# Numerical control

## HISTORICAL DEVELOPMENT

- 15th century - machining metal.
- 18th century - industrialization, production-type machine tools.
- 20th century - F.W. Taylor - Tool metal - HSS

Automated production equipment -

- Screw machines
- Transfer lines
- Assembly lines
- using cams and preset stops

Programmable automation -

- NC
- PLC
- Robots

# A Definition of NC

- Numerical Control is a system in which actions are controlled by the direct insertion of numerical data at some point.
- In other words, Programmable automation in which the mechanical actions of a 'machine tool' are controlled by a program
- or
- It is defined Method of programmable automation in which various functions of machine tools are controlled by numbers , letters and symbols.

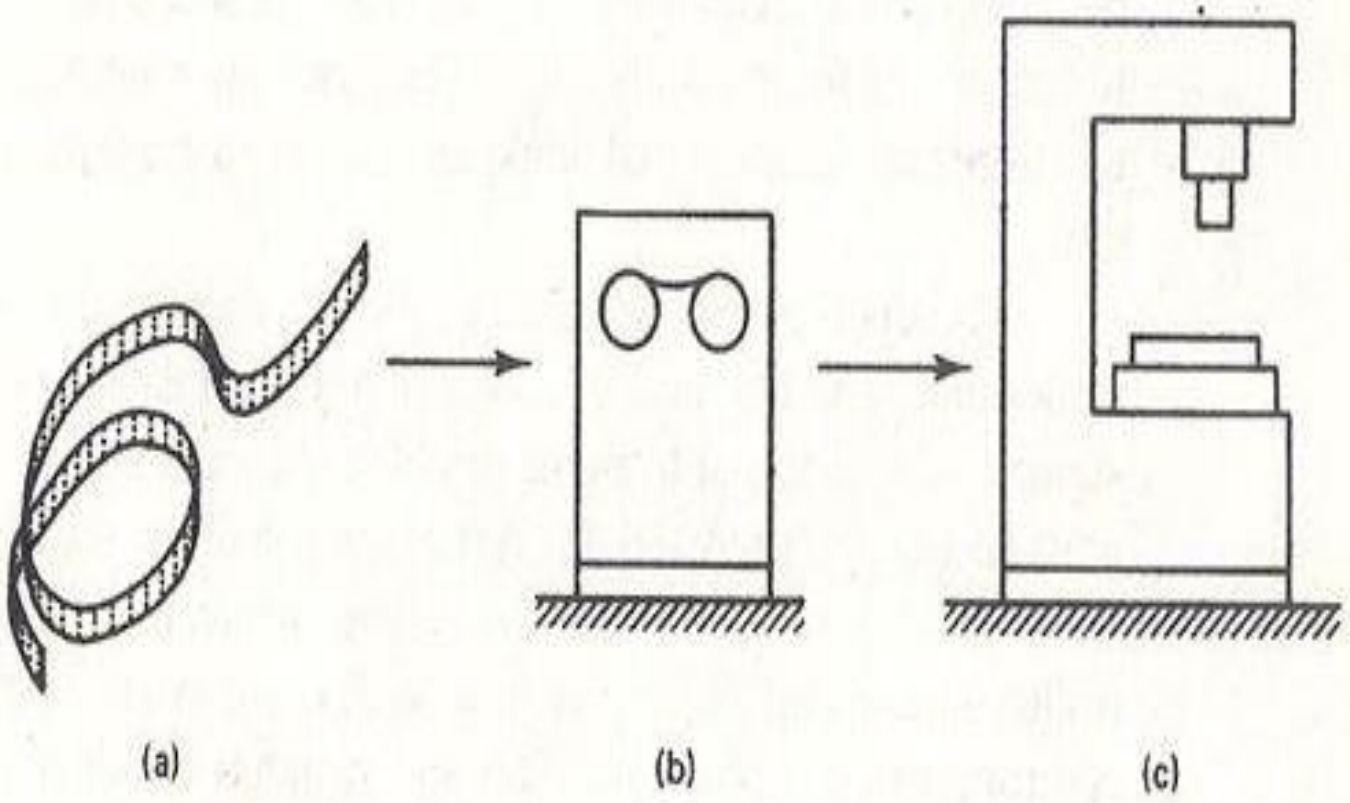



FIGURE 7.1 Three basic components of a numerical control system: (a) program of instruction; (b) controller unit; (c) machine tool.

# NUMERICAL CONTROL ELEMENTS

1.Program of instructions.

2.Machine control unit (MCU).

3.NC machine tool.

4.NC Cutting tools.

## 1.Program of Instructions:

1. The program of instructions is the detailed step - by - step of operations which are implemented by MCU .
2. The program is coded in alphanumerical form on an input medium to the MCU
3. The input medium is a punched tape or a magnetic tape .
4. Two method are used to program for NC
   I. Manual part programming
   II. Computer – aided part programming

# 2. Machine Control Unit (MCU)

- NC machine tool has a main unit, which is known as Machine Control Unit.
- It consists of some electronic hardware that reads theNC programme, interprets it and conversely translates it for mechanical actions of the machine tool.
- MCU consists of two parts : Data processing unit (DPU) and
- control loops unit (CLU).
- Function of DPU: read the decode the instructions available on the tape & to provide the decoded data to the control loops unit (CLU).
- Function of (CLU): To control the drives attached to the axes and receive the feedback signals from machine tool
- CLU also prompts a signal that the previous data segment is completed and that the DPU can read the next block of the part program.

## 3 .NC Machine Tool :

•Machine tool is the main components of a numerical control system, which executes the operations.

•It may consist of worktable, cutting tools, jigs and fixtures, motors for driving spindle and coolant and lubricating system.

•The latest development in the numerical control machine tool is the versatile machining center.

•This is a single machine capable of doing a number of operations such as milling, boring, drilling, reaming, and tapping by Automatic Tool Changer (ATC) under the control of tool selection instruction.

# Problems with Conventional NC

- Part programming mistakes
- Nonoptimal speeds and feeds
- Punched tape
- Tape reader
- Controller
- Management information

# Computer Numerical Control

- Conventional hard-wired NC controller unit replaced by computer.
- NC system that utilizes stored programs in a dedicated computer to perform some or all NC functions
- Soft-wired
- Flexibility

# CNC

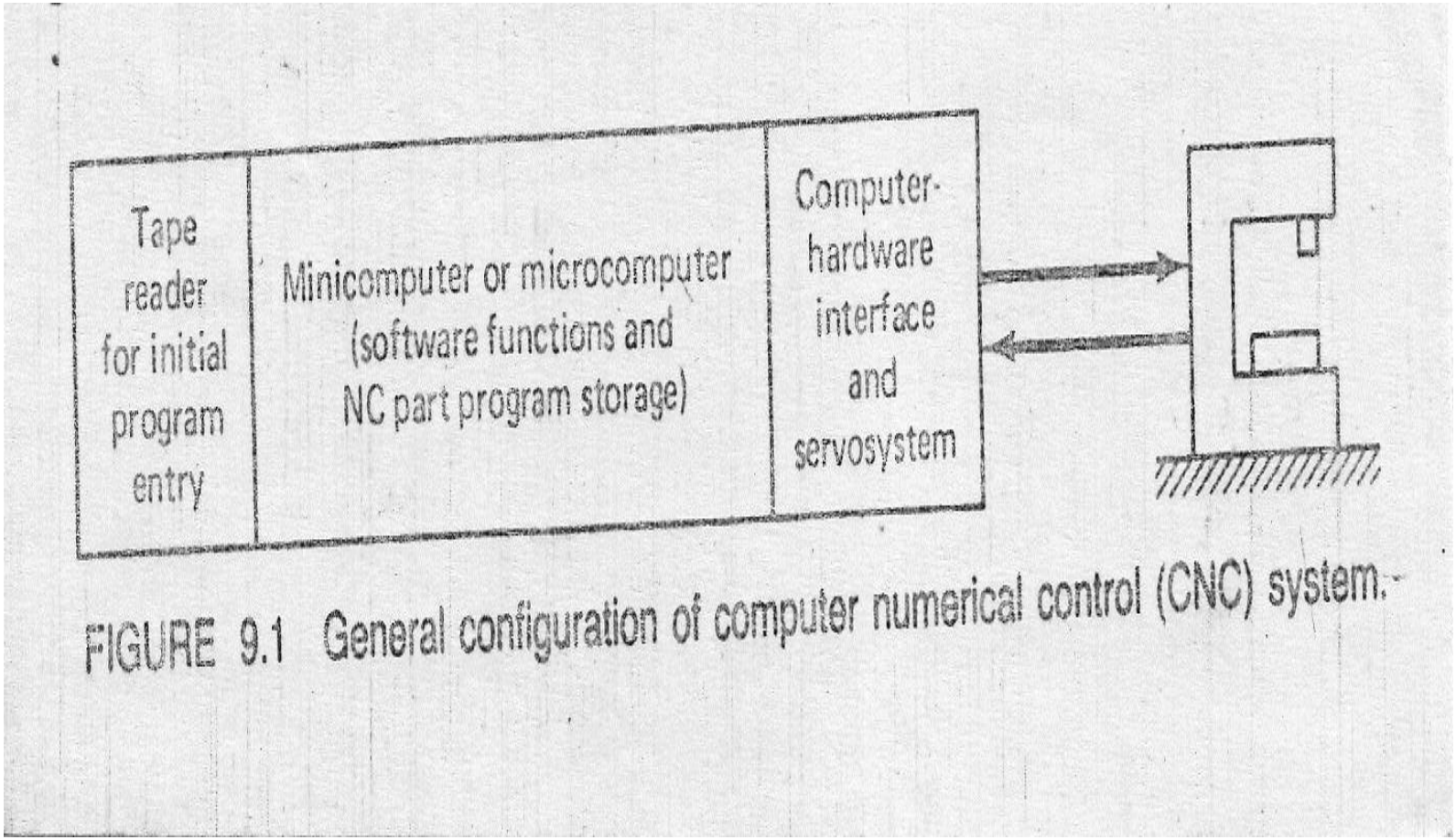



FIGURE 9.1 General configuration of computer numerical control (CNC) system.

# Functions of CNC

**1.Machine tool control**

- Hybrid CNC –Hard-wired logic circuits for functions like feed rate generation , circular interpolation etc. in addition to computer

Mass production of circuits and less expensive computer

- Straight CNC –Computer to perform all NC functions

# Hybrid CNC

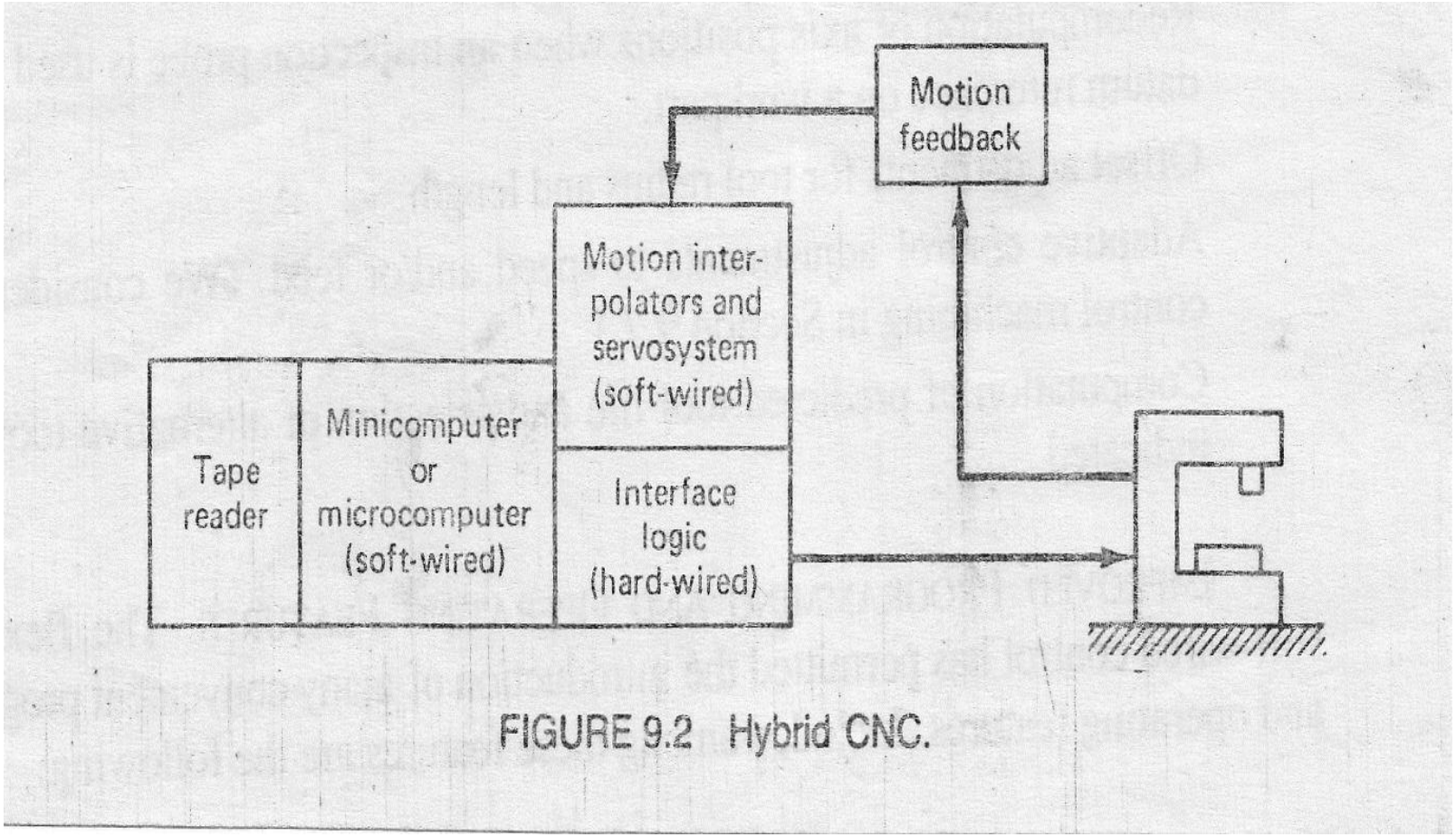



FIGURE 9.2 Hybrid CNC.

# Straight CNC

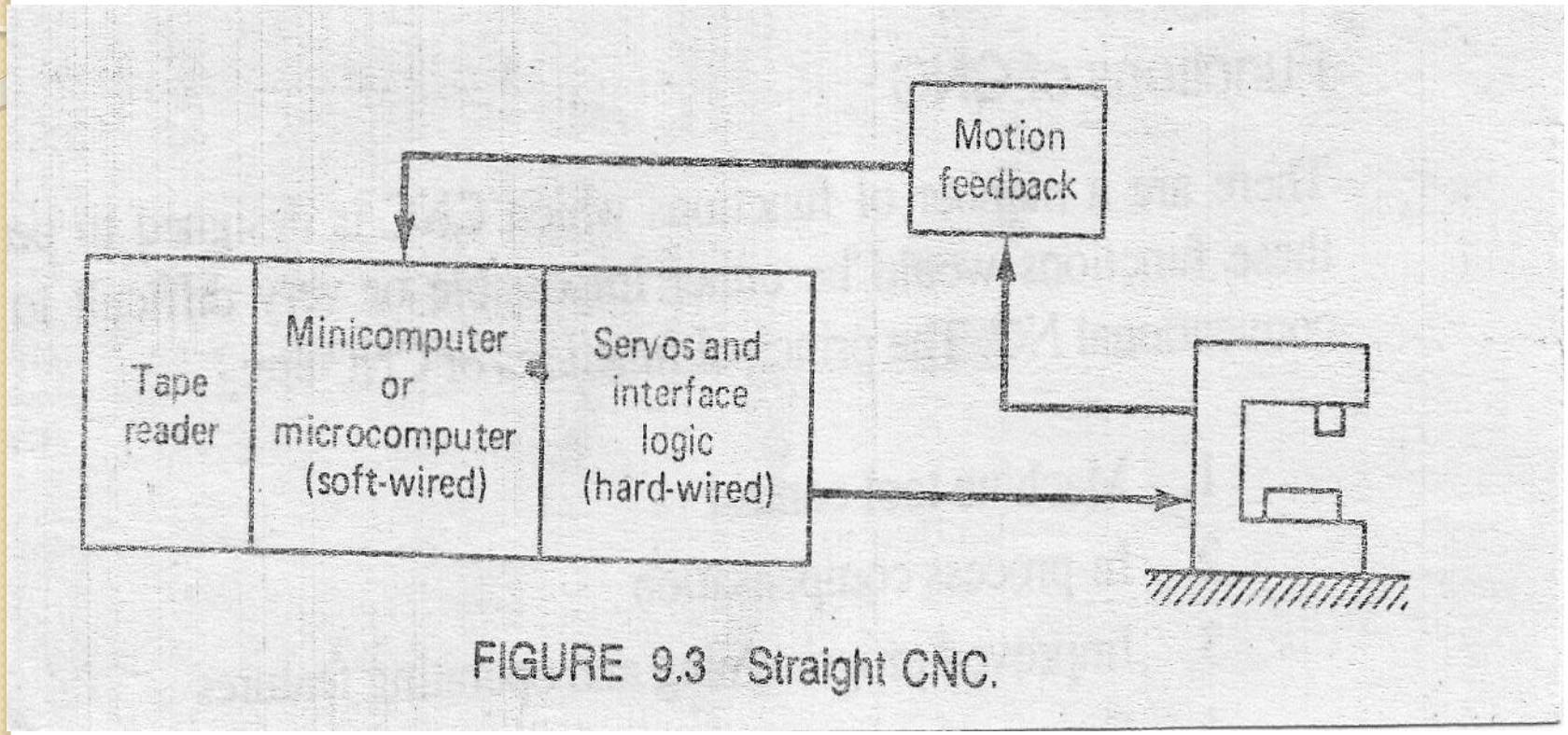



FIGURE 9.3 Straight CNC.

# Functions of CNC

**2.In-process compensation**–Dynamic correction of machine tool motion for changes or errors that occur during processing

- Adjustment of errors sensed by in-process inspection probes and gauges
- Recomputation of axis positions when an inspection probe is used to locate a datum reference on the work part
- Offset adjustments for tool radius and length
- Adaptive control adjustments to sped and feed
- Computation of predicted tool life and selection of alternate tooling when indicated.

# Functions of CNC

3. **Improved programming and operating features**

- Use of tape and tape reader only once
- On-line editing of part programs at the machine
- Special canned cycles.
- Graphic display of tool path to verify the tape
- Various types of interpolation: circular, parabolic, cubic
- Support of various units. Conversion from one unit to another unit.
- Use of specially written subroutines or macros
- Manual data input (MDI)
- Several part programs in bulk can be stored

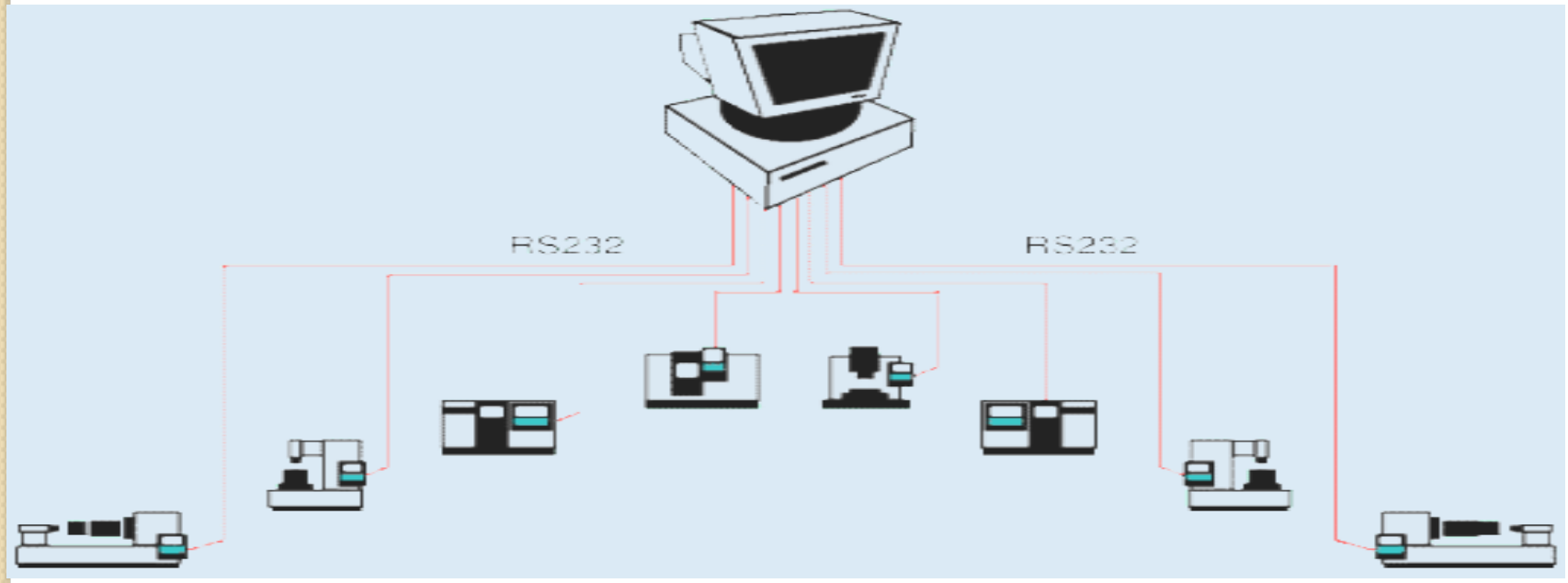
# Functions of CNC

**4.Diagnostics-**Equipped with diagnostic capability to assist in maintaining and repairing the system

- Identification of reason for downtime
- Indication of imminent failure of certain component
- Redundancy of components

# Direct Numerical Control (DNC)

A manufacturing system in which no. of machines are controlled by a computer through direct connection and in real time.



# Direct Numerical Control (DNC)

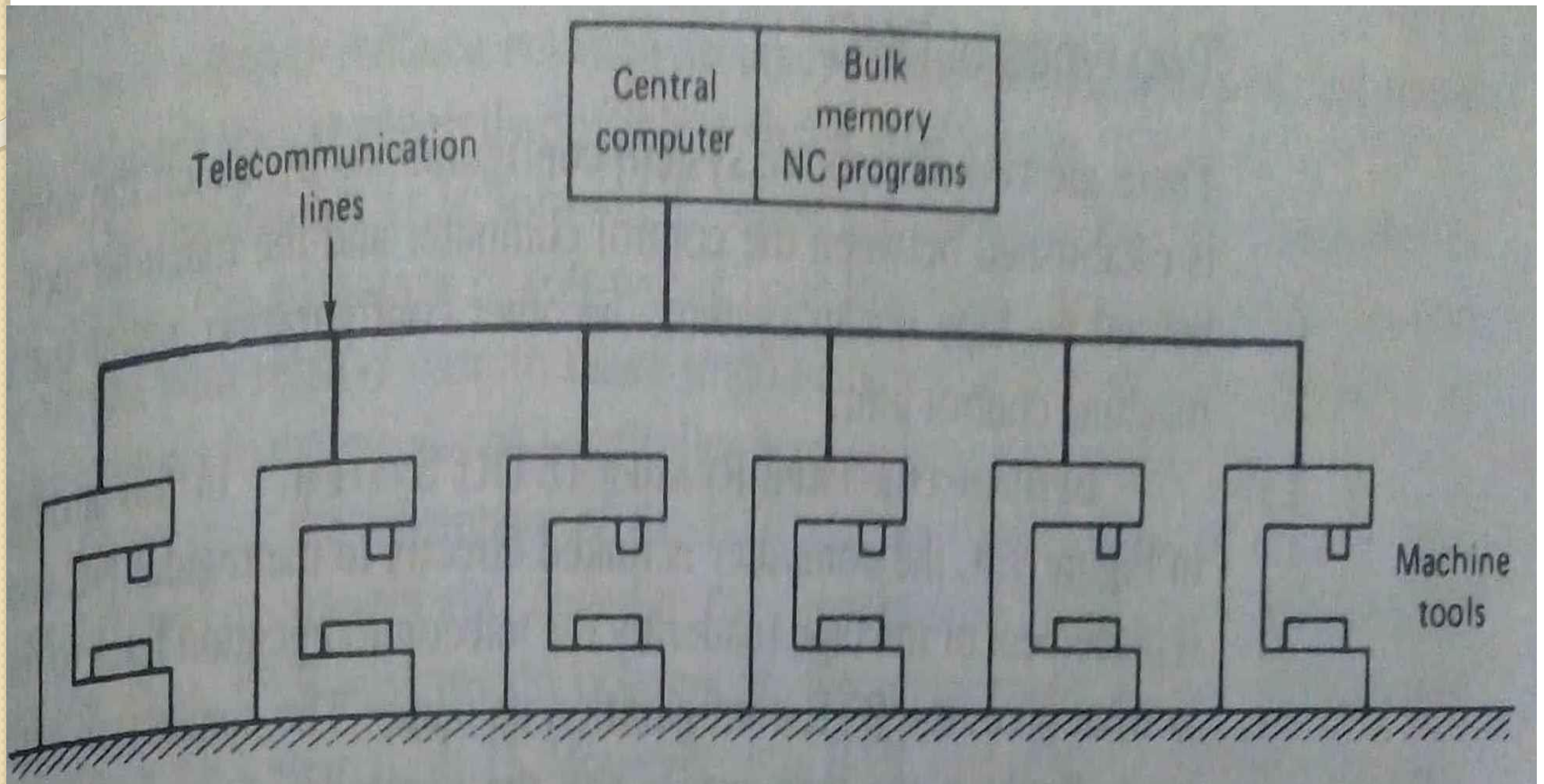



FIGURE 9.4 General configuration of direct numerical control (DNC) system.

# DNC with satellite computer

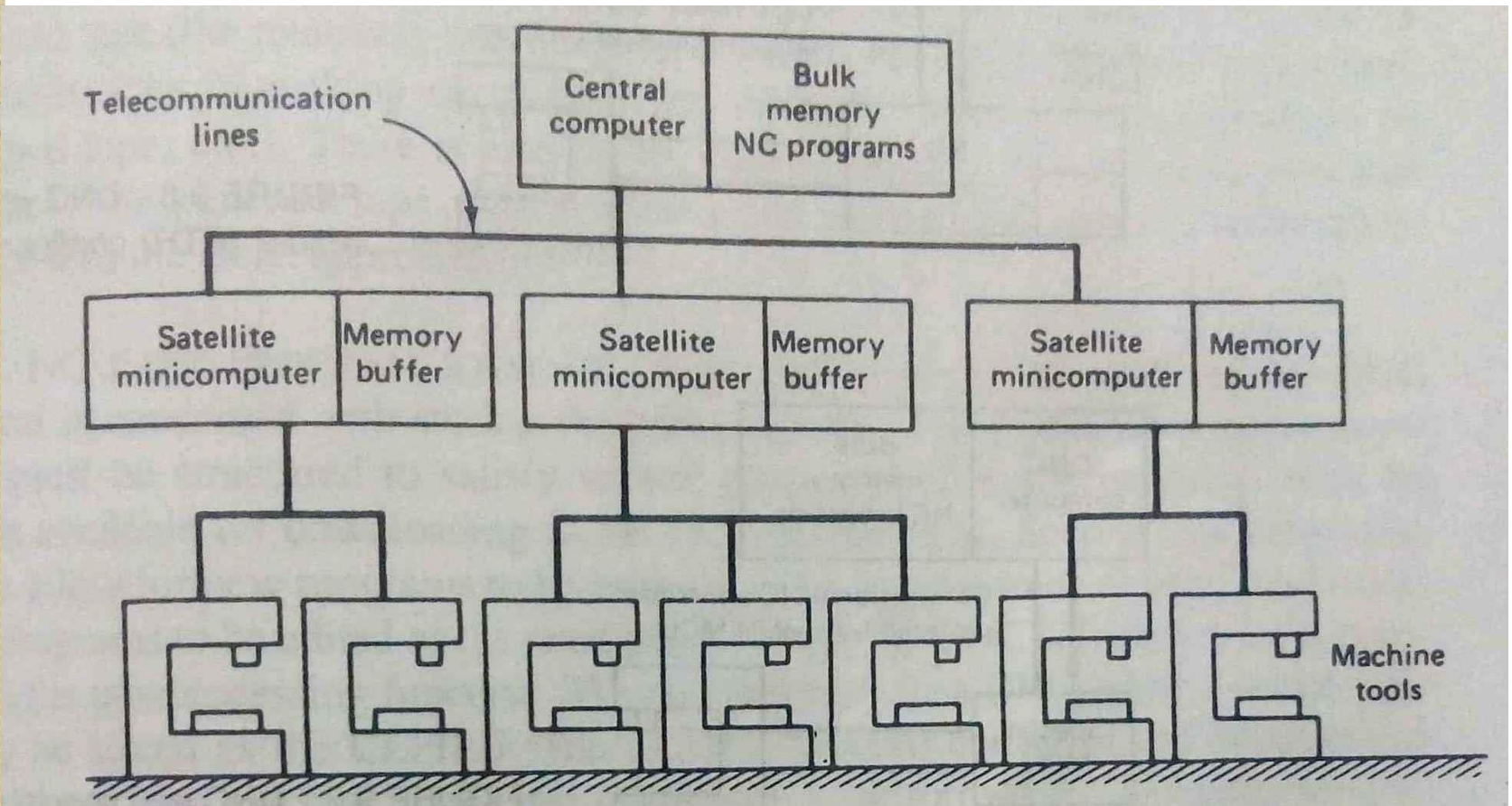
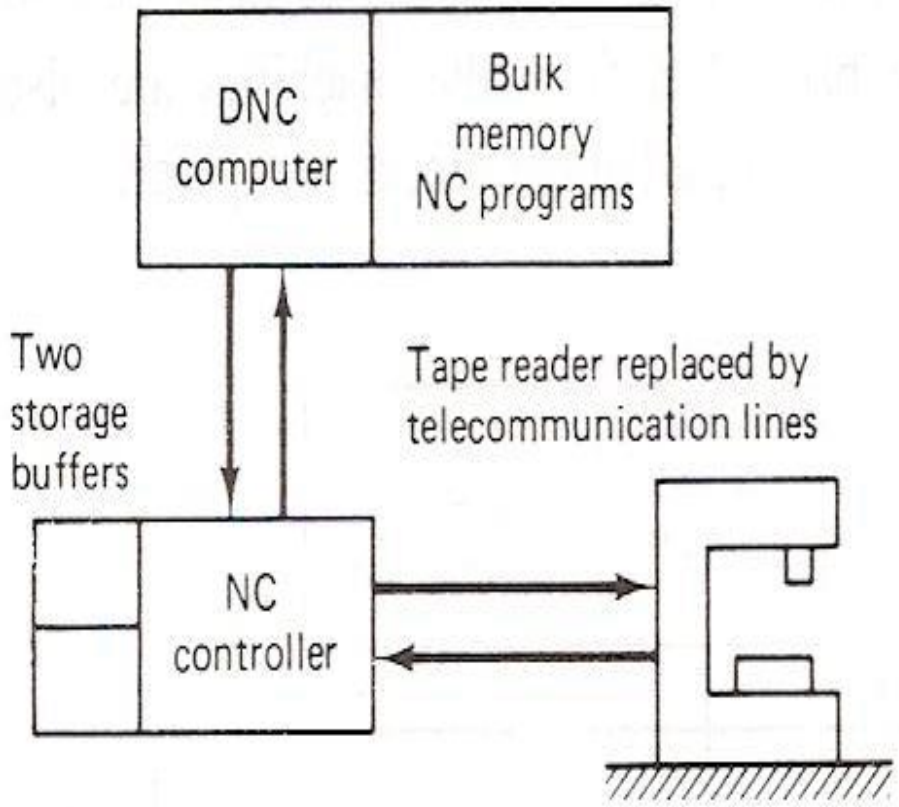


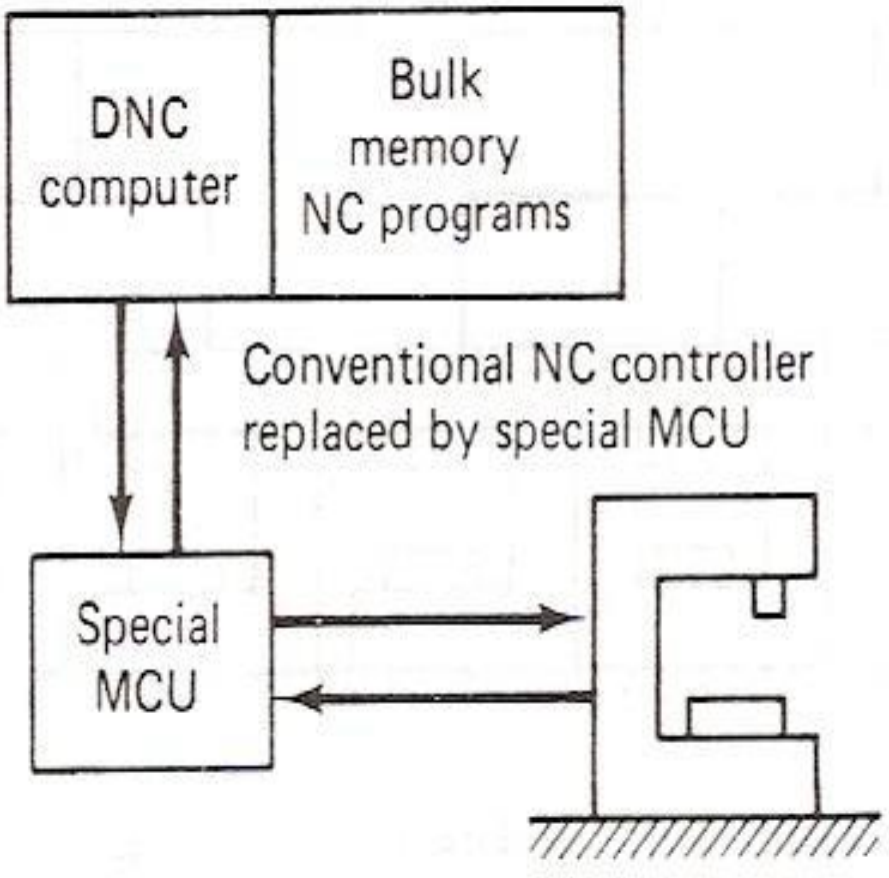

FIGURE 9.5 DNC with satellite minicomputers.

# Behind the Tape Reader (BTR)




- Computer is linked directly to regular NC controller unit
- The connection is made behind the tape reader
- Two temporary storage buffers
- Less cost

# Special Machine Control Unit




- Regular NC controller is replaced by special MCU
- More accuracy in circular interpolation and fast material removal rates than BTR systems
- Most CNC machines are sold with computer

# NC, CNC and DNC

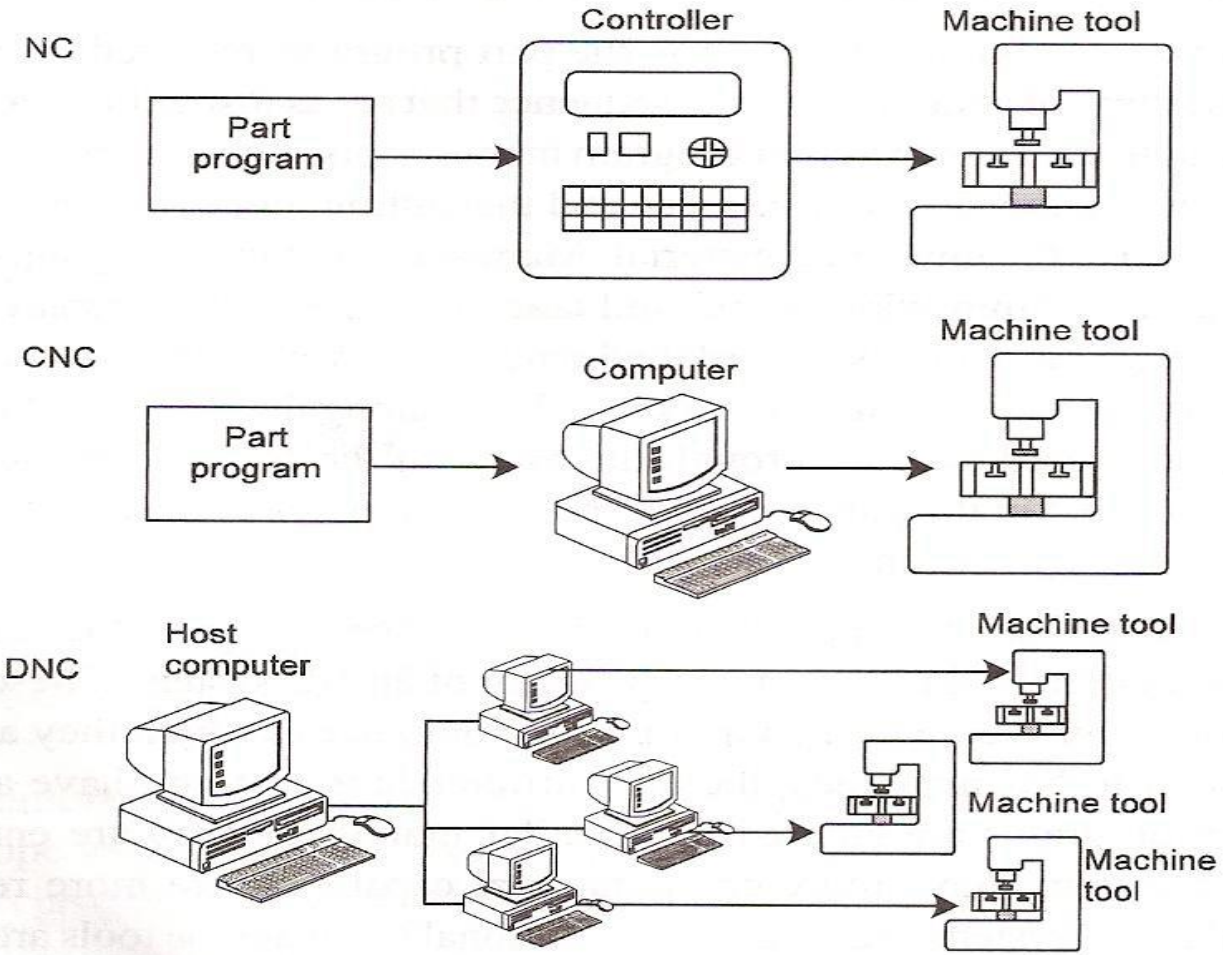



*Figure 1.7: NC, CNC, and DNC Systems*

# Functions of DNC

**1.NC without punched tape**

**2.NC part program storage**

- Programs must be made available for downloading to CNC machine tools
- Part program can be uploaded after editing from CNC machine
- Entry of new programs. Editing of programs , deletion of programs
- Tool management
- Tool offsets can be downloaded in to MCU
- Postprocessor
- Data processing and management functions

# Functions of DNC

**3.Data collection, Processing and reporting**

- Monitor production in the factory

Data processing and report generation by DNC computer

- Getting the data about health of the machine in the form of sensor signals or diagnostic messages which can be used for preventive/predictive maintenance
- Metrological data in the form of dimensional acceptance

# Functions of DNC

**4.Communications**

- Central computer and machine tools
- Central computer and NC part programmer terminals
- Central computer and bulk memory, which stores the NC programs
- CAD system
- Shop floor control system
- Corporate data processing
- Remote maintenance diagnostics system

# Adaptive Control

- A control system that measures certain output process variables like spindle deflection, force, torque, cutting temperature, vibration amplitude, horse power and uses them to control speed or feed
- NC reduces non productive time in a machining operation
- AC determines proper speeds and feeds during machining as a function of variation in work piece hardness, width or depth of cut, air gaps in part geometry etc.
- Increased metal removal rate and reduced cost per volume of metal removed

# Where to use adaptive control?

- In-process time consumes significant portion of the machining cycle time. (>40%)
- Significant sources of variability in the job
- Higher cost of operation of machine tool
- Work material –steel, titanium, high strength alloys

# Sources of variability in machining

1. Variable depth/width of cut
2. Variable workpiece hardness and variable machinability
3. Variable workpiece rigidity
4. Toolwear
5. Air gaps during cutting

# Adaptive Control Optimization (ACO)

- Index of performance is a measure of overall process performance such as production rate or cost per volume of metal removed.
- Objective is to optimize the index of performance by manipulating speed or feed in the operation
- IP = MRR/TWR
- MRR –Material removal rate
- TWR –Tool wear rate
- Sensors for measuring IP not available

# Adaptive control Constraint (ACC)

- Less sophisticated and less expensive than research ACO systems
- Objective is to manipulate speed or feed

so that measured process variables are maintained at or below their constraint limit values.

# Operation of ACC system

- Profile or contour milling on NC machine tool
- Feed is controlled variable
- Cutter force and horsepower are used as measured variables
- **Hardware components**

1.Sensors mounted on the spindle to measure cutter force

2.Sensors to measure spindle motor current

3.Control unit and display panel to operate the system

4.Interface hardware to connect the AC system to existing NC/CNC system

# Relationship of AC software to APT program

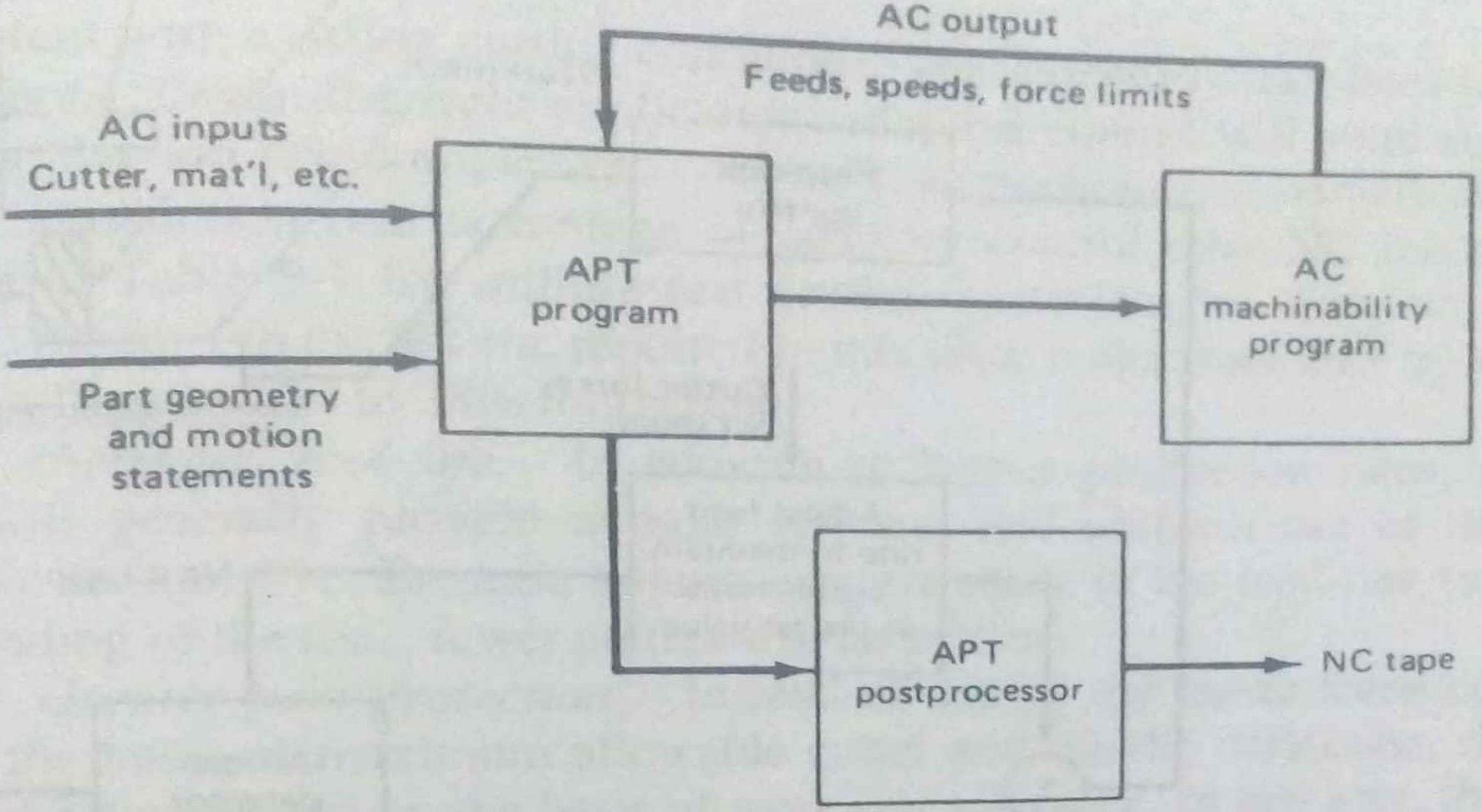



FIGURE 9.8 Relationship of adaptive control (AC) software to APT program.

# Operation of ACC system during machining process

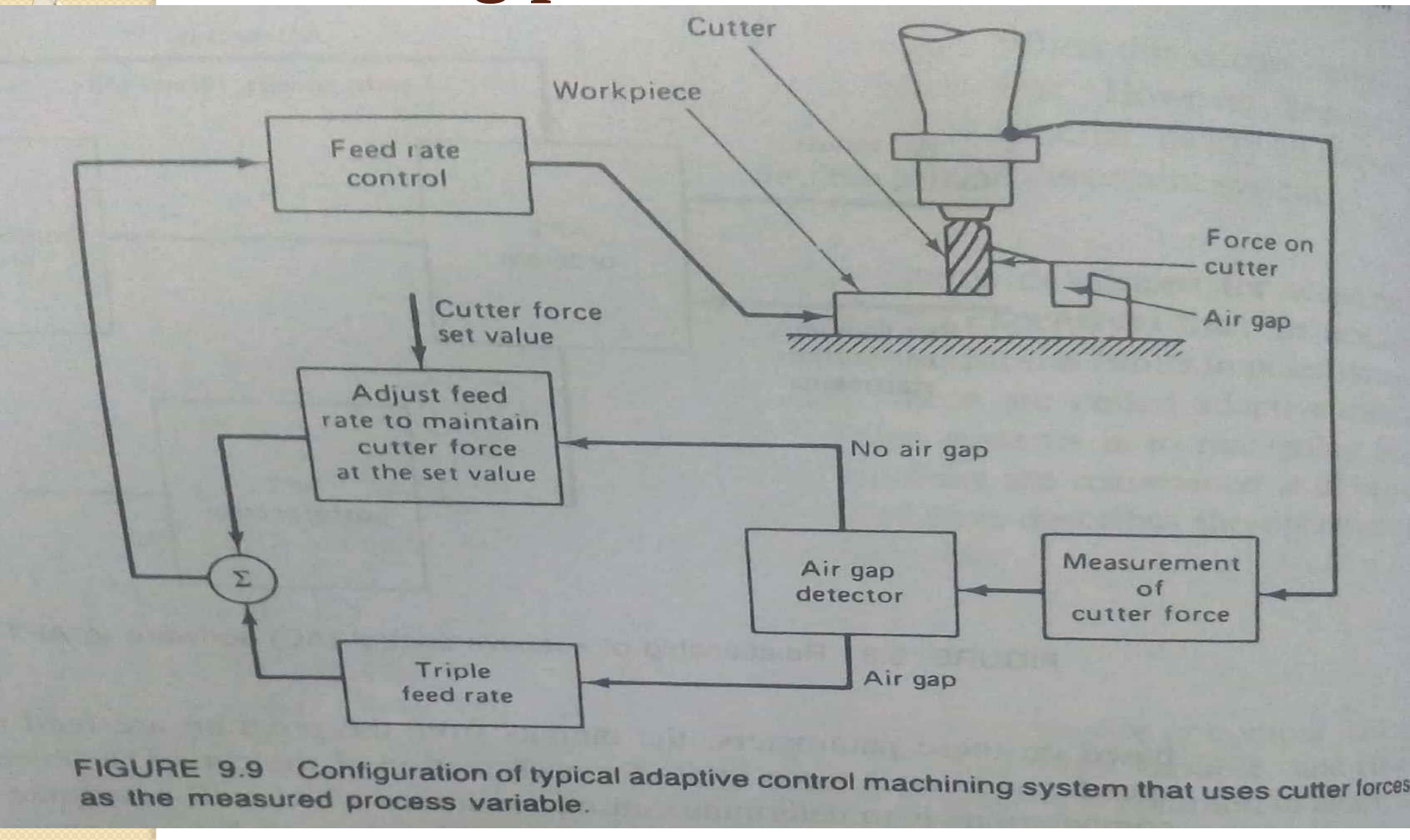
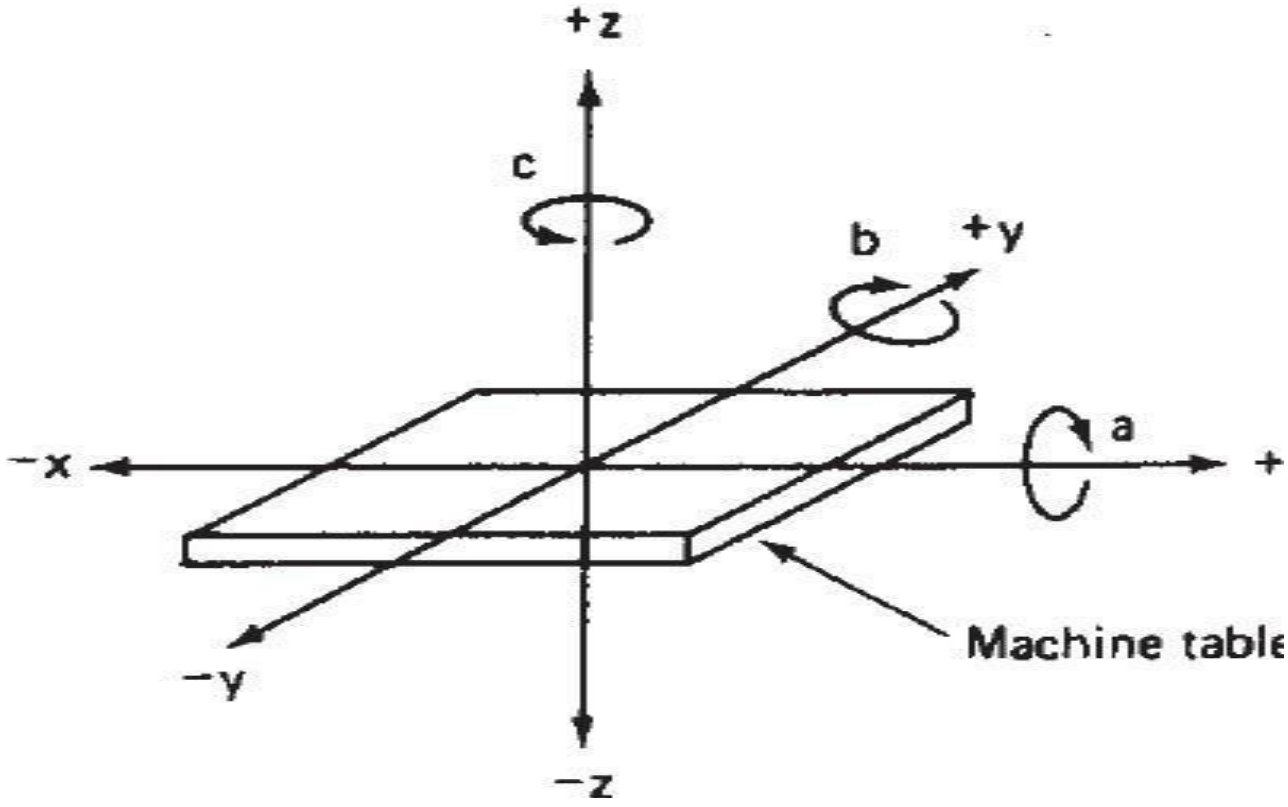



FIGURE 9.9 Configuration of typical adaptive control machining system that uses cutter forces as the measured process variable.

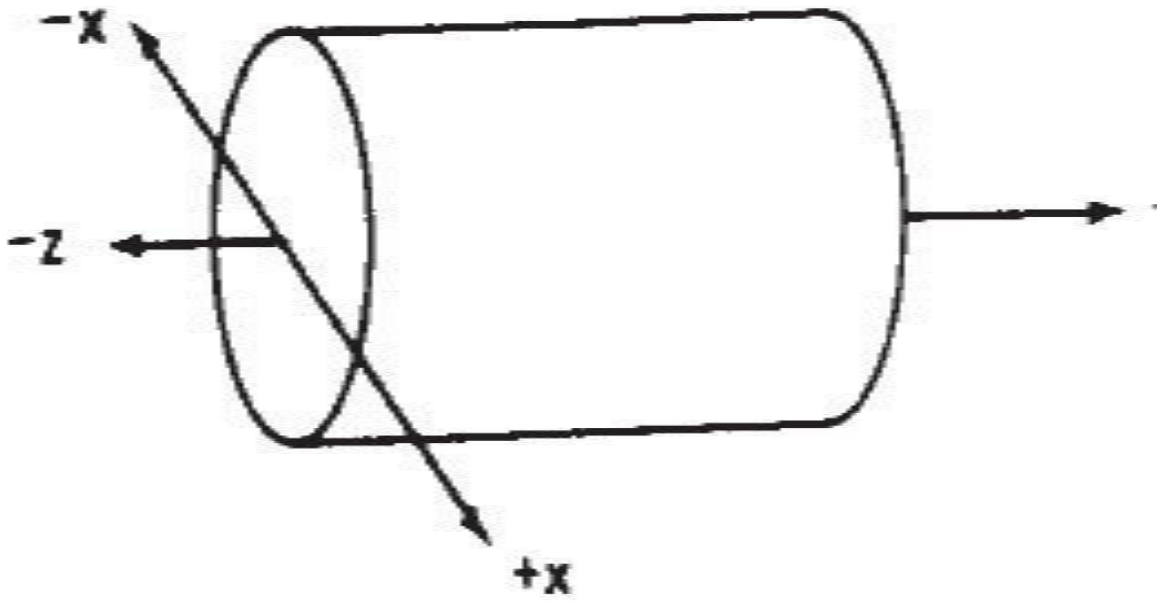
# Benefits of AC

1. Increased production rate
2. Increased tool life
3. Greater part protection
4. Increases machine life
5. Less operator intervention
6. Easier part programming

# NC COORDINATE SYSTEMS

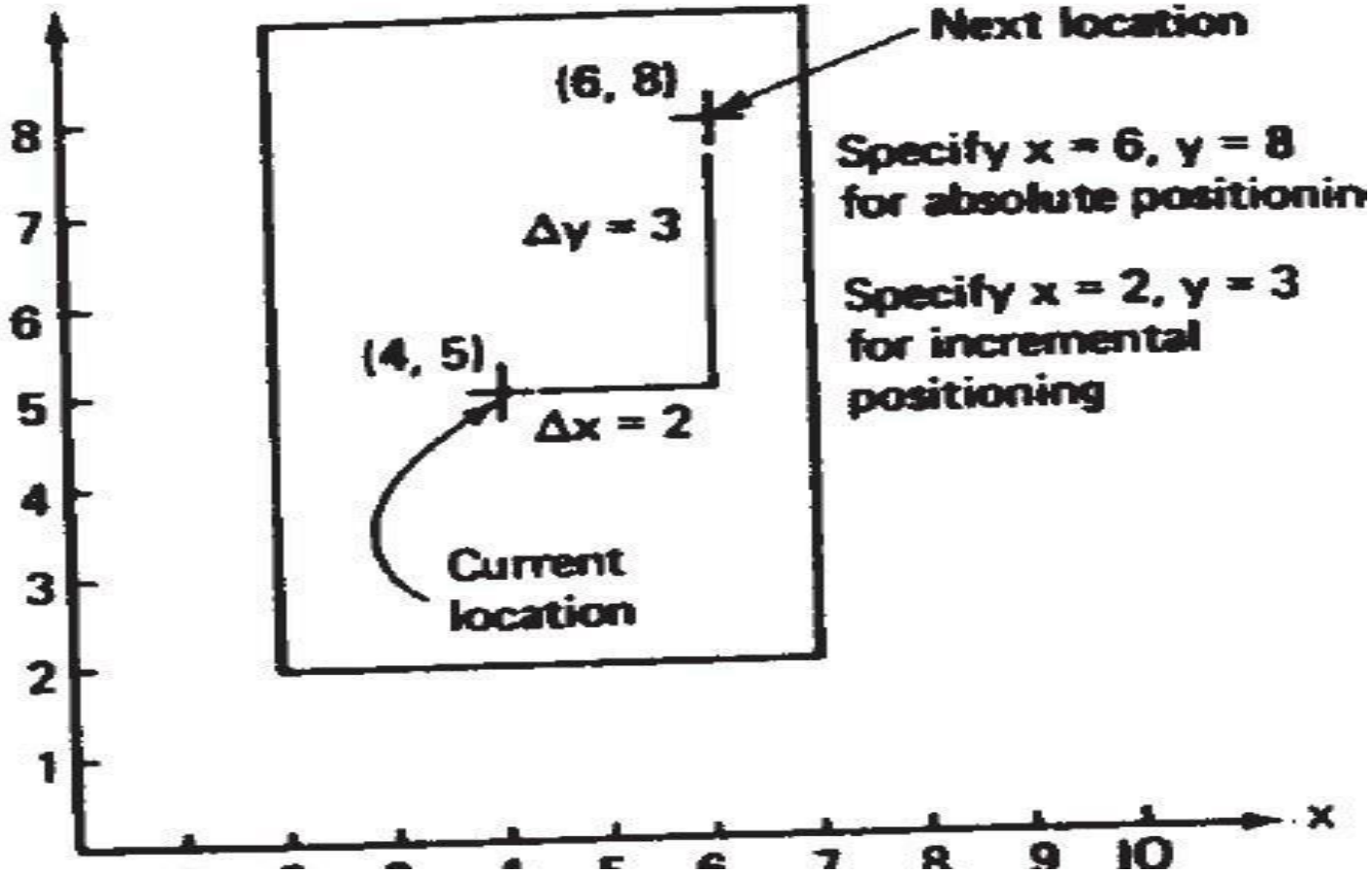



**NC machine tool axis system for milling and drilling operations.**




**NC machine tool axis system for turning operation.**

# Absolute versus incremental positioning.



# Fixed zero and floating zero

- The programmer must determine the position of the tool relative to the origin (zero point) of the coordinate system. NC machines have either of two methods for specifying the zero point. The first possibility is for the machine to have a fixed zero.
- The second and more common feature on modern NC machines allows the machine operator to set the zero point at any position on the machine table. This feature is called floating zero.

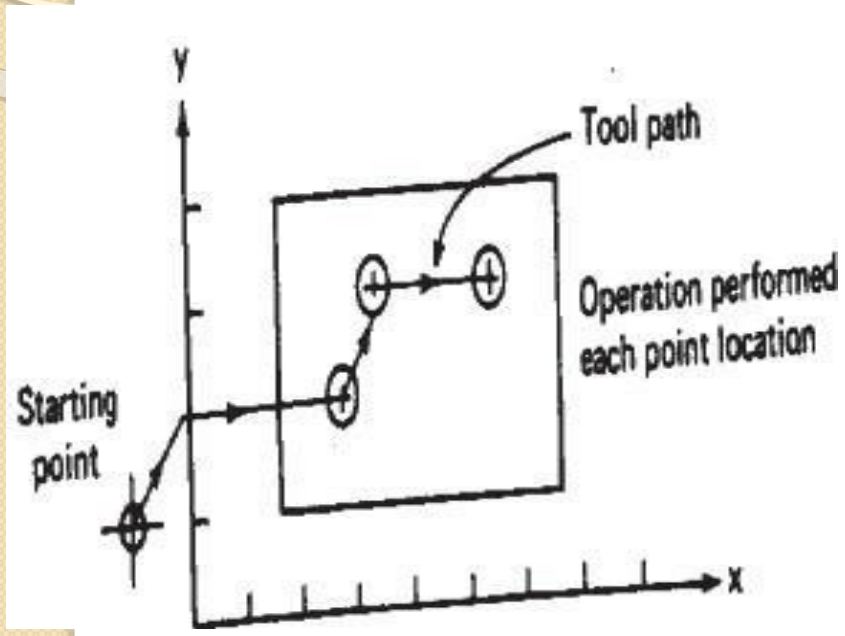
# NC MOTION CONTROL SYSTEMS

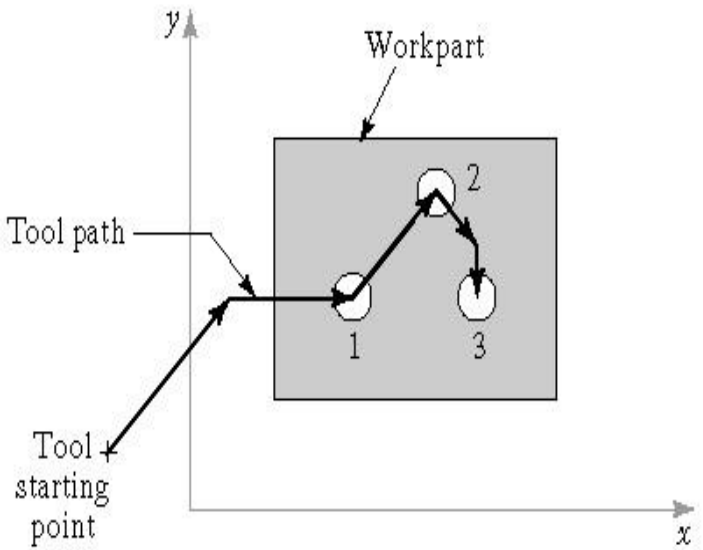
l. Point-to-point

2. Straight cut

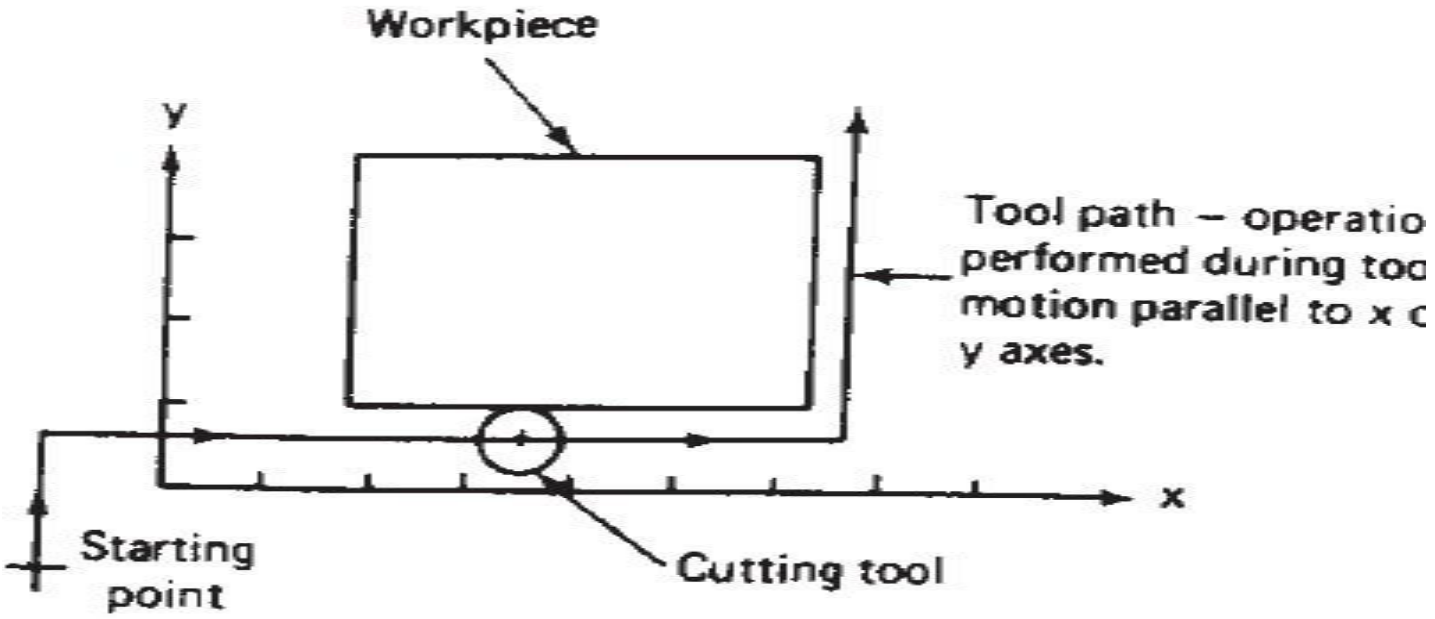
3. Contouring

# Point-to-point (positioning) NC system.

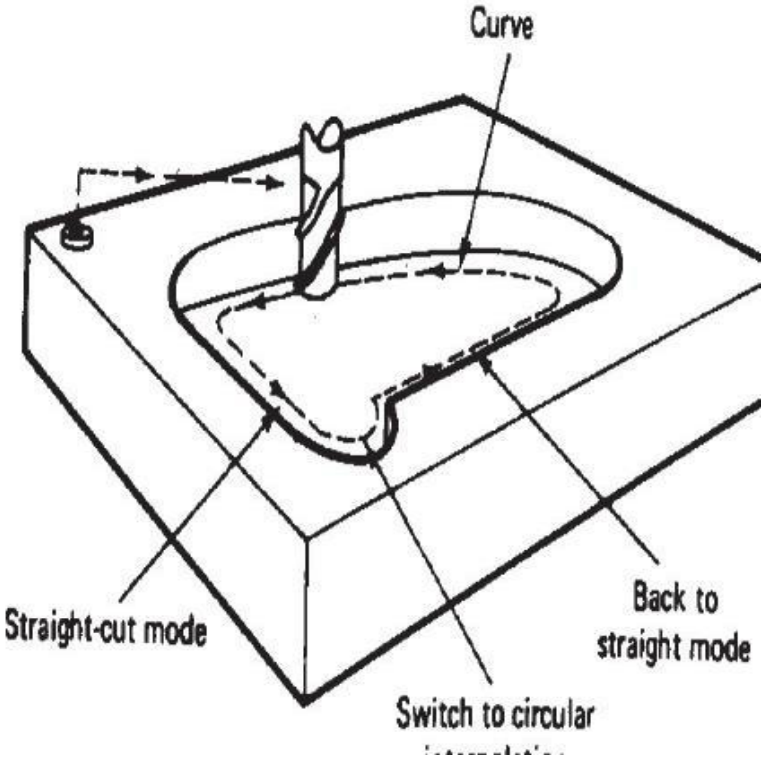


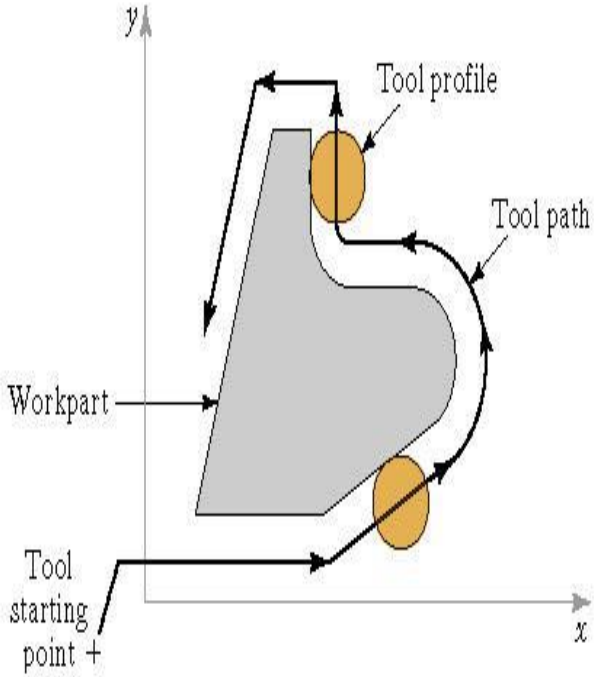



# Straight-cut system.



# Contouring (continuous path) NC system for two-dimensional operations



# CNC Machining Centers

- Industrial surveys in 1960's showed smaller machine components requiring several operations tool long time to complete
  - Part sent to several machines before finished
  - There was much "operator intervention" during machining process
- In late 1960s and early 70s, begin to design machine that would perform several operations and do 90% of machining on one machine

# Types of Machining Centers

- Three types: horizontal, vertical and universal

**Factors to determine type and size**

1. Size and weight of largest piece machined
2. Maximum travel of three primary axes
3. Maximum speeds and feeds available
4. Horsepower of spindle
5. Number of tools automatic tool changer can hold

# Two Types of Horizontal Machining Centers

- Traveling-column
  - One or usually two tables where work mounted
    - Column and cutter move toward work on one table while operator changes work piece on other table
- Fixed-column
  - Equipped with pallet (removable table)
    - After work piece machined, pallet and work piece moved off receiver onto shuttle; shuttle rotated, bringing new pallet into position for shuttle and finished work pallet into position for unloading

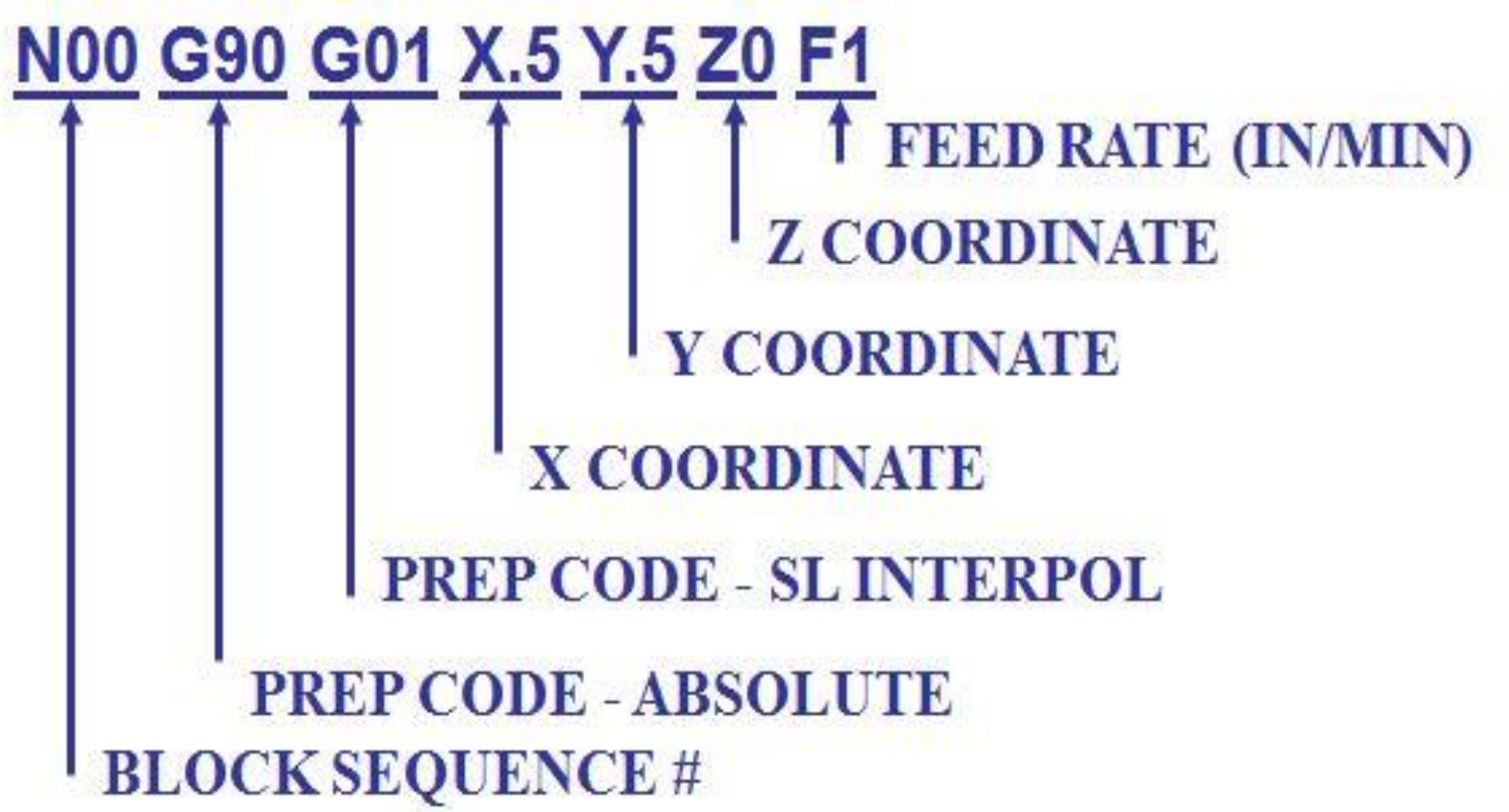
# Vertical Machining Center

- Saddle-type construction with sliding bed ways that use a sliding vertical head instead of quill movement
- Generally used to machine flat parts held in vise or simple fixture
- Versatility increased by addition of rotary accessories

# Universal Machining Center

- Combines features of vertical and horizontal machining centers
  - Spindle can be programmed in both vertical and horizontal positions
    - Allows for machining all side of a part in one setup
- Useful for small and medium batch parts
- Has additional accessories such as indexible pallets and rotary-tilt tables

# Sample NC Program Block



# NC CODES

- Block Number (N)
- Preparatory  Codes (G)
- Miscellaneous Codes (M)
- Primary X Motion (X)
- Primary Y Motion (Y)
- Primary Z Motion (Z)

# Preparatory Codes

- G90 - Absolute Coordinates
- G91 - Relative Coordinates
- G00 - Rapid Traverse (non-cutting move)
- G01 - Straight Line Interpolation (cutting move
- G02 - Circle Interpolation (clockwise)
- G03 - Circle Interpolation (c-clockwise)
- G04 - Dwell (wait) Pause between motions on all axis. Time in seconds - G04F2 - pause for 2 sec.
- G05 - Pause - waits for user intervention.

# M Codes - Miscellaneous

- M00 - Pause
- M01 - Optional stop
- M02 - End of Program
- M03 - Spindle on
- M05 - Spindle off
- M06 - Tool Change
- M08 / M09 - Accessory # 1 on / off
- M10 / M11 - Accessory # 2 on / off

The following is a list of commonly used G Codes for CNC machines and their functionality.

| CODE | DESCRIPTION |
|---|---|
| G00 | Rapid Linear Positioning |
| G01 | Linear Feed Interpolation |
| G02 | CW Circular Interpolation |
| G03 | CCW Circular Interpolation |
| G04 | Dwell |
| G07 | Imaginary Axis Designation |
| G09 | Exact Stop |
| G10 | Offset Value Setting |
| G17 | XY Plane Selection |
| G18 | ZX Plane Selection |
| G19 | YZ plane Selection |
| G20 | Input In Inches |
| G21 | Input In Millimeters |
| G22 | Stored Stroke Limit On |
| G23 | Stored Stroke Limit Off |
| G27 | Reference Point Return Check |
| G28 | Return To Reference Point |
| G29 | Return From Reference Point |
| G30 | Return To 2nd, 3rd and 4th Ref. Point |
| G31 | Skip Cutting |

| | |
|---|---|
| G31 | Skip Cutting |
| G33 | Thread Cutting |
| G40 | Cutter Compensation Cancel |
| G41 | Cutter Compensation Left |
| G42 | Cutter Compensation Right |
| G43 | Tool Length Compensation + Direction |
| G44 | Tool Length Compensation - Direction |
| G45 | Tool Offset Increase |
| G46 | Tool Offset Double |
| G47 | Tool Offset Double Increase |
| G48 | Tool Offset Double Decrease |
| G49 | Tool Length Compensation Cancel |
| G50 | Scaling Off |
| G51 | Scaling On |
| G52 | Local Coordinate System Setting |
| G54 | Work Coordinate System 1 Selection |
| G55 | Work Coordinate System 2 Selection |

| CODE | DESCRIPTION |
|---|---|
| G56 | Work Coordinate System 3 Selection |
| G57 | Work Coordinate System 4 Selection |
| G58 | Work Coordinate System 5 Selection |
| G59 | Work Coordinate System 6 Selection |
| G60 | Single Direction Positioning |
| G61 | Exact Stop Mode |
| G64 | Cutting Mode |
| G65 | Custom Macro Simple Call |
| G66 | Custom Macro Modal Call |
| G67 | Custom Macro Modal Call Cancel |
| G68 | Coordinate System Rotation On |
| G69 | Coordinate System Rotation Off |
| G73 | Peck Drilling Cycle |
| G74 | Counter Tapping Cycle |
| G76 | Fine Boring |
| G80 | Canned Cycle Cancel |
| G81 | Drilling Cycle, Spot Boring |
| G82 | Drilling Cycle, Counter Boring |
| G83 | Peck Drilling Cycle |
| G84 | Tapping Cycle |

| | |
|---|---|
| G85 | Boring Cycle |
| G86 | Boring Cycle |
| G87 | Back Boring Cycle |
| G88 | Boring Cycle |
| G89 | Boring Cycle |
| G90 | Absolute Programming |
| G91 | Incremental Programming |
| G92 | Programming Of Absolute Zero |
| G94 | Feed Per Minute |
| G95 | Feed Per Revolution |
| G96 | Constant Surface Speed Control |
| G97 | Constant Surface Speed Control Cancel |
| G98 | Return To Initial Point In Canned Cycles |
| G99 | Return To R Point In Canned Cycles |

| CODE | DESCRIPTION |
|---|---|
| M00 | Program Stop |
| M01 | Optional Stop |
| M02 | End of Program |
| M03 | Spindle On CW |
| M04 | Spindle On CCW |





| CODE | DESCRIPTION |
|---|---|
| M05 | Spindle Stop |
| M06 | Tool Change |
| M07 | Mist Coolant On |
| M08 | Flood Coolant On |
| M09 | Coolant Off |

| | |
|---|---|
| M19 | Spindle Orientation On |
| M20 | Spindle Orientation Off |
| M21 | Tool Magazine Right |
| M22 | Tool Magazine Left |
| M23 | Tool Magazine Up |
| M24 | Tool Magazine Down |
| M25 | Tool Clamp |
| M26 | Tool Unclamp |
| M27 | Clutch Neutral On |
| M28 | Clutch Neutral Off |
| M30 | End Program, Stop and Rewind |
| M98 | Call Sub Program |
| M99 | End Sub Program |

**2016 IISc Bangalore**

**8.15** Match the following part programming codes with their respective function

| Part programming codes | Functions |
|---|---|
| P. G01 | I. Spindle stop |
| Q. G03 | II. Spindle rotation, clockwise |
| R. M03 | III. Circular interpolation, anticlockwise |
| S. M05 | IV. Linear interpolation |

## 2010 IIT Guwahati

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

(A) Circular interpolation in counter clockwise direction and incremental dimension

(B) Circular interpolation in counter clockwise direction and absolute dimension

(C) Circular interpolation in clockwise direction and incremental dimension

(D) Circular interpolation in clockwise direction and absolute dimension

## 2009 IIT Roorkee

**8.12** Match the following:

| NC Code | Definition |
|---|---|
| P. M05 | 1. Absolute coordinate system |
| Q. G01 | 2. Dwell |
| R. G04 | 3. Spindle stop |
| S. G90 | 4. Linear interpolation |

**2014 IIT Kharagpur**

**8.13** For the CNC programing, match Group A with Group B :

**Group A**

P : Circular interpolation, counter clockwise

Q : Dwell

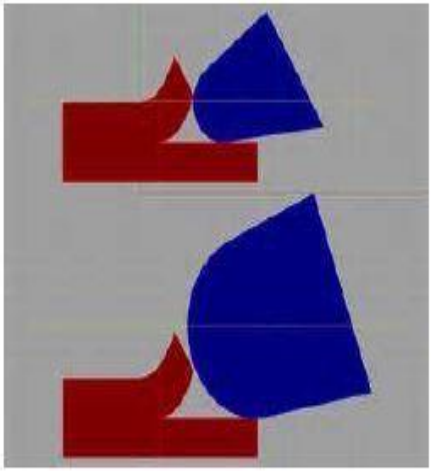
R : Circular interpolation, clockwise

S : Point to point countering

**Group B**

| | | | |
|---|---|---|---|
| 1. | G02 | 2. | G03 |
| 3. | G04 | 4. | G00 |

# What is tool compensation??



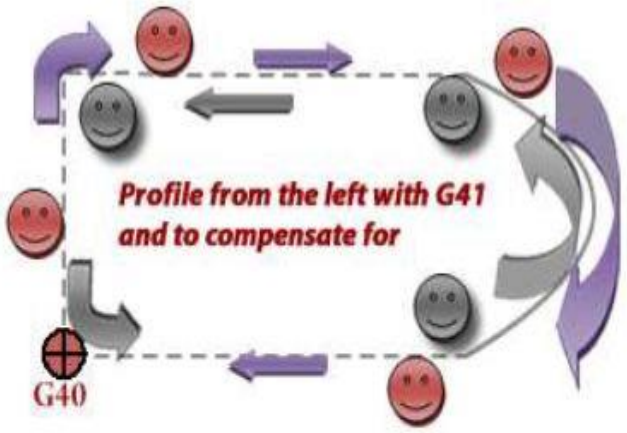
# And, why do we really need it?

- Ability to manufacture accurate parts
- Dimensional errors can be immediately detected and avoided
- The world market for machine tools in 2007 is estimated to be 71 billion US dollars, which represents a growth of 18% compared to 2006.

# Types of tool compensation

- Cutter Radius(or Diameter) Compensation
- Tool Nose Radius Compensation
- Tool Length Compensation

# Tool Diameter Compensation



# Why Cutter Diameter Compensation?

- When machining finished surfaces with the side of a milling cutter (generally called profiling), the accuracy of the finished surface depends on the cutter accuracy and how closely the cutter diameter matches the programmed size.
- Cutters wear causing size changes in profiled surfaces.
- Reground endmills are always smaller than nominal size.

Note: this feature is also frequently called Cutter Radius Compensation. We use Diameter Compensation to avoid confusion with turning center operation.

# Cutter Diameter Compensation Codes

| Code | Application |
|---|---|
| G40 | Cancel cutter diameter compensation. |
| G41 | Compensate for the cutter to the LEFT of the programmed path. |
| G42 | Compensate for the cutter to the RIGHT of the programmed path. |
| Dtt | tt is the tool number. D tells the controller where to find the cutter's diameter. |

# Determining G41 or G42

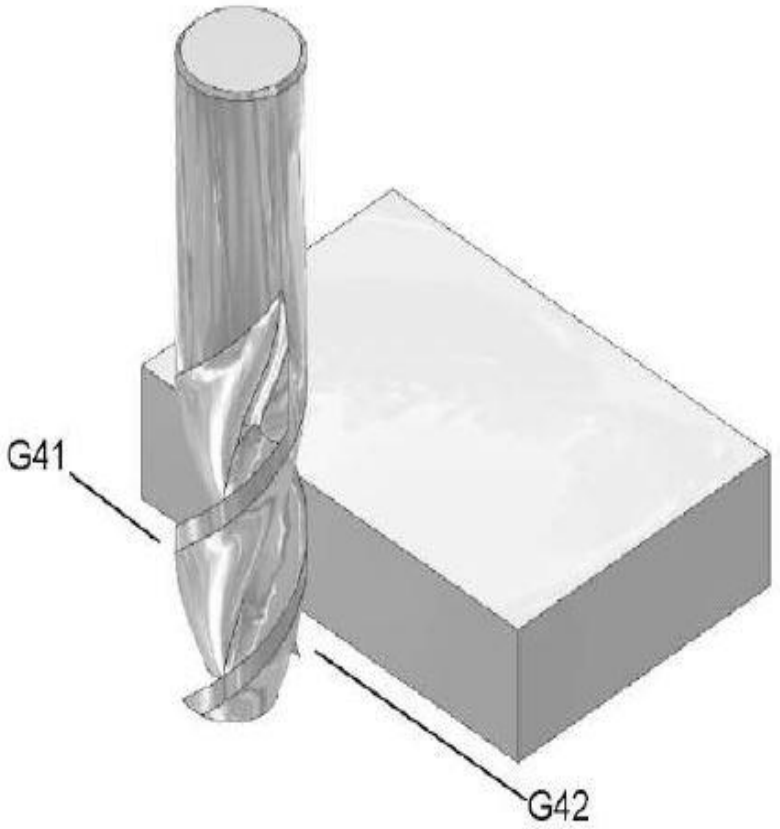
G41: the cutter is to the left of the part when looking in the direction of the cut.

G42: the cutter is to the right of the part when looking in the direction of the cut.

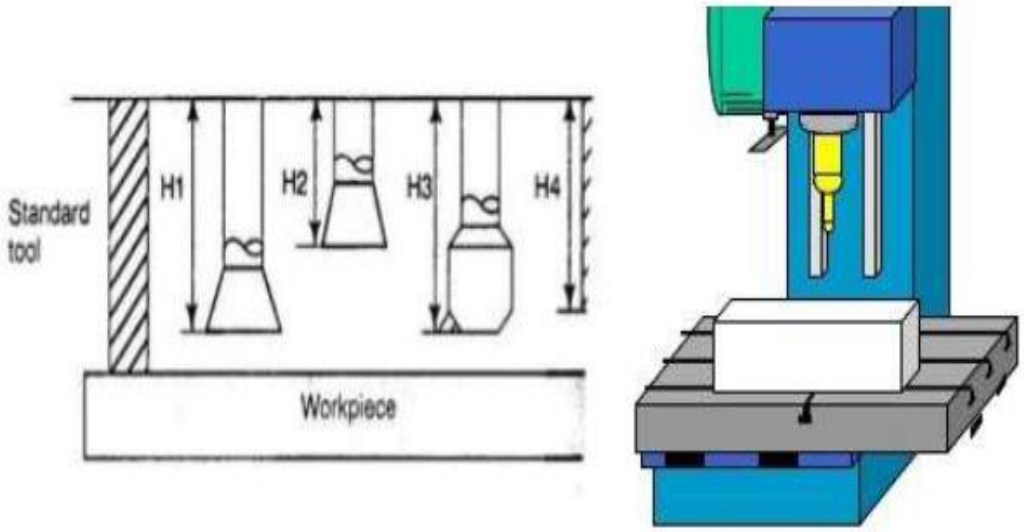
Climb milling features: use G41.

Conventional milling features: use G42.

Since we normally climb mill, we will generally use G41 on a machining center.



# Tool Length Compensation

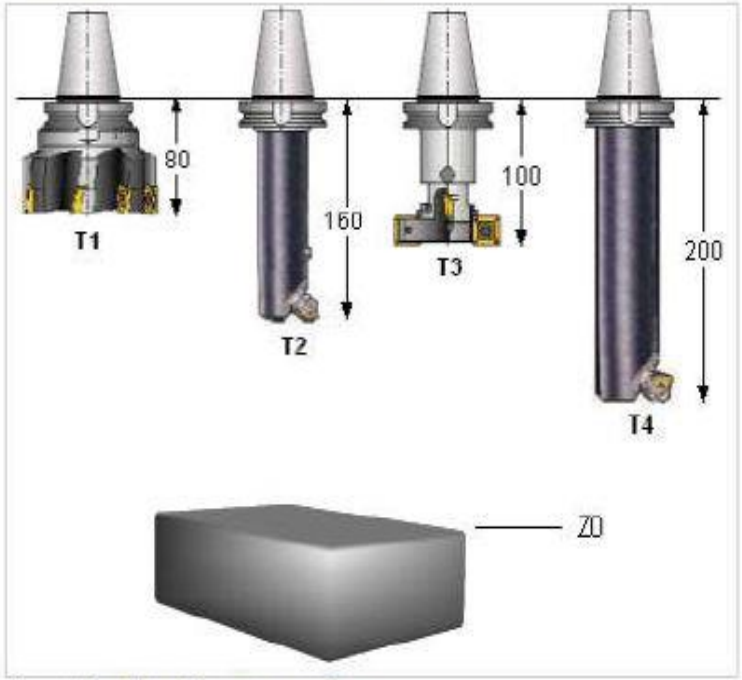


# Why Tool Length Compensation is Needed

- No Two Tools Will Have Exactly the Same Length
- Tool's Length Will Vary from Each Time it is Assembled
- Tool Data is Entered Separately from the Program
- Sizing and Trial Machining Must Often be Done

**Tool length compensation**

Tools used in machining a part are of different lengths. It would be extremely tedious to write the program with these lengths taken into consideration.




**Necessity of length compensation**

In this picture, for example, to move to the position Z0, the programed coordinate would be Z80, Z160, Z100 and Z200 for tools T1 to T4 respectively. Each time that a tool got worn out and you had to change it , you would have to change the Z coordinates in the whole program.

To eliminate this problem, machines have a length compensation feature. The program is written for the drawing coordinates, without considering tool lengths lengths. The lengths are entered in the controller's memory. The controller does the job of adjusting for the tool length. A rapid motion to the

# Tool length compensation G Codes

- G43 - Tool length offset plus
- G44 - Tool length offset minus
- G49 - Tool length compensation cancel

# TOOL TURRET

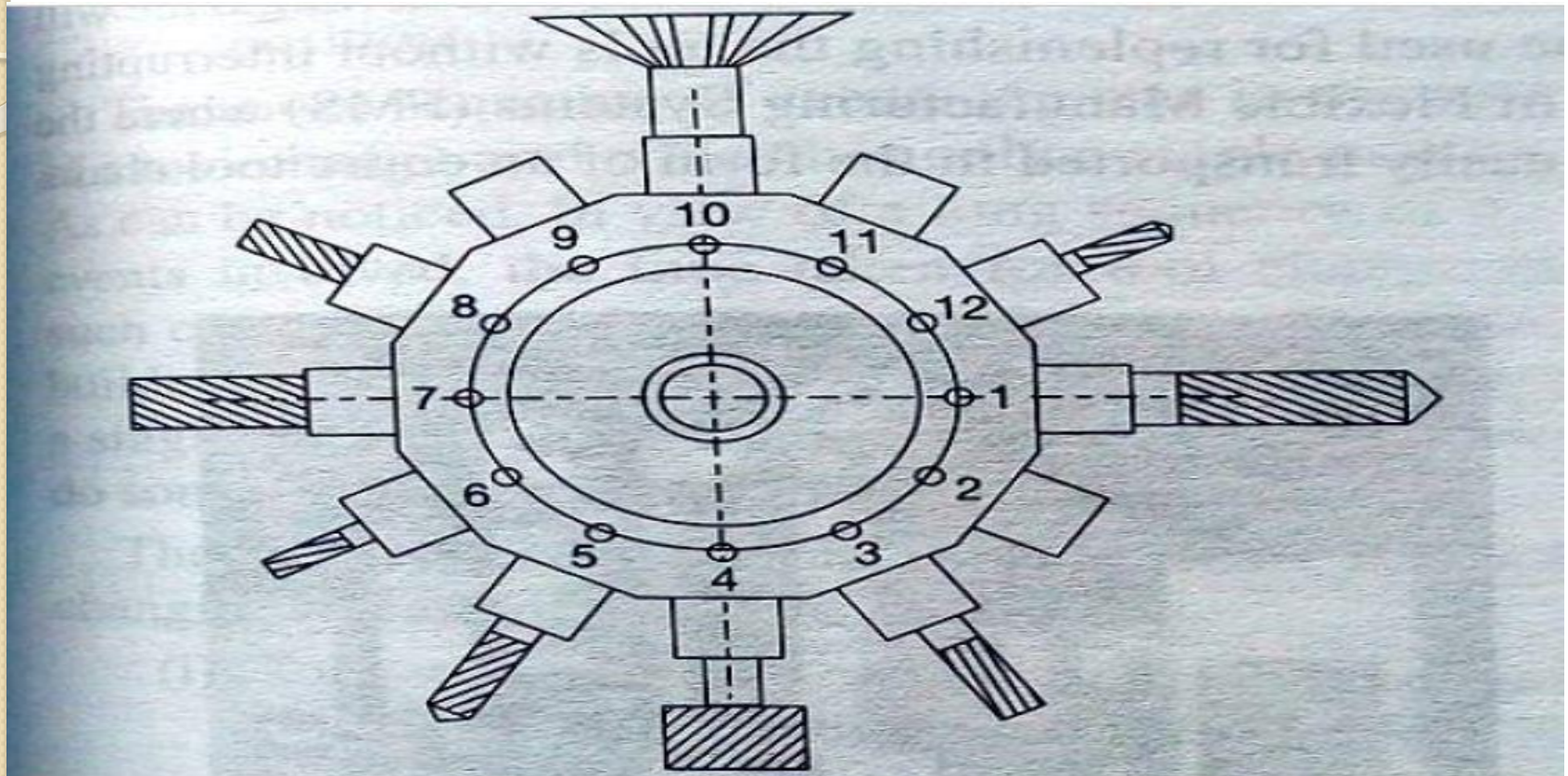



**Fig. 11.14** *Typical tool turret used in CNC drilling/milling machines*

# CHAIN TYPE MAGZINE

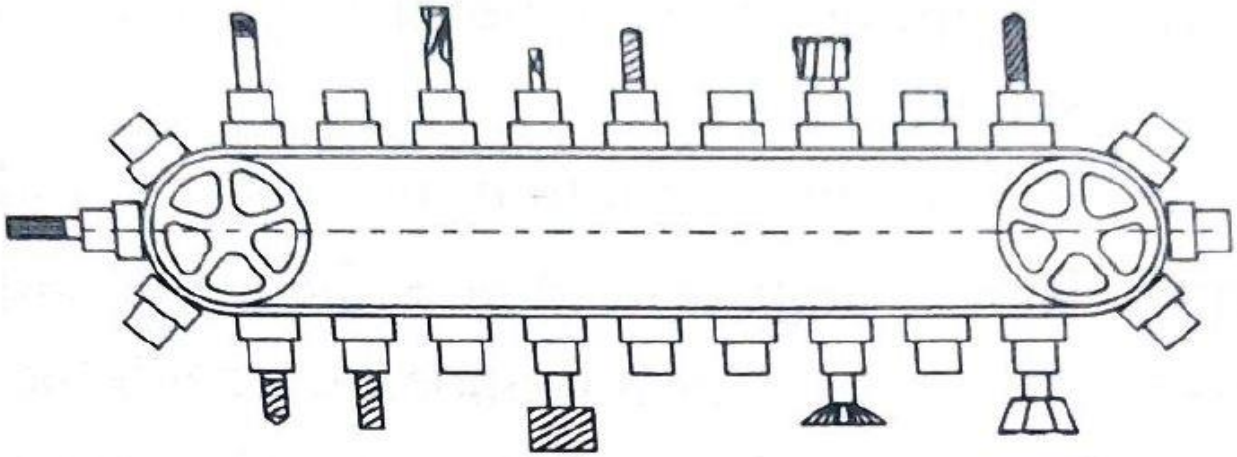


**Fig. 11.17** *Chain-type tool magazine for holding larger number of spindle tooling used in CNC machining centres*

# TOOL CHANGE ARAM

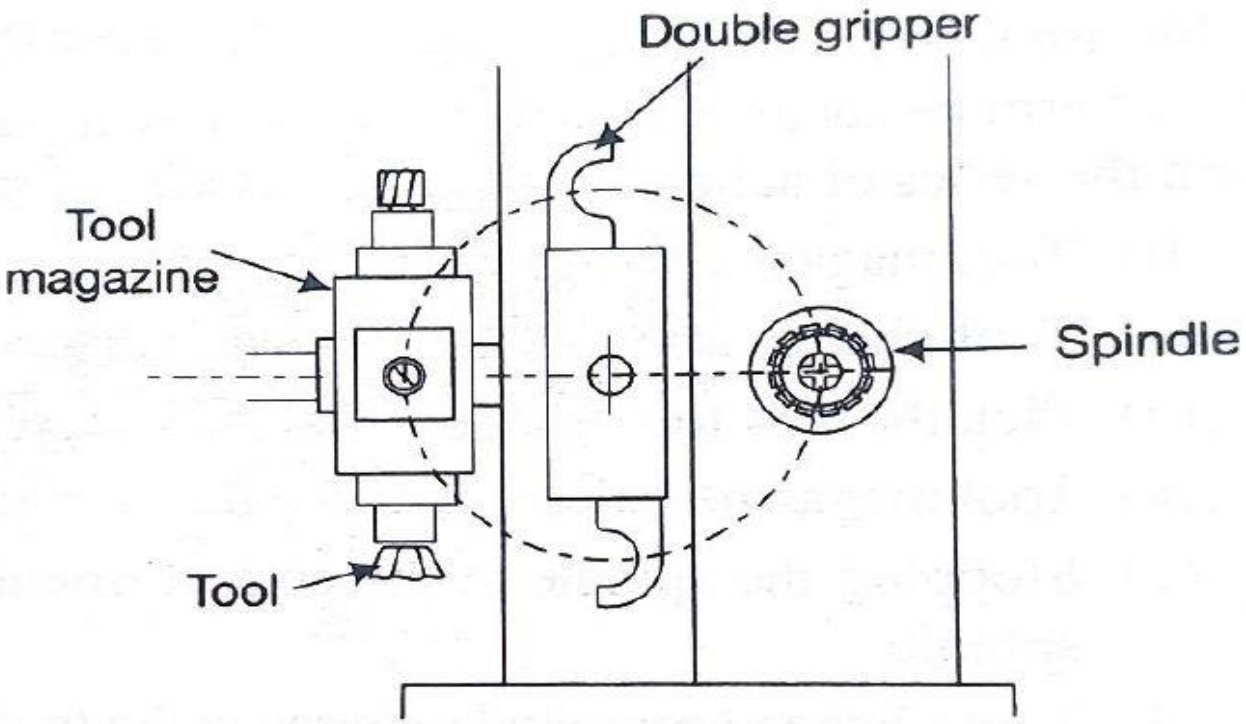



**Fig. 11.20** *One common type of tool-change arm used for tool changing with a double gripper*

# TOOL CHANGE PROCEDURE

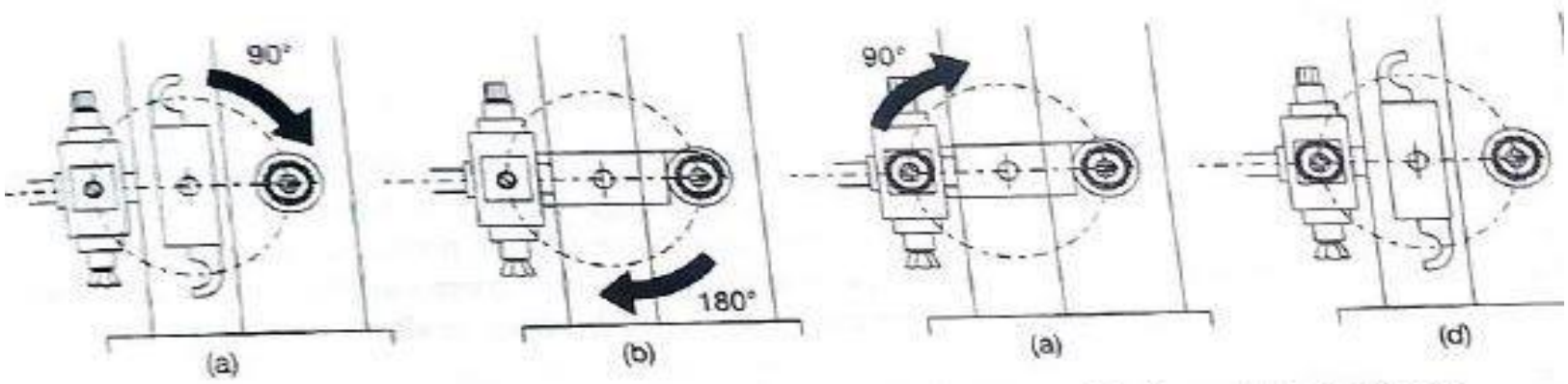



**Fig. 11.21** *Tool-change procedure with a tool-change arm having a double gripper*

# Work holding devices

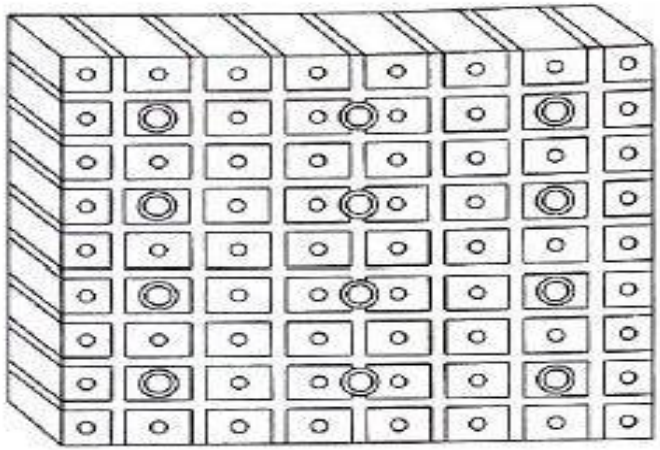


**Fig. 11.24** *Grid plate with holes which can be used as a machine table*

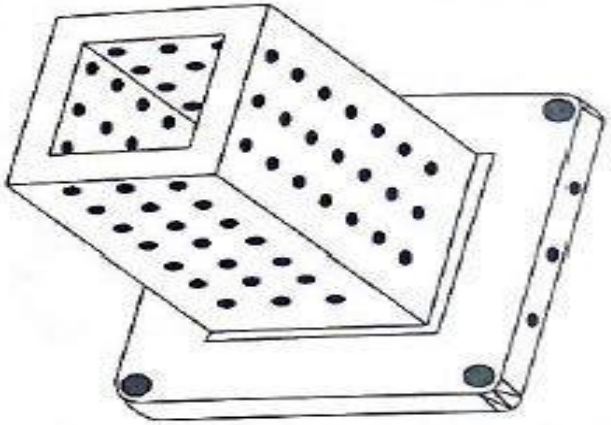


**Fig. 11.25** *Tombstone for mounting multiple components on different places*

In addition to these standard fixture bases, a large number of fixture elements such as angle blocks and base elements (Fig. 11.26) are used to quickly clamp the workpieces in position.

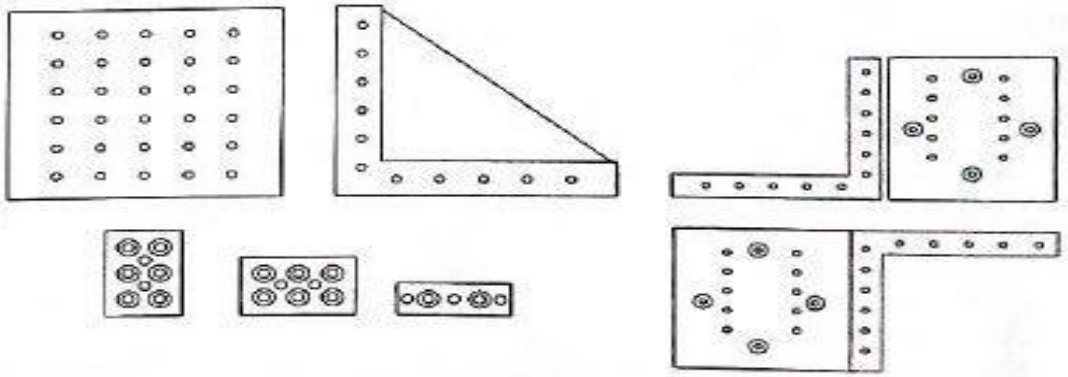


**Fig. 11.26** *Modular fixture elements used for supporting complex workpieces*

# ACTUATION SYSTEMS

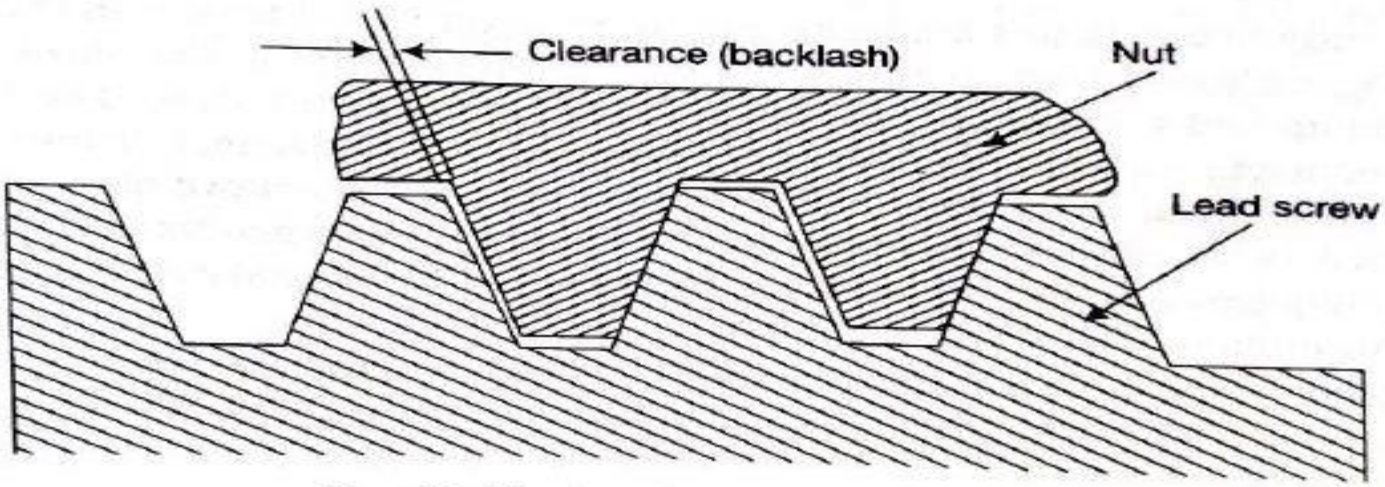



**Fig. 10.11** *Lead screw with Acme nut*

# Recirculating ball screw & nut

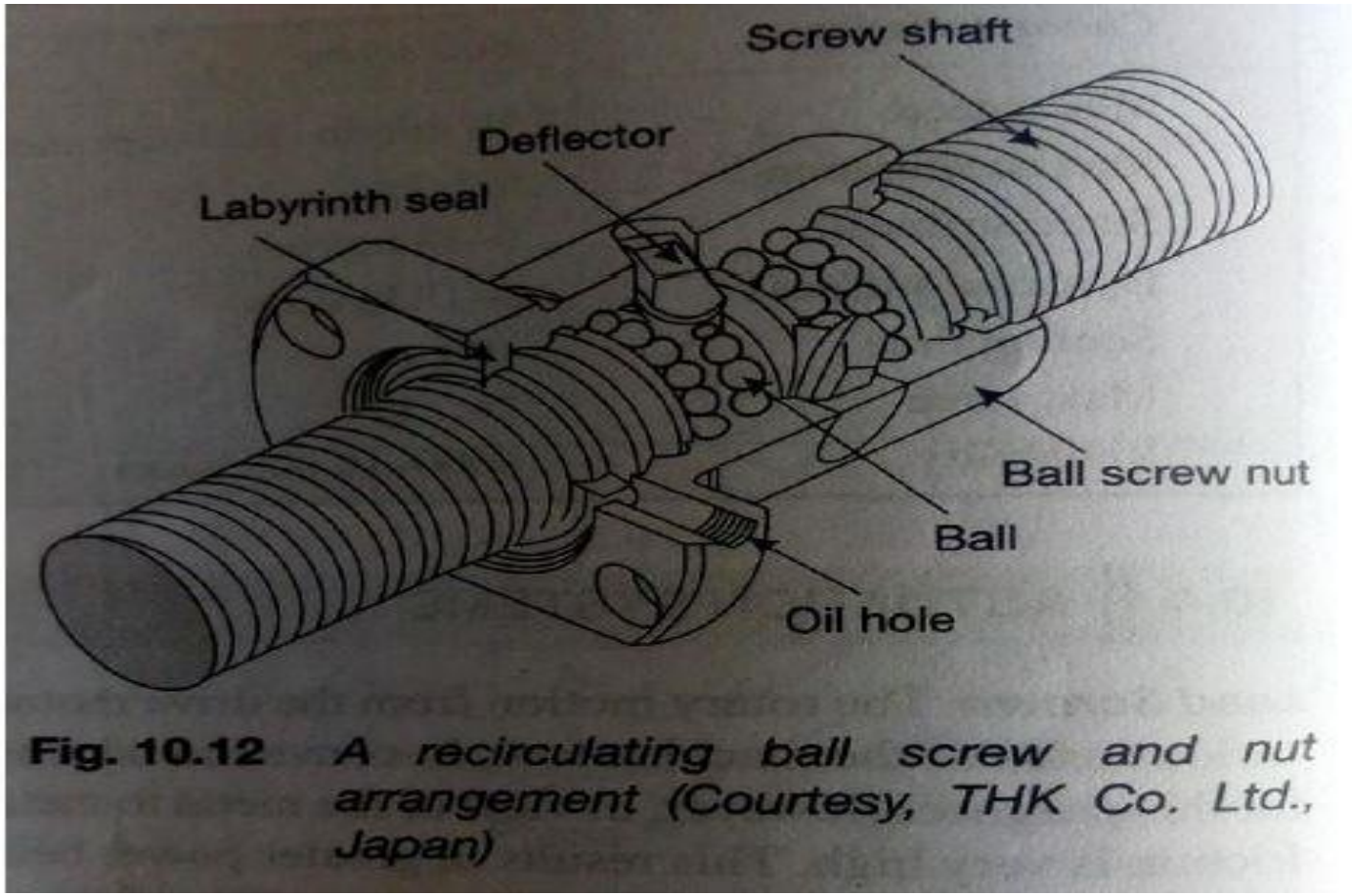



**Fig. 10.12** *A recirculating ball screw and nut arrangement (Courtesy, THK Co. Ltd., Japan)*

# Recirculating ball screw and nut arrangement with external return tube

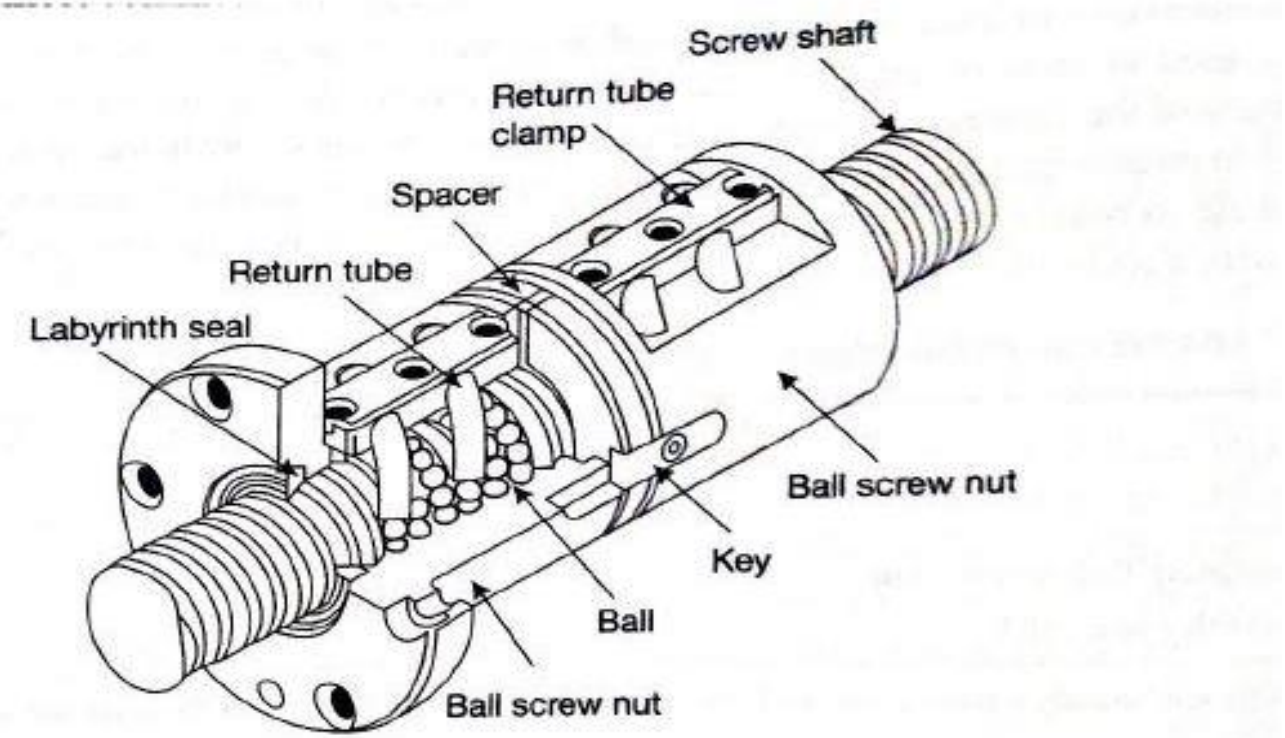



**Fig. 10.13** *A recirculating ball screw and nut arrangement with external return tube (Courtesy, THK Co. Ltd., Japan)*

# Preloading of the recirculating ball screw and nut arrangement

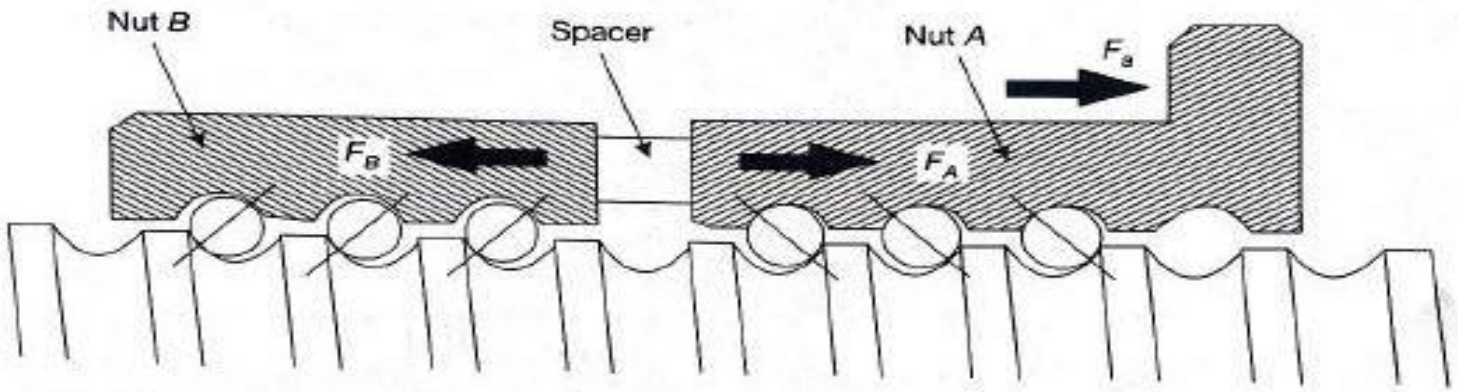



**Fig. 10.14** *Preloading of the recirculating ball screw and nut arrangement*

# Slide ways

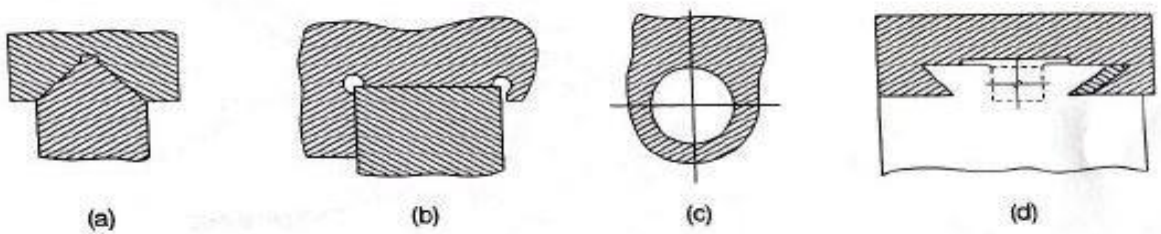



**Fig. 10.15** *Conventional slideway systems used in machine tools*

# Antifriction guide ways

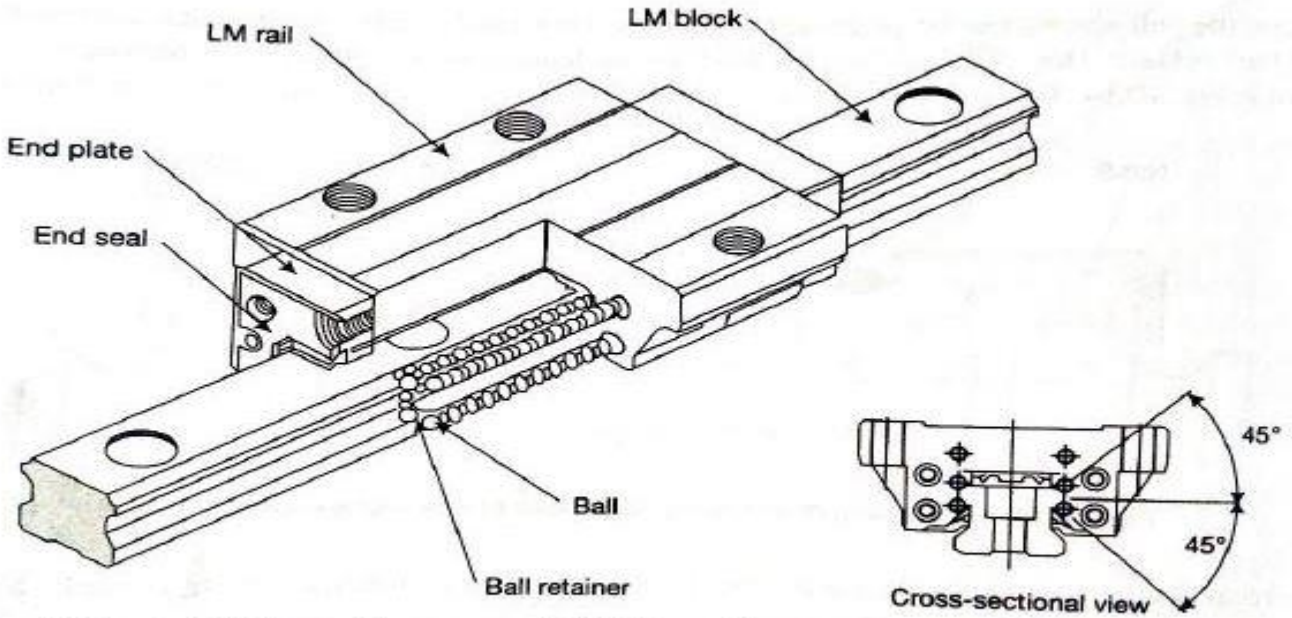



**Fig. 10.16** *Antifriction guideways used in CNC machine tools (Courtesy, THK Co. Ltd., Japan)*