



# Introduction to NC



Arvind Deshpande

# Numerical Control

---

- Types of manufacturing
  1. Continuous process industries – Sugar industry, chemical industry etc.
  2. Mass production industries – Automobiles, Consumer goods etc.
  3. Batch production industries – Machines, aircrafts etc.
  4. Job production industries – Prototypes, heat exchangers , chemical reactors etc.
- Automation is technology concerned with the application of complex mechanical, electrical, electronic , hydraulic, pneumatic or computer based system in the operation and control of production
- Competition, cost, quality
- Flexible automation

# Numerical Control

---

- **Form of programmable automation in which process is controlled by numbers, letters and symbols.**
- One or more of the following functions may be automatic
  1. Starting or stopping of machine
  2. Controlling the spindle speed
  3. Positioning the tool tip at desired locations and guiding it along the desired paths by automatic control of the motion of the slides.
  4. Controlling the rate of movement of the tool tip (feed)
  5. Changing the tools in the spindle

# History of CNC

---

- 1947 - John Parsons and US Air Force define a need to develop a machine tool capable of machining complex and close tolerance aircraft parts with the same quality time after time (repeatability).
- 1952 - MIT is the subcontractor and builds the machine for the project.
- 1959 - MIT announces Automatic Programmed Tools (APT) programming language
- 1960 - Direct Numerical Control (DNC). This eliminates paper tape punch programs and allows programmers to send files directly to machine tools

# History of CNC

---

- 1968 - Kearney & Trecker machine tool builders market first machining center
- 1970's - CNC machine tools & Distributed Numerical Control
- 1980's - Graphics based CAM systems introduced. Unix and PC based systems available
- 1990's - Price drop in CNC technology
- 1997 - PC- Windows/NT based “Open Modular Architecture Control (OMAC)” systems introduced to replace “firmware” controllers.

# Motivation and uses

---

To manufacture complex curved geometries in 2D or 3D was extremely expensive by mechanical means (which usually would require complex jigs to control the cutter motions)

Machining components with repeatable accuracy

Unmanned machining operations

# History

---

- **First generation – Vacuum tubes**

Bulky, high power consumption and poor reliability

- **Second generation – Transistors**

Reduced size

Large no. of components and connections, poor reliability

- **Third generation – Integrated circuits**

Reliable and compact

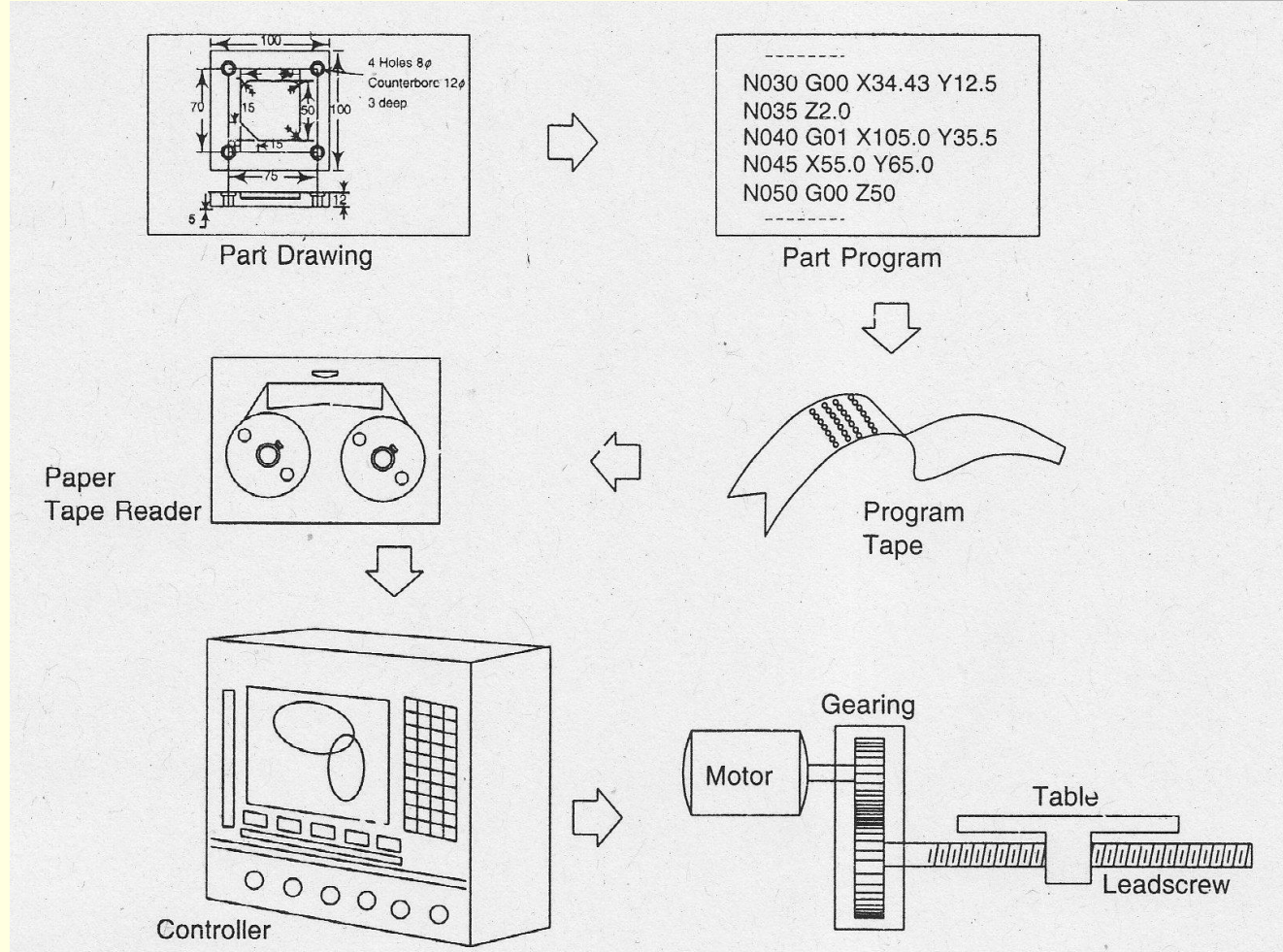
Use of computers, DNC

Easier programming, flexibility

- **Fourth generation - Microprocessors**

Simplified logic, control and design

# Basic components of NC system





N0 4 G01 X 7600 F150  
 N0 5 Y 40000

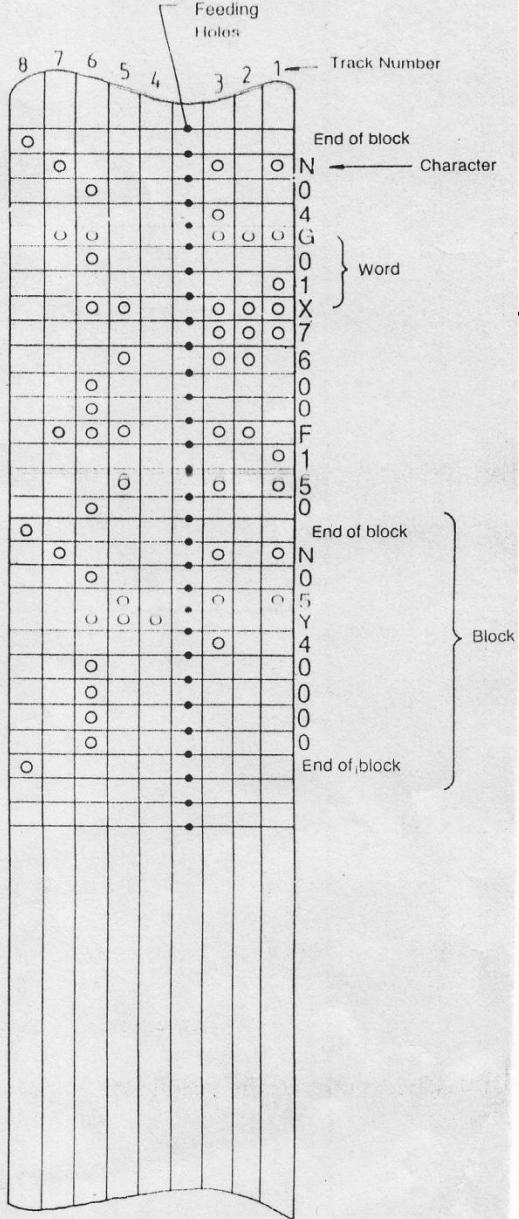
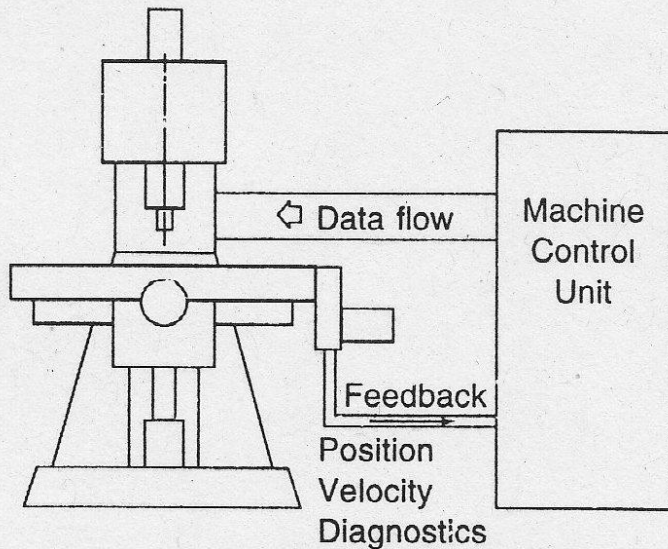


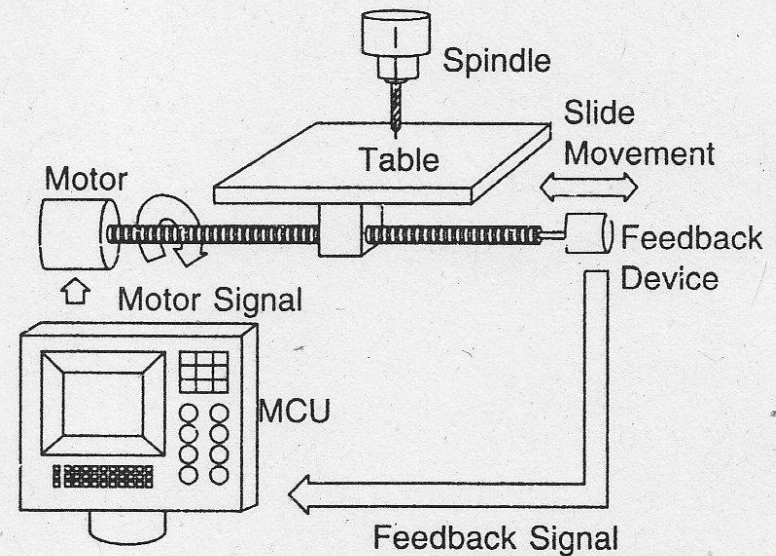
Fig. 4.48 EIA coded program on a Punched Tape

# Punched Tape

# Closed loop control system

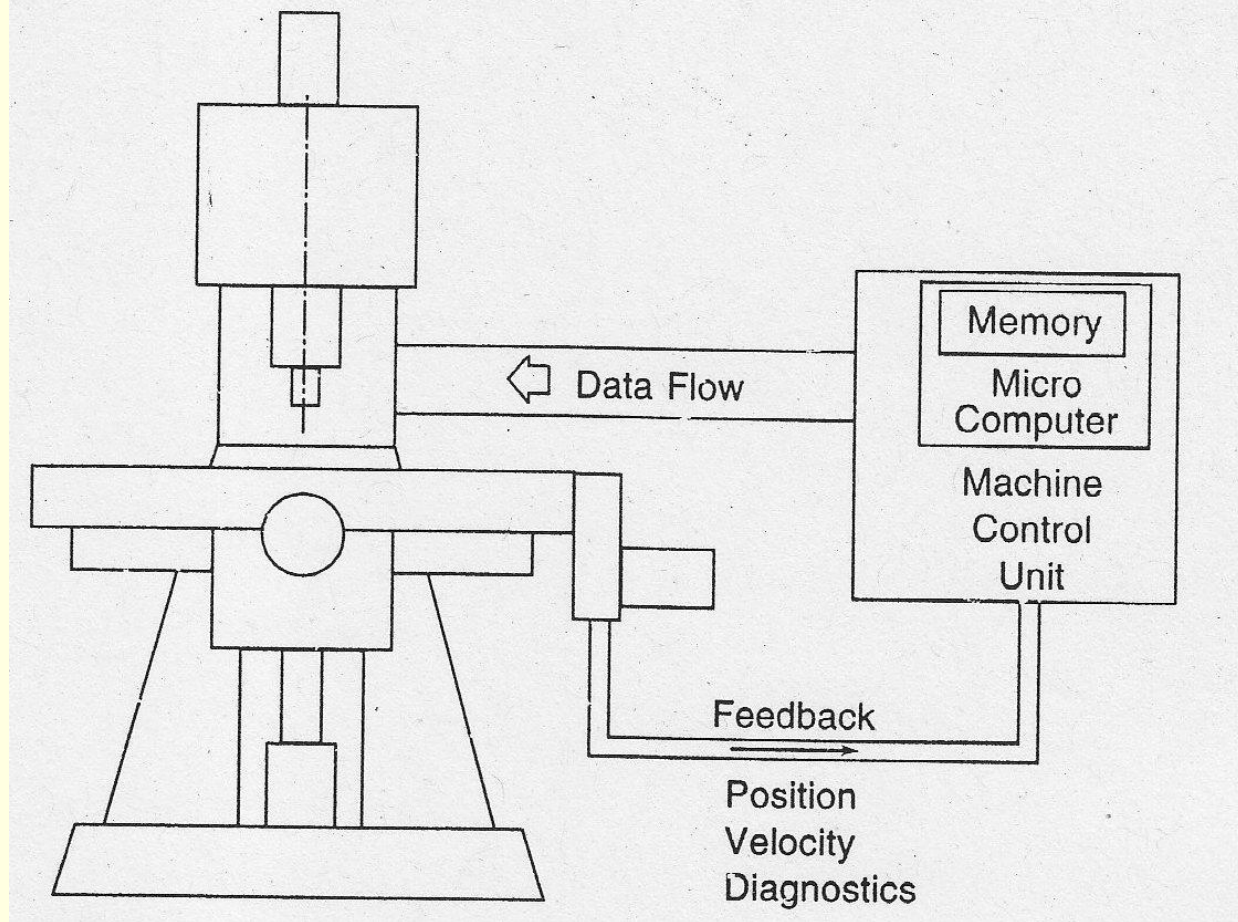


**Fig. 9.4** *The Data Processing in a CNC Machine Tool in Closed Loop Control*



**Fig.9.5** *The Data Processing in a CNC Machine Tool in Closed Loop Control*

# Operation of CNC machine



# Basic components of NC system

---

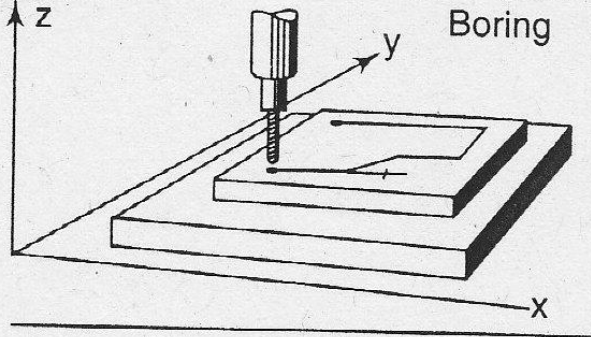
- 1) **Program of instructions** - Step by step set of directions which tell machine tool what to do.
- 2) **Controller unit (MCU)** - Electronic and computerized interface between operator and machine that read and interpret program of instructions and convert it into mechanical actions of machine tool.

**Elements** – Tape reader, Data buffer, Signal output channels, feedback channels from machine tool and sequence control to coordinate operation, control panel which contains dials, switches by which machine operator controls the NC machine.

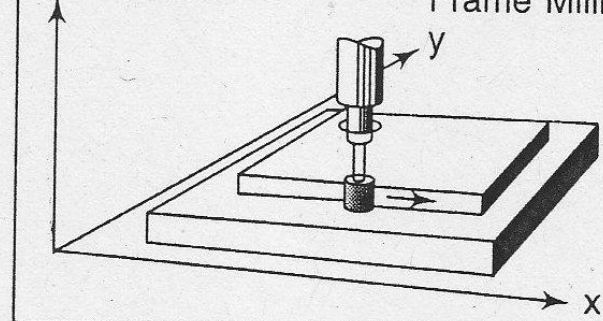
- 3) **Machine tool or other controlled process**  
Machining centre, Welding, Assembly etc.

# NC motion control systems

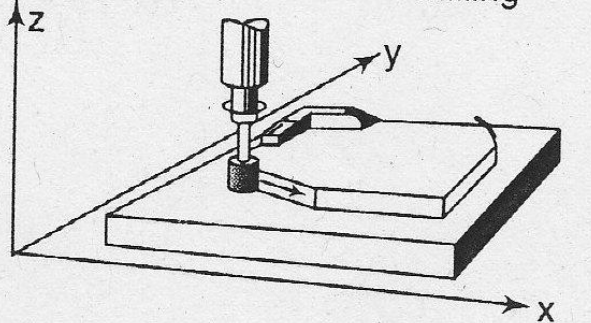
Point-to-point



Point-to-point  
Straight Line

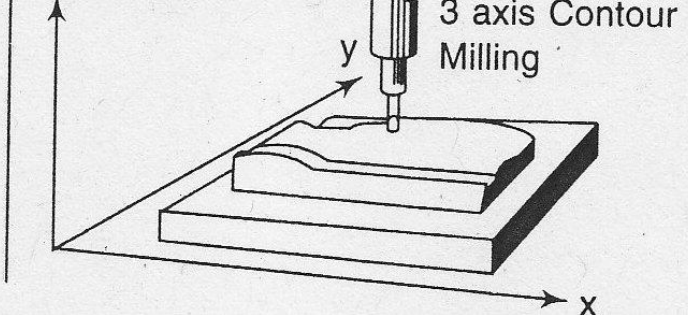


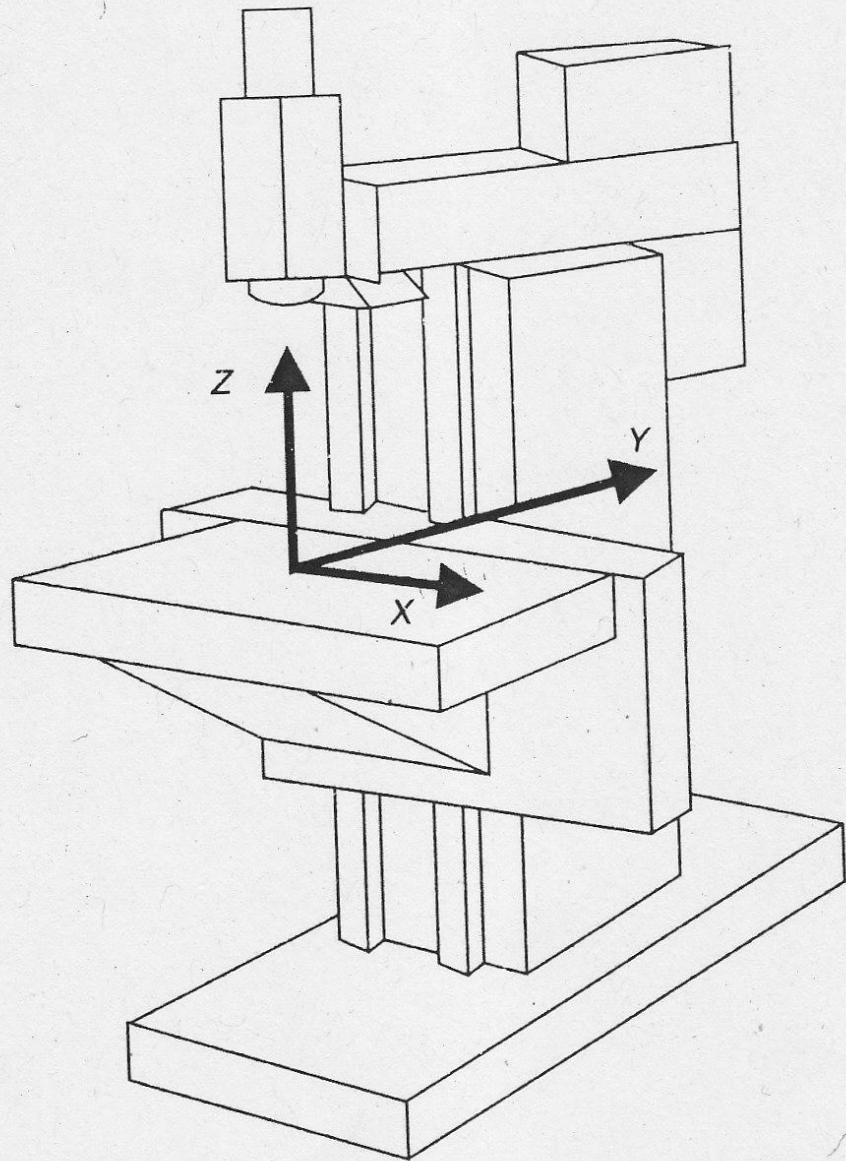
2 axis Contouring  
with Switchable Plane



2 axis Contour  
Milling

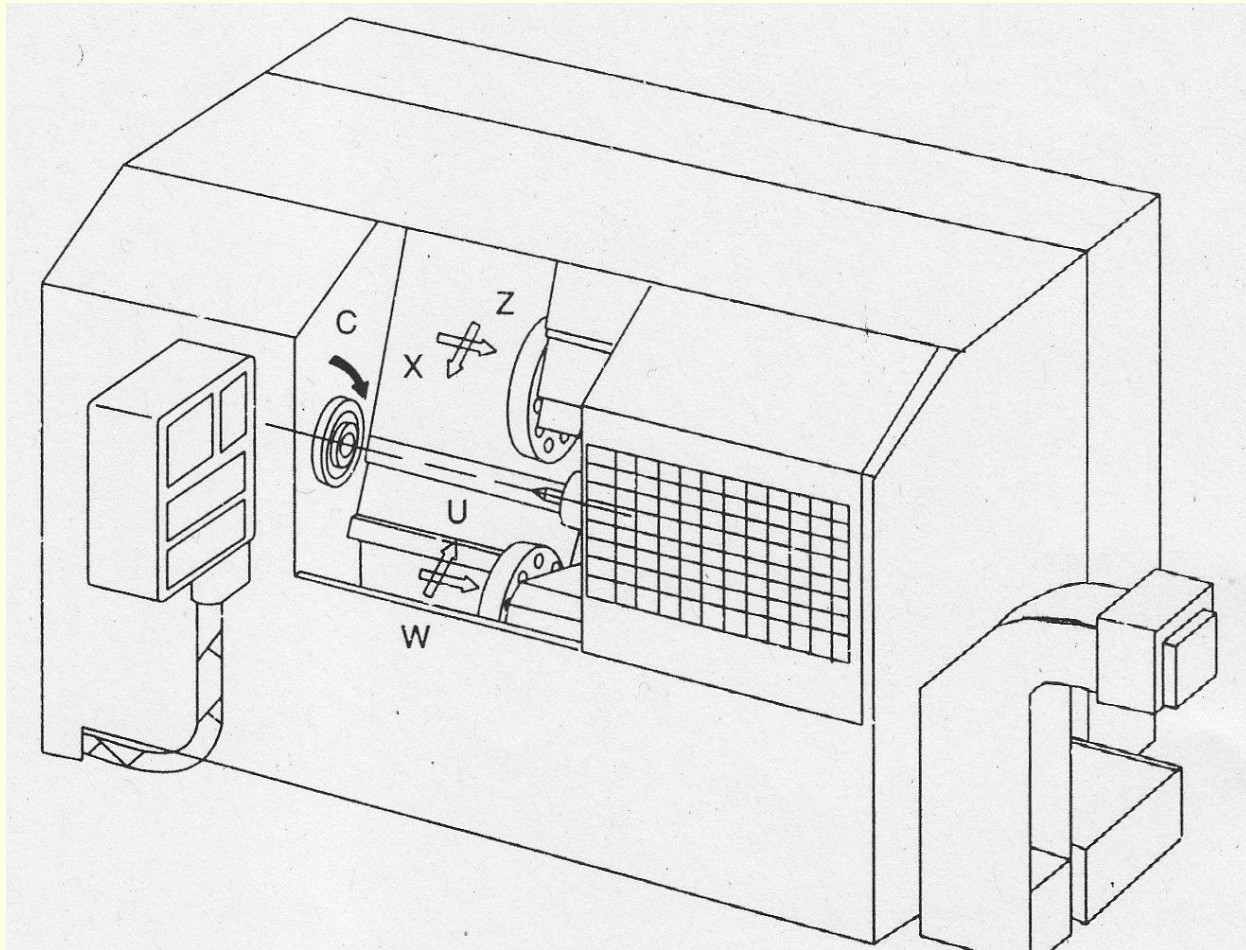
3 axis Contouring  
Continuous Path



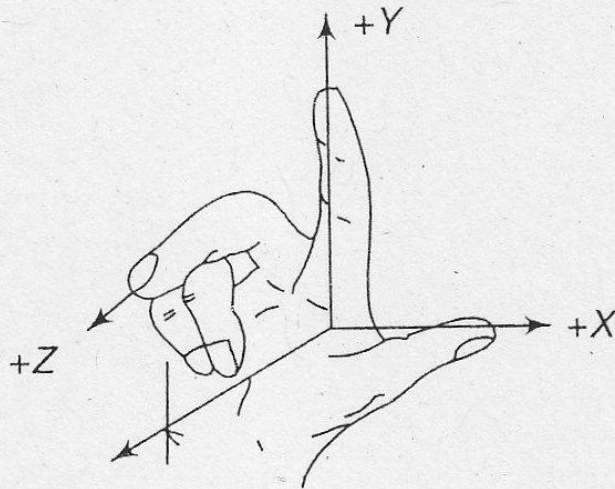


10.28 Vertical Axis Milling Machine or Machining Centre

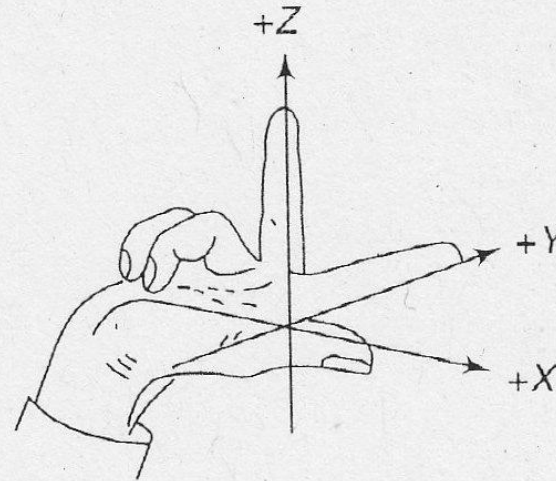
# CNC lathe



# Direction



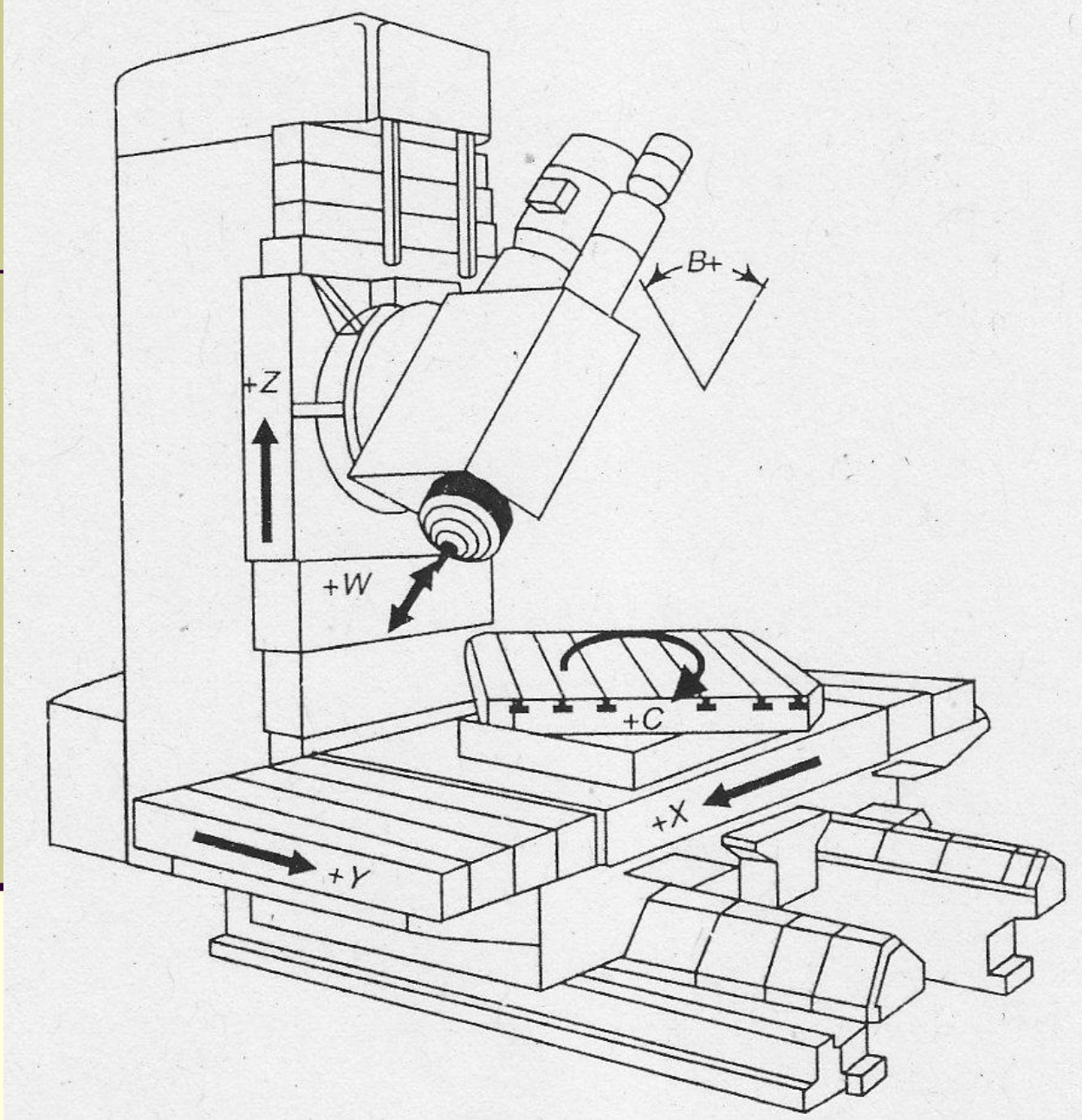
(a) Axis Designation for Horizontal Z



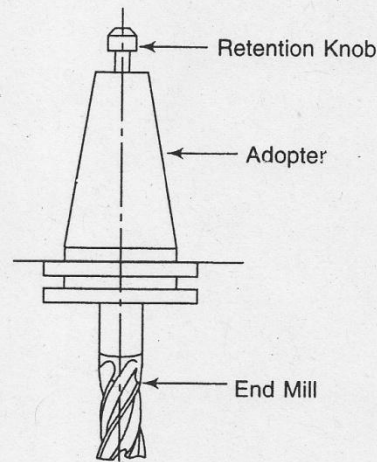
(b) Axis Designation for Vertical Z


*Finding Directions in a Right Hand Co-ordinate System and also the Positive Directions for Rotary Motions*

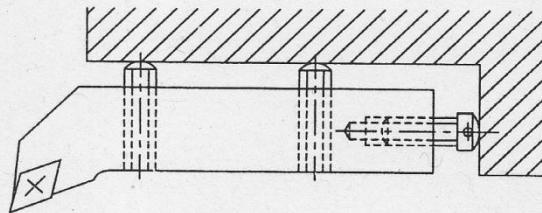





# NC tooling



 **Fig. 9.9** *Typical Spindle Tooling Holding an End Mill*



 **Fig. 9.10** *Typical Preset Tooling used in CNC Turning Machines*

# NC procedure

---

1. Process Planning
2. Part programming
  - Manual part programming
  - Computer Assisted part programming
3. Tape preparation
4. Tape verification
5. Production

# Numerical control

---

- Fixed zero and floating zero
- Absolute programming and incremental positioning
- Inside tolerance, outside tolerance
- Subroutine programming / Macro
- Parametric programming
- Canned cycles
- Diagnostic features – Real time, On-line, Off-line

# Applications

---

1. Machining e.g.milling, drilling, turning, boring, grinding
2. Machining centre – Horizontal, vertical, Universal
3. Gear hobbing / shaping
4. Press working
5. Welding
6. Tube bending
7. Flame cutting
8. Laser beam /Electron beam process
9. Plasma arc cutting
10. Automatic riveting
11. Assembly
12. Inspection
13. Coordinate measuring machine

# Advantages of NC

---

1. Higher productivity - Reduction in no. and time of set-up, Work piece handling time, tool changing time
2. Lower cost of production – Reduced fixturing, reduced manufacturing lead time, Complex geometry is produced as cheaply as simple ones
3. Greater manufacturing flexibility
4. Improved quality control - Avoids human errors, Usually generates closer tolerances than manual machines
5. Reduced inventory
6. Reduced floor space requirements
7. NC machines are safer to operate

# Disadvantages

---

1. Higher investment cost
2. Higher maintenance cost
3. Finding and training of NC personnel

- 
- NC is suitable for components with complex shapes, batch production, numerous or costly set-ups, close tolerances, design changes, high inspection cost
  - Development in NC was major factor for evolution of CAD/CAM and CIM



# Manual CNC programming

---

- **Part program:** A computer program to specify
  - Which tool should be loaded on the machine spindle?
  - What are the cutting conditions (speed, feed, coolant ON/OFF etc)?
  - The start point and end point of a motion segment?
  - how to move the tool with respect to the machine?

# Word address format

Sequence and format of words:

N5 G2 X<sub>±</sub>53 Y<sub>±</sub>53 Z<sub>±</sub>53 U V W I J K F5 S4 T4 M2

sequence no

destination coordinates

dist to center of circle

tool

preparatory function

feed rate

spindle speed

miscellaneous function

# Word address format

---

- Each instruction to the machine consists of a letter followed by a number.
- Each letter is associated with a specific type of action or piece of information needed by the machine.
- **G-codes**: Preparatory Functions – involve actual tool moves.
- **M-codes**: Miscellaneous Functions – involve actions necessary for machining (i.e. spindle on/off, coolant on/off).

# Letter Codes (G Codes)

**G00 Rapid traverse**

**G01 Linear interpolation**

**G02 Circular interpolation,  
CW**

**G03 Circular interpolation,  
CCW**

**G04 Dwell**

**G08 Acceleration**

**G09 Deceleration**

**G17 X-Y Plane**

**G18 Z-X Plane**

**G19 Y-Z Plane**

**G20 Inch Units (G70)**

**G21 Metric Units (G71)**

**G40 Cutter compensation –  
cancel**

**G41 Cutter compensation –  
left**

**G42 Cutter compensation-  
right**

**G70 Inch format**

**G71 Metric format**

**G80 Fixed-cycle cancel**

**G81-G89 Fixed cycles**

**G90 Absolute dimensions**

**G91 Incremental dimensions**

# Letter Codes (M Codes)

---

M00 Program stop

M01 Optional program stop

M02 Program end

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

M30 Program stop, reset to start

# Letter Codes

---

- **N-codes:** Gives an identifying number for each block of information. It is generally good practice to increment each block number by 5 or 10 to allow additional blocks to be inserted if future changes are required.
- **X, Y, and Z** codes are used to specify the coordinate axis.
- Number following the code defines the coordinate at the end of the move relative to an incremental or absolute reference point.
- The number may require that a specific format be used (i.e. 3.4 means three numbers before the decimal and four numbers after the decimal).

# Letter Codes

---

- **I, J, and K** codes are used to specify the coordinate axis when defining the center of a circle.
- Number following the code defines the respective coordinate for the center of the circle.
- The number may require that a specific format be used (i.e. 3.4 means three numbers before the decimal and four numbers after the decimal).
- **F-code**: used to specify the feed rate
- **S-code**: used to specify the spindle speed
- **T-code**: used to specify the tool identification number associated with the tool to be used in subsequent operations.

# Letter Codes

---

- **R-code:**
- Retract distance when used with G81, 82, and 83.
- Radius when used with G02 and G03.
- **P-code:** Used to specify the dwell time associated with G04.



# Program Start Flag and Program Identification Number

---

- % Program start flag. Placed at the beginning of each program.
- **:1025** Program identification number.
- Follows the program start flag at the beginning of each program.
- Only two commands that don't start with a letter.

# Block Format

---

- **Sample Block**

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

- 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 feedrate
- 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

# Modal G-Codes

---

- Most G-codes set the machine in a “mode” which stays in effect until it is changed or cancelled by another G code.
- These commands are called “modal”.

# Modal G-Code List

- G00 Rapid Transverse**
- G01 Linear Interpolation**
- G02 Circular Interpolation, CW**
- G03 Circular Interpolation, CCW**
- G17 XY Plane**
- G18 XZ Plane**
- G19 YZ Plane**
- G20/G70 Inch units**
- G21/G71 Metric Units**
- G40 Cutter compensation cancel**
- G41 Cutter compensation left**
- G42 Cutter compensation right**
- G43 Tool length compensation (plus)**
- G44 Tool length compensation (minus)**
- G49 Tool length compensation cancel**
- G80 Cancel canned cycles**
- G81 Drilling cycle**
- G82 Counter boring cycle**
- G83 Deep hole drilling cycle**
- G90 Absolute positioning**
- G91 Incremental positioning**