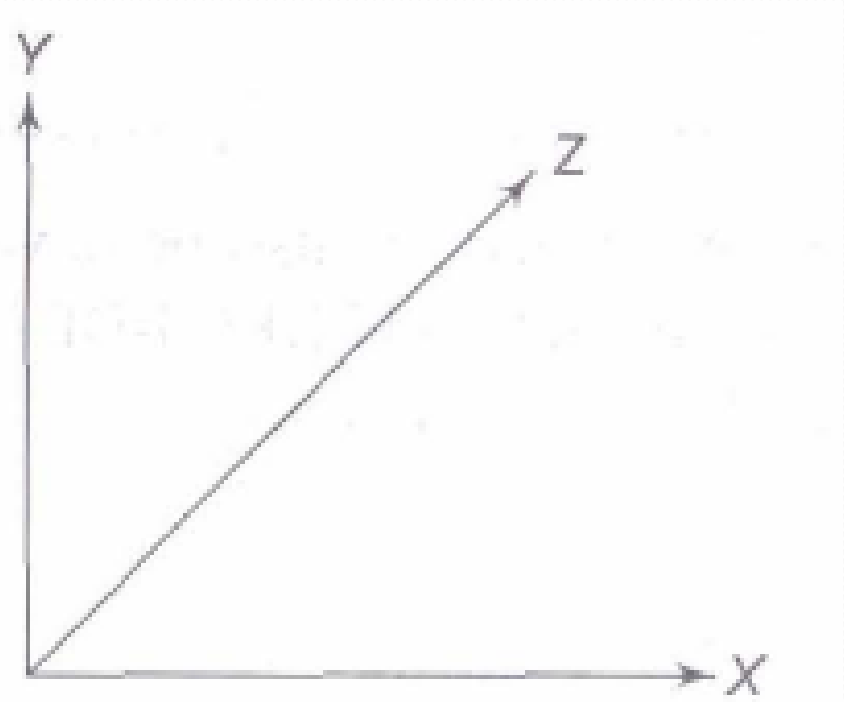
(b) AW, D, LC and M

(c) D, LC, P and SW

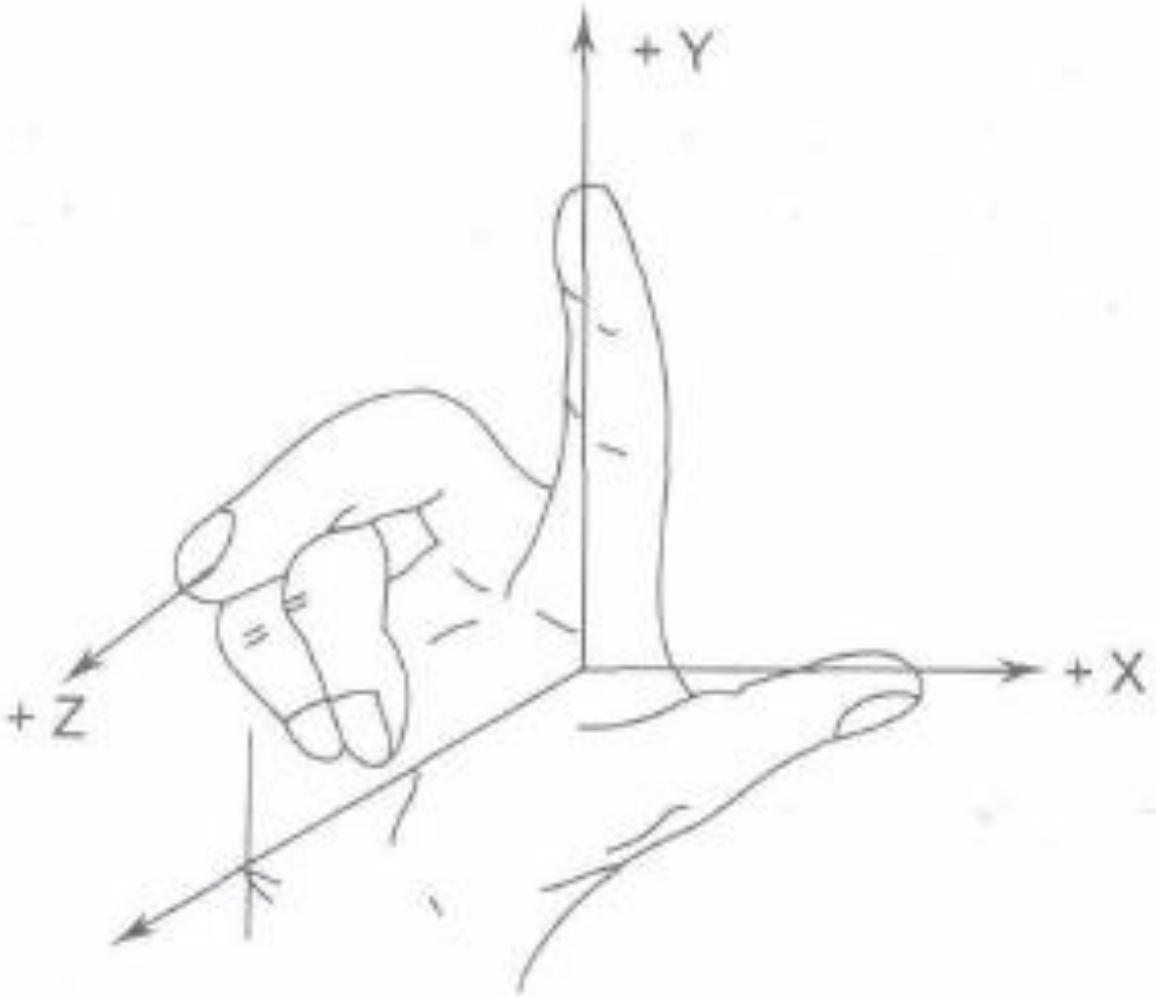
(d) D, LC, and SW

Co-ordinate system

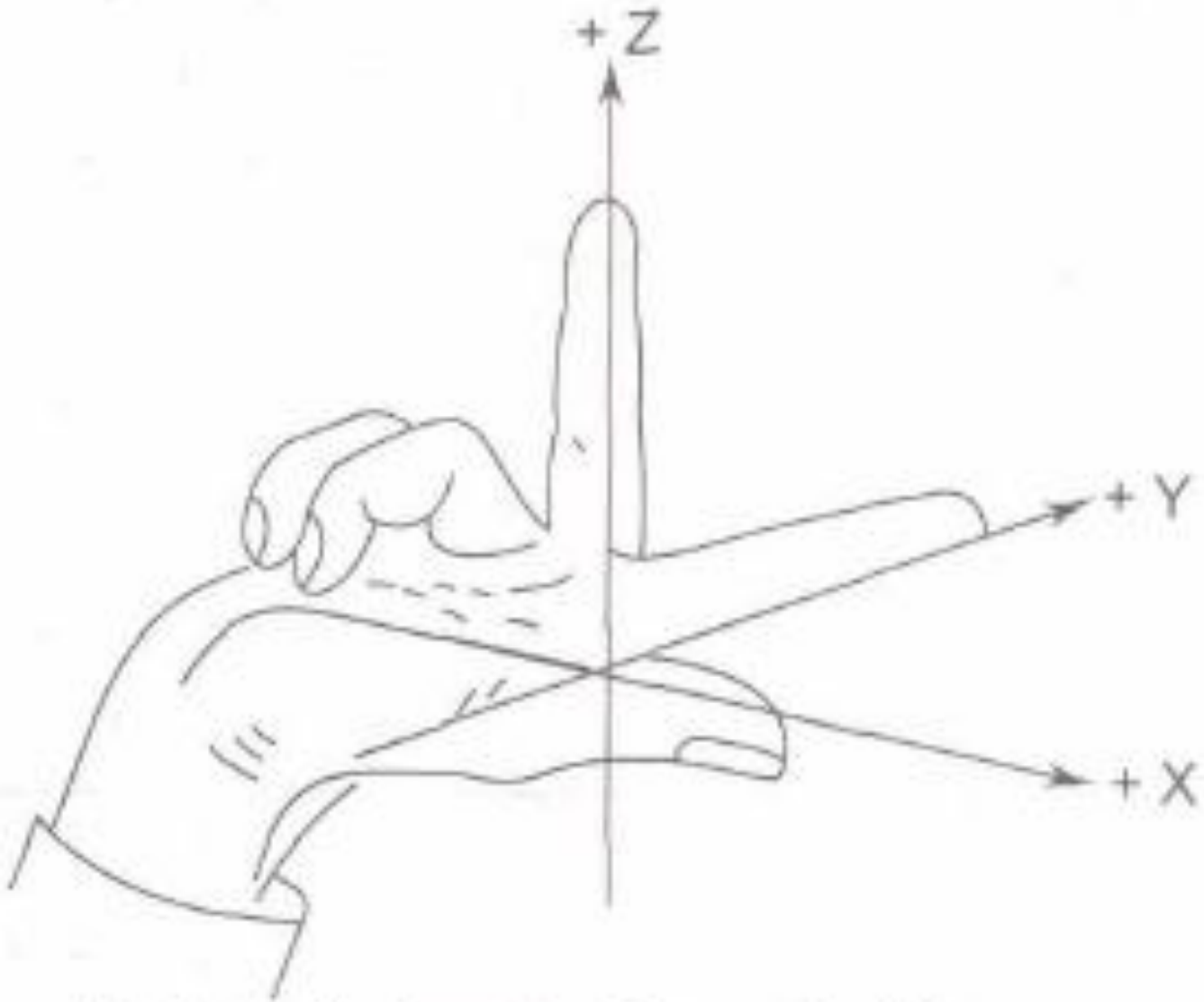
- All the machine tool use Cartesian Co-ordinate system.
- The first axis to be identified is the Z – axis, This is followed by X and Y axes respectively.



Right-hand coordinate systems



(a) Axis designation for horizontal Z



(b) Axis designation for vertical Z

IES - 2000

Assertion (A): The axis of an NC drilling machine spindle is denoted as z-axis.

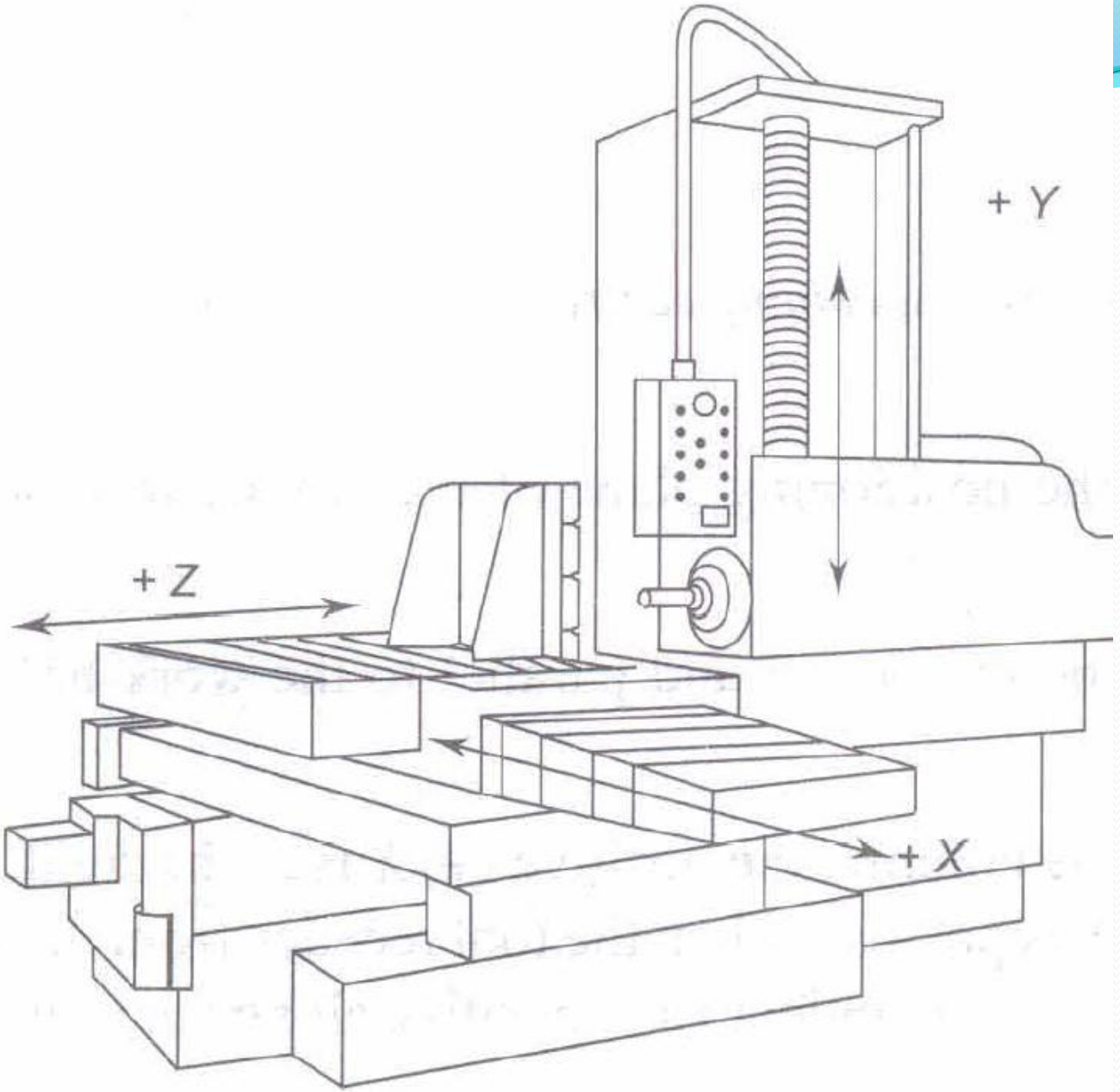
Reason (R): In NC machine tool, the axis perpendicular to both x- and y-axis is designated as z-axis

(a) Both A and R are individually true and R is the correct explanation of A

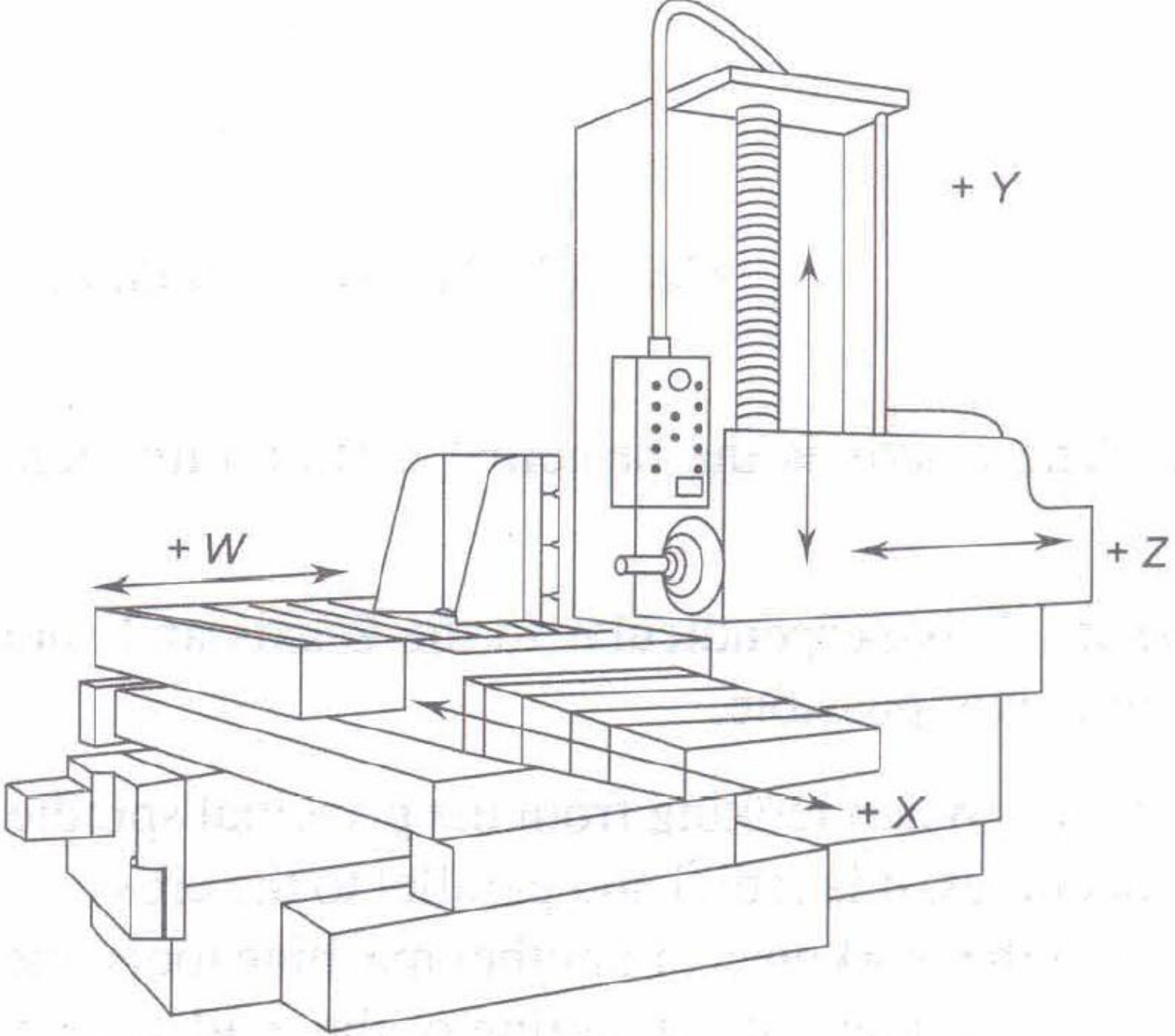
(b) Both A and R are individually true but R is **not** the correct explanation of A

(c) A is true but R is false

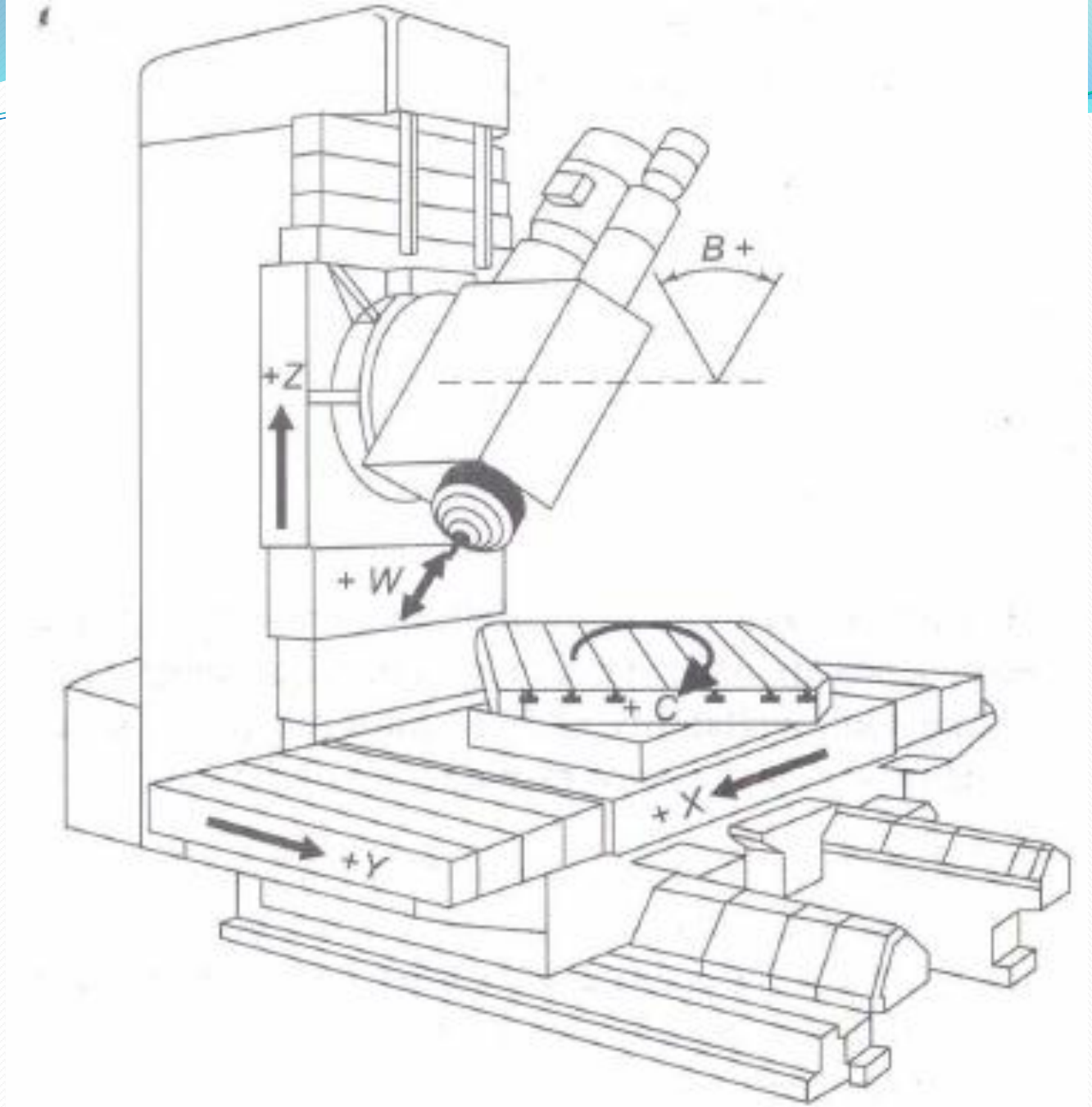
(d) A is false but R is true



(a) Three-axis boring mill



(b) Five axis machine



5 axes CNC vertical axis machining centre configuration

IES - 1996

Assertion (A): Numerically controlled machines having more than three axes do not exist.

Reason (R): There are only three Cartesian coordinates namely x-y-z.

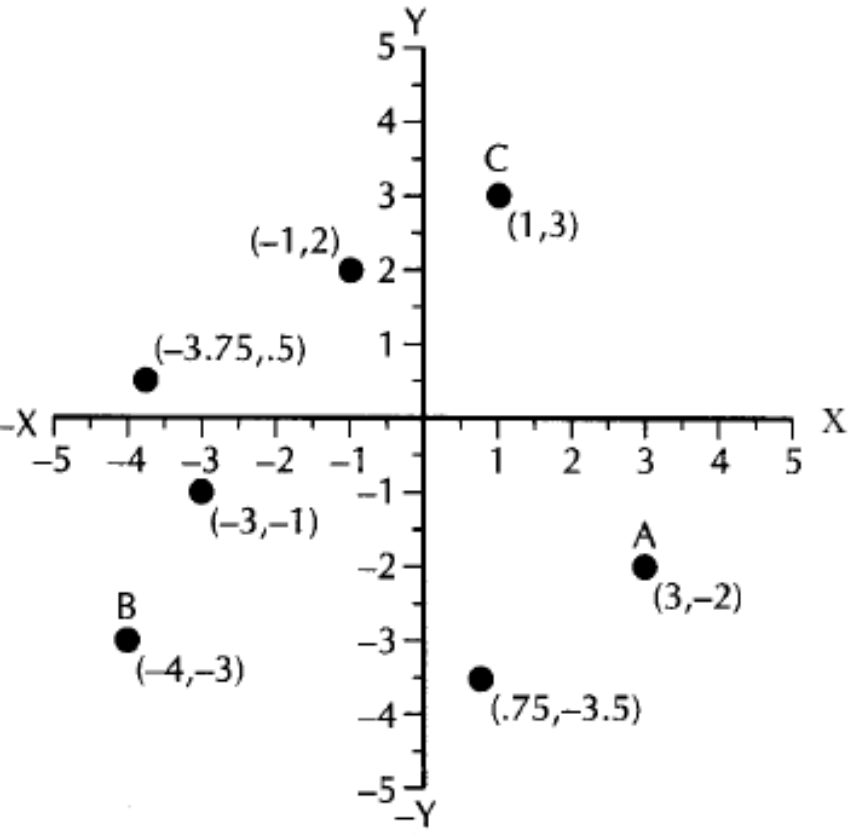
(a) Both A and R are individually true and R is the correct explanation of A

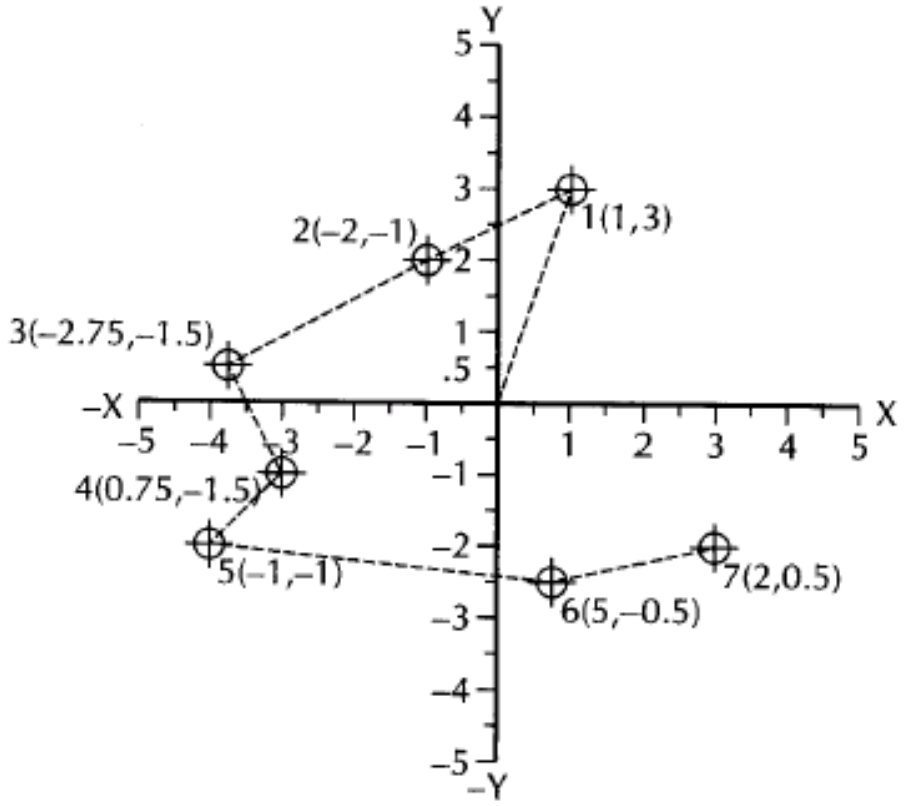
(b) Both A and R are individually true but R is **not** the correct explanation of A

(c) A is true but R is false

(d) A is false but R is true

Absolute and Incremental Coordinate System





Absolute Coordinate System

Incremental Coordinate System

GATE -2012 Same Q in GATE-2012 (PI)

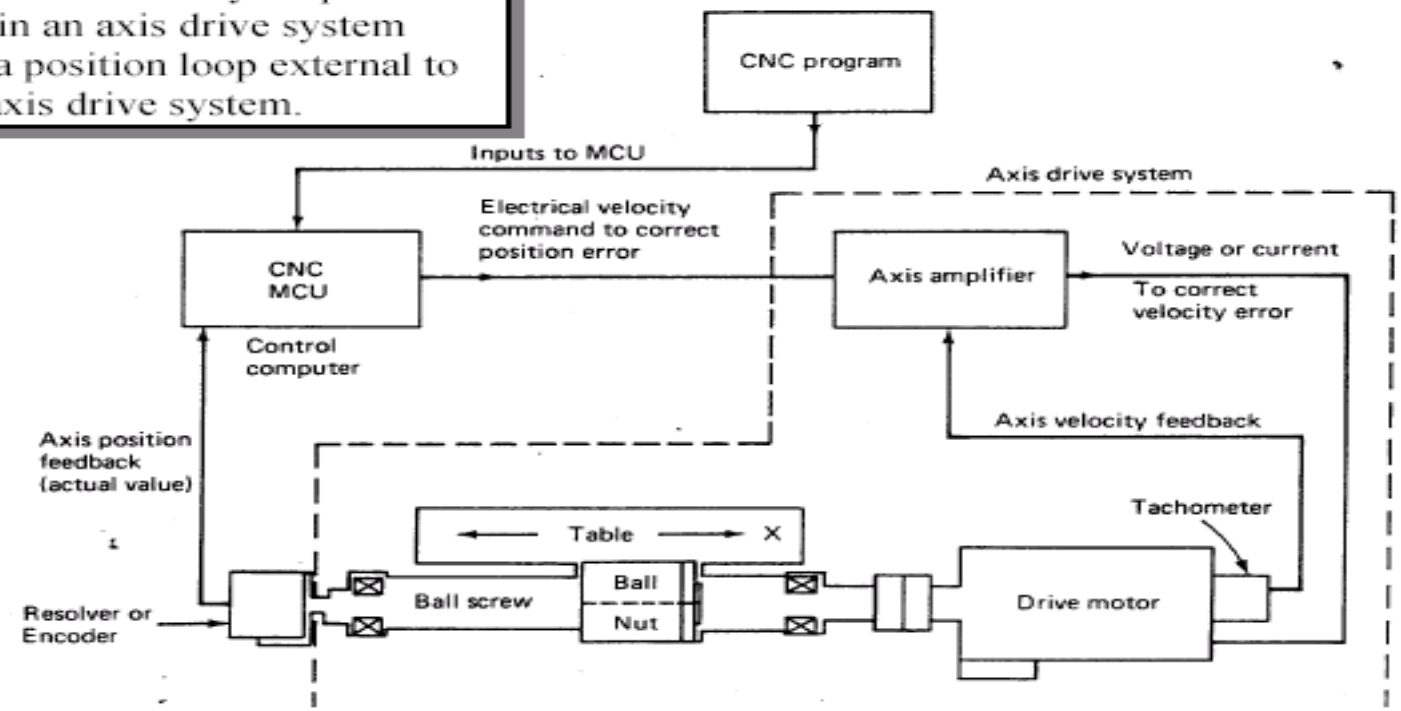
A CNC vertical milling machine has to cut a straight slot of 10 mm width and 2 mm depth by a cutter of 10 mm diameter between points (0, 0) and (100, 100) on the XY plane (dimensions in mm). The feed rate used for milling is 50 mm/min. Milling time for the slot (in seconds) is

(a) 120 (b) 170 (c) 180 (d) 240

The following are the steps to be followed while developing the CNC part programs.

- Process planning
- Axes selection
- Tool selection
- Cutting process parameters planning
- Job and tool setup planning
- Machining path planning
- Part program writing
- Part program proving

includes a velocity loop within an axis drive system and a position loop external to the axis drive system.



- For a CNC machine control unit (MCU) decides cutting speed, feed, depth of cut, tool selection , coolant on off and tool paths. The MCU issues commands in form of numeric data to motors that position slides and tool accordingly.

Part Programming

- FANUC CONTROLL
- SIEMENS CONTROLL

CNC programming

Important things to know:

- Coordinate System
- Units, incremental or absolute positioning
- Coordinates: X,Y,Z, RX,RY,RZ
- Feed rate and spindle speed
- Coolant Control: On/Off, Flood, Mist
- Tool Control: Tool and tool parameters

Programming Key Letters

- O - Program number (Used for program identification)
- N - Sequence number (Used for line identification)
- G - Preparatory function
- X - X axis designation
- Y - Y axis designation
- Z - Z axis designation
- R - Radius designation
- F – Feed rate designation
- S - Spindle speed designation
- H - Tool length offset designation
- D - Tool radius offset designation
- T - Tool Designation
- M - Miscellaneous function

Table of Important G codes

Code	Meaning	Format
G00	Rapid Transverse	N__G00 X___ Y___ Z___
G01	Linear Interpolation	N__G01 X___ Y___ Z___ F___
G02	Circular Interpolation, CW	N__G02 X__ Y__ Z___ R___ F___ N__G02 X___ Y__Z__I ___J __K __ F __
G03	Circular Interpolation, CCW	N__G03 X___ Y___ Z__R__F___ N__G03 X__ Y__Z__I __J __K __ F __
G04	Dwell	N__G04P___
G17	XY Plane	
G18	XZ Plane	
G19	YZ Plane	

Table of Important G codes

Code	Meaning	Format
G20/G70	Inch Unit	
G21/G71	Metric Unit	
G28	Automatic Return to Reference Point	
G40	Cutter compensation cancel	
G41	Cutter compensation left	N__G41D__
G42	Cutter compensation right	N__G42D__
G43	Tool length compensation (plus)	N__G43H__

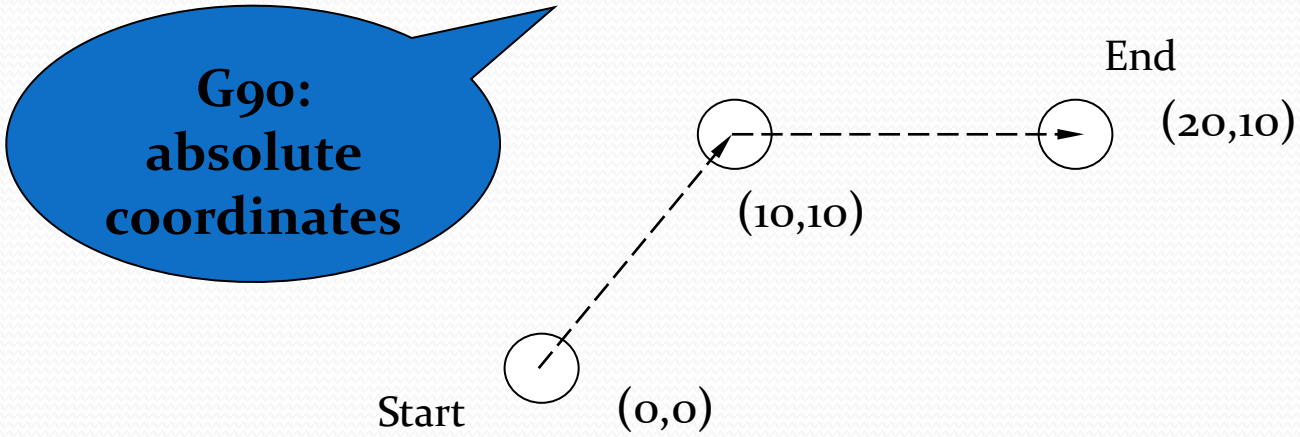
Table of Important G codes

Code	Meaning	Format
G44	Tool length compensation (minus)	N__G44H__
G49	Tool length compensation cancel	
G80	Cancel canned cycles	
G81	Drilling cycle	N__G81 Z__R__F__
G90	Absolute positioning	
G91	Incremental positioning	
G92	Absolute preset, change the datum position	N__G92X__Y__Z__

Rapid traverse: G00

- G00:
  - to make the machine move at maximum speed.
  - It is used for positioning motion.

G90 G00 X20.0 Y10.0

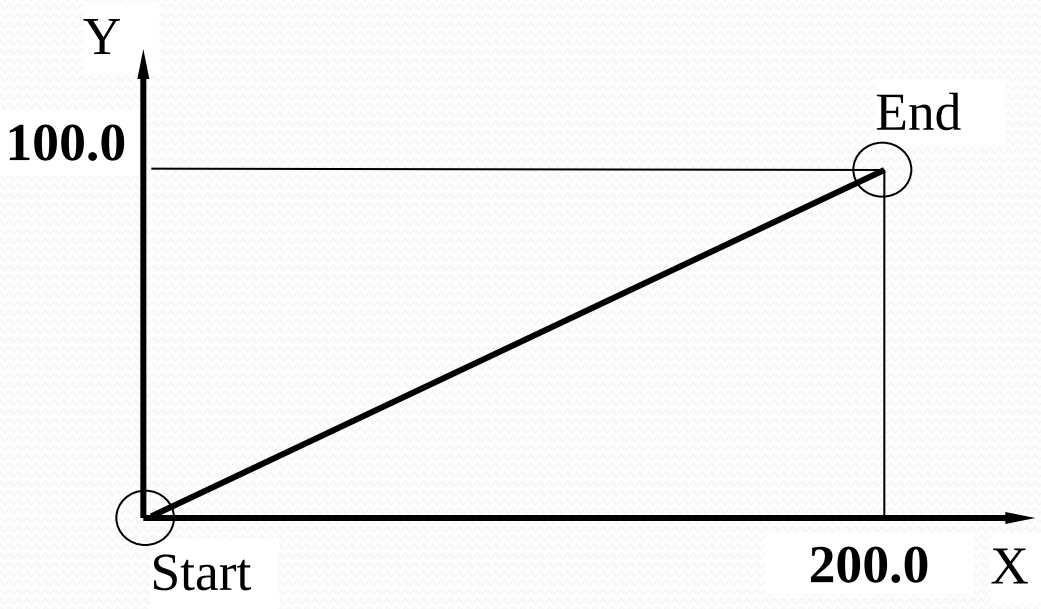


Linear interpolation: G01

- G01:
  - linear interpolation at feed speed.

G91 G01 X200.0 Y100.0 F200.0

G91: incremental coordinates



Circular interpolation: G02, G03

- G02, G03:
  - For circular interpolation, the tool destination and the circle center are programmed in one block
  - G02 is clockwise interpolation, G03 is counterclockwise interpolation

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_\_\,Y\_\_\begin{Bmatrix}R\\I\_\_\quad J\_\_\end{Bmatrix}F\_\_;$$

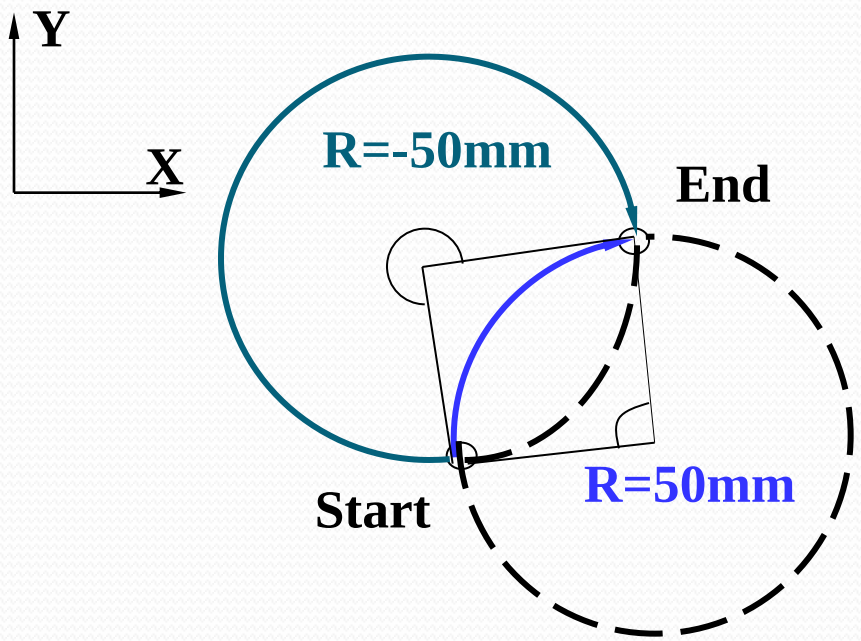
$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_\_\,Z\_\_\begin{Bmatrix}R\\I\_\_\quad K\_\_\end{Bmatrix}F\_\_;$$

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_\_\,Z\_\_\begin{Bmatrix}R\\J\_\_\quad K\_\_\end{Bmatrix}F\_\_;$$

End point

Circle center, radius

Circular interpolation: G02, G03



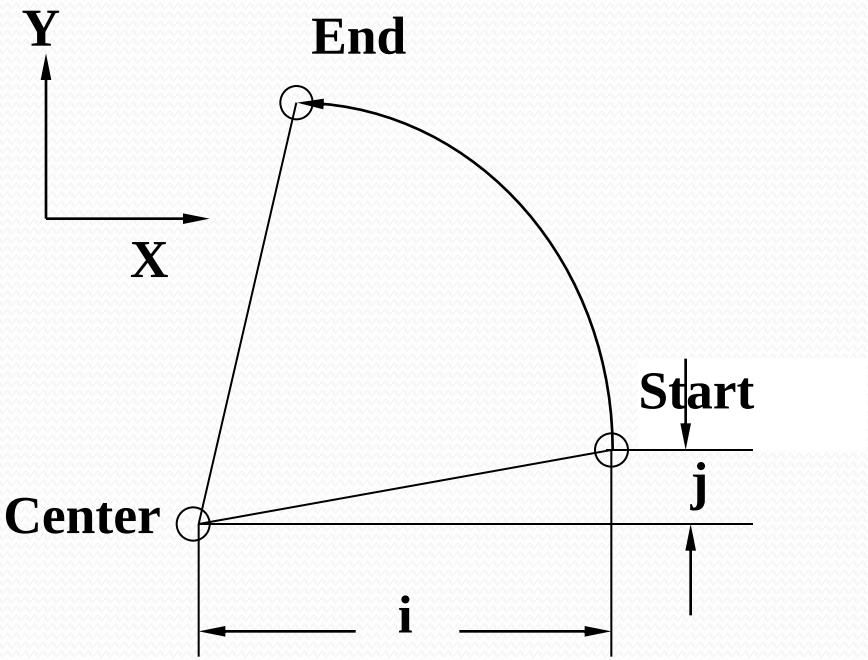
Specify R with sign before it:

≤180° +R

>180° -R

G91 G02 X60.0 Y20.0 R50.0 F300.0
G91 G02 X60.0 Y20.0 R-50.0 F300.0

Circular interpolation: G02, G03



- Specify Center with I, J, K
  - I, J, K are the incremental distance from the start of the arc;
  - Viewing the start of arc as the origin, I, J, K have positive or negative signs.

Circular interpolation: G02, G03

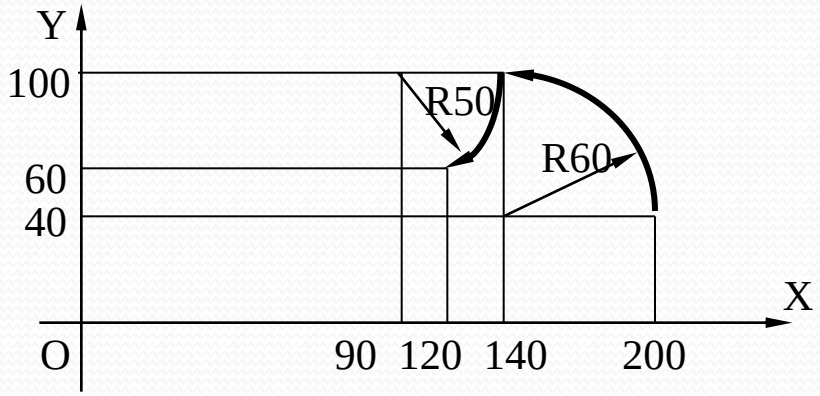
N0010 G92 X200.0 Y40.0 Z0 ;
N0020 G90 G03 X140.0 Y100.0 I -60.0 F300;
N0030 G02 X120. 0 Y60.0 I- 50.0;

Or

N0010 G92 X200.0 Y40.0 Z0;
N0020 G90 G03 X140.0 Y100.0 R60.0 F300;
N0030 G02 X120.0 Y60.0 R50.0;

G92: To define working coordinate

G90: absolute coordinates



Circular interpolation: G02, G03

Annotation for Circular Interpolation

- I0.0, J0.0, and K0.0 can be omitted.
- If X,Y,Z are all omitted in the program, that means start and end of arc are same points.

  N0020 G02 I20.0 (a full circle)

- If I, J, K, and R all appears in circular interpolation instruction, R is valid and I, J, and K are invalid

GATE - 2004

During the execution of a CNC part program block

N020 G02 X45.0 Y25.0 R5.0 the type of tool motion will be

(a) Circular Interpolation – clockwise

(b) Circular Interpolation - counter clockwise

(c) Linear Interpolation

(d) Rapid feed

GATE - 2010

In a CNC program block, N002 G02 G91 X40 Z40…, G02 and G91 refer to

(a) Circular interpolation in counterclockwise direction and incremental dimension

(b) Circular interpolation in counterclockwise direction and absolute dimension

(c) Circular interpolation in clockwise direction and incremental dimension

(d) Circular interpolation in clockwise direction and absolute dimension

GATE - 2001

In an NC machining operation, the tool has to be moved from point (5, 4) to point (7, 2) along a circular path with centre at (5, 2). Before starting the operation, the tool is at (5, 4). The correct G and M code for this motion is

(a) N010 G03 X7.0 Y2.0 I5.0 J2.0

(b) N010 G02 X7.0 Y2.0 I5.0 J2.0

(c) N010 G01 X7.0 Y2.0 I5.0 J2.0

(d) N010 G00 X7.0 Y2.0 I5.0 J2.0

GATE - 2005

The tool of an NC machine has to move along a circular arc from (5, 5) to (10,10) while performing an operation. The centre of the arc is at (10, 5). Which one of the following NC tool path commands performs the above mentioned operation?

(a) N010G02 X10 Y10 X5 Y5 R5

(b) N010G03 X10 Y10 X5 Y5 R5

(c) N010G01 X5 Y5 X10 Y10 R5

(d) N010G02 X5 Y5 X10 Y10 R5

Tool Compensation

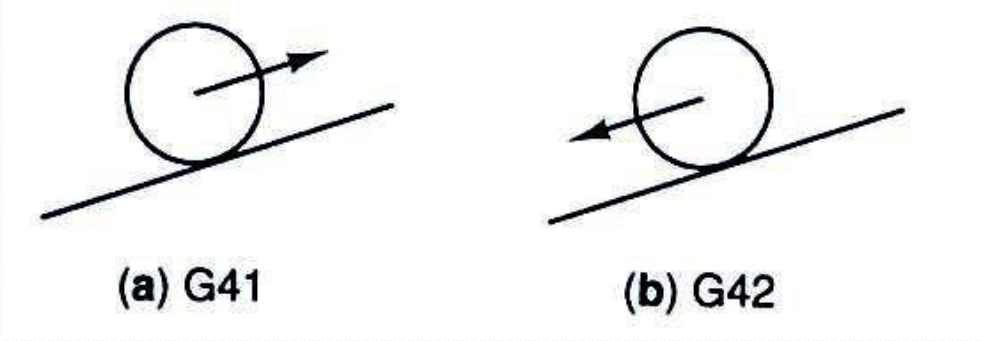
- Tool-Radius Compensation
  - Left hand G41
  - Right hand G42
  - Cancel tool-radius compensation G40
- Tool-Height Compensation
  - Positive G43
  - Negative G44
  - Cancel tool-height compensation G49

Tool-Radius Compensation

- Tool-radius compensations make it possible to program directly from the drawing, and thus eliminate the tool-offset calculation

G41 (G42) D××

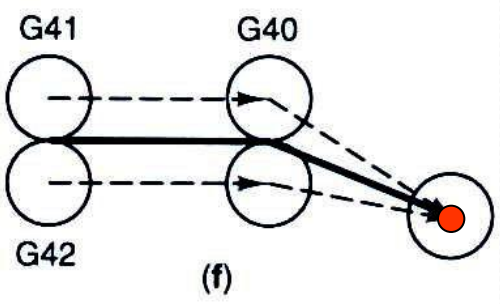
- D××: the radius of tool to compensate is saved in a memory unit that is named D××
- G41/G42 is directly related with direction of tool movement and which side of part is cut.



Cancel Tool Compensation: G40

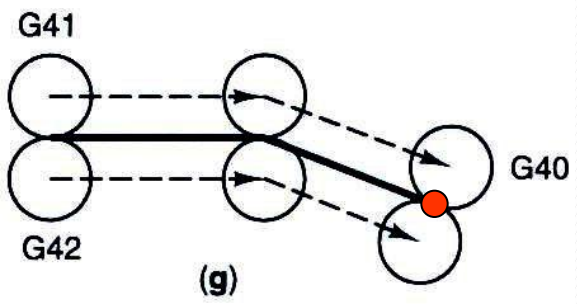
- Note the difference between two ways

N0060 **G40 G01** X2.000 Y1.700 M02



ramp off block

N0060 **G01** X2.000 Y1.700
N0070 **G40** M02



effective to the end point

GATE - 2000

In finish machining of an island on a casting with CNC milling machine, an end mill with 10 mm diameter is employed. The corner points of the island are represented by (0, 0), (0, 30), (50, 30), and (50, 0). By applying cutter radius right compensation, the trajectory of the cutter will be

(a) (-5, 0), (-5, 35), (55, 35), (55, -5), (-5, -5)

(b) (0, -5), (55, -5), (55, 35), (-5, 35), (-5, -5)

(c) (5, 5), (5, 25), (45, 25), (45, 5), (5, 5)

(d) (5, 5), (45, 5), (45, 25), (5, 25), (5, 5)

Tool-Height Compensation

G43 (G44) H××

- H××: specified memory unit used to save height compensation of tool.
- Positive compensation (G43):

  real position = specified position + value saved in H××
- Negative compensation (G44):

  real position = specified position - value saved in H××

Tool-Height Compensation

- Example:
  - N0010 G91 G00 X12.0 Y80.0
  - N0020 G44 Z-32.0 H02;

  - If we put 0.5mm into H02,
  - real position = -32.0 - 0.5 = -32.5

- Cancel tool-height compensation: G49

Table of Important M codes

- M00 Program stop
- M01 Optional program stop
- M03 Spindle on clockwise
- M04 Spindle on counterclockwise
- M05 Spindle stop
- M06 Tool change
- M08 Coolant on
- M09 Coolant off
- M10 Clamps on
- M11 Clamps off
- M02 or M30 Program stop, reset to start

Rules for programming

Block Format

N135 G01 X1.0 Y1.0 Z0.125 F5

Sample Block

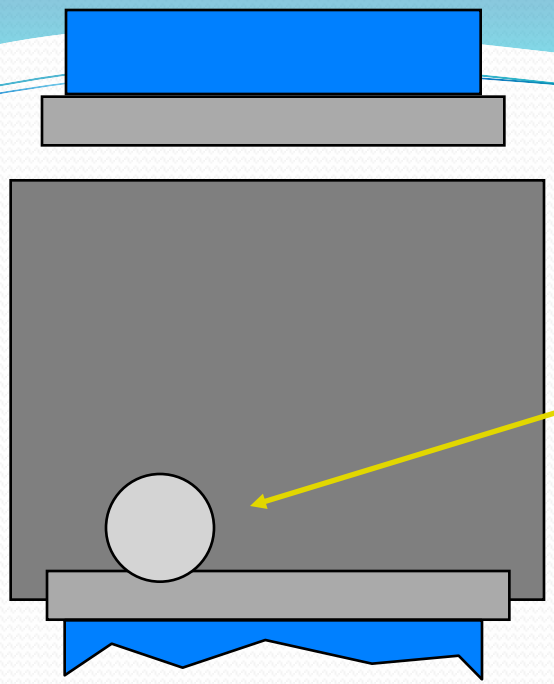
- Restrictions on CNC blocks
- Each may contain only one tool move
- Each may contain any number of non-tool move G-codes
- Each may contain only one feed rate
- Each may contain only one specified tool or spindle speed
- The block numbers should be sequential
- Both the program start flag and the program number must be independent of all other commands (on separate lines)
- The data within a block should follow the sequence shown in the above sample block

Example of CNC Programming

- *What Must Be Done To Drill A Hole On A CNC Vertical Milling Machine*

Tool Home

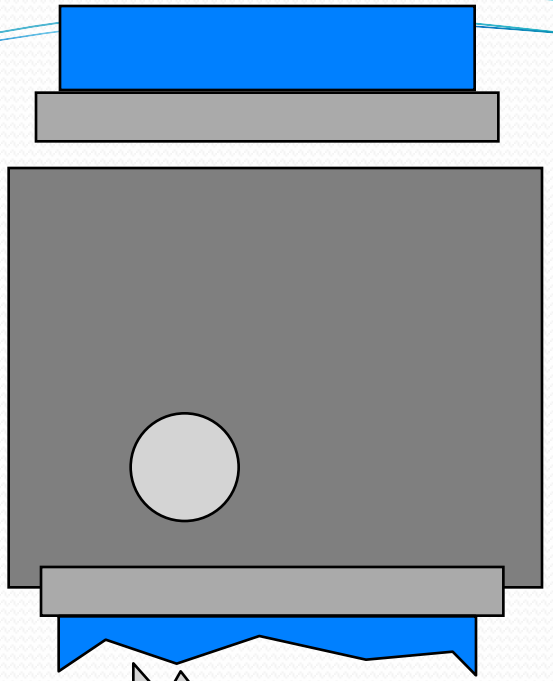
Top View



1.) X & Y Rapid To Hole Position

Front View

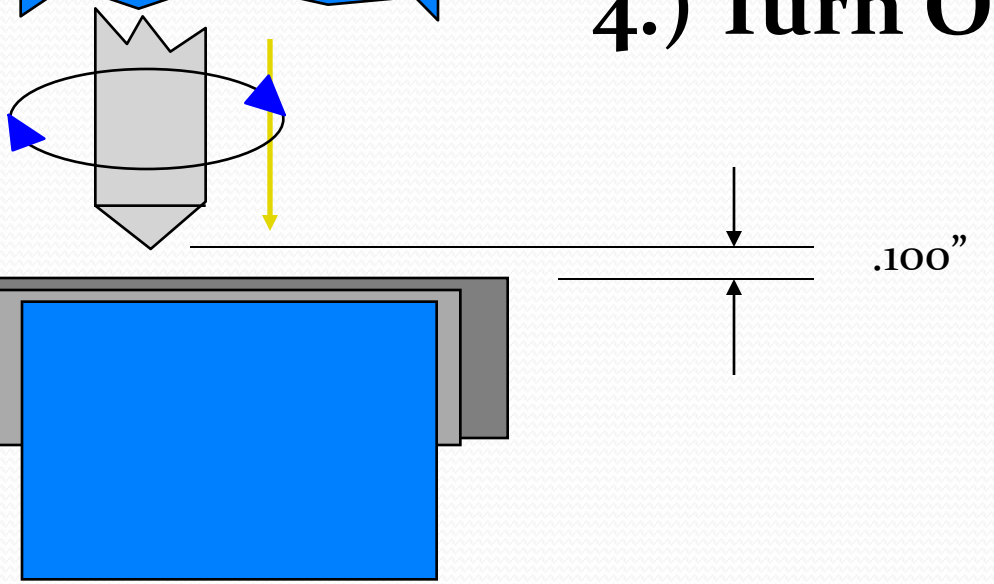
Top View



2.) Z Axis Rapid Move Just Above Hole

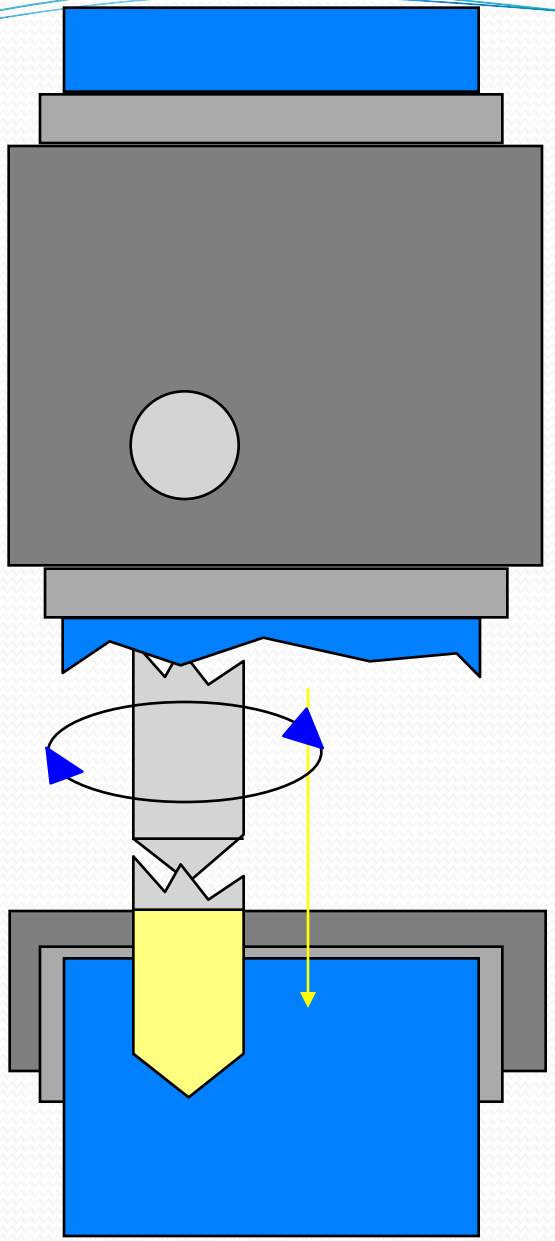
3.) Turn On Coolant

4.) Turn On Spindle



Front View

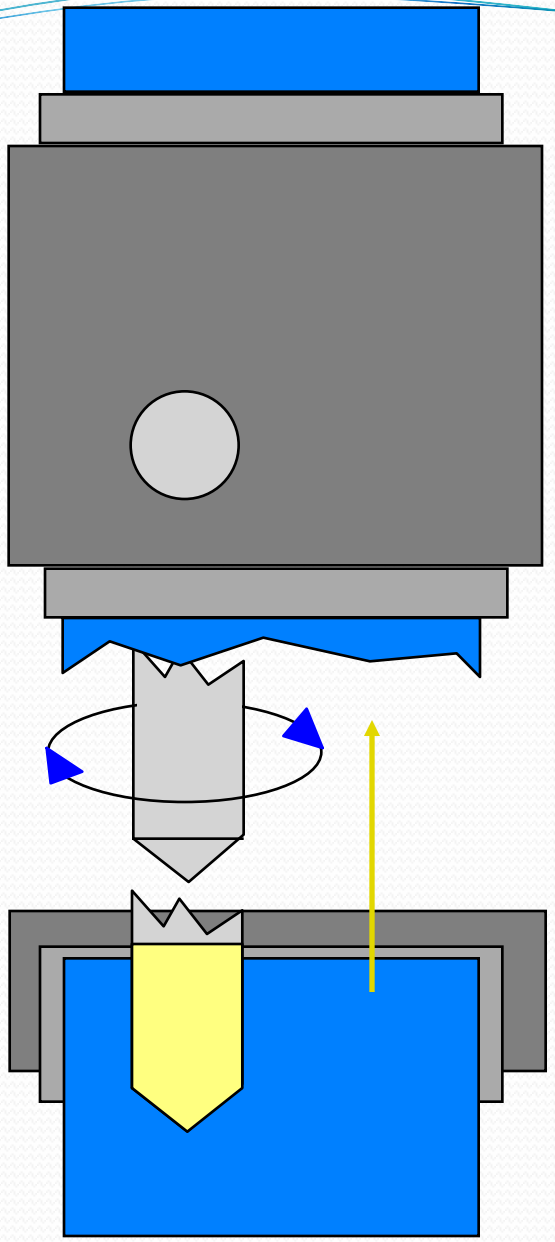
Top View



Front View

5.) Z Axis Feed Move to Drill Hole

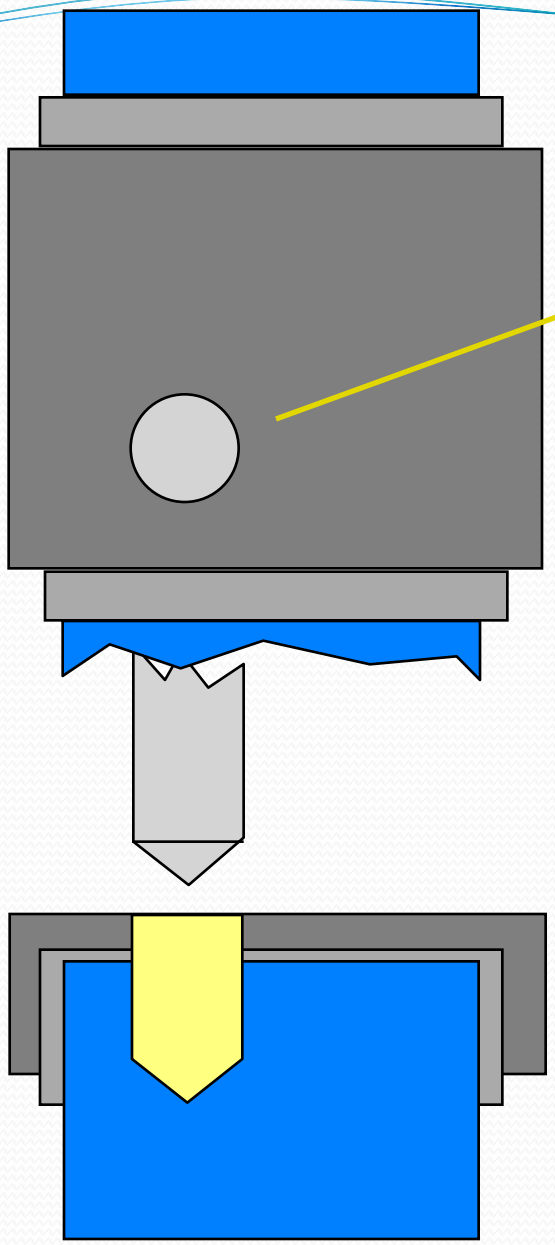
Top View



Front View

6.) Rapid Z Axis Move Out Of Hole

Top View



Front View

7.) Turn Off Spindle

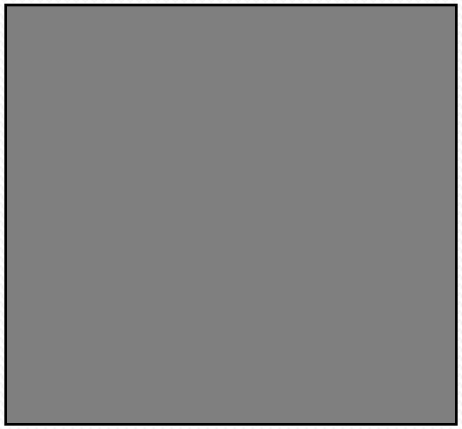
8.) Turn Off Coolant

9.) X&Y Axis Rapid Move Home

Here's The CNC Program!

Tool At Home

Top View

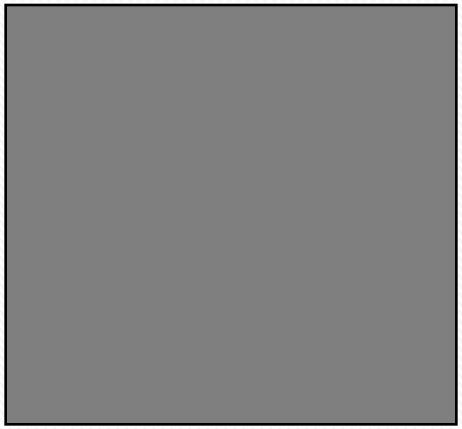


Front View

```
O0001
  N005 G54 G90 S600 M03
  N010 G00 X1.0 Y1.0
  N015 G43 H01 Z.1 M08
 N020 G01 Z-.75 F3.5
 N025 G00 Z.1 M09
 N030 G91 G28 X0 Y0 Z0
 N035 M30
```

Tool At Home

Top View



O0001

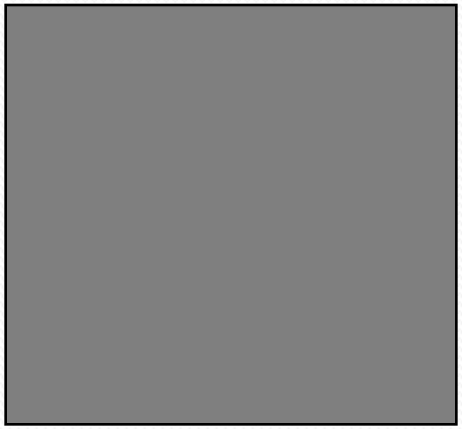
O0001

Number Assigned to this program

Front View

Tool At Home

Top View



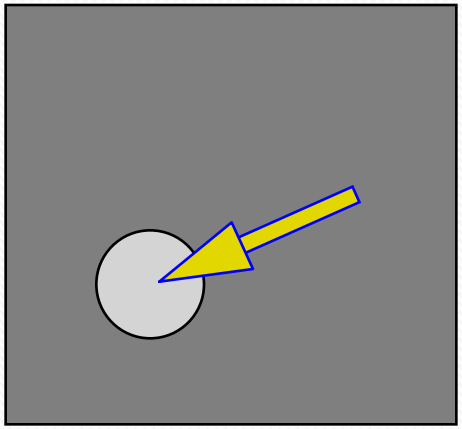
O0001

N005 G54 G90 S600 M03

N005	Sequence Number
G54	Fixture Offset
G90	Absolute Programming Mode
S600	Spindle Speed set to 600 RPM
M03	Spindle on in a Clockwise Direction

Front View

Top View



O0001

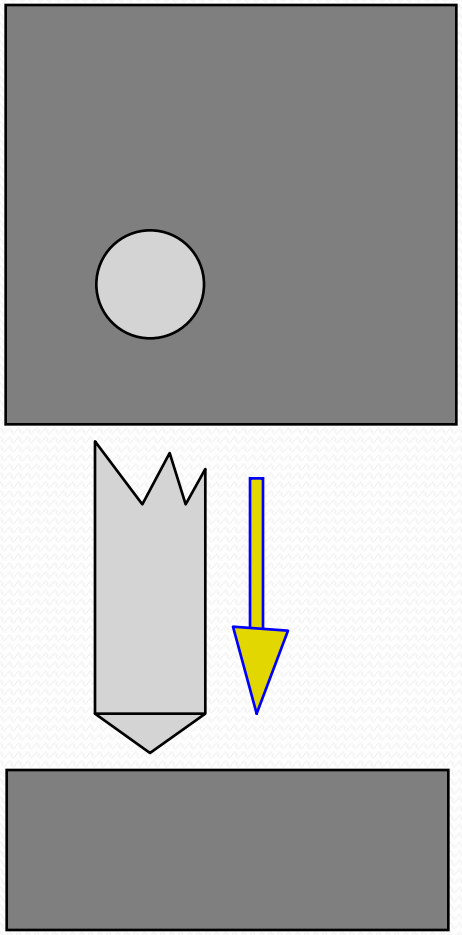
N005 G54 G90 S600 M03

N010 G00 X1.0 Y1.0

G00	Rapid Motion
X1.0	X Coordinate 1.0 in. from Zero
Y1.0	Y Coordinate 1.0 in. from Zero

Front View

Top View



Front View

O0001

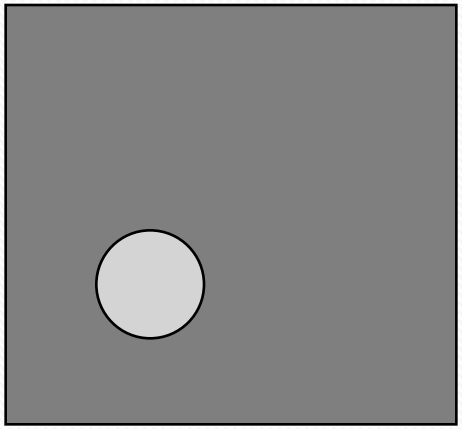
N005 G54 G90 S600 M03

N010 G00 X1.0 Y1.0

N015 G43 H01 Z.1 M08

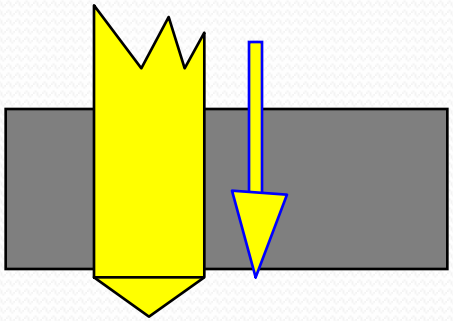
G43	Tool Length Compensation
H01	Specifies Tool length compensation
Z.1	Z Coordinate .1 in. from Zero
M08	Flood Coolant On

Top View



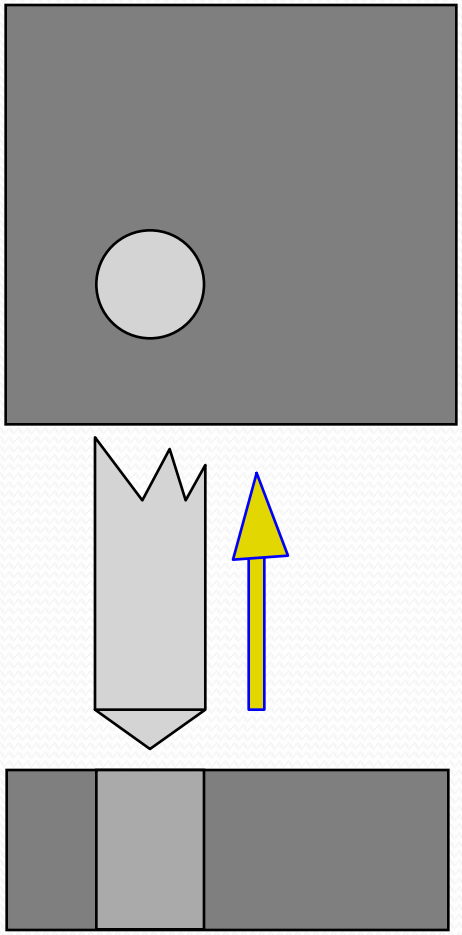
```
O0001
  N005 G54 G90 S600 M03
  N010 G00 X1.0 Y1.0
  N015 G43 H01 Z.1 M08
 N020 G01 Z-.75 F3.5
```

Front View



G01	Straight Line Cutting Motion
Z-.75	Z Coordinate -.75 in. from Zero
F3.5	Feed Rate set to 3.5 in/min.

Top View



Front View

O0001

N005 G54 G90 S600 M03

N010 G00 X1.0 Y1.0

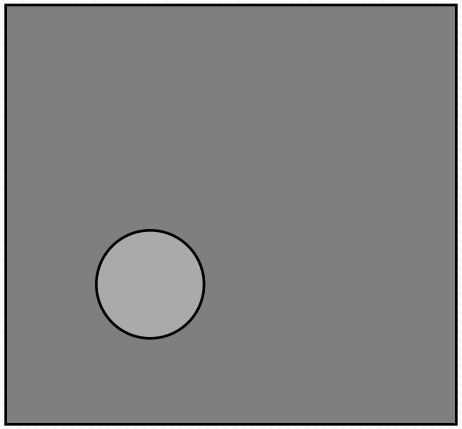
N015 G43 H01 Z.1 M08

N020 G01 Z-.75 F3.5

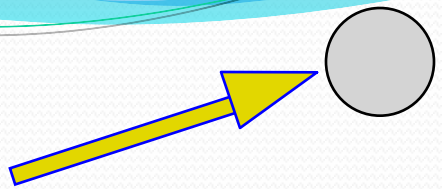
N025 G00 Z.1 M09

G00	Rapid Motion
Z.1	Z Coordinate .1 in. from Zero
M09	Coolant Off

Top View



Front View



O0001

N005 G54 G90 S600 M03

N010 G00 X1.0 Y1.0

N015 G43 H01 Z.1 M08

N020 G01 Z-.75 F3.5

N025 G00 Z.1 M09

N030 G91 G28 X0 Y0 Z0

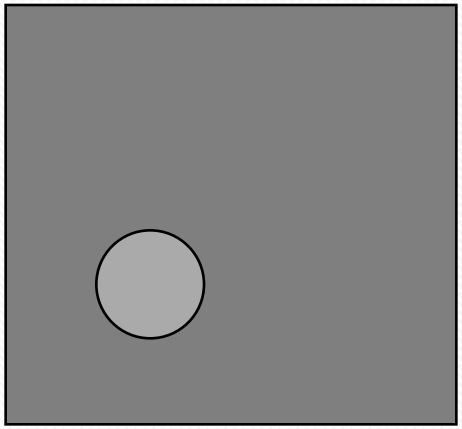
G91 Incremental Programming Mode

G28 Zero Return Command

X0, Y0, Z0

X,Y,& Z Coordinates at Zero

Top View



Front View

O0001

N005 G54 G90 S600 M03

N010 G00 X1.0 Y1.0

N015 G43 H01 Z.1 M08

N020 G01 Z-.75 F3.5

N025 G00 Z.1 M09

N030 G91 G28 X0 Y0 Z0

N035 M30

M30 End of Program

IES - 1995

Match List I with List II and select the correct answer using the codes given below the lists:

List I (A function connected with NC m/c tool)	**List II (Associated parameter)**
A. Interpolation	1. Tape preparation
B. Parity check	2. Canned cycle
C. Preparatory function	3. Drilling
D. Point to point control	4. Contouring
	5. Turning

Code:	**A**	**B**	**C**	**D**
(a)	4	1	2	3
(b)	4	1	2	5
(c)	5	1	3	2
(d)	1	4	3	2

APT Language

APT Language

- APT (Automatically Programmed Tools)
- The APT language consists of many different types of statements made up of the following valid letters, numerals and punctuation marks.
- Letters: **ABCDEFGHIJKLMNOPQRSTUVWXYZ**
- Numerals: 0 1 2 3 4 5 6 7 8 9

/ A slash divides a statement into two sections. eg., GO/PAST,

, A comma is used as a separator between the elements in a statement generally to the right of the slash.

= An equals is used for assigning an entity to a symbolic name, e.g., P1 = POINT/25,50,30.

Words

- The words to be used in the statements are built up from one to six letters or numerals with the first one being a letter. No special character is allowed in the words.

IES - 1998

Which of the following are the rules of programming NC machine tools in APT language?

1. Only capital letters are used
2. A period is placed at the end of each statement
3. Insertion of space does not affect the APT word

Select the correct answer using the codes given below:

(a) 1 and 2 (b) 2 and 3

(c) 1 and 3 (d) 1 alone

The complete APT part program consists of the following four types of statements

- **Geometry**
- **Motion**
- **Post processor**
- **Compilation control**

Other Part Programming Languages

- **ADAPT** (ADaptation APT) was the first attempt to adapt APT programming system for smaller computers
- **AUTOSPOT** (AUTOmatic Sytem for POsitioning Tools) was developed by IBM and first introduced in 1962
- **EXAPT** (EXtended subset of APT) was developed jointly in German in about 1964 by several universities to adapt APT for European use. It is compatible with APT and thus can use the same processor as APT
- **COMPACT** was developed by Manufacturing Data Systems, Inc. (MDSI)
- **SPLIT** (Sundstrand Processing Language Internally Translated) was developed by Sundstrand Corporation, intended for its own machine tools
- **MAPT** (Micro-APT) is a subset of APT, to be run on the microcomputers

APT Language

Additional statements:

- **MACHIN/DRILL, 2**
- **COOLNT/**

For example: **COOLNT/MIST COOLNT/FLOOD COOLNT/OFF**

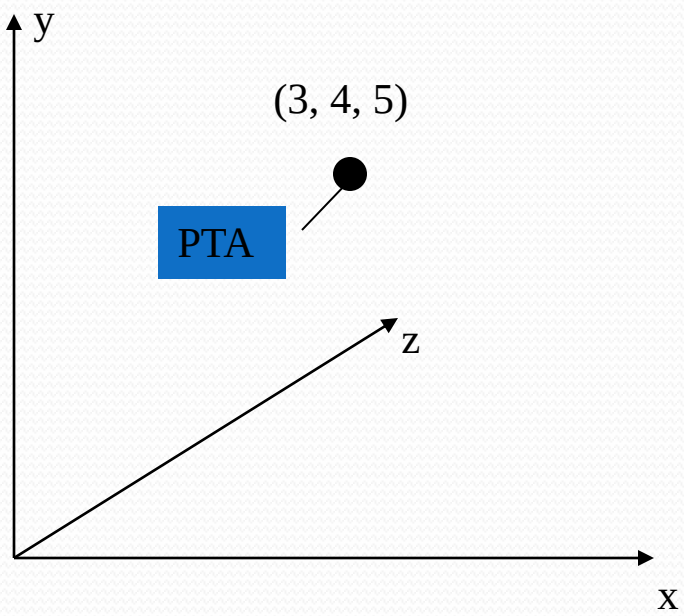
- **FEDRAT/**
- **SPINDL/**

For example: **SPINDL/ON SPINDL/1250, CCLW**

- **TOOLNO/**
- **TURRET/**
- **END**

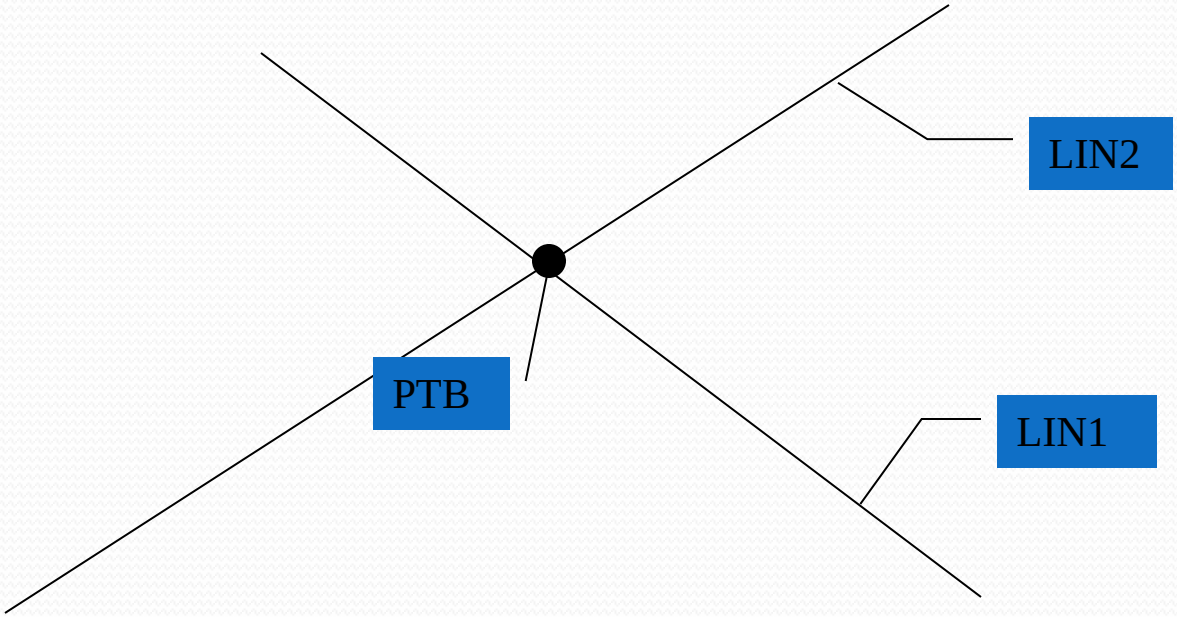
Point (POINT)

PTA = POINT/ 3,4,5



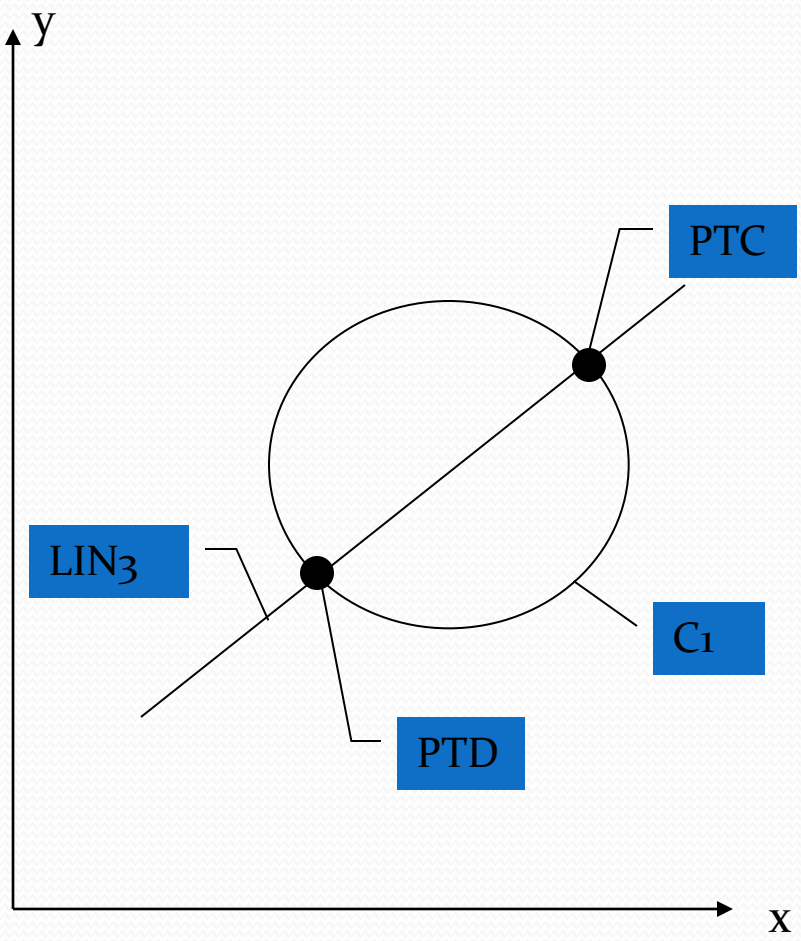
Point (POINT)

PTB = POINT/ INTOF, LIN1, LIN2



Point (POINT)

PTD = POINT/ YSMALL, INTOF, LIN3, C1
PTD = POINT/ XSMALL, INTOF, LIN3, C1
PTC = POINT/ YLARGE, INTOF, LIN3, C1
PTC = POINT/ XLARGE, INTOF, LIN3, C1



Point (POINT)

PTE = POINT/ YLARGE, INTOF, C1, C2

PTE = POINT/ XLARGE, INTOF, C1, C2

PTF = POINT/ YSMALL, INTOF, C1, C2

PTF = POINT/ XSMALL, INTOF, C1, C2