



Govt. of Bihar

Department of Science & Technology  
Government Polytechnic Vaishali

*PRODUCTION PROCESS*  
*Semester-IV (Mechanical Engineering)*

Unit -1

**CNC Lathe & Part Programming**

*by*

**Prof. Jitendra Kumar**

Lecturer ,Department of Mechanical  
Engineering

G. P. Vaishali Bihar-844118

# CONTENTS

- POWERPOINT PRESENTATION
- NOTES
- EXAMPLES
- ASSIGNMENTS

# Learning Objectives

Students will be able to:

- Understand the NC CNC & DNC.
- Advantages of CNC
- Positioning of CNC Lathe
- Programming Format
- G codes and M codes
- CNC Programming

# 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 equipped 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.

# History of CNC

- 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.

# 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?

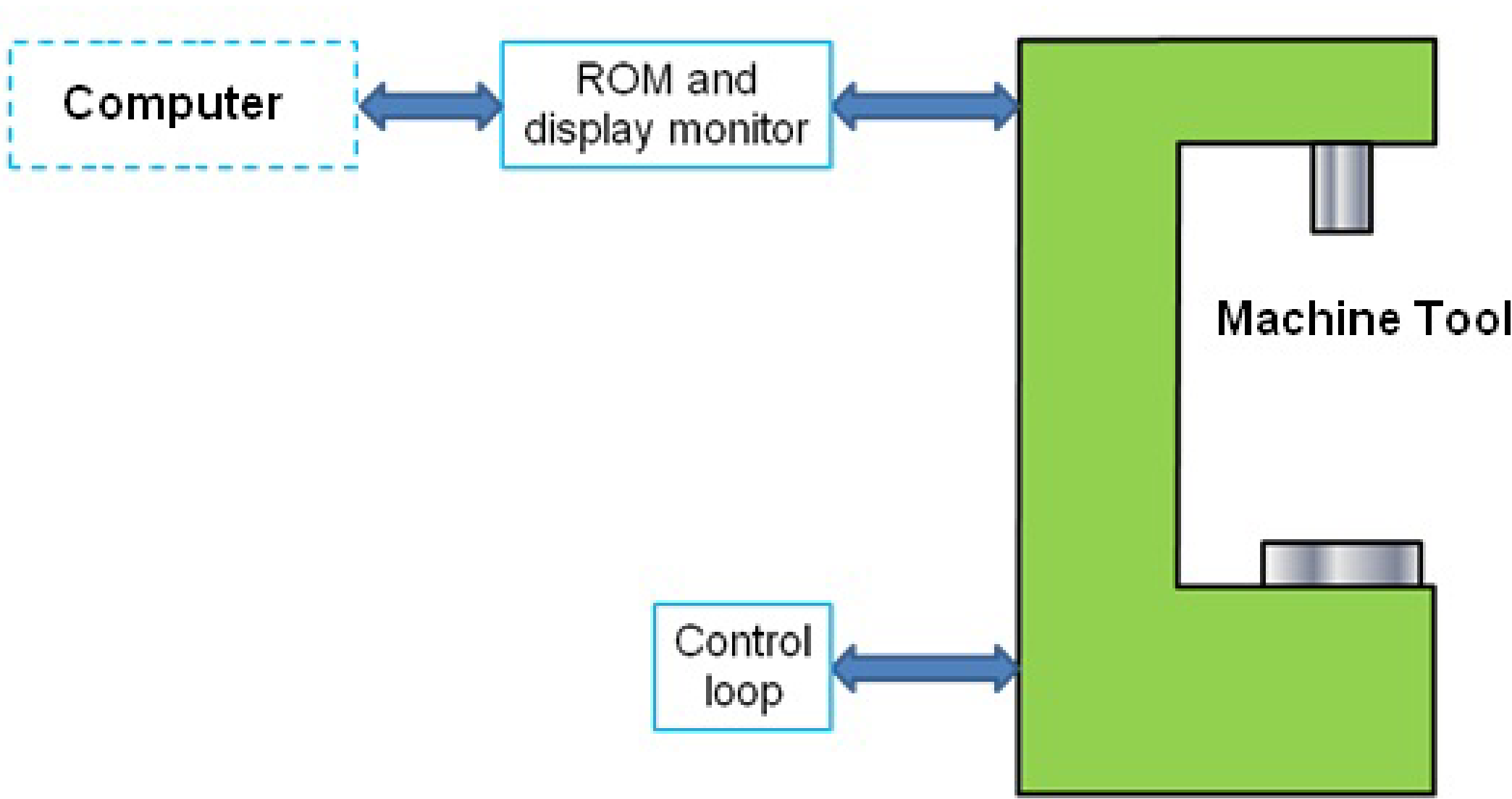
- Yes, they can be. But like anything else it depends on who is running the machine, how well the machines has been maintained, quality of setup and so on.

# Advantages of NC/CNC Machines

- 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



**CNC Machine Tool**

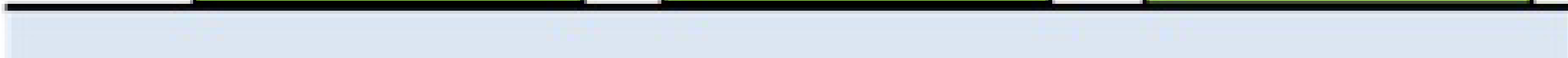
# DNC

Central computer

Communication network

CNC  
machine  
tools

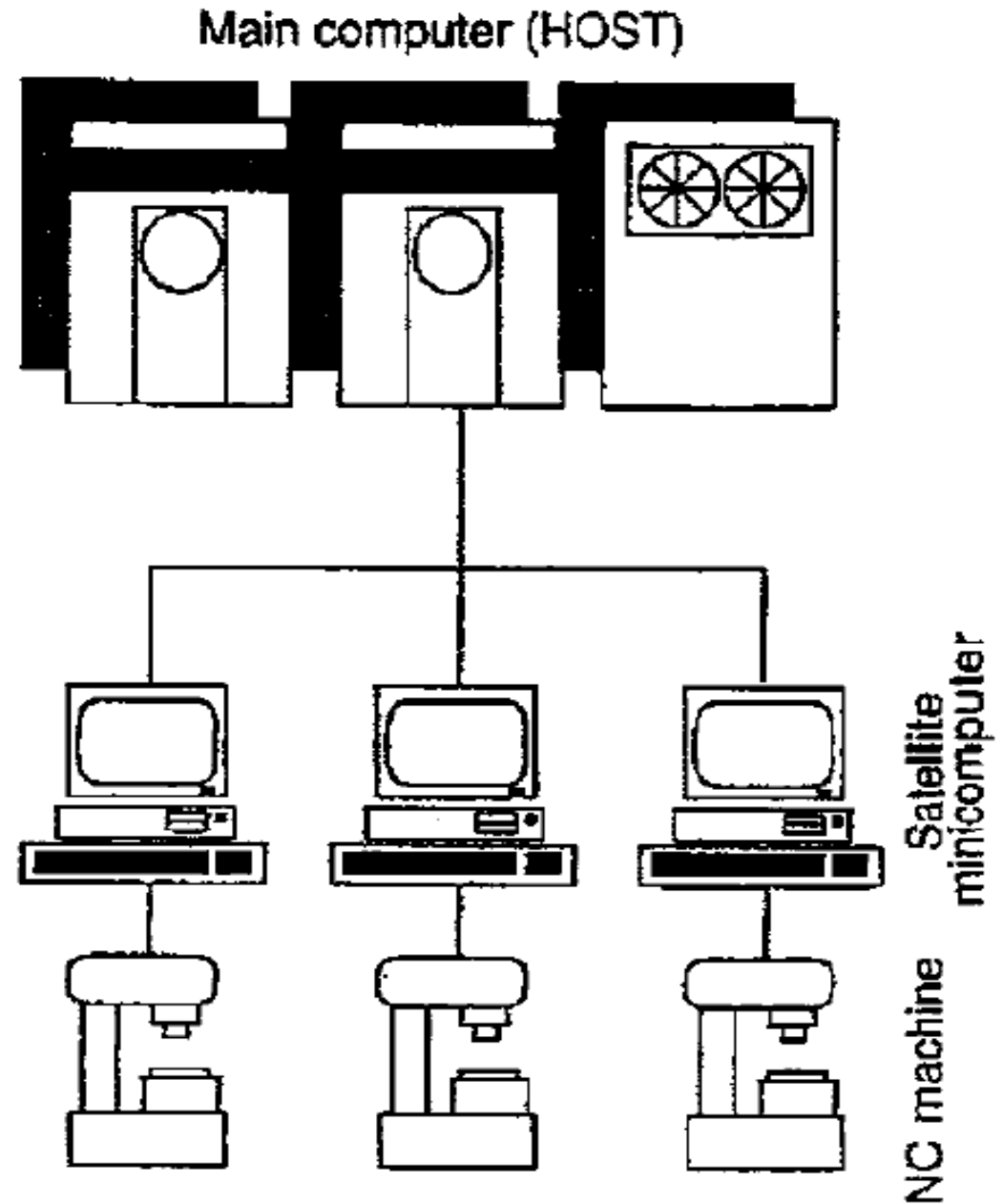
...



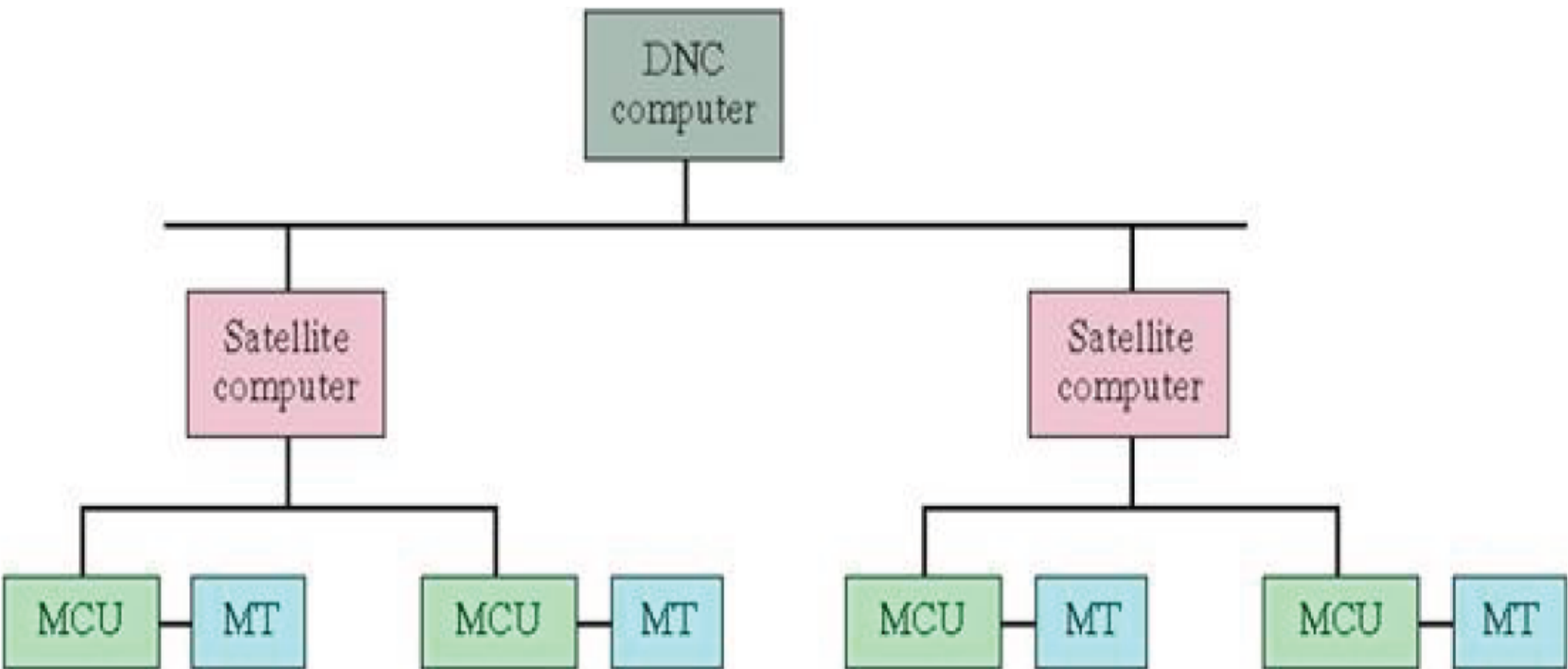
# Distributed numerical control (DNC)

- In early 1980s, Distributed numerical control (DNC) comes into picture.
- Now, many CNC machines together with robots, programmable logic controllers, and other computer-based controllers have been integrated into DNC systems to make automated manufacturing systems possible.

# DNC Configuration-1



# DNC Configuration-2





# Components of DNC

- Central computer
- Bulk memory for storing programs
- Communication network
- CNC machines

# Advantages of DNC

- Greater computational capability and flexibility
- Central computer and bulk memory, which stores the NC programs.
- Shop floor control
- Establishes the framework for evolution of future automation, CIM

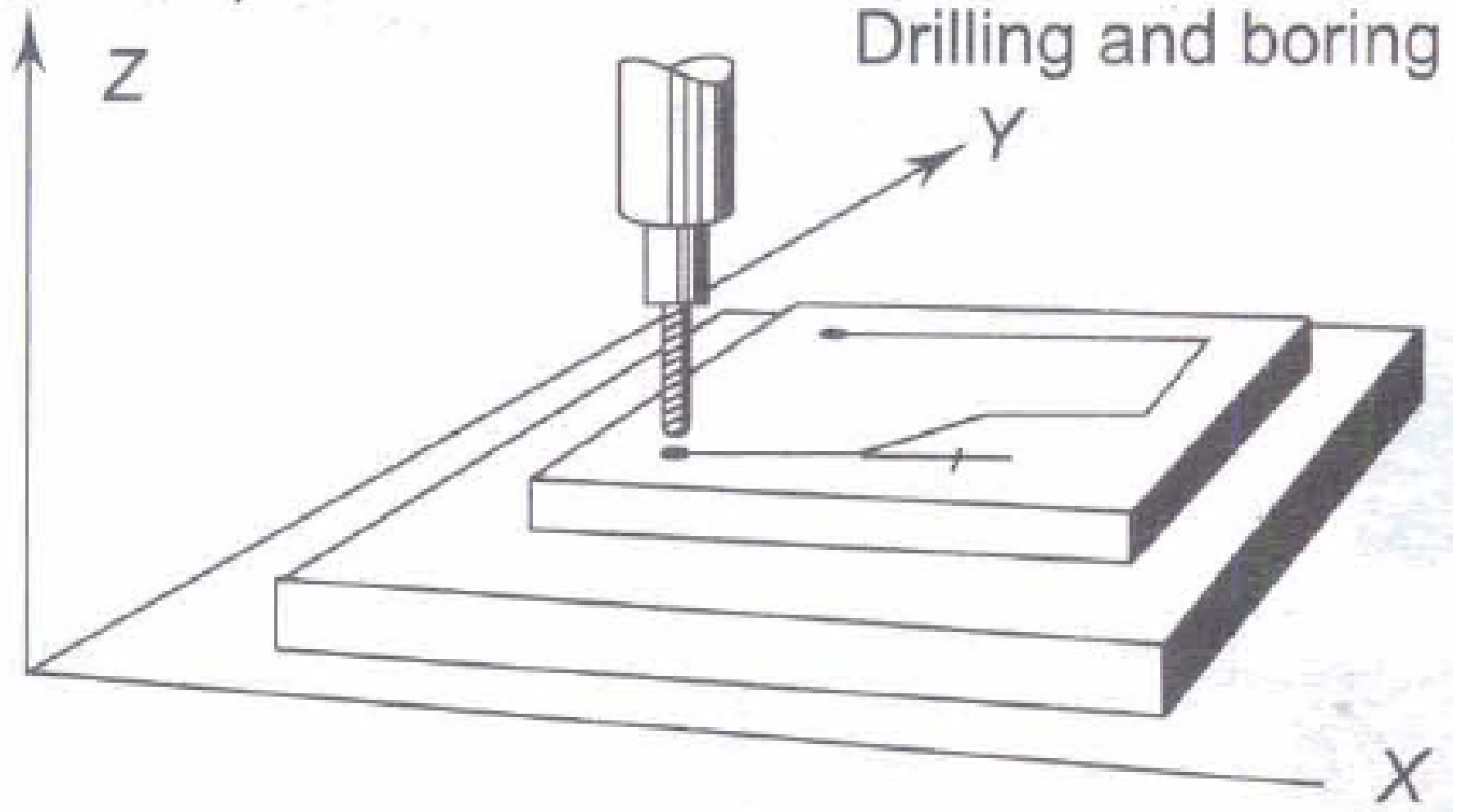
# Satellite Computers

- Satellite computers are minicomputers and they serve to take some of the burden of central computer, Each satellite computer control several machine tools.

# Control Systems possible in CNC Machine

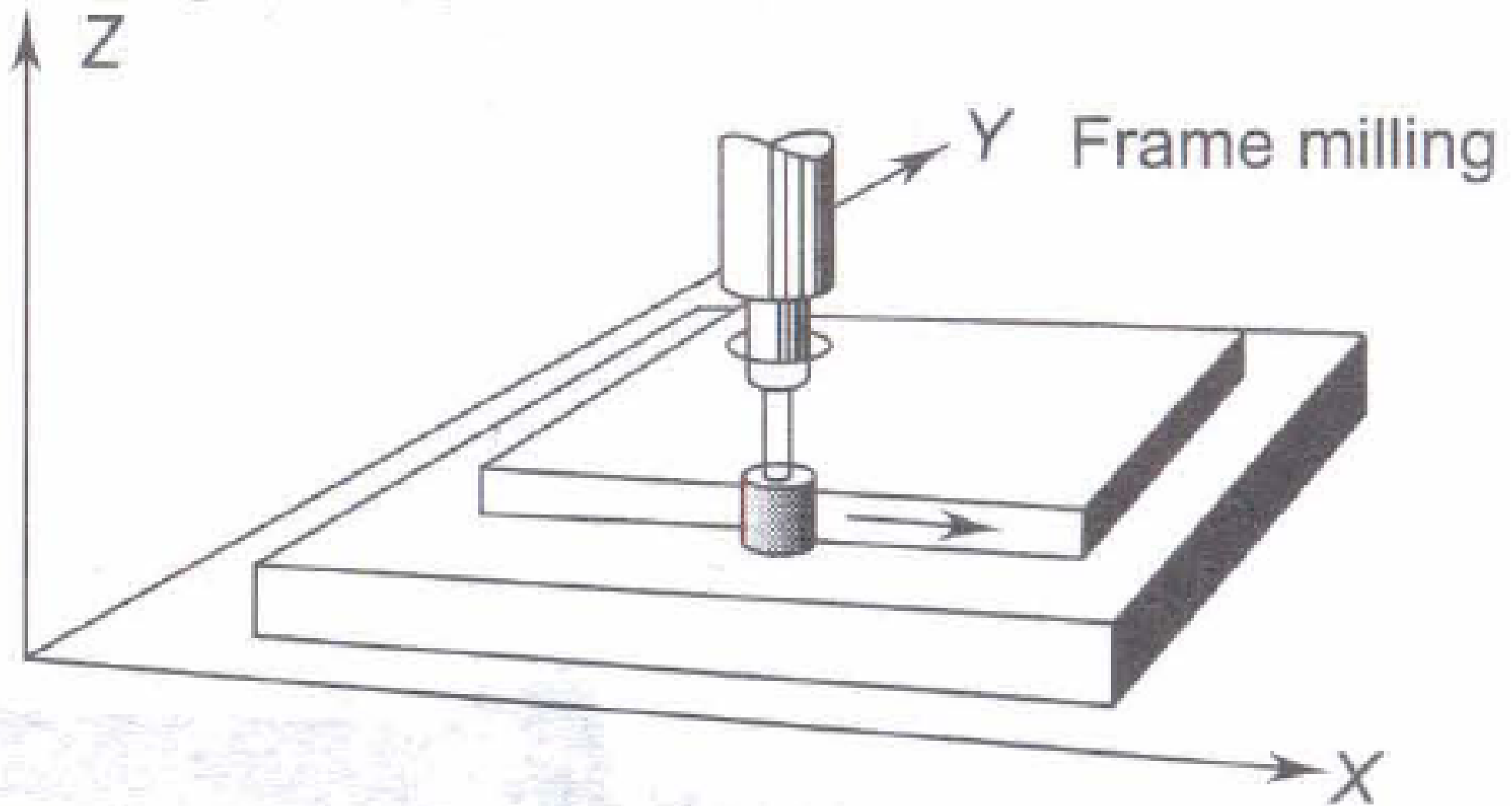
- Point to point mode:

Point-to-point



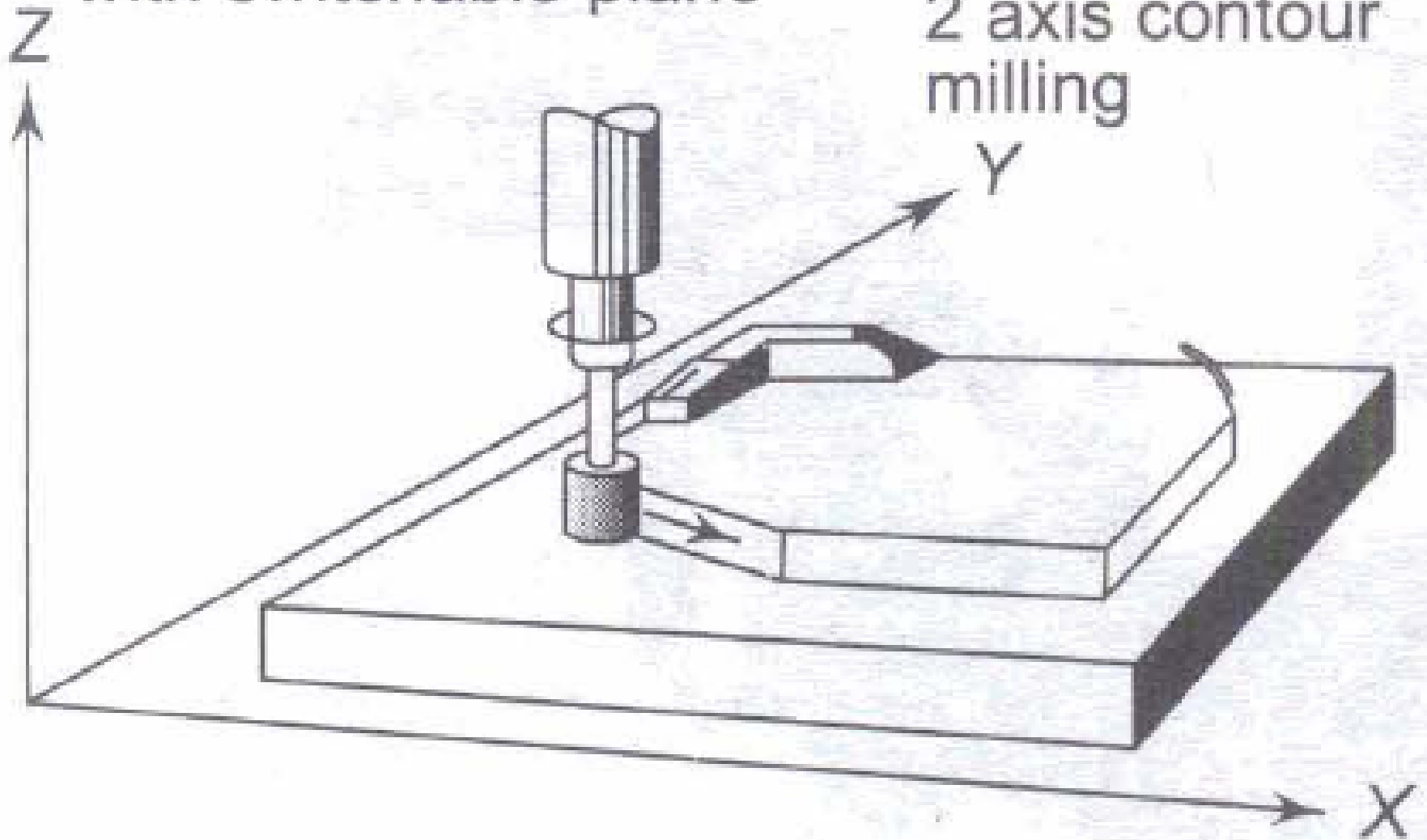
# Point-to-point straight line mode

Point-to-point  
straight line



2 axis contouring  
with switchable plane

2 axis contour  
milling



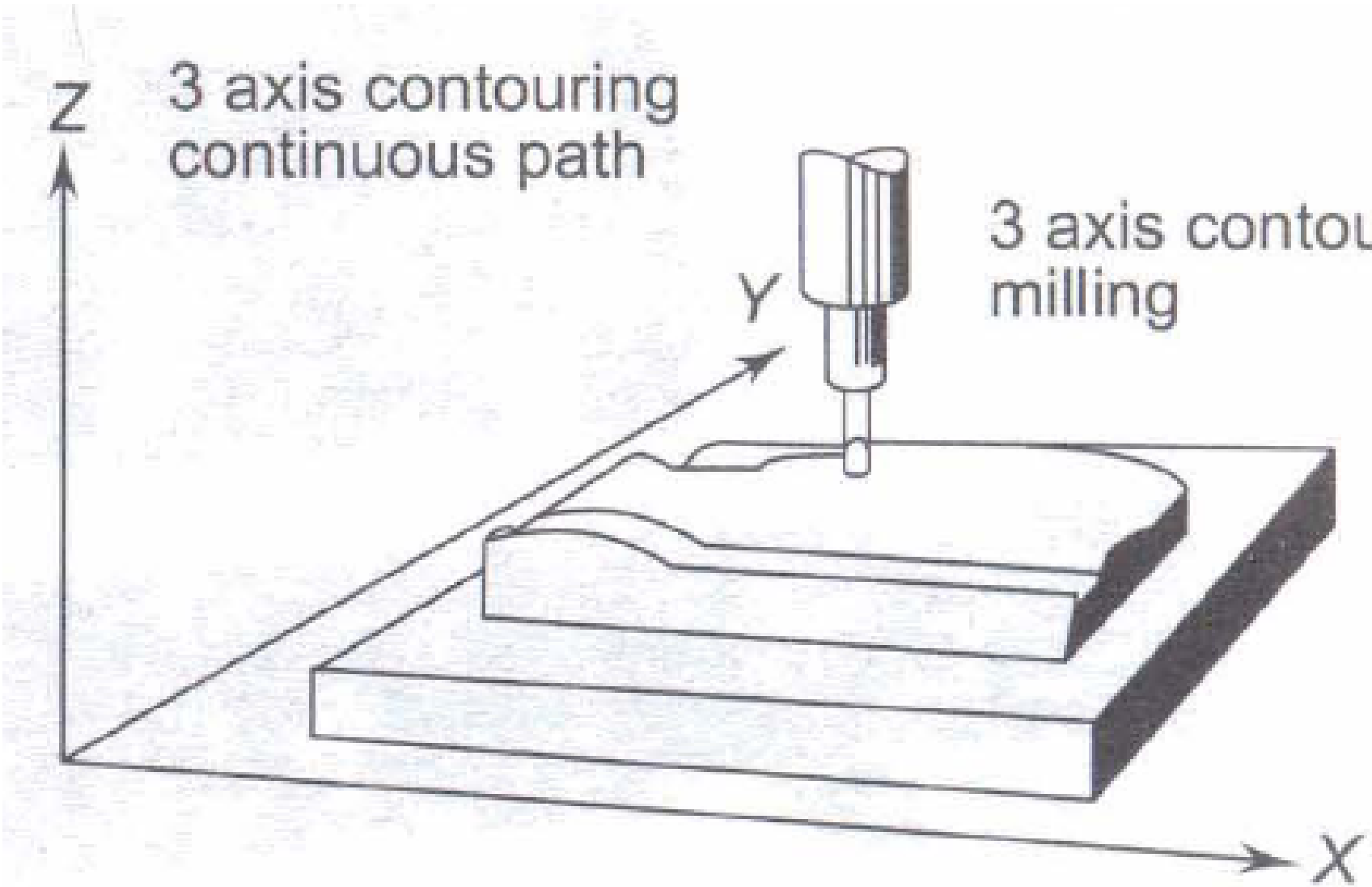
Z

3 axis contouring  
continuous path

Y

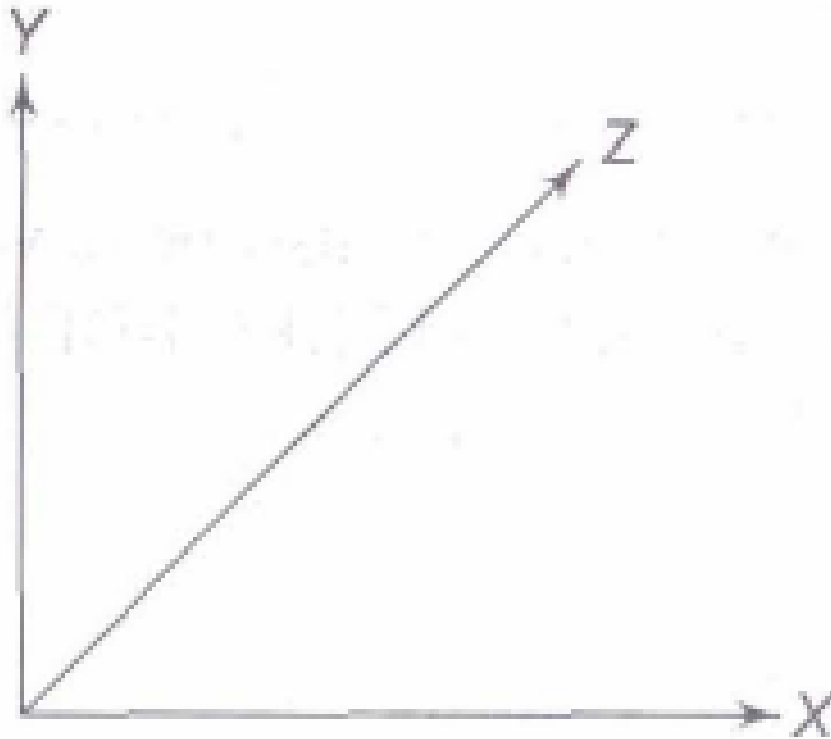
3 axis contour  
milling

X



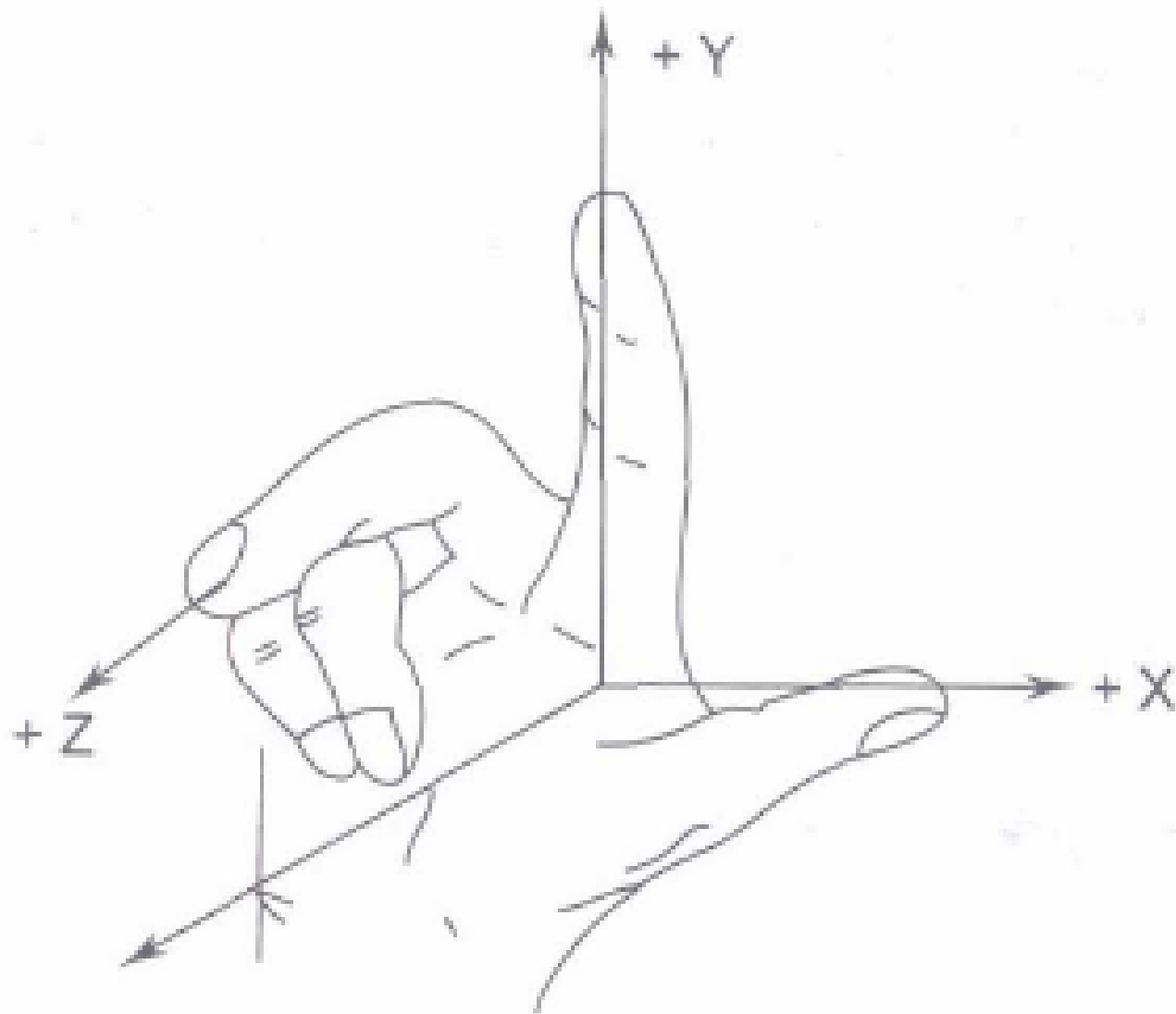
# Co-ordinate system

- 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.

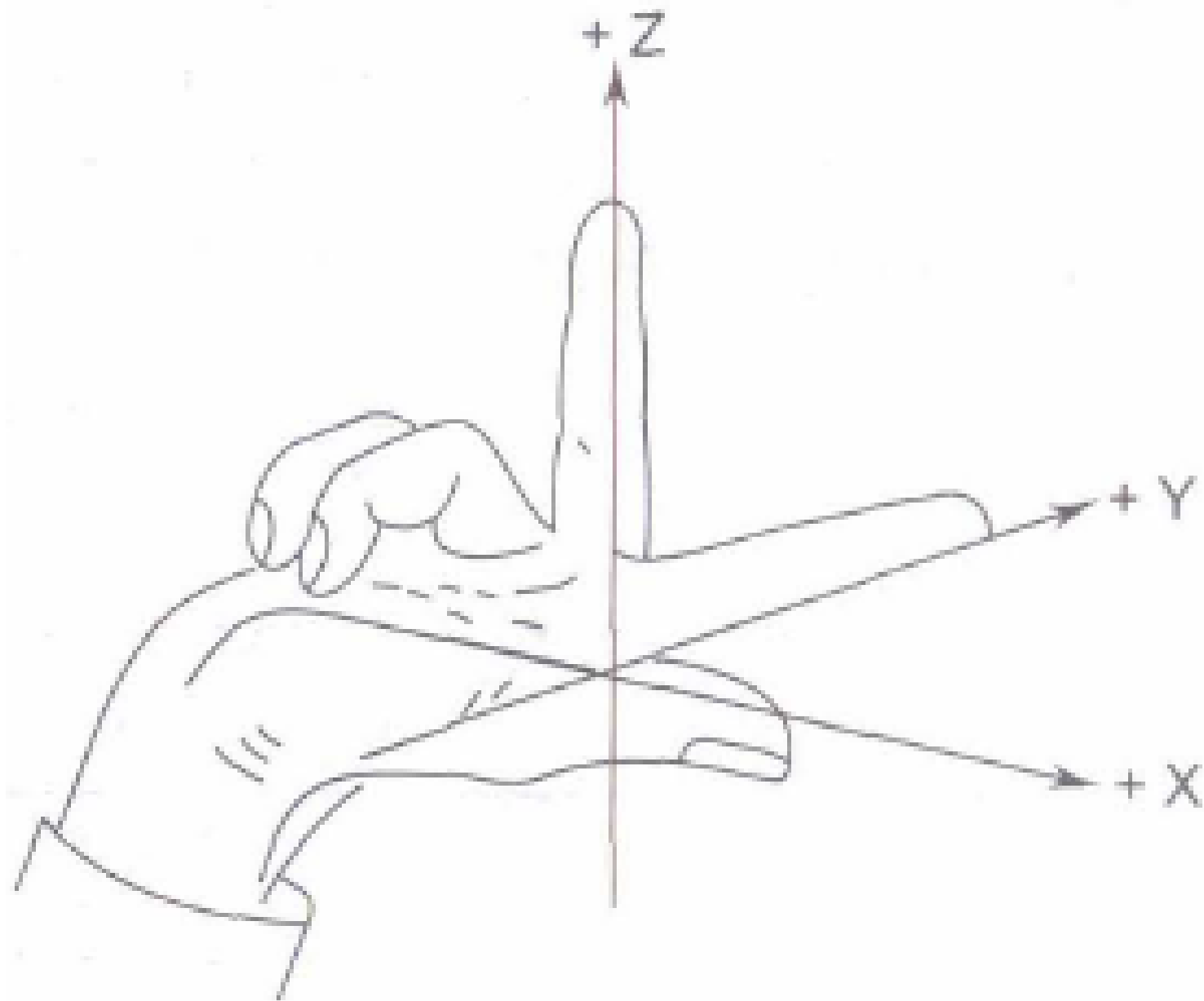


Right-hand coordinate systems

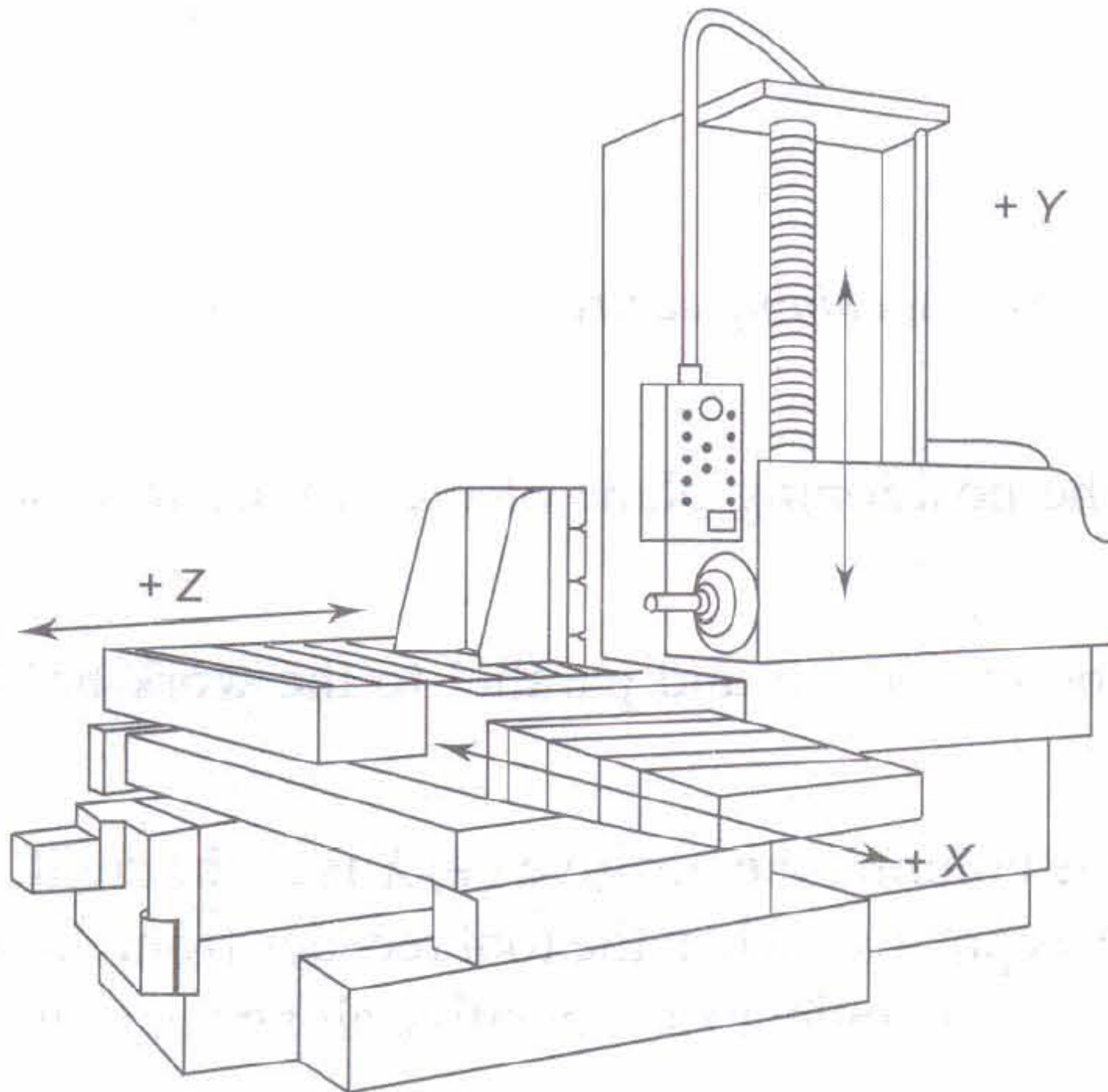




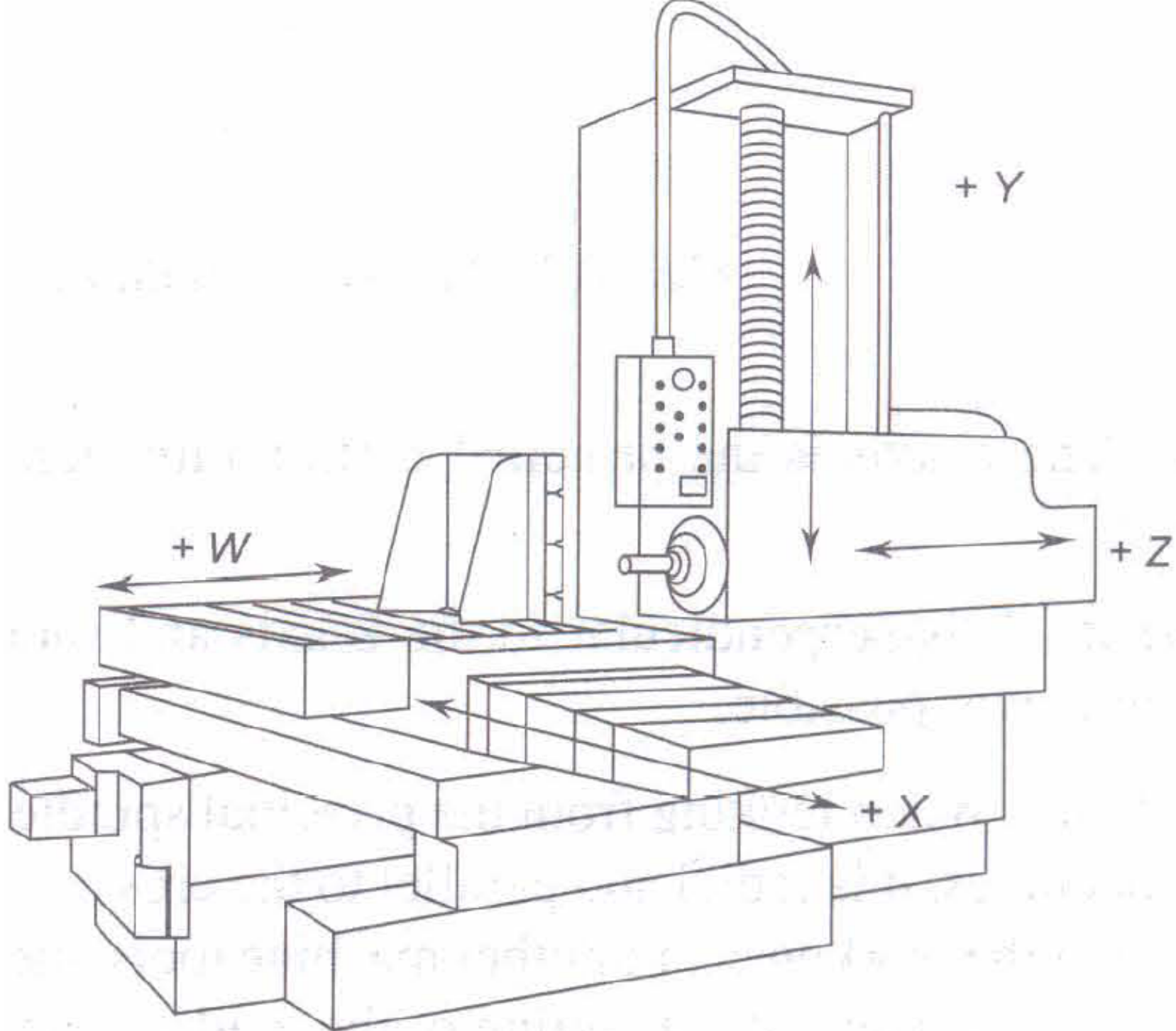
(a) Axis designation for horizontal Z



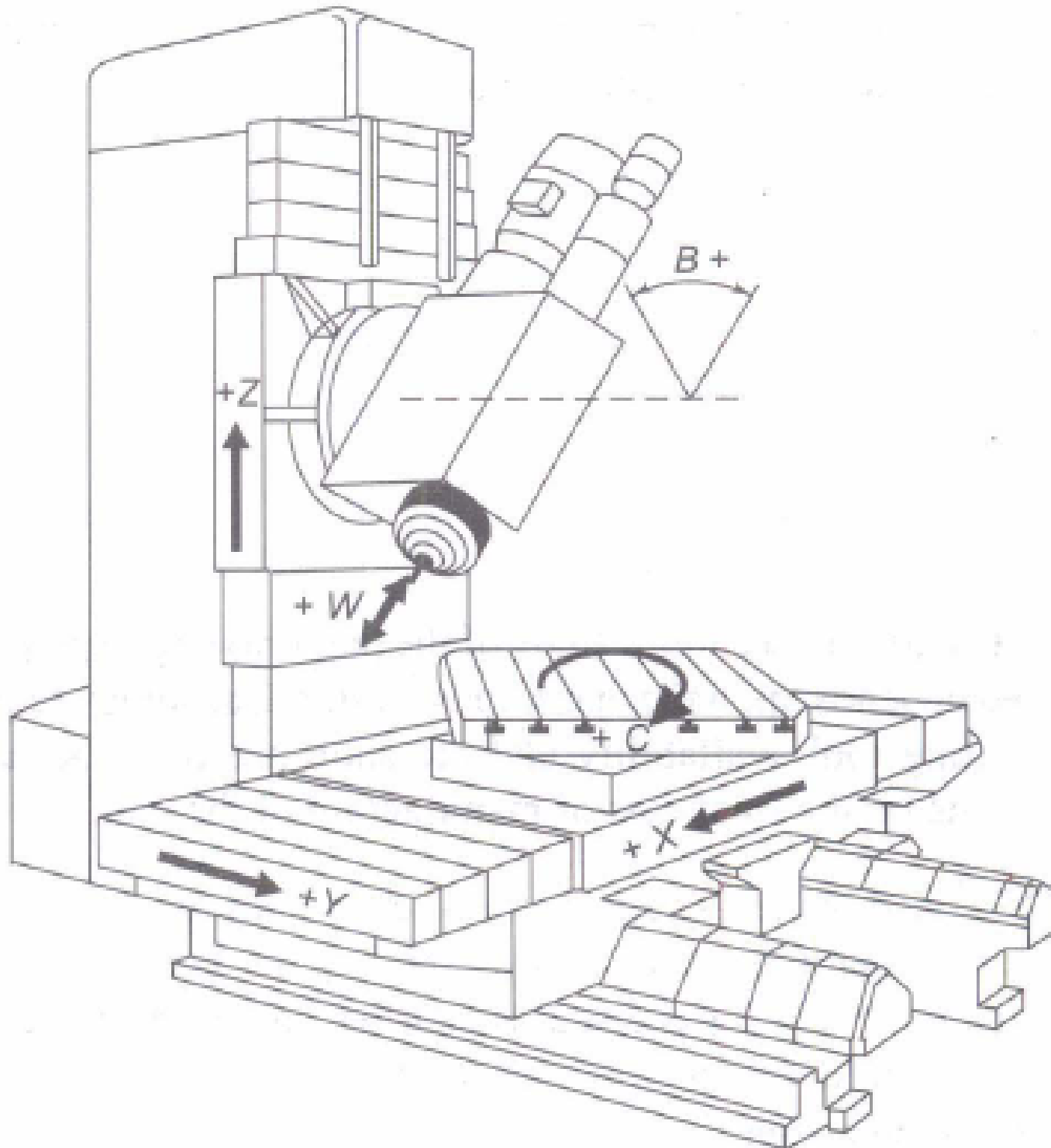
(b) Axis designation for vertical Z



(a) Three-axis boring mill

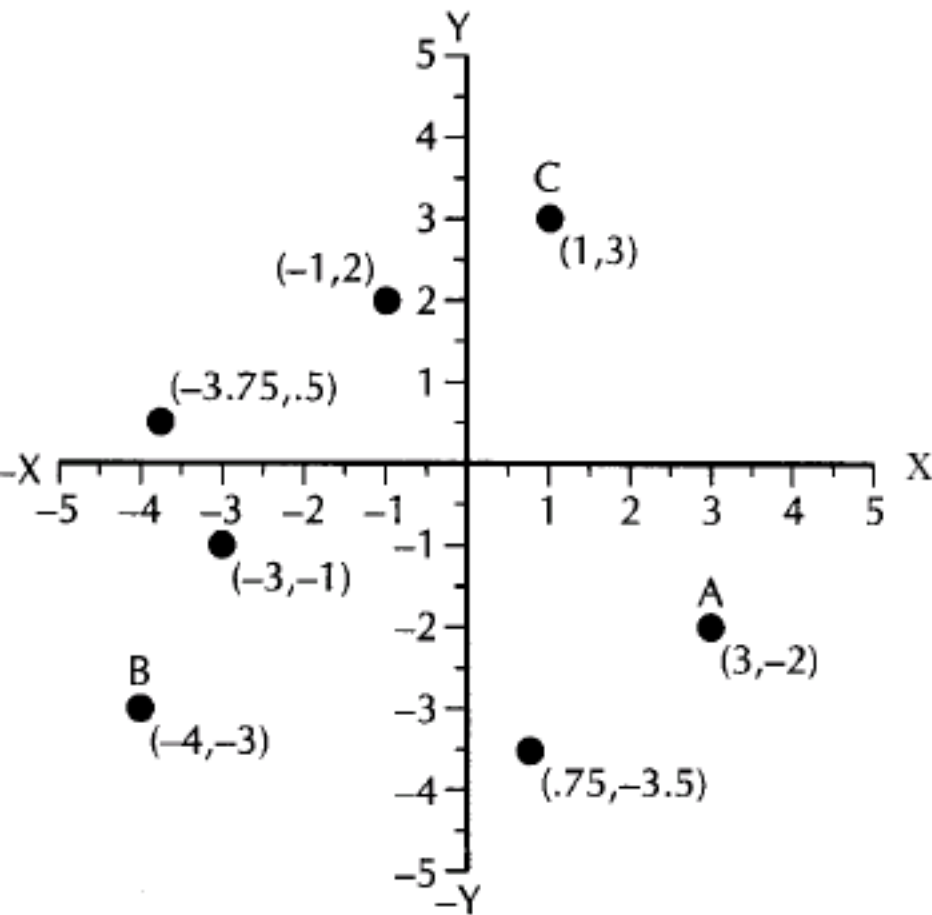


(b) Four-axis boring mill

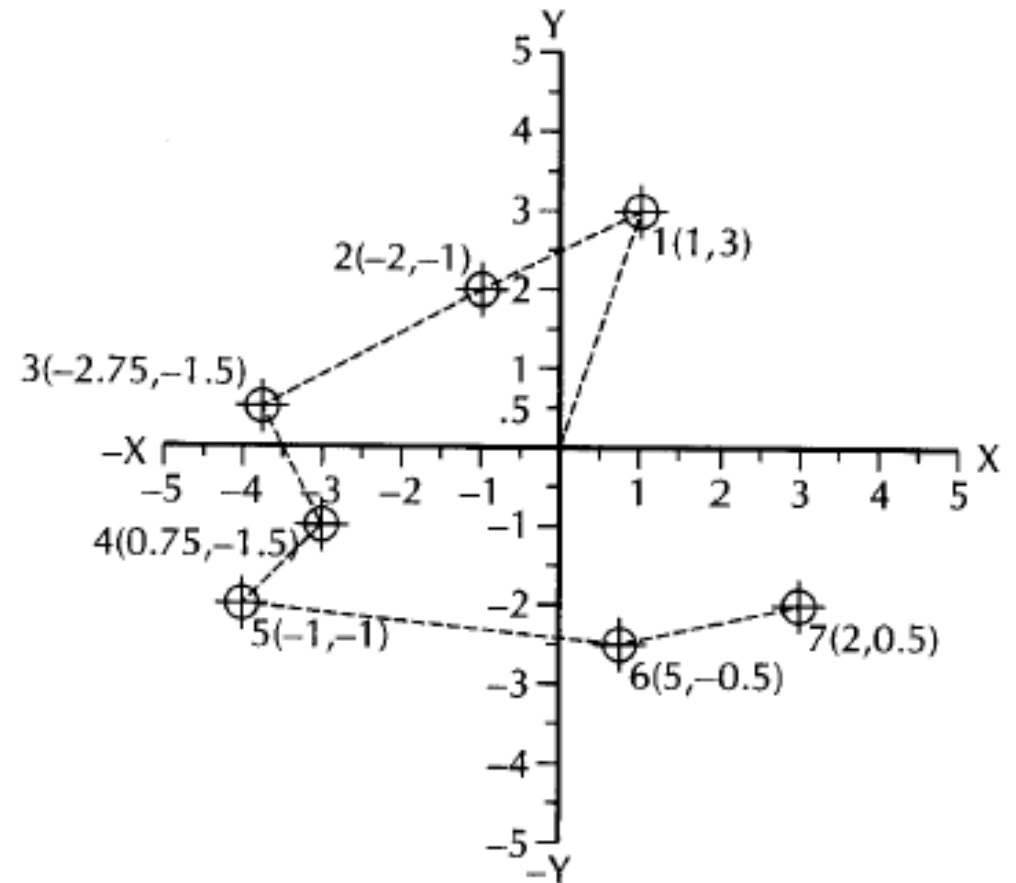


**5 axes CNC vertical axis machining centre configuration**

# Absolute and Incremental Coordinate System



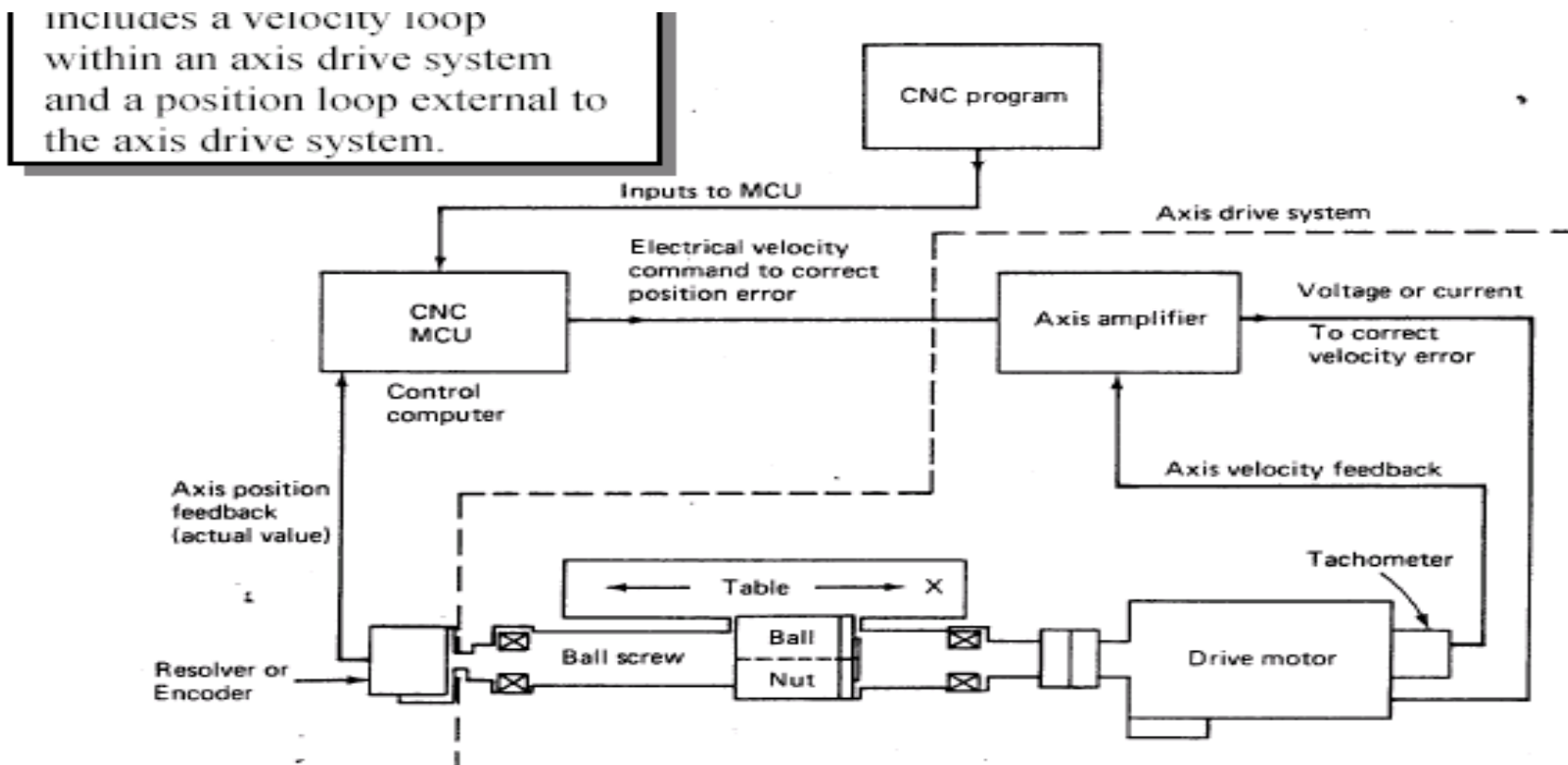
Absolute Coordinate System



Incremental Coordinate System

# Machine Control Unit

- The machine control unit (MCU) is the backbone of CNC systems. Following six functions are being done by MCU:
- Read coded instructions
- Decode coded instructions
- Implement interpolations to generate axis motion commands
- Feed axis motion commands to amplifier circuits to drive axis mechanisms
- Receive the feed back signals of position and speed for each drive axis
- Implement auxiliary control functions such as coolant ON/OFF, spindle ON/OFF, and tool change



- 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.



# **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**

# 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

## Codes 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__

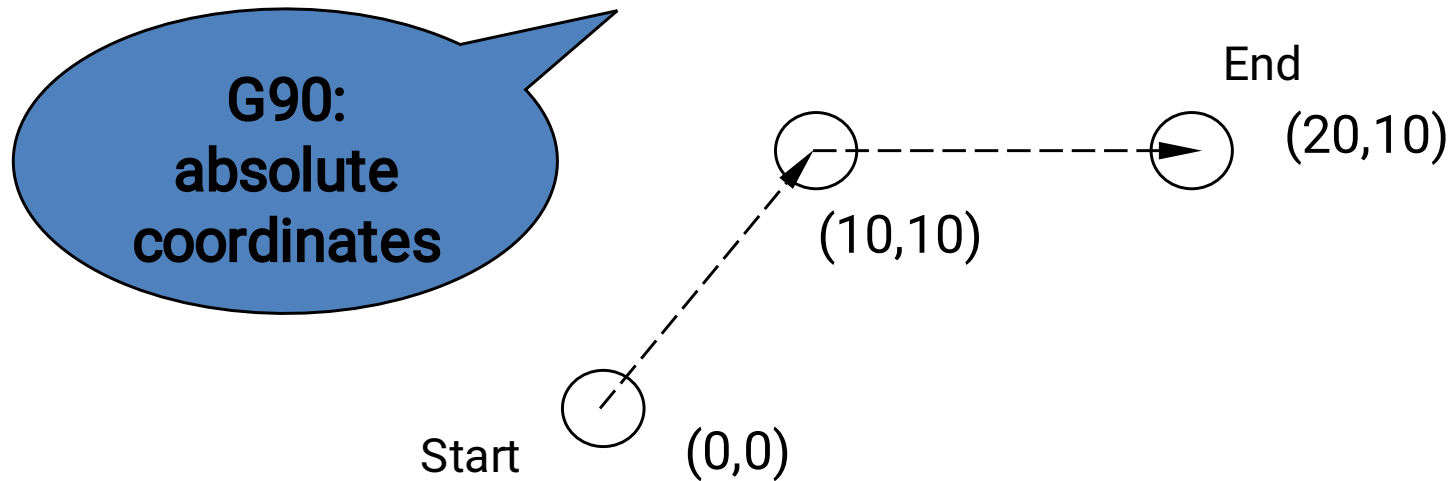
# Table of Important G codes

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__
G82	Counter Bore Cycle	N_G82Z__R__P__F__
G83	Deep Hole Drilling Cycle	N_G83Z__ R__ Q__F__
G90	Absolute positioning	
G91	Incremental positioning	
G92	Absolute preset, change the datum position, Reposition Origin Point	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

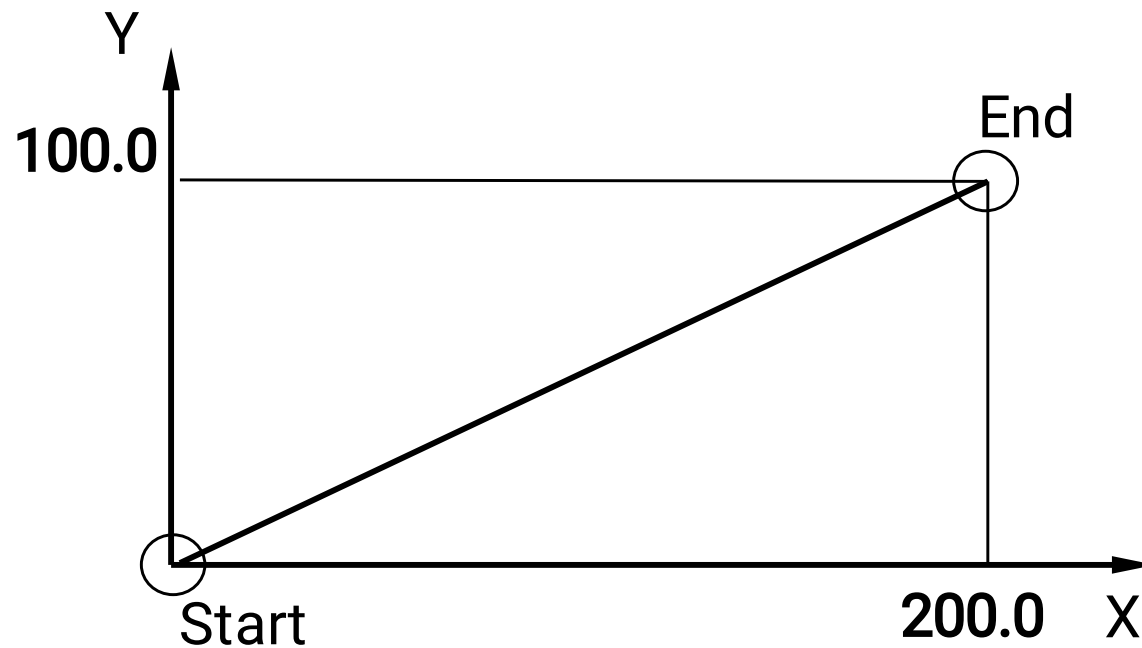


# Linear interpolation: G01

- G01:
  - linear interpolation at feed speed.

G91 G01 X200.0 Y100.0 F200.0

G91:  
incremental  
coordinates





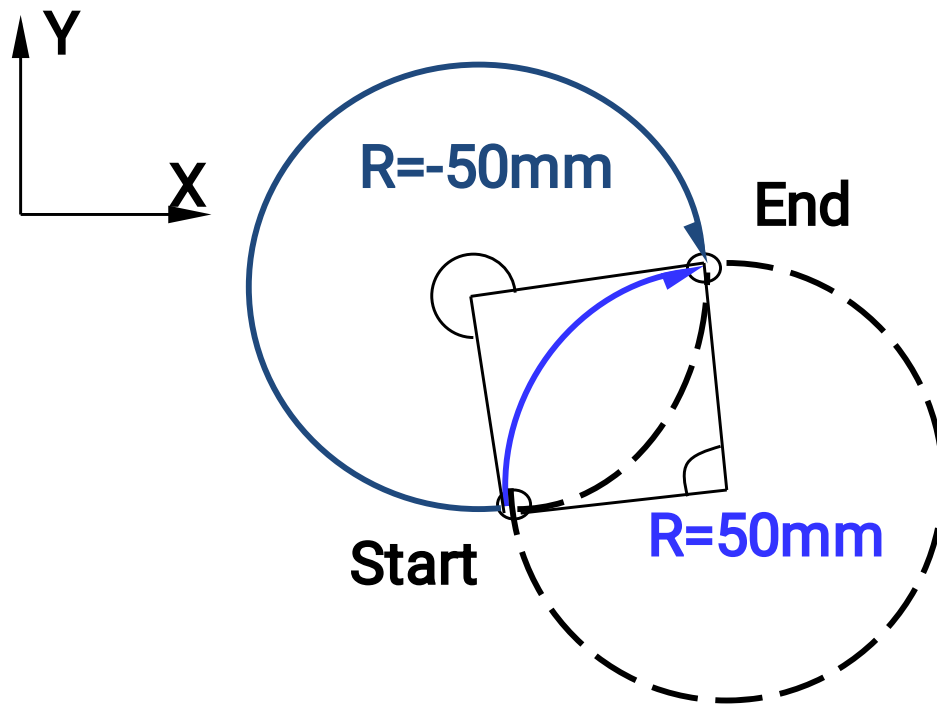
# Circular interpolation: G02, G03

- 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

$$G17 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X \_ Y \_ \left\{ \begin{array}{l} R \\ I \_ J \_ \end{array} \right\} F \_ ;$$
$$G18 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X \_ Z \_ \left\{ \begin{array}{l} R \\ I \_ K \_ \end{array} \right\} F \_ ;$$
$$G19 \left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} \underline{Y \_ Z \_} \left\{ \begin{array}{l} R \\ \underline{J \_ K \_} \end{array} \right\} F \_ ;$$

End point    Circle center, radius

# Circular interpolation: G02, G03



Specify R with  
sign before it:

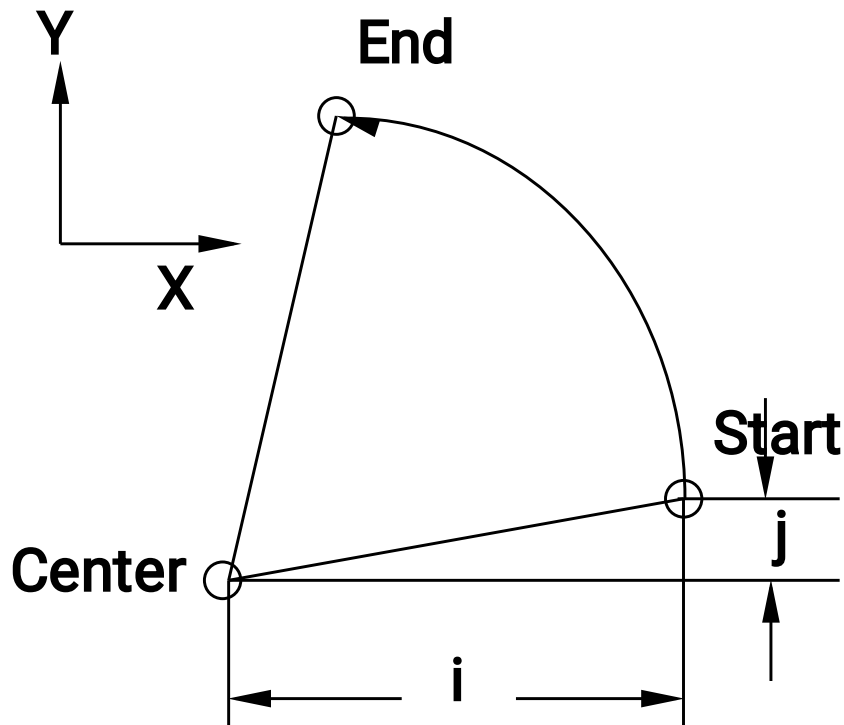
$\leq 180^\circ$  +R

$> 180^\circ$  -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

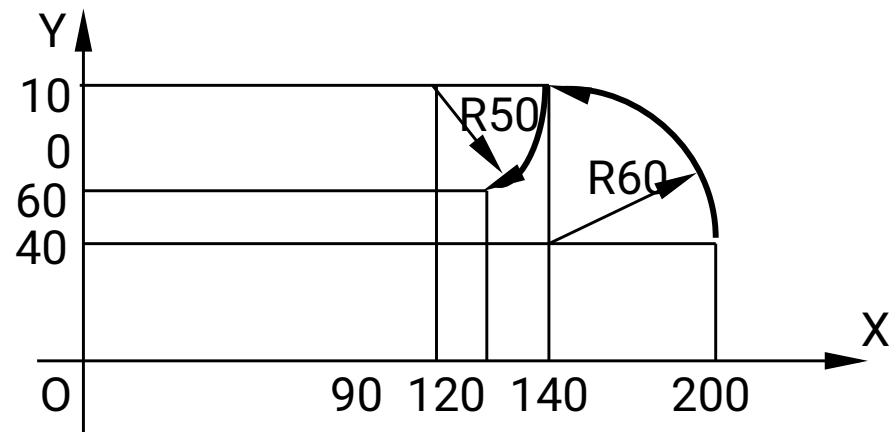
```
N0010 G92 X200.0 Y40.0 Z0 ;  
N0020 G90 G03 X140.0 Y100.0 I-60.0 F300 ;  
N0030 G02 X120.0 Y60.0 I-50.0 ;
```

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

# Tool Compensation

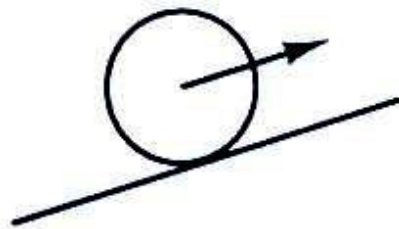
- Tool-Radius Compensation
  - Left hand G41
  - Right hand G42
  - Cancel tool-radius compensation G40
- Tool-Height Compensation
  - Positive G43
  - Negative G44
  - Cancel tool-height compensation G49

# Tool-Radius Compensation

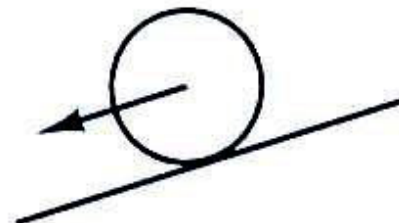
- Tool-radius compensations make it possible to program directly from the drawing, and thus eliminate the tool-offset calculation

G41 (G42) D<sub>xx</sub>

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



(a) G41



(b) G42

# Tool-Height Compensation

G43 (G44) H<sub>xx</sub>

- H<sub>xx</sub>: specified memory unit used to save height compensation of tool.
- Positive compensation (G43):  
real position = specified position + value saved in H<sub>xx</sub>
- Negative compensation (G44):  
real position = specified position - value saved in H<sub>xx</sub>



# 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 =  $-32.0 - 0.5 = -32.5$



**G91:**  
incremental  
coordinates

- 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
- M07 Coolant on -1
- M08 Coolant on-2
- M09 Coolant off
- M10 Clamps on
- M11 Clamps off
- M02 or M30 Program stop, reset to start

# Block Format

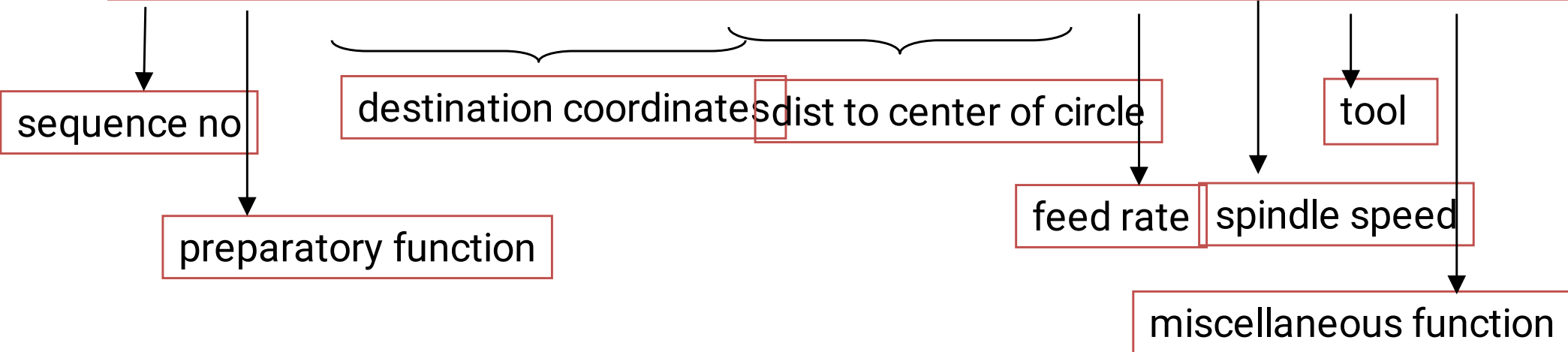
```
N135 G01 X1.0 Y1.0 Z0.125 F5
```

## 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

## Sequence and format of words:

N3 G2 X+1.4 Y+1.4 Z+1.4 I1.4 J1.4 K1.4 F3.2 S4 T4 M2



# Programming Formats

- **Word Address Format or variable block format**

This type of format uses alphabets called address, identifying the function of numerical data followed.

This format is used by most of the NC machines.

```
N20 G00 X1.200 Y0.100 F325 S1000 T03 M09 <EOB>
```

The MCU uses this alphabet for addressing a memory location in it.

# Programming Formats

- **TAB Sequential Format or interchangeable format**

Here the alphabets are replaced by a Tab code, which is inserted between two words. The MCU reads the first Tab and stores the data in the first location then the second word is recognized by reading the record Tab. If the word remains same in the succeeding block, the word need not be repeated but TAB is required to maintain the sequence of words.

>20 >00 >1.200 >0.100 >325 >1000 >03 >09

# Programming Formats

- **Fixed Block Format**

In fixed block format no letter address or Tab code are used and none of words can be omitted. The main advantage of this format is that the whole instruction block can be read at the same instant, instead of reading character by character. All instructions must be given in every block, including those instructions which remain unchanged from the preceding blocks. This format can only be used for positioning work only.

20 00 1.200 0.100 325 1000 03 09 <EOB>

## Writing a part program:

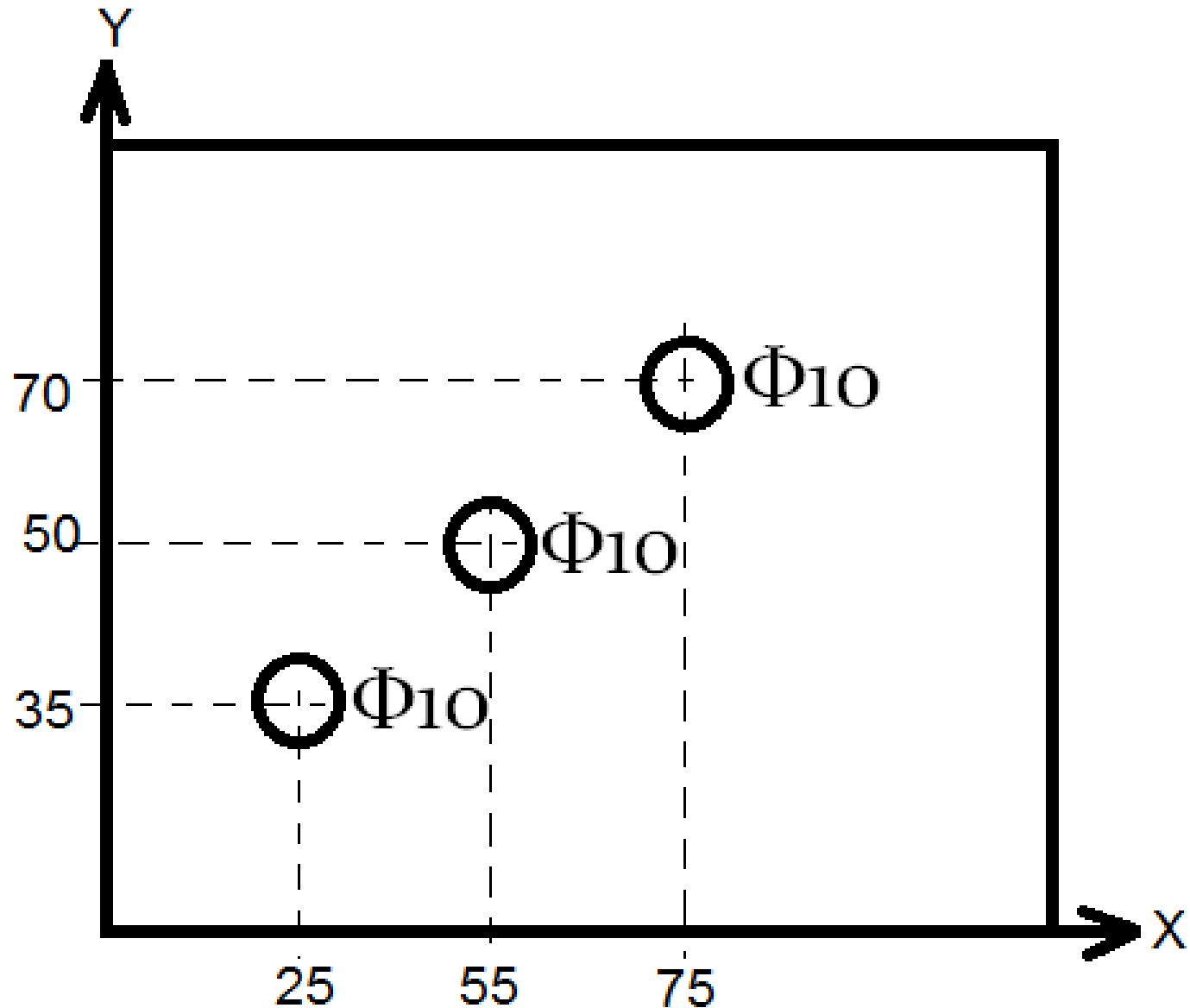
The first instruction in any part program is to inform the control system about the various set-up conditions for the machining task to be taken up and should specify the following.

1. Block Number (Sequence number - N)
2. Coordinate value – absolute (or) incremental
3. Dimensional units – inches or metric
4. Tool Number ( T – word)
5. Spindle speed (S – Word)
6. Feed function (F – Word) – (mm/min or mm/rev) Each block is terminated by typing EOB character.



# Example-1

Part program for Drilling three holes without using Can



%

01001 <EOB> (Program number)

N010 G92 X0 Y0 Z50 <EOB> (Absolute presetting at A)

N020 G90 <EOB> (Absolute programming)

N030 G71 <EOB> (Dimension in metric unit)

N040 T01 M06 <EOB> (Tool change, M06-Tool Change)

N050 S500 M03 M07 M10 <EOB> (Speed 500 rpm,

M03 – Spindle on, CW

M04 – Spindle on, CCW

M05 – Spindle stop)

M07 – Coolant 1 on M10- Clamp on

N060 G49 <EOB> (Cancel tool-height compensation)

N070 G43 H25.00<EOB> (Positive compensation (G43)

real position = specified position + value saved in H<sub>xx</sub>

N075 G00 X25.00 Y35.0 Z2.0 <EOB> (Rapid travel X25.0, Y35.0, Z27.0 real position = 2 + 25)

N080 G01 Z-18.0 F125 <EOB> (X and Y not changing no need so specify, Z = -15 (thickness of plate) - 3 (Breakthrough distance or cone height) G01-linear interpolation cutting from X25.00 Y35.0 Z2.0 to X25.00 Y35.0 Z-18.0)

N090 G00 Z2.0 <EOB> (Rapid return to X25.00 Y35.0 Z2.0)

N100 X55.0 Y50.0 <EOB> (Rapid travel to X55.00 Y50.0 Z2.0)

N110 G01 Z-18.0 F125 <EOB> (X and Y not changing no need so specify, Z = -15 (thickness of plate) - 3 (Breakthrough distance or cone height) G01-linear interpolation cutting from X55.00 Y50.0 Z2.0 to X55.00 Y50.0 Z-18.0)

N120 G00 Z2.0<EOB>(Rapid return to X55.00 Y50.0 Z2.0)  
N130 X75.0 Y70.0<EOB>(Rapid travel to X75.00Y70.0Z2.0)  
N140 G01 Z-18.0 F125 <EOB>(X and Y not changing no need so specify, Z = -15 (thickness of plate) - 3 (Breakthrough distance or cone height) G01- linear interpolation cutting from X75.00 Y70.0 Z2.0 to X75.00 Y70.0 Z-18.0)  
N150 G00 Z2.0<EOB>(Rapid return to X75.00 Y00.0 Z2.0)  
N160 M05 M09 <EOB>(M05 – spindle off, M09 – Coolant off)  
N170 X0 Y0 Z50<EOB> (Rapid travel to X75.00 Y00.0 Z2.0)  
N180 M11 <EOB> (Unclamp)  
N190 M30 <EOB> (Program Stop)

## Format of Canned cycle

N\_\_G81 X\_\_Y\_\_Z\_\_R\_\_F\_\_

R is Position of the clearance plane (same as the z- axis position of the clearance plane)

F is the feed rate.

G80 is cancel Canned cycle

# ASSIGNMENT

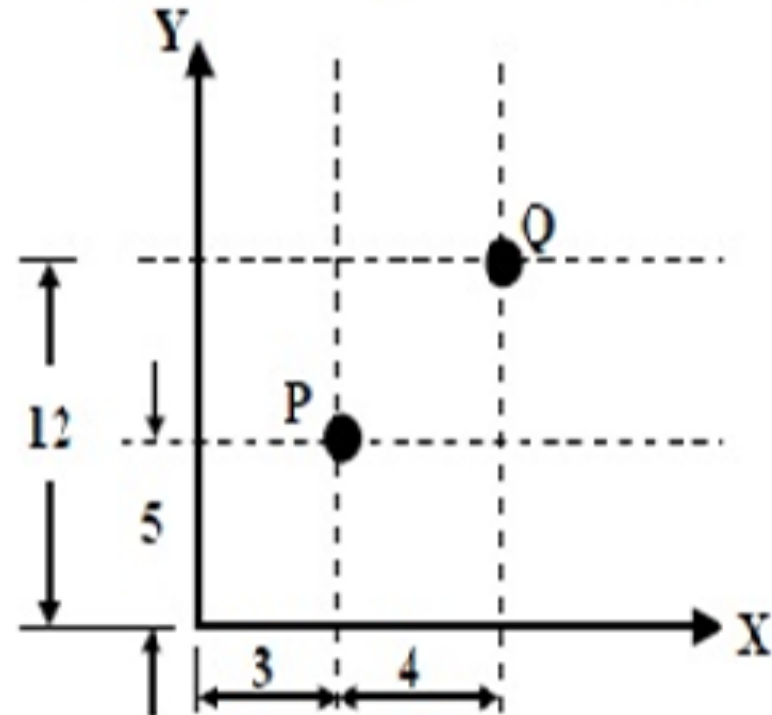
**Q-1. NC contouring is an example of**

- (a) Continuous path positioning
- (b) Point-to-point positioning
- (c) Absolute positioning
- (d) Incremental positioning

## Question-2

A drill is positioned at point P and it has to proceed to point Q. The coordinates of point Q in the incremental system of defining position of a point in CNC part program will be

- a) (3,12)
- b) (5,7)
- c) (7,12)
- d) (4,7)



# Question-3

For the CNC part programming, match Group A with Group B:

Group A	Group B
P: circular interpolation, counter clock wise	I: G02
Q: dwell	II: G03
R: circular interpolation, clock wise	III: G04
S: point to point counterling	IV: G00

**PQRS PQRS**

(a) II III I IV (b) I III III IV

(c) I IV III III (d) III III IV



## Question-4

The interpolator in a CNC machine controls

- (a) Spindle Speed
- (b) Coolant flow
- (c) Feed rate
- (d) Tool change

# Question-5

**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

# Question-6

**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

## Question-7

**The function of CNC machine controller is to**

- a) Control spindle speed
- b) Coordinate feed rates of axes
- c) Control tool rapid approach speed
- d) Perform miscellaneous(M) functions (tool change, coolant control etc.)

# Question-8

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

**THANK YOU**



Govt. of Bihar

Department of Science & Technology  
Government Polytechnic Vaishali

*PRODUCTION PROCESS*  
*Semester-IV (Mechanical Engineering)*

Unit -2  
**Drilling**

*by*

**Prof. Jitendra Kumar**  
Lecturer ,Department of Mechanical  
Engineering

G. P. Vaishali Bihar-844118

# CONTENTS

- POWERPOINT PRESENTATION
- NOTES
- EXAMPLES
- QUESTION BANK



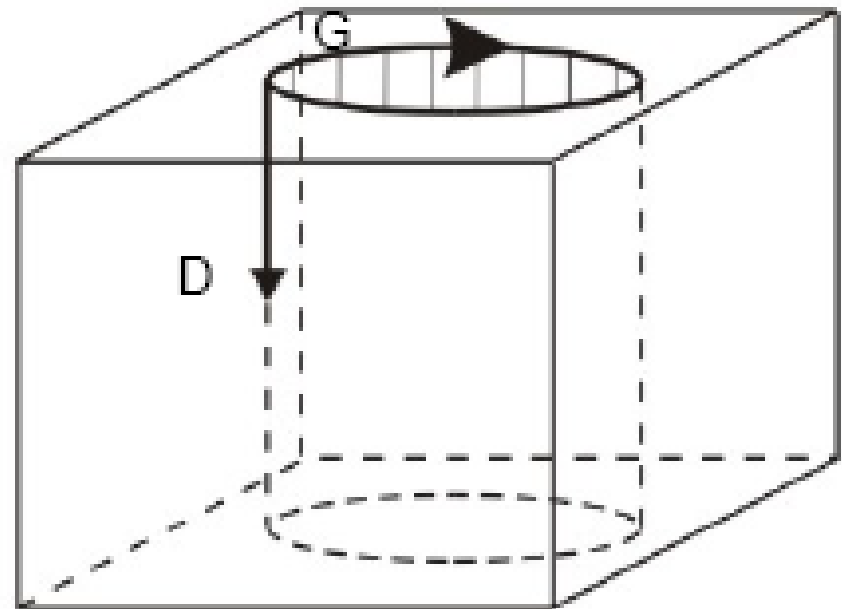
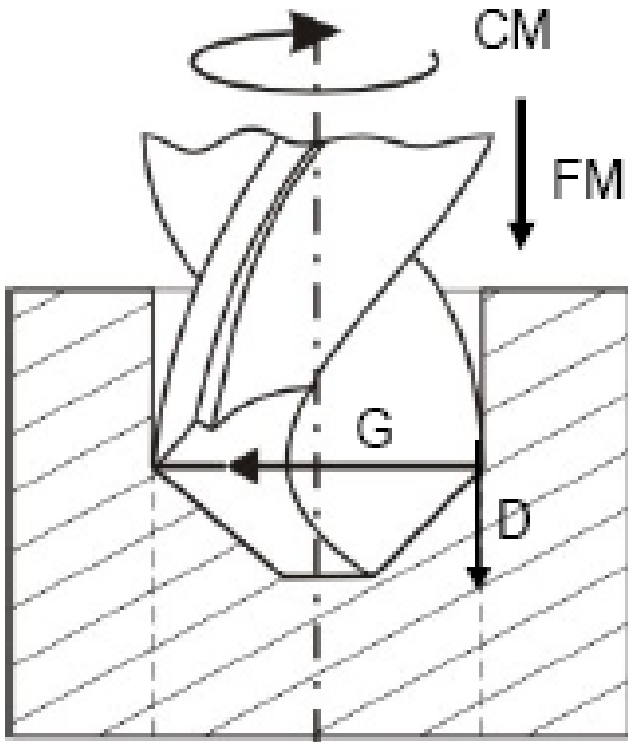
# Learning Objectives

Students will be able to:

- Understand drilling process
- Types of Drilling process
- Drilling Terminology
- Numerical Problems

# Drilling

- Drilling is a operation that cuts cylindrical holes.



# DRILLING

INTRODUCTION: A drilling machine was primarily designed to drill a hole, but it can also perform a No. of similar operations. In a drilling machine holes may be drilled and at low cost. The hole is generated by the rotating edge of a cutting tool known as the drill which exerts large force on the work clamped on the table. The cutting motion is provided by rotating the drill and feeding is done by giving rectilinear motion to the drill in the axial direction. Here the drill has two cutting edges called lips. used



# TYPES OF DRILLING

## (1) Portable drilling machine:

This type of D.M. can be operated with ease anywhere in the work shop and is used for drilling holes in work pieces in any position which cannot be drilled in a standard D.M. The entire D.M. including the motor is compact and small in size. The max. size of the drill that can accommodate is not more than 12 to 18 mm.



(2) Sensitive D.M. It is a small machine designed for drilling small holes at high speed in light and small jobs. The base of the machine may be mounted on a bench or on the floor.



(3) Upright D.M.: This is designed for handling medium sized W.P. It is similar to a S.D.M. but is heavier and larger than S.D.M. and is powered with feed arrangement.

a) Round Column Section (or) Pillar D.M  
It consists of round column and a round table. The table can be moved up and down on the column for accommodating heights. The

W.P. of different table may be rotated about its own centre. The max. size of the hole that can be drilled is not more than 50mm.



(4) Radial D.M.: It is intended for drilling medium large and heavy W.P. It consists of a heavy, round vertical column mounted on a large base. The column supports a radial arm which can be raised and lowered to accommodate work pieces of difference heights. The arm may be swung around to any position over the work bed. The drill head containing mechanism for rotating and feeding the drill is mounted on the radial arm and can be moved horizontally on the guide ways and clamped at any desired position. This can be further classified as ☒

(a) Plain RDM:- It has the movements explained above





(5) Gang D.M.: When a No. of single spindle D.M. columns are placed side by side on a common base have a worktable, in a G.D.M. 4 to 6 spindles may be mounted side by side. The speed and feed of spindles are controlled independently.



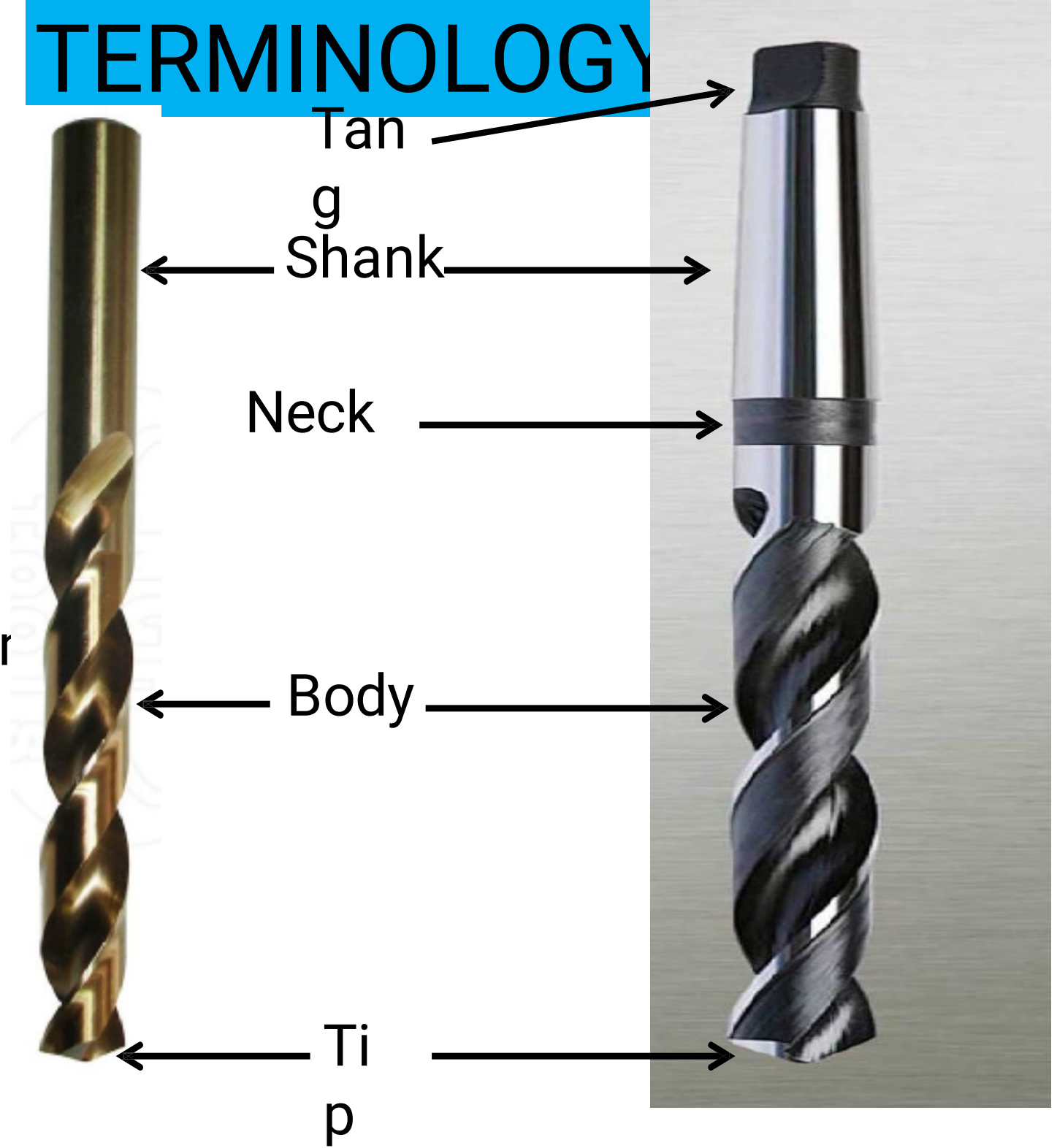
# Specifications

1. Max. size of drill that the machine can operate,
2. Max. spindle travel
3. Table diameter / size
4. Morse taper No. of the drill spindle
5. No. of spindle speeds and feeds available.
6. Floor space required
7. Net wt. of the machine

# TERMINOLOGY

Drills are manufactured as:

1. Straight shank drills (up to  $\phi 13.5$  mm)
2. Taper shank Drills ( $\phi 14.0$  mm onwards)





Drilling Machine Spindle



Drift

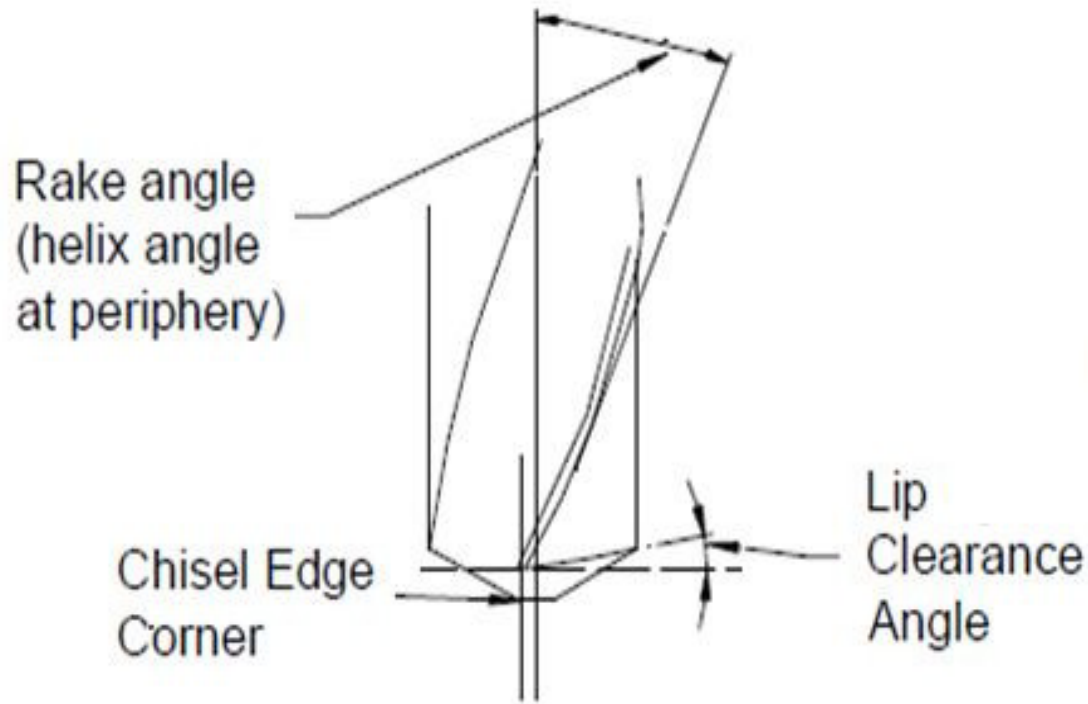


Drill Chuck with Chuck key  
*Morse taper is provided on all drilling accessories and inside drilling machine spindle*

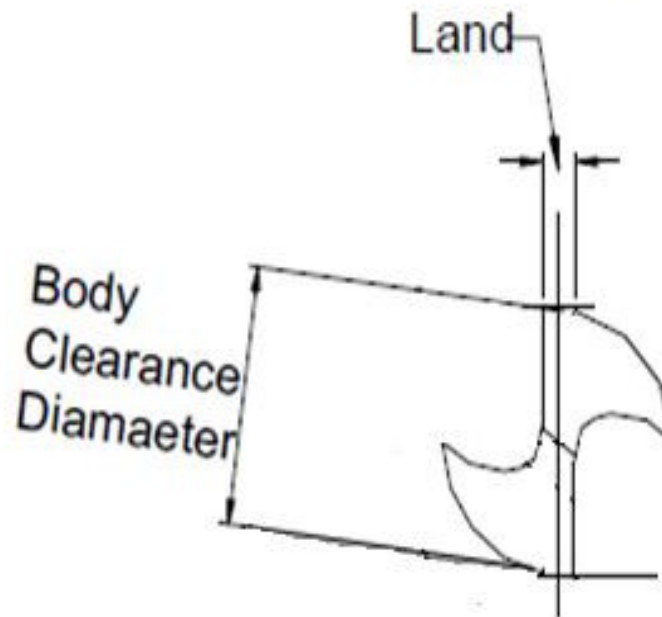
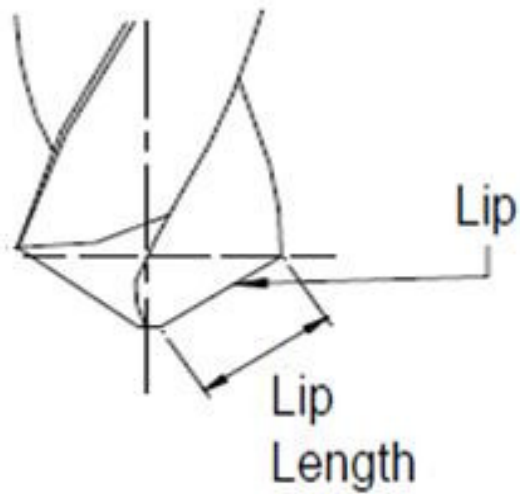
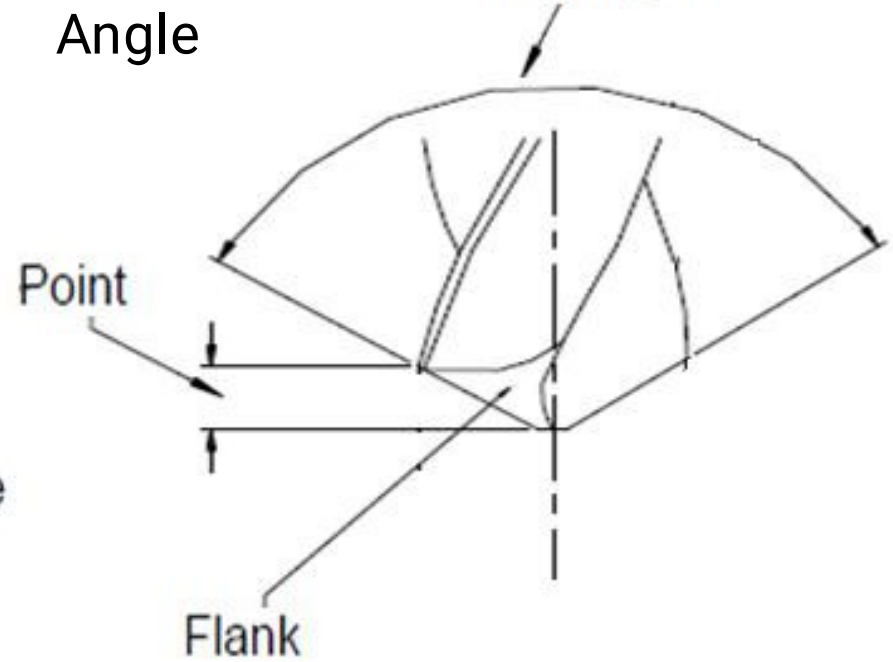


Sleeve

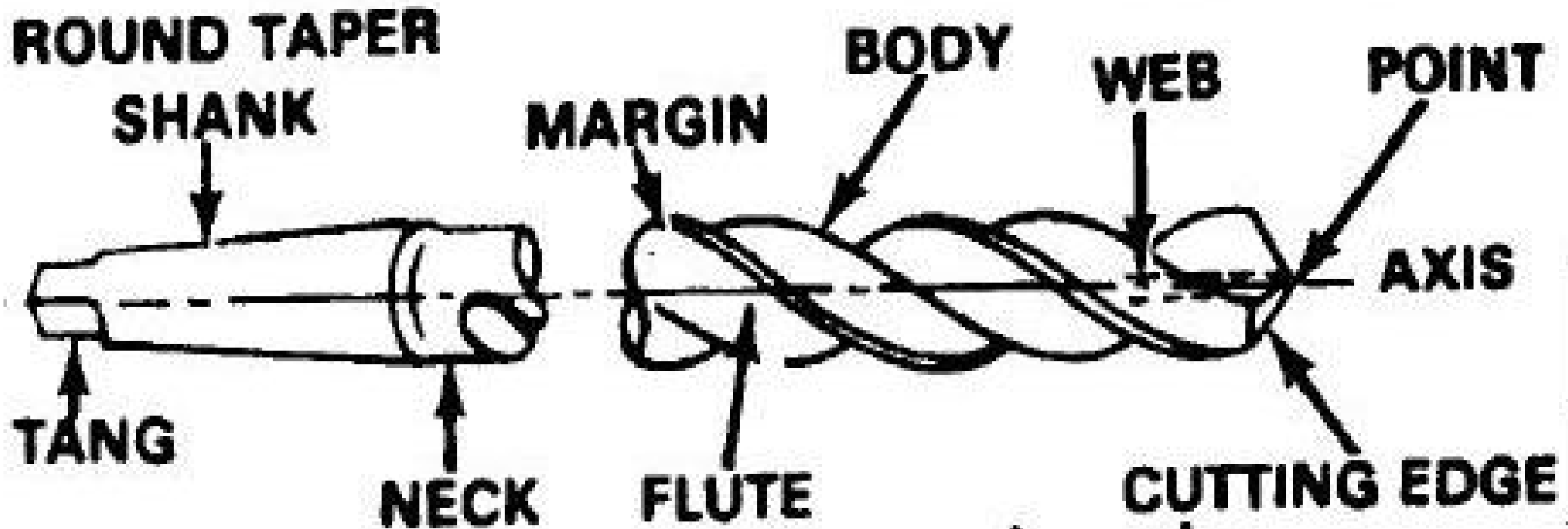
# DRILL GEOMETRY



Lip angle/ Tip Angle/ Point  
Angle



# Drill



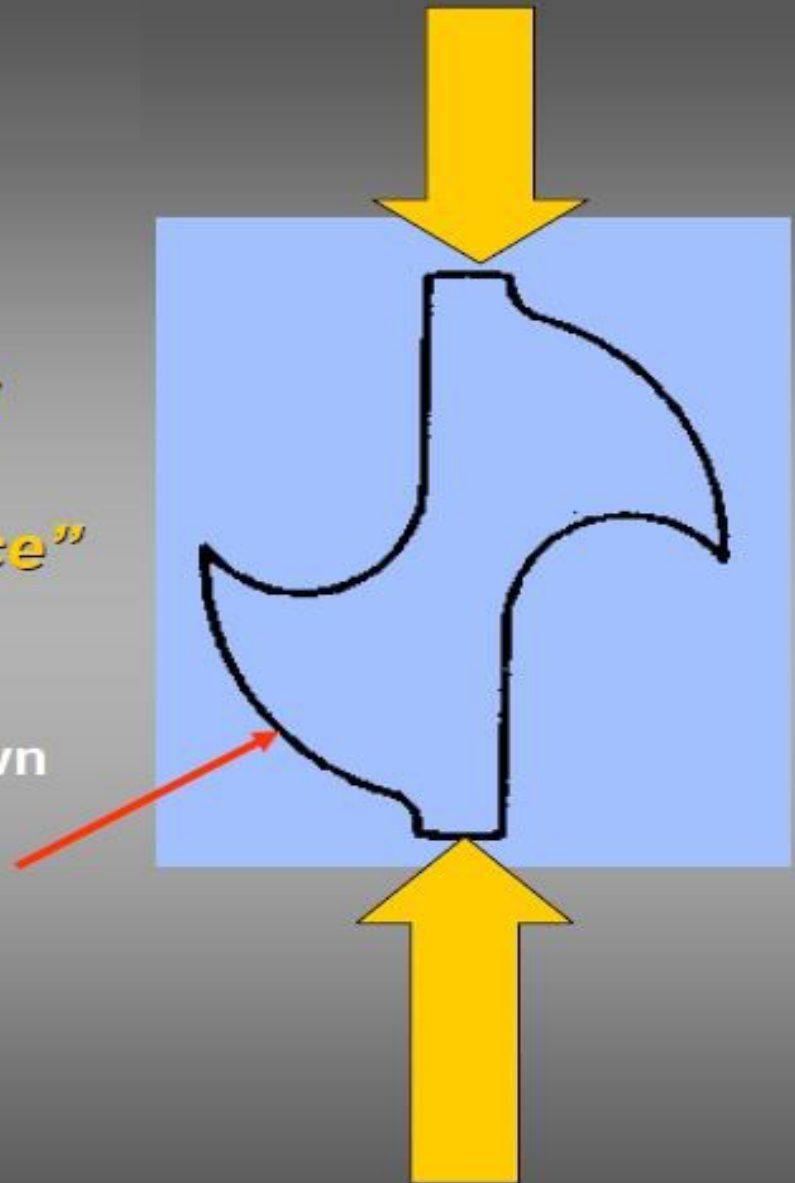
Land: Land maintains the alignment of the drill so that hole is straight and to the right size.

- Margins:

*“The cylindrical portion of the land that is not cut away to provide clearance”*

- The balance of the land is reduced in diameter, known as “cleared diameter” or “body clearance”

- **Body clearance prevents excessive rubbing and friction**



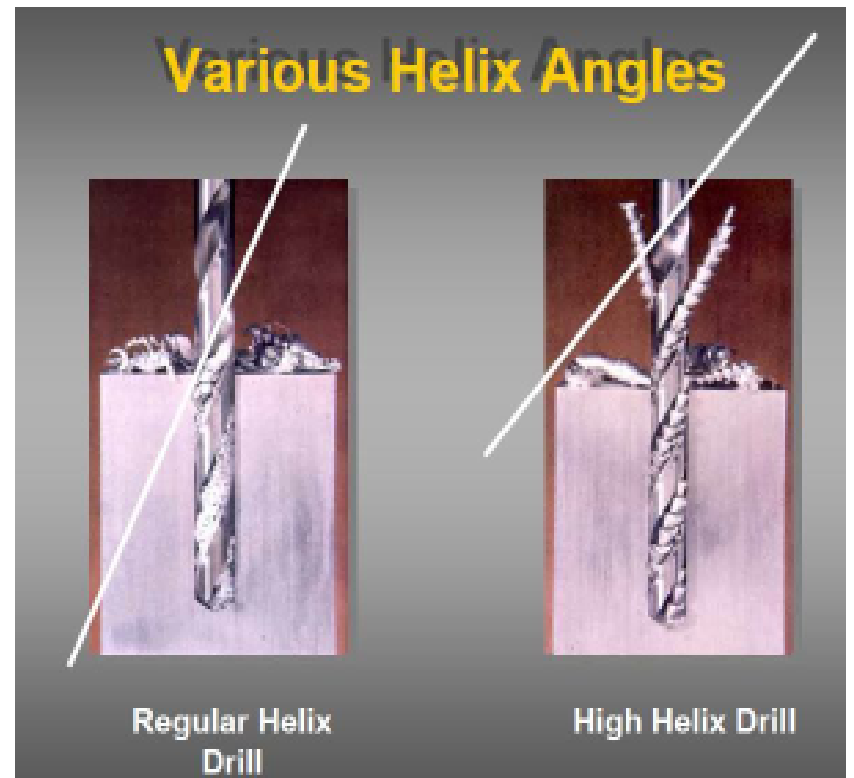
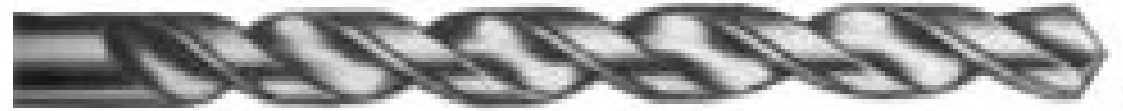
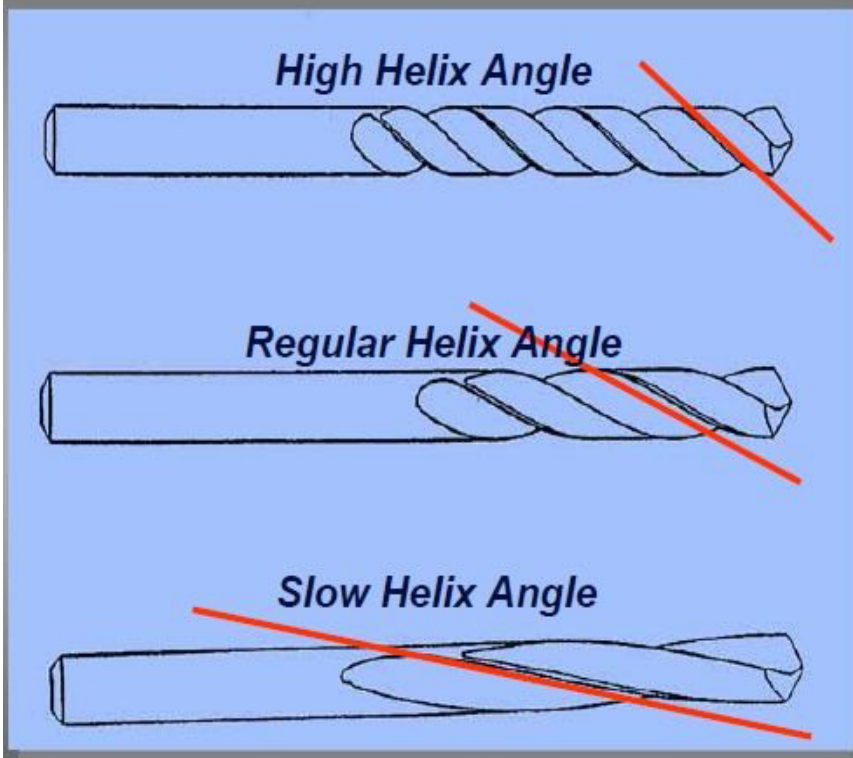
Helix angle: Angle formed b/w a plane containing drill axis and the leading edge of land. Based on the value of the angle the drills can be classified as

1) Slow spiral series:  $12^{\circ}$  to  $22^{\circ}$  - Used for brass, bronze, CI that produce broken chips. They provide less lifting power, but are stronger, used for shallow holes. Also used in horizontal applications where drill is not rotating.

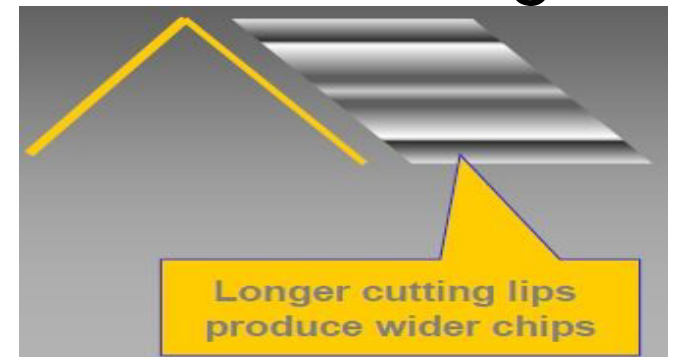
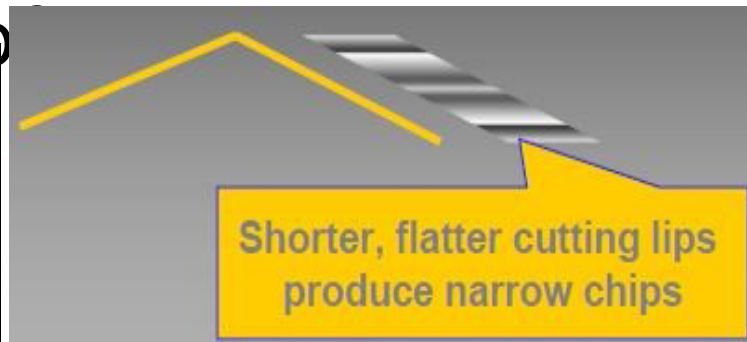
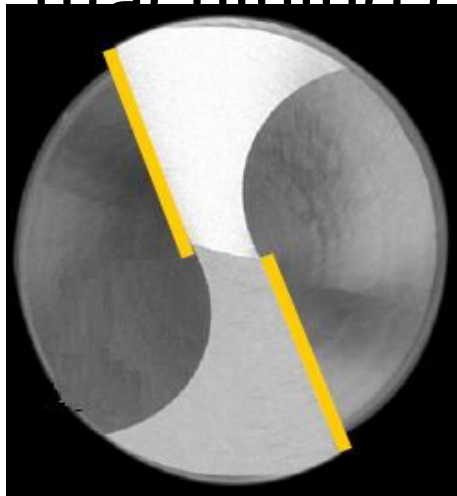
2) Regular spiral series:  $28^{\circ}$  to  $32^{\circ}$  - most widely used

3) Fast/High spiral series:  $34^{\circ}$  to  $38^{\circ}$  - Used for softer ferrous and non-ferrous materials producing long string like chips. They provide great lifting power, but are weak, used for deep holes.





**Lip angle:** Angle formed b/w the cutting edges. Smaller point angles results in lower effective rake. Effect of change in effective rake is negligible on drill performance. Less point angles generate wider and thin chips. Higher point angles generate narrow and thick chips. Increase in point angle increases the cutting efficiency of the drill because most materials are cut efficiently in the form of thick chips. Smaller the point angle, longer is the lip length and hence reduces load per unit length of the lip and helps in resisting the wear caused by abrasive action during machining.

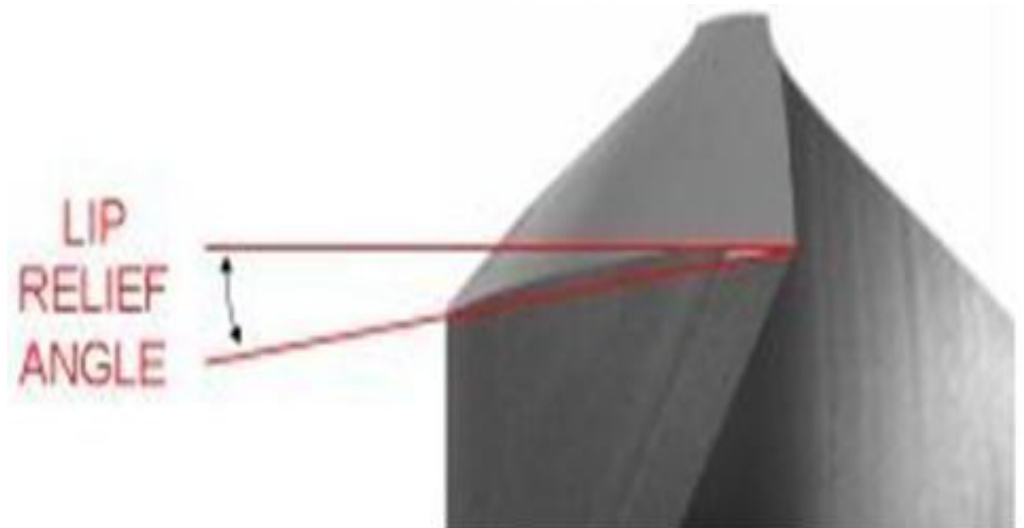
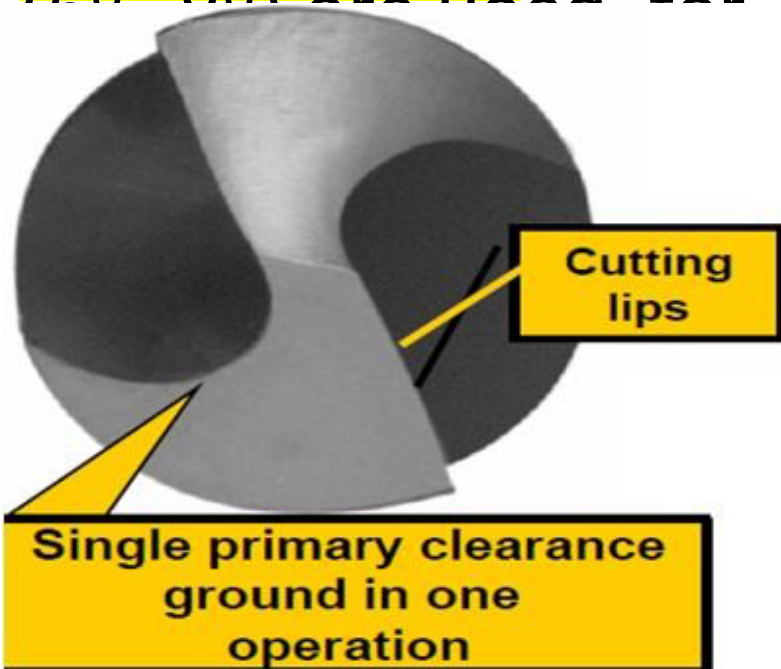
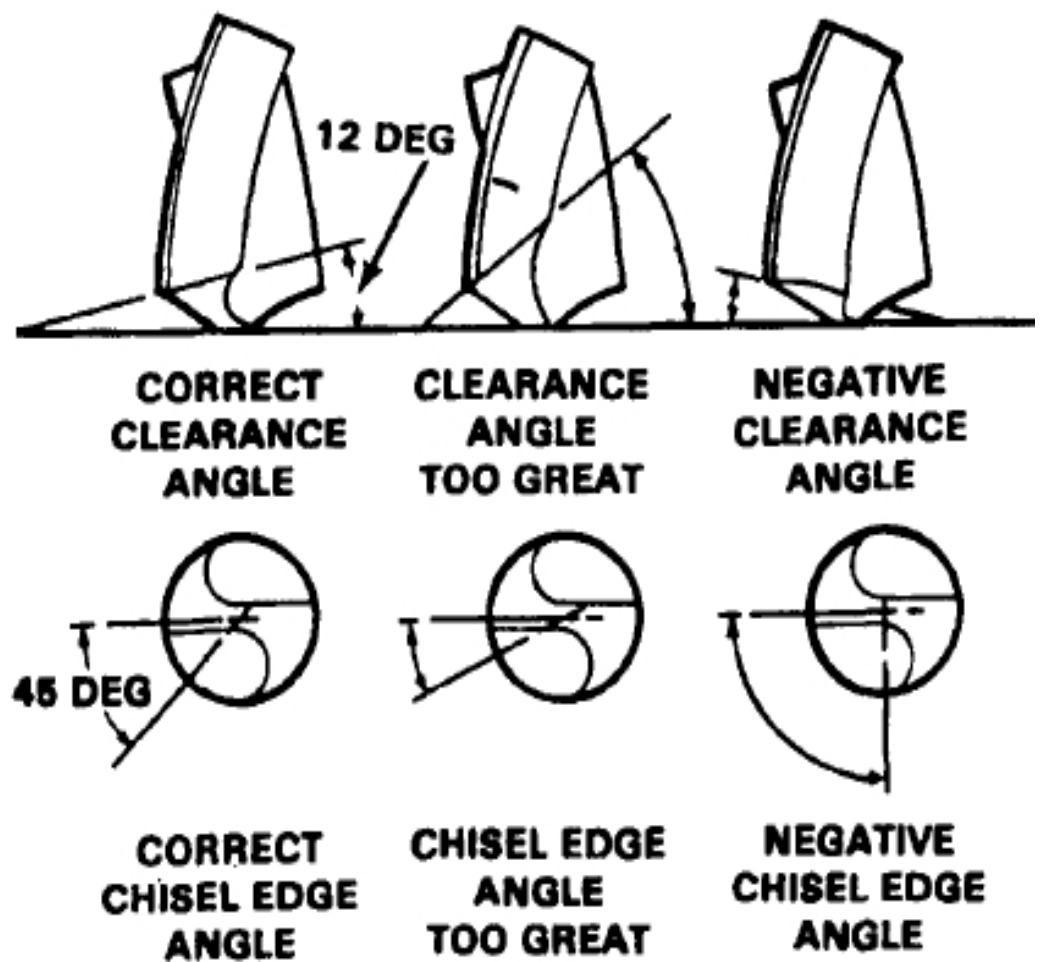


Lips are the cutting edges that extend from the centre of the drill to the outer

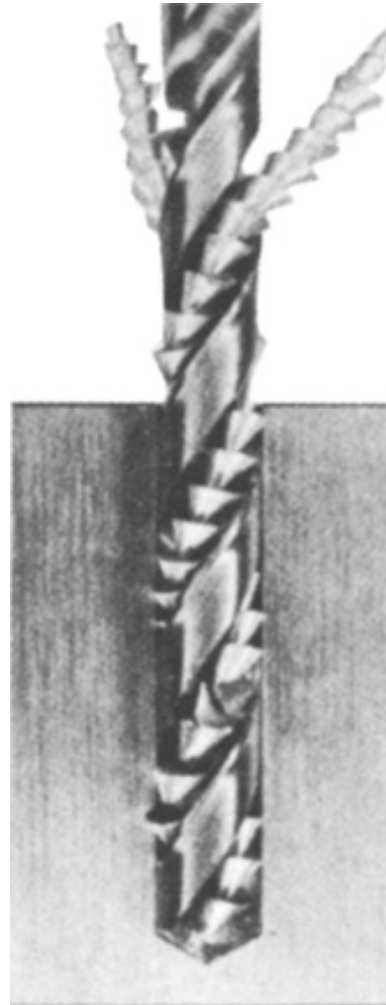
- |                     |                                  |
|---------------------|----------------------------------|
| 1) M.S.             | ⊠ $118^0$ (< 180 HB)             |
| 2) Steel            | ⊠ $118^0$ (180 - 280 HB)         |
| 3) Steel            | ⊠ $135^0 - 140^0$ (280 – 380 HB) |
| 4) Grey C.I.        | ⊠ $90^0$ (< 180 HB)              |
| 5) Grey C.I.        | ⊠ $118^0$ (180 - 280 HB)         |
| 6) Chilled C.<br>I. | ⊠ $135^0 - 140^0$ (> 350 HB)     |
| 7) Aluminum         | ⊠ $118^0$                        |
| 8) Copper           | ⊠ $118^0$                        |
| 9) Bronze           | ⊠ $118^0$                        |
| 10) Brass           | ⊠ $111^0$                        |

# Clearance / Lip relief

Angle: Angle formed b/w flank and a plane normal to drill axis at the tip of the drill. Large clearance angles ( $8^{\circ}$ – $12^{\circ}$ ) are used for ductile materials to compensate elastic recovery. Small angles ( $6^{\circ}$ – $9^{\circ}$ ) are used for



# Chip formation of a drill



# Cutting Speed in Drilling

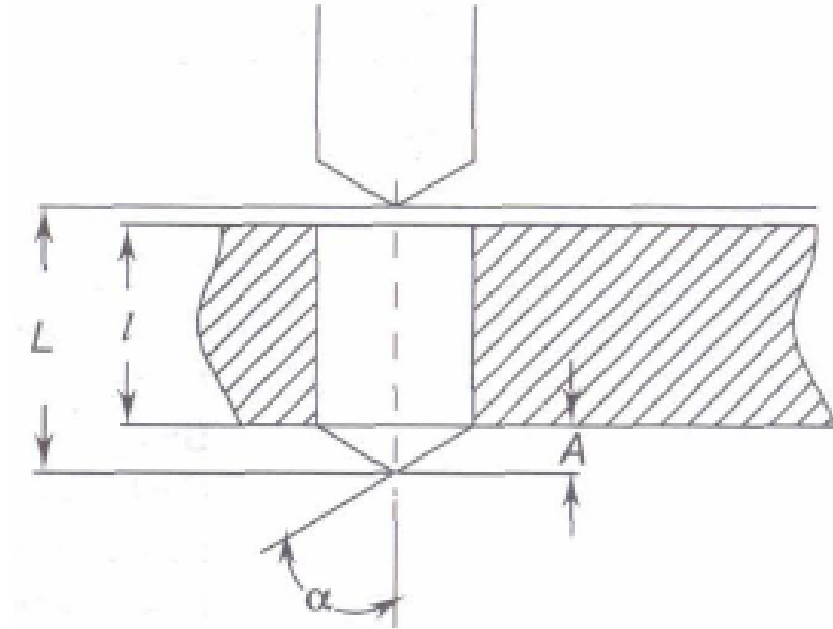
- The cutting speed in drilling is the surface speed of the twist drill.

$$V = \frac{\pi D N}{1000} \quad m / \text{min}$$

# Drilling Time

- Time for drilling the hole

$$T = \frac{L}{fN}, \text{ min}$$



# MRR in Drilling

$$MRR = \left( \frac{\pi D^2}{4} \right) fN, \quad \text{mm}^3 / \text{min}$$



# Example

A hole with 40-mm diameter and 50-mm depth is to be drilled in mild steel component. The cutting speed can be taken as 65 m/min and the feed rate as 0.25 mm/rev. Calculate the machining time and the material removal rate.

# QUESTION BANK

**The rake angle in a drill**

**(GATE-1996)**

- (a) Increases from centre to periphery
- (b) decreases from centre to periphery
- (c) Remains constant
- (d) Is irrelevant to the drilling operation

# IES - 1997

## The rake angle in a twist drill

- (a) Varies from minimum near the dead centre to a maximum value at the periphery
- (b) Is maximum at the dead centre and zero at the periphery
- (c) Is constant at every point of the cutting edge
- (d) Is a function of the size of the chisel edge.

# IES - 1992

**A drill for drilling deep holes in aluminum should have**

- (a) High helix angle
- (b) Taper shank
- (c) Small point angle
- (d) No lip

# IES - 1992

**Low helix angle drills are preferred for drilling holes in**

(a) Plastics (b) Copper

(c) Cast steel (d) Carbon steel

# GATE- 2002

The time taken to drill a hole through a 25 mm thick plate with the drill rotating at 300 r.p.m. and moving at a feed rate of 0.25 mm/revolution is

- (a) 10 sec      (b) 20 sec  
(c) 60 sec      (d) 100 sec

# IES - 2002

The arm of a radial drilling machine is being raised at a speed of 3.9 m/min by single start square threads of 6 mm pitch and 30 mm diameter. The speed of the screw

- (a) Is 650 rpm
- (b) Is 180 rpm
- (c) Is 130 rpm
- (d) Cannot be determined as the data is insufficient

# IES - 1992

**A drill for drilling deep holes in aluminum should have**

- (a) High helix angle
- (b) Taper shank
- (c) Small point angle
- (d) No lip



**THANK YOU**