Introduction to Numerical Control (NC) Machines

V.Gunasegaran

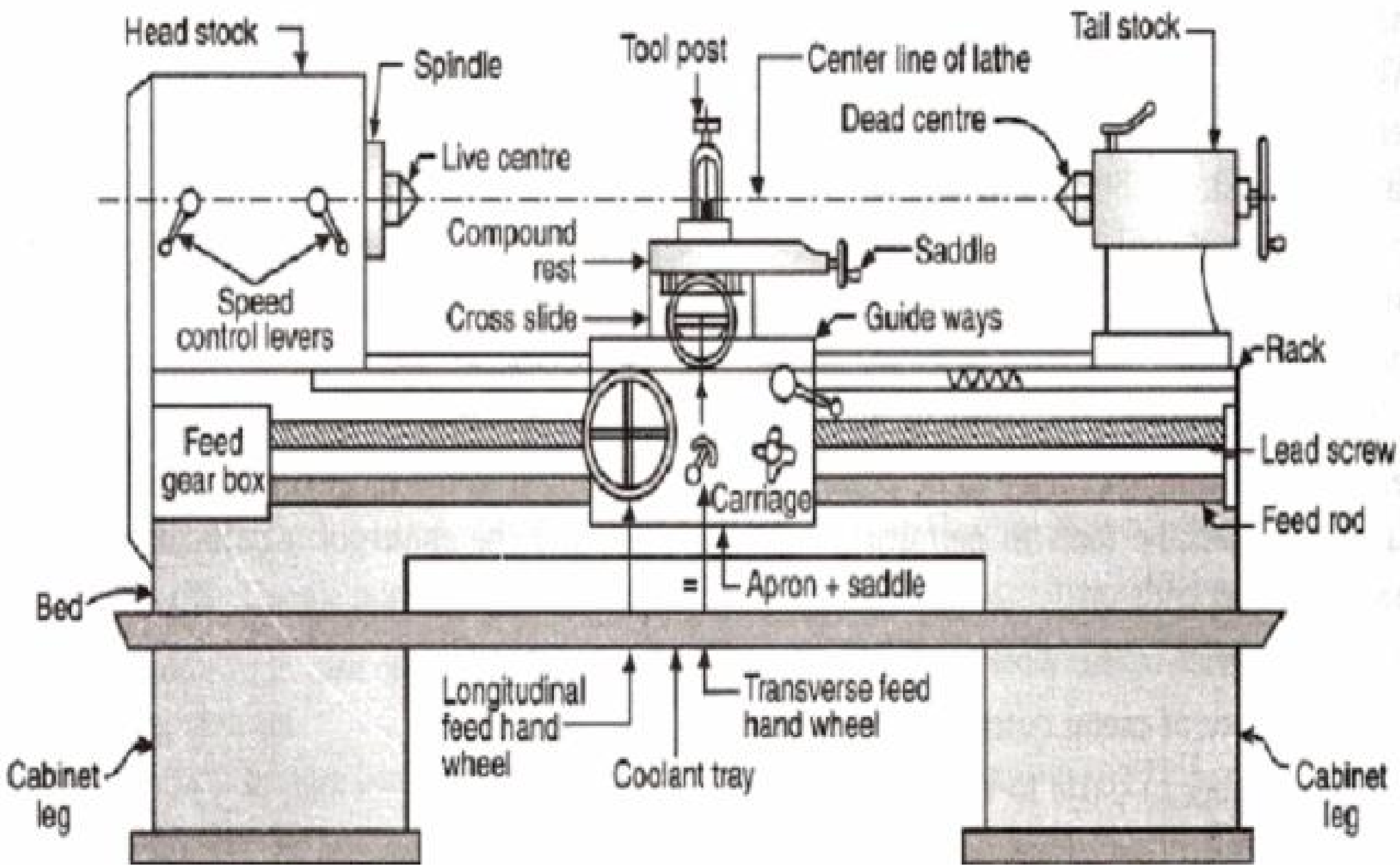
Assistant Professor

Department of Mechanical Engineering

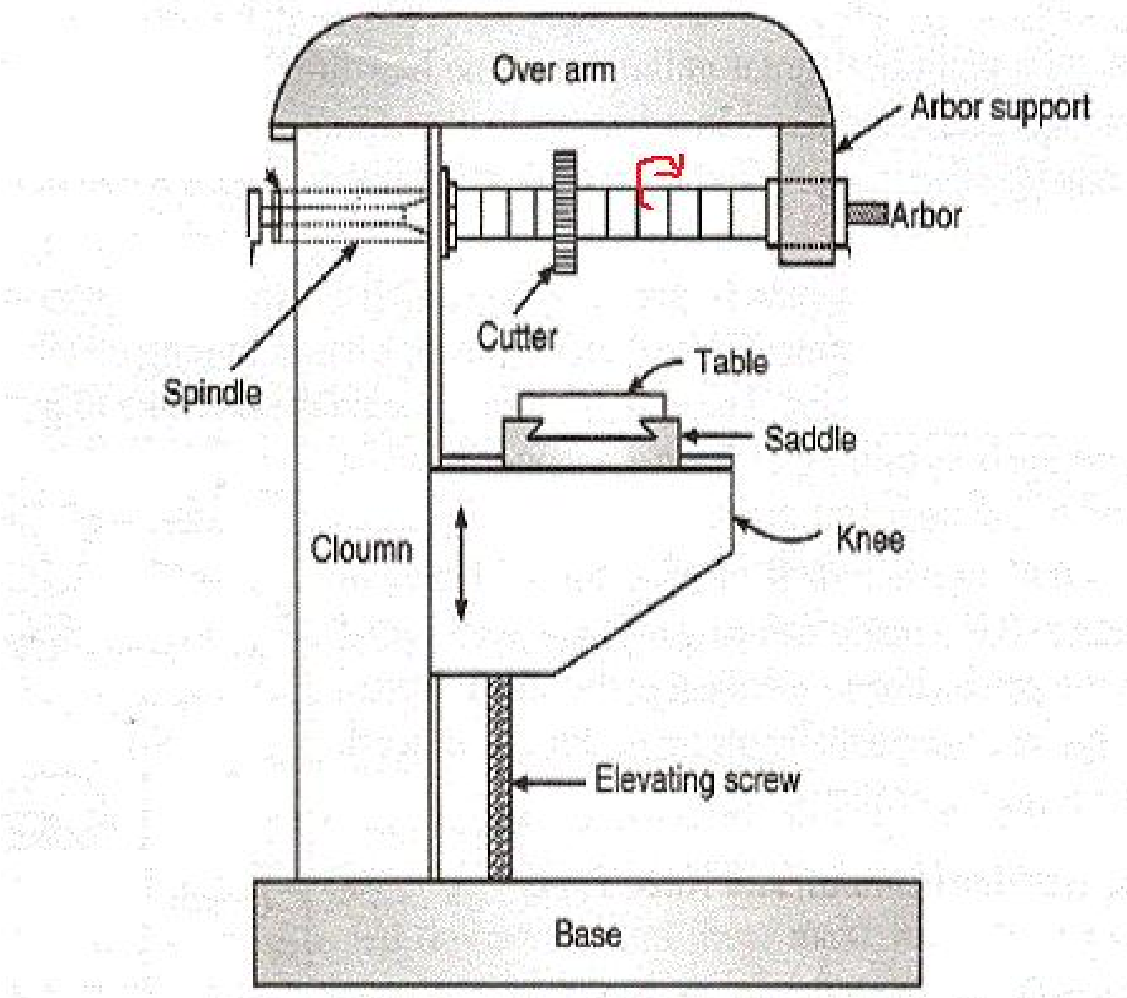
School of Mechanical Sciences

BSAU, Chennai - 48

Conventional Lathe machine

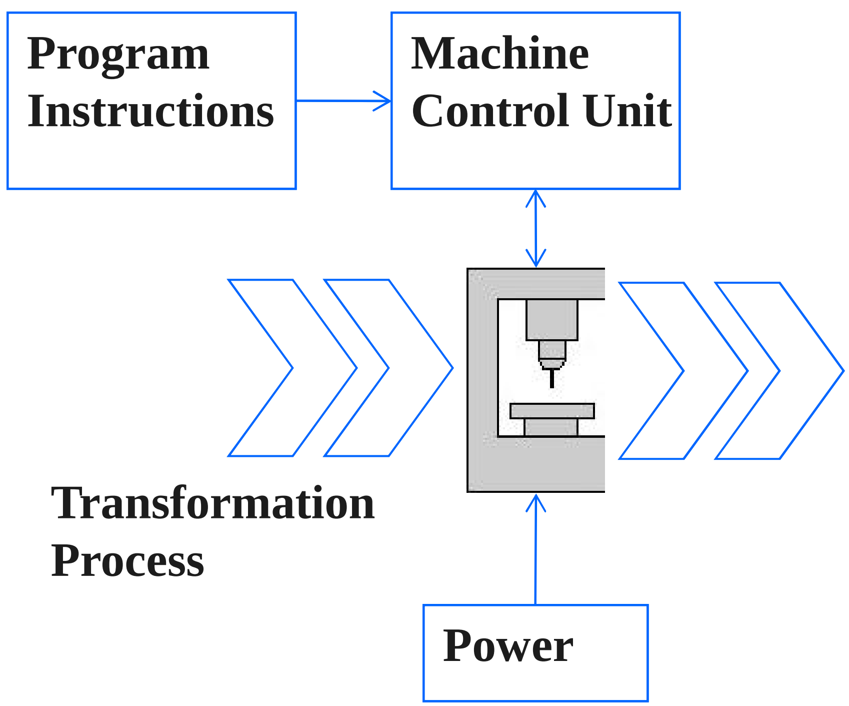


Conventional Milling machine



Numerical Control machine

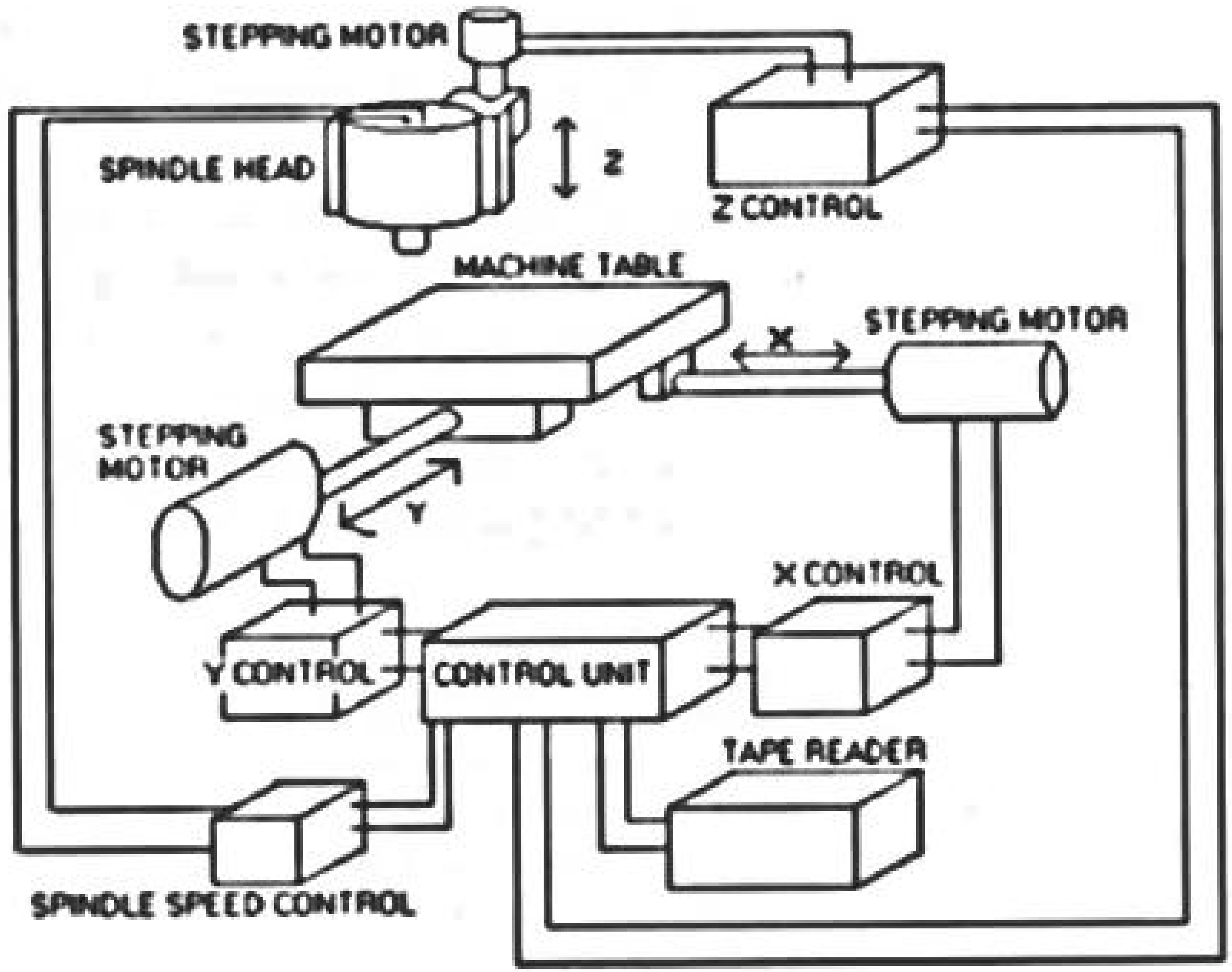
- ✓ Programmable automation in which the mechanical actions of a 'machine tool' are controlled by a program containing coded alphanumeric data that represents relative positions between a cutting tool and a work part.



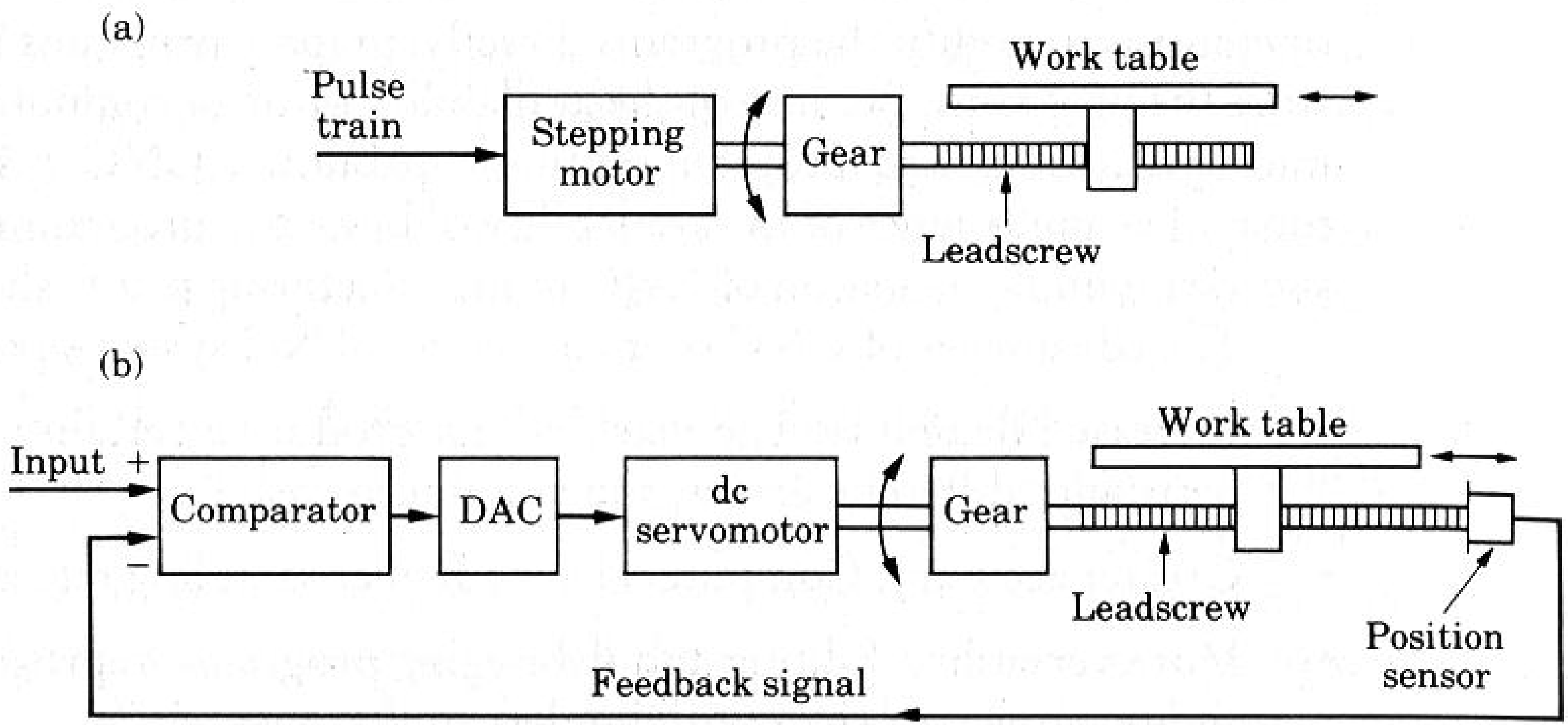
Difference between Conventional m/c & NC m/c

Item	Conventional machine	NC machine
1. Movement	Acme screw	Ball screw
2. Feed	manual	motor
3.measurement	manual	Linear scale

Control System



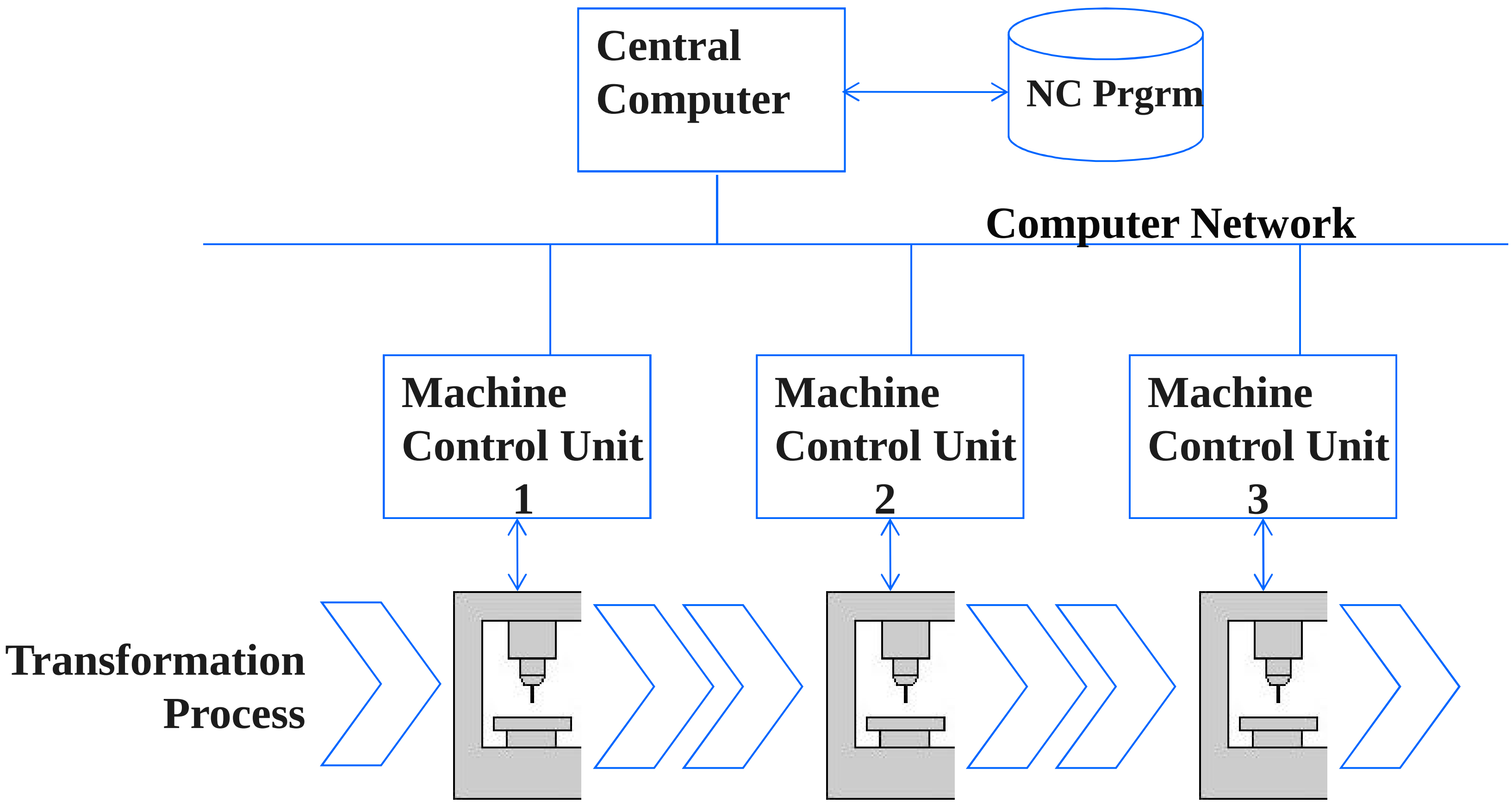
Open and Closed loop



Direct Numerical Control

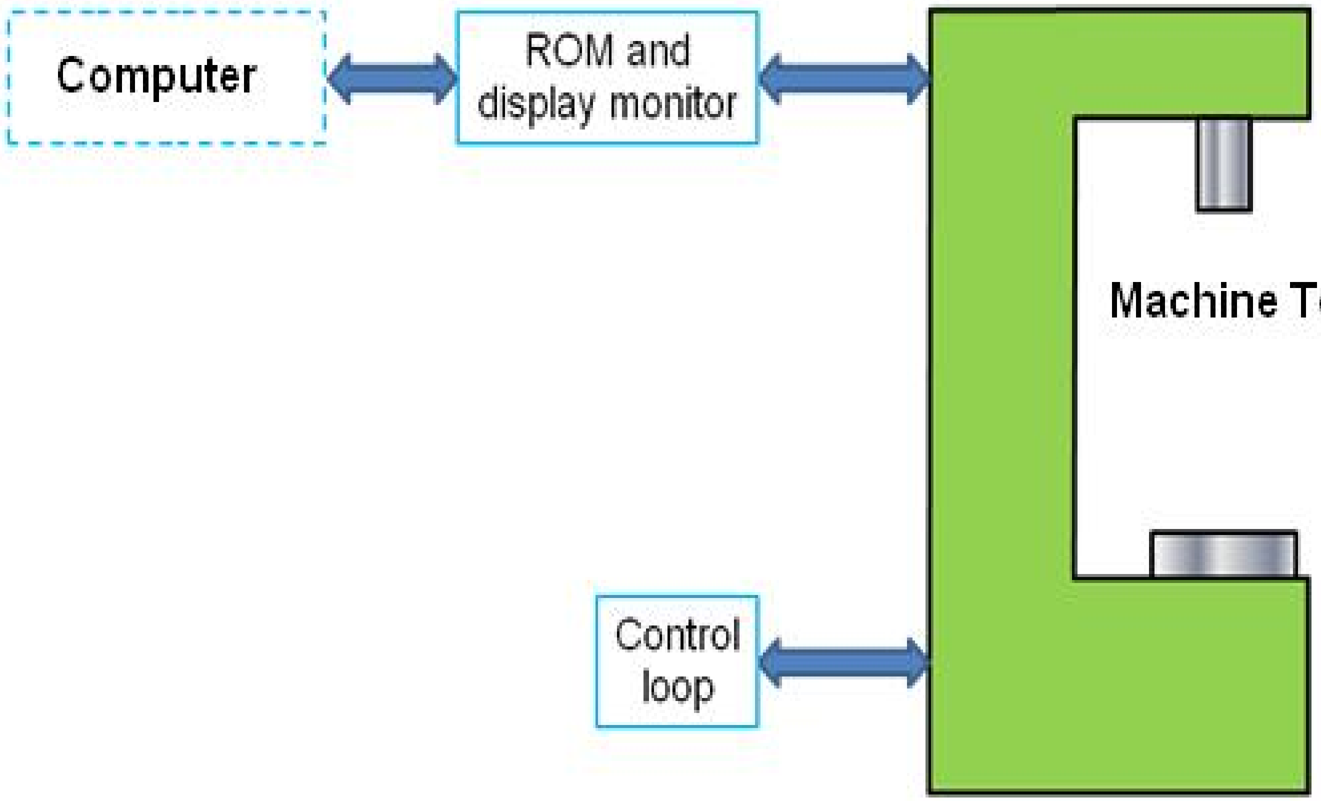
- ✓ Direct numerical control (DNC) – control of multiple machine tools by a single (mainframe) computer through direct connection and in real time
  - 1960s technology
  - Two way communication

DNC – contd.,

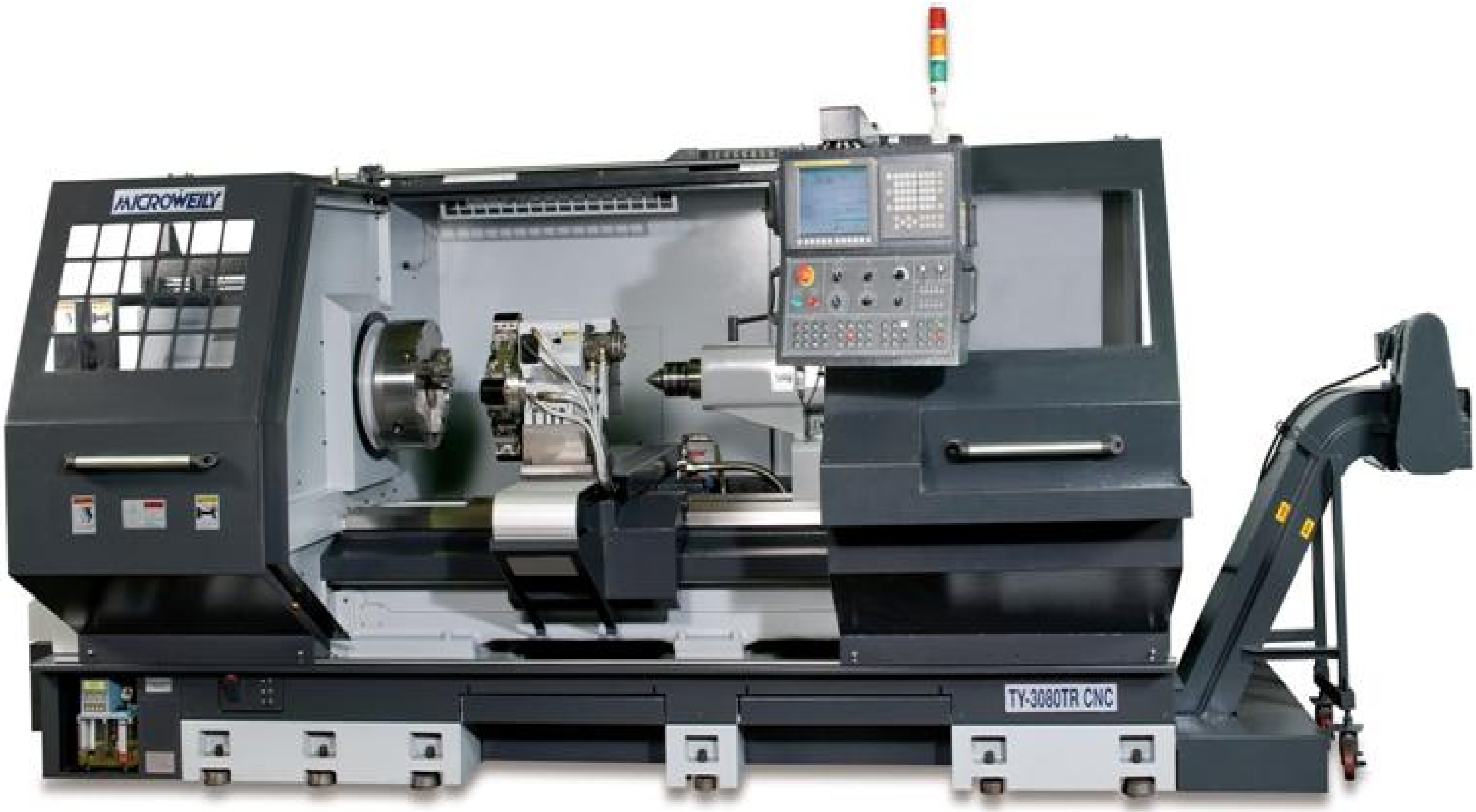

Central Computer
NC Prgrm
Computer Network
Machine Control Unit 1
Machine Control Unit 2
Machine Control Unit 3
Transformation Process

Computer Numerical Control (CNC) Machines

Basic CNC Principle



Photography of CNC turner

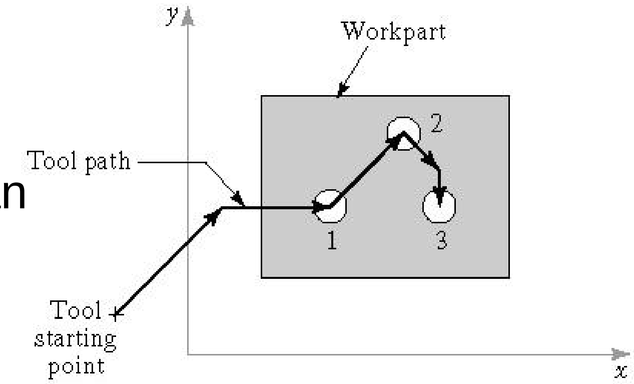


Photography of CNC miller

Motion Control methods

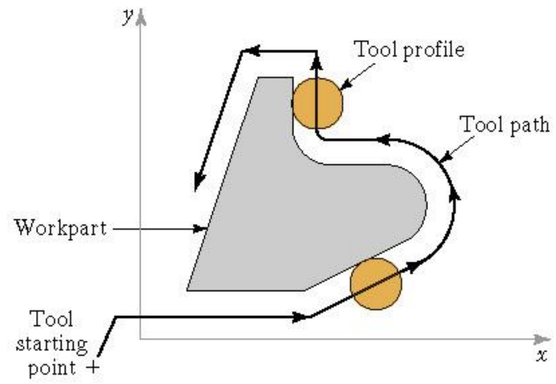
Point-to-Point systems

- ✓ Also called position systems
- ✓ System moves to a location and performs an operation at that location (e.g., drilling)
- ✓ Also applicable in robotics



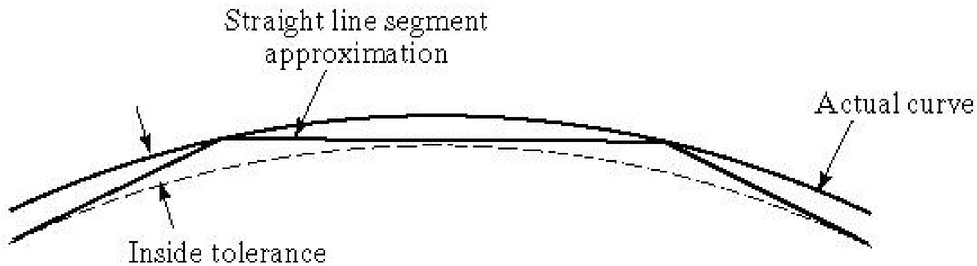
Continuous path systems

- ✓ Also called contouring systems in machining
- ✓ System performs an operation during moven (e.g., milling and turning)



Interpolation methods

1. Linear interpolation
   – Straight line between two points in space
2. Circular interpolation
   – Circular arc defined by starting point, end point, center or radius, and direction
3. Helical interpolation
   – Circular plus linear motion
4. Parabolic and cubic interpolation
   – Free form curves using higher order equations



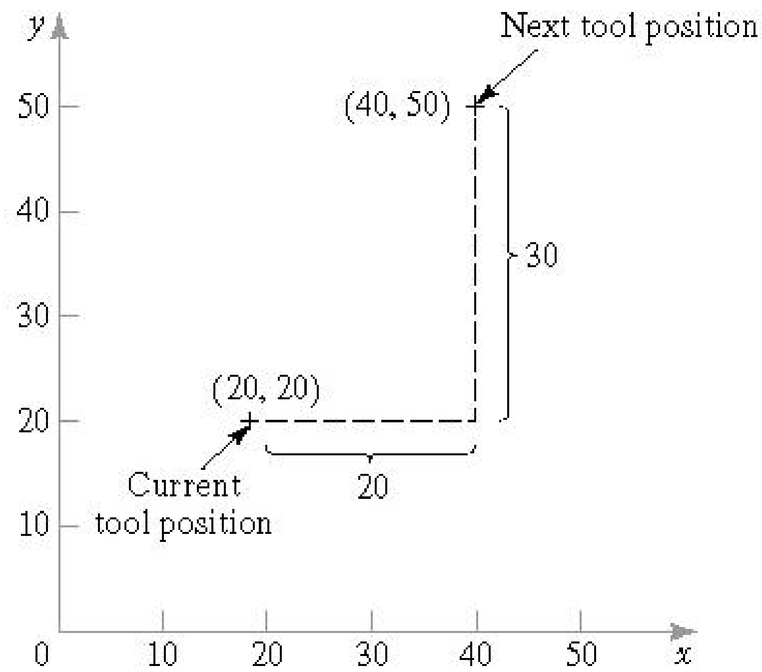
Absolute vs. Incremental Positioning

- ✓ Absolute positioning

  Move is: x = 40, *y* = 50

- ✓ Incremental positioning

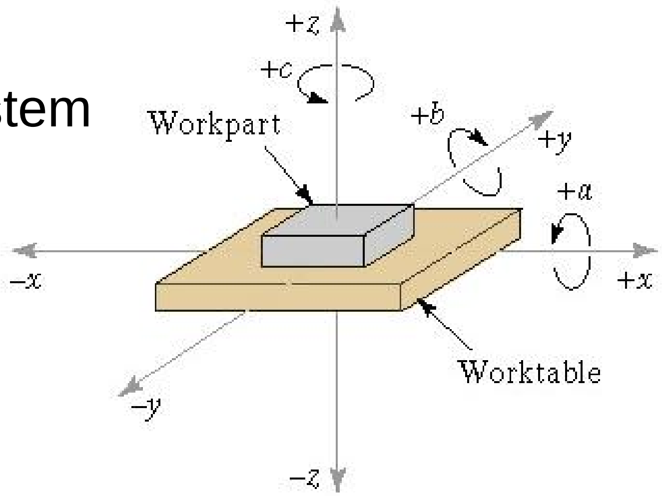
  Move is: x = 20, *y* = 30



Coordinate Systems

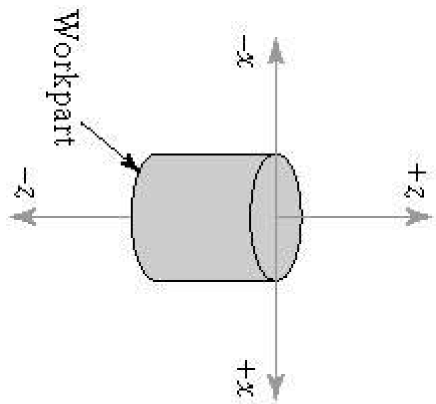
For flat and prismatic (block-like) parts:

- ✓ Milling and drilling operations
- ✓ Conventional Cartesian coordinate system
- ✓ Rotational axis about each linear axis

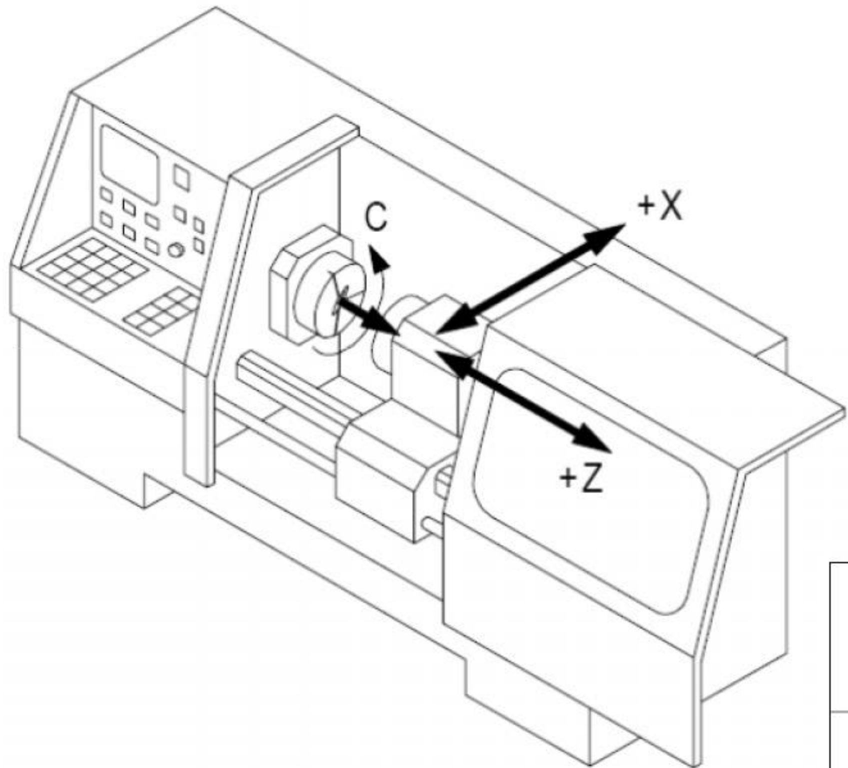


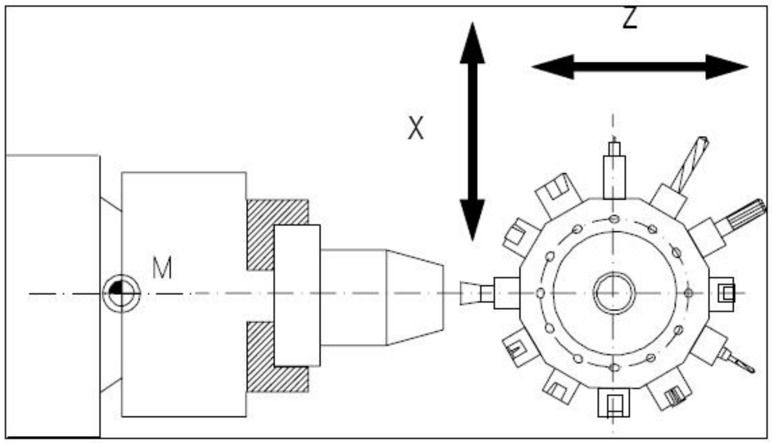
For rotational parts:

- ✓ Turning operations
- ✓ Only *x*- and *z*-axis

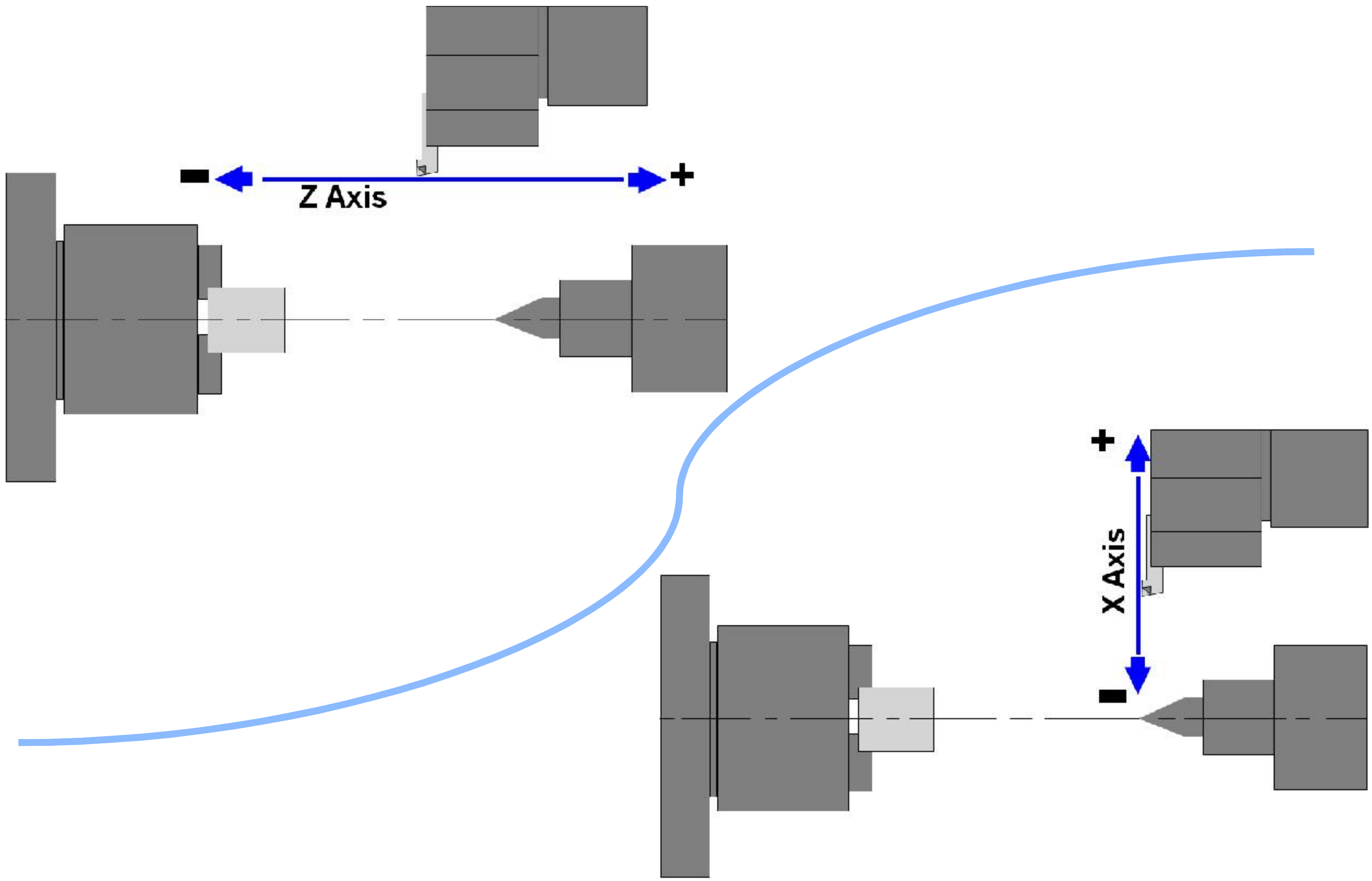


Axes on Lathe machine tool

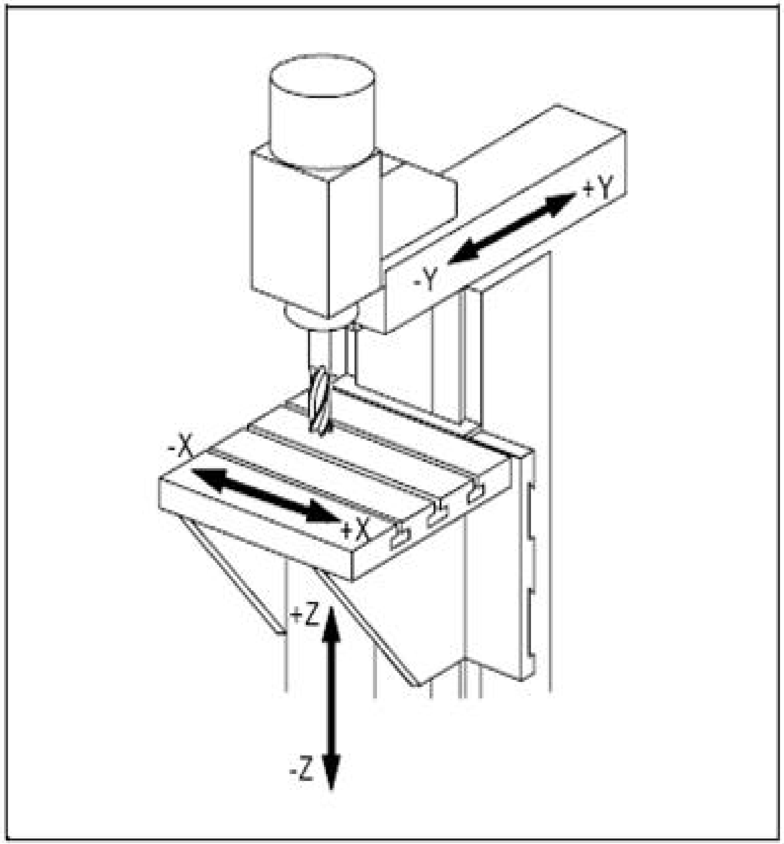




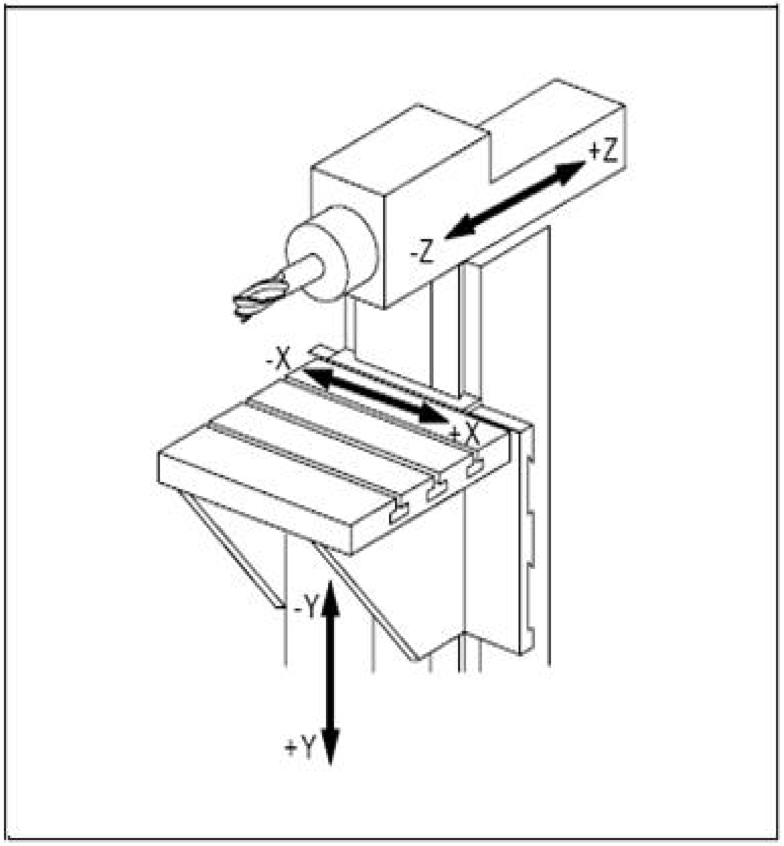
Contd.,



Axes on Milling machine tool

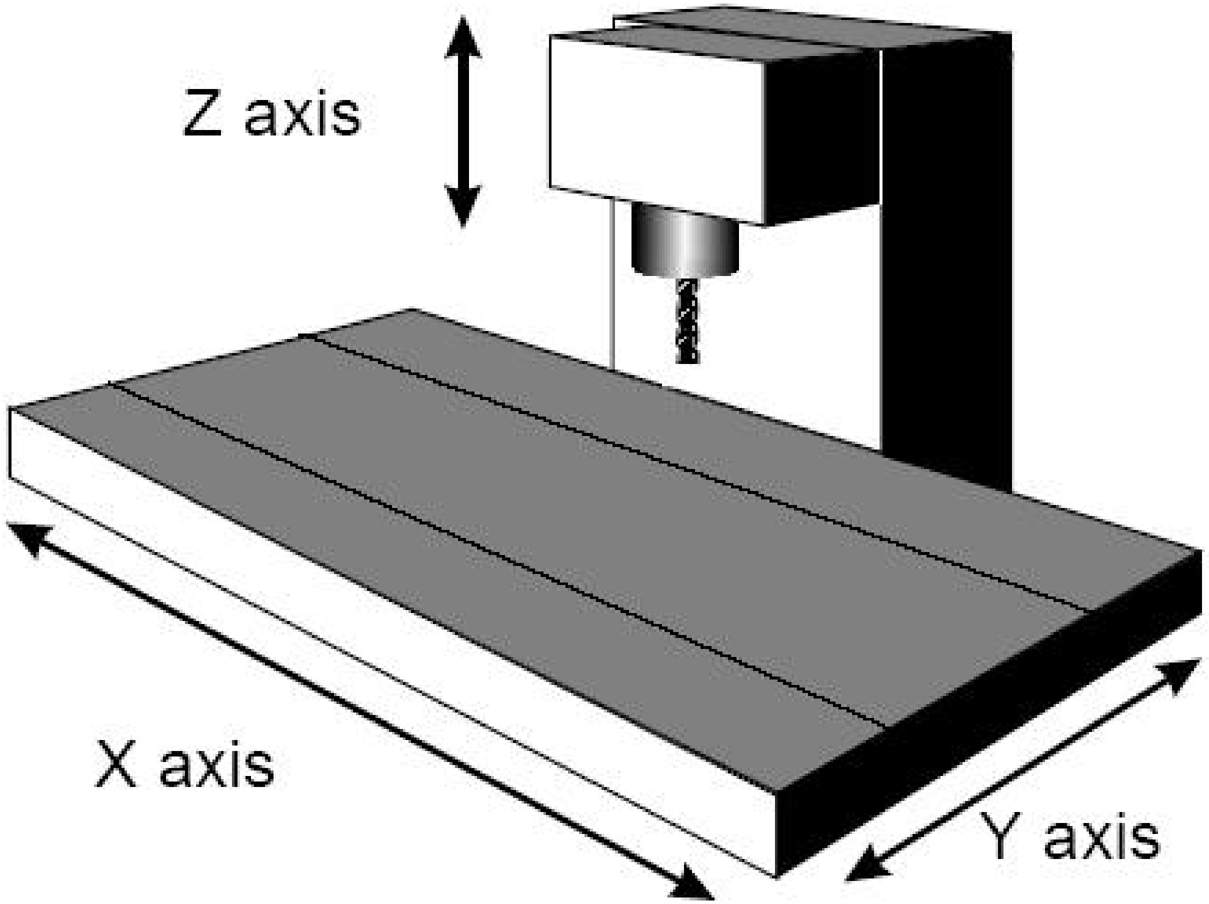


Axis on the vertical milling machine



Axis on the horizontal milling machine

Contd.,



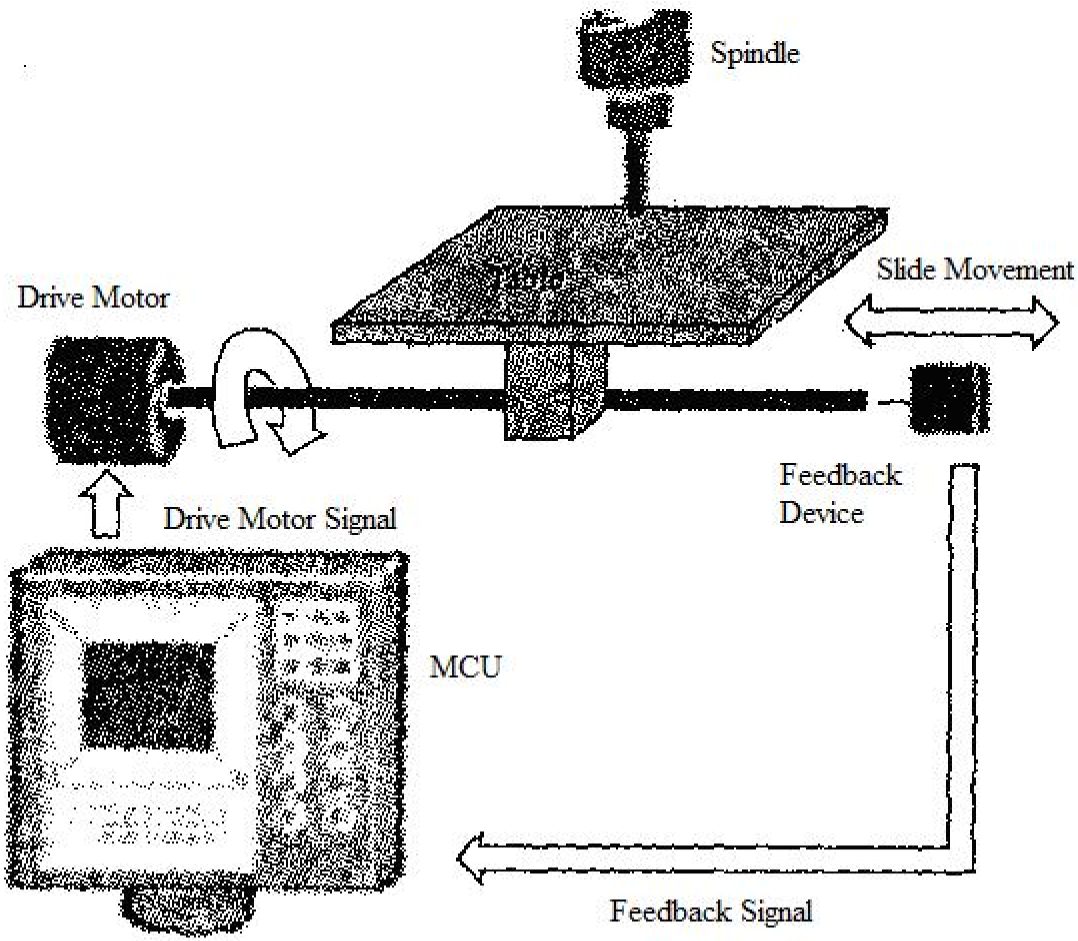
Classifications

- ✓ Based on Machine centre:
  Turn mill centre / Vertical mill centre
- ✓ Based on Motion Type:
  Point-to-Point / Continuous path
- ✓ Based on Control Loops:
  Open loop / Closed loop
- ✓ Based on Input power:
  Electric / Hydraulic / Pneumatic
- ✓ Based on Positioning System
  Incremental / Absolute

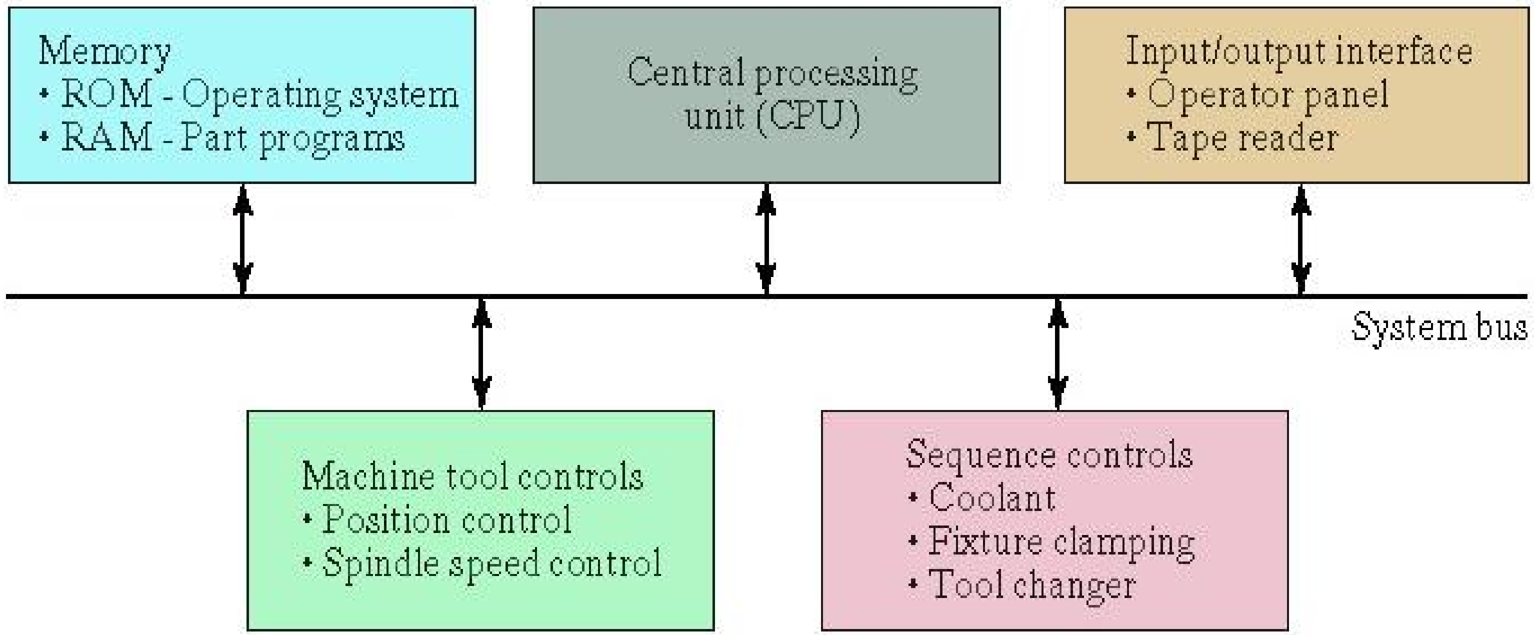
Constructional features of CNC

- ✓ The tool / work part (table) moves
- ✓ Tools can operate in 1-5 axes
- ✓ Larger machines have a machine control unit (MCU) which manages operations
- ✓ Movement is controlled by servo motors (actuators)
- ✓ Feedback is provided by sensors (transducers)
- ✓ Tool magazines are used to change the tools automatically
- ✓ Relative movement of the tool with respect to workpiece or vice versa can be guided and controlled by a Part program

Feed control system



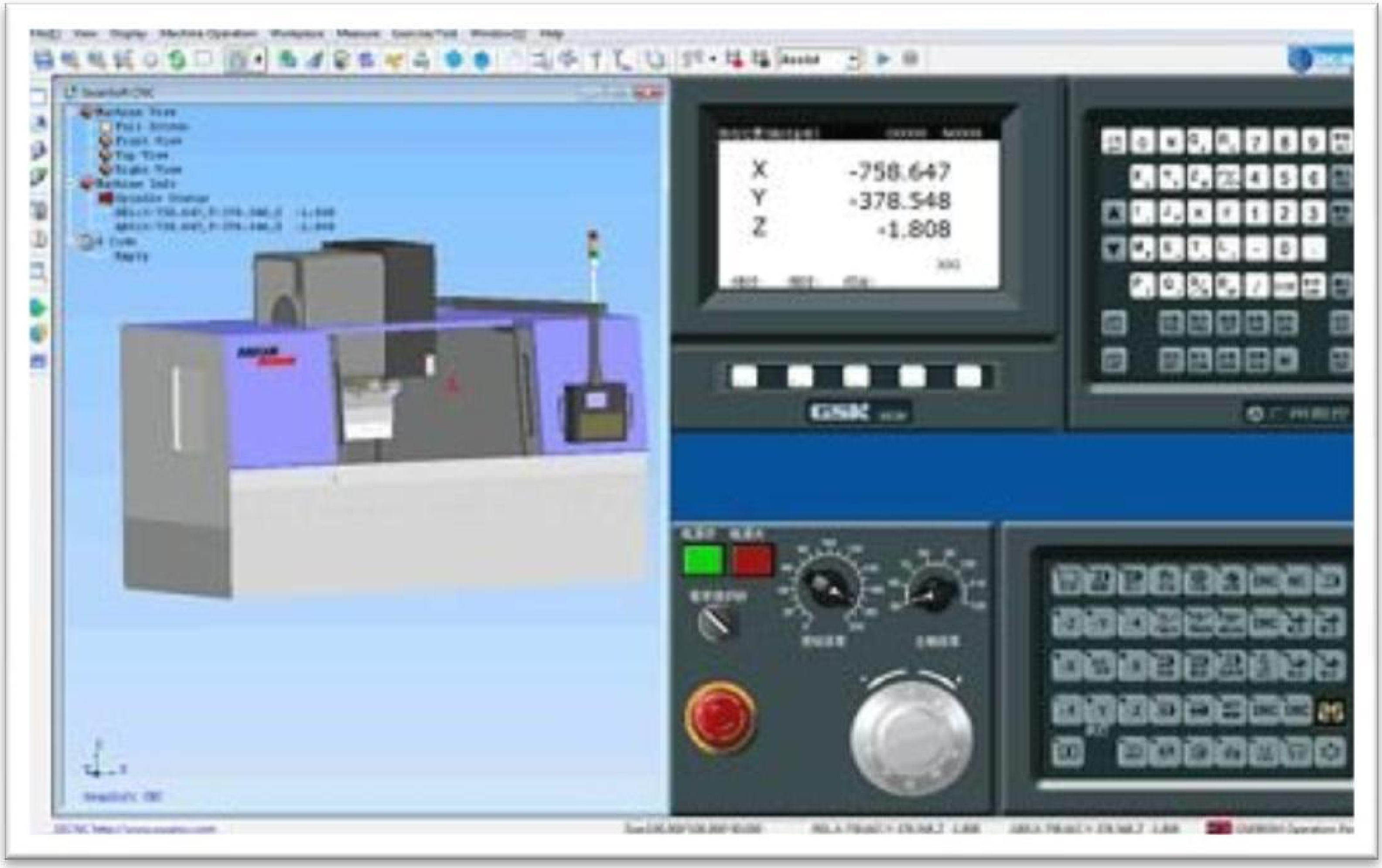
Machine Control Unit



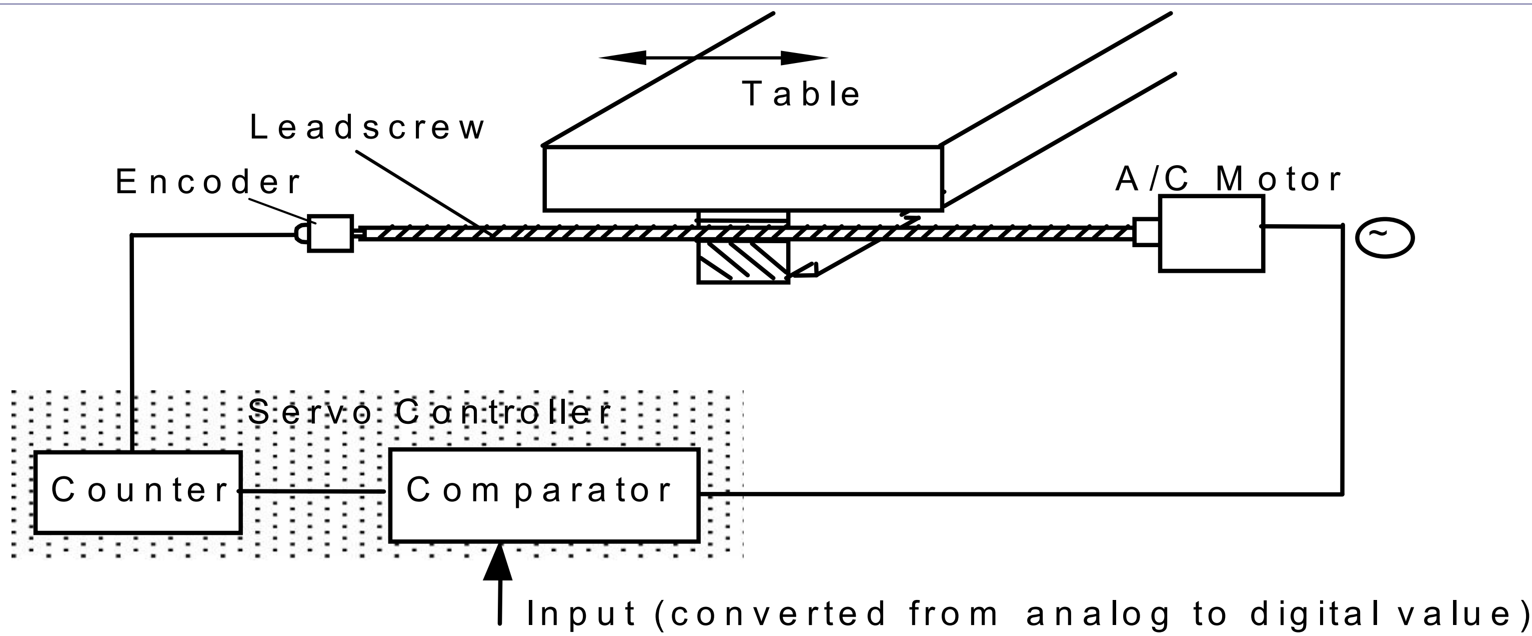
Components and its Functions

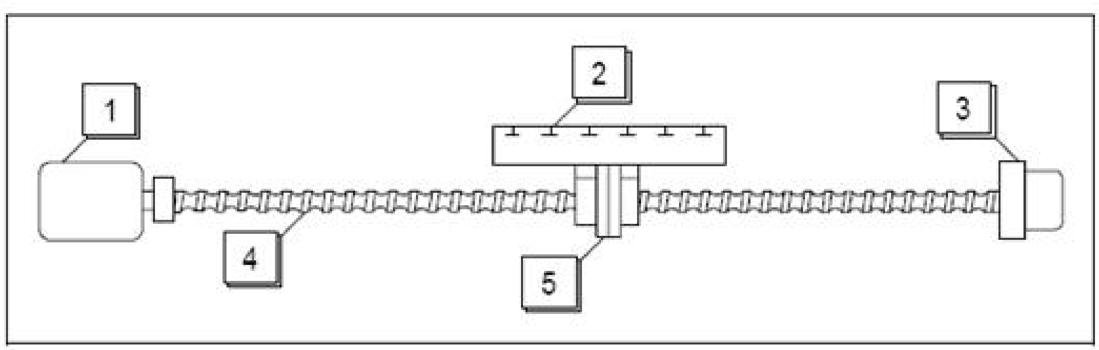
- ✓ On board computer
- ✓ Feed drive
- ✓ Measuring system
- ✓ Work spindle / Work table
- ✓ Cooling system
- ✓ Tool magazine (Turret)

On board computer



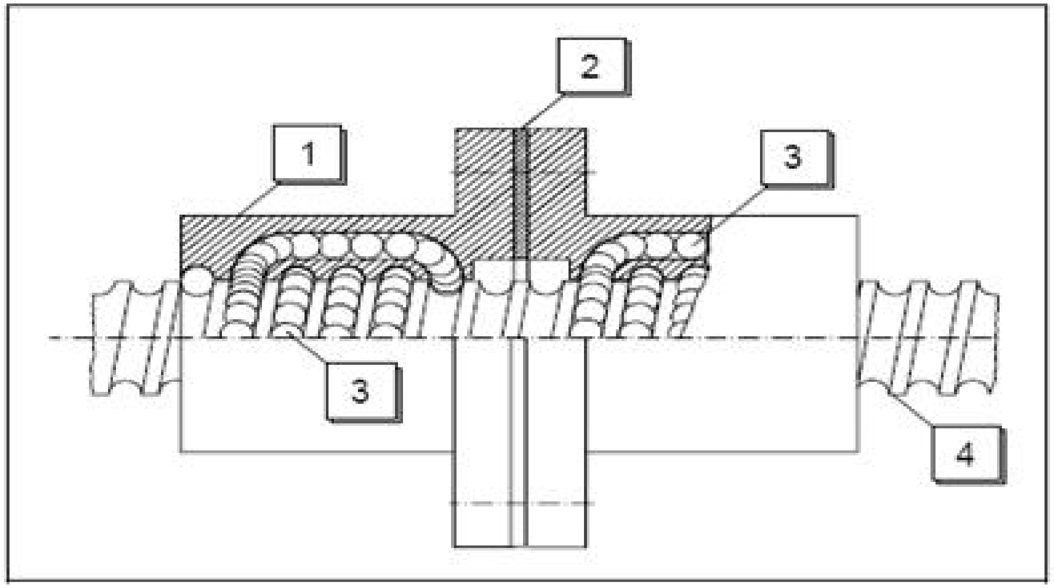
Feed drive



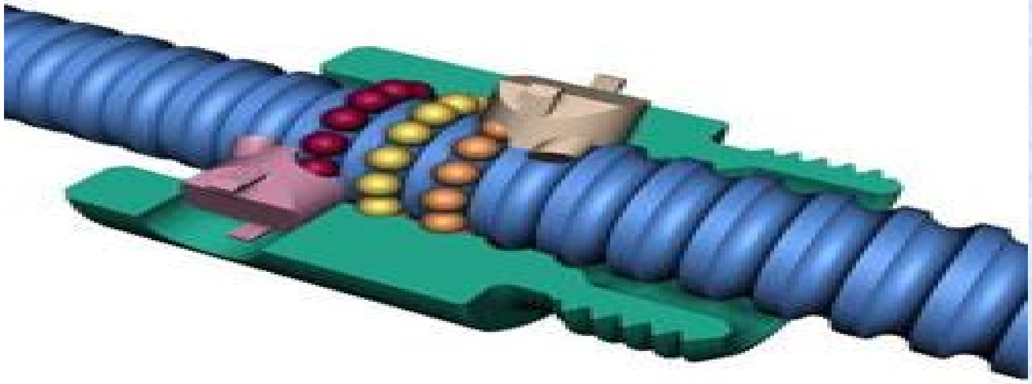


1 feed drive
2 work table
3 measuring system
4 ball screw
5 ball screw nut

Recirculating ball mechanism

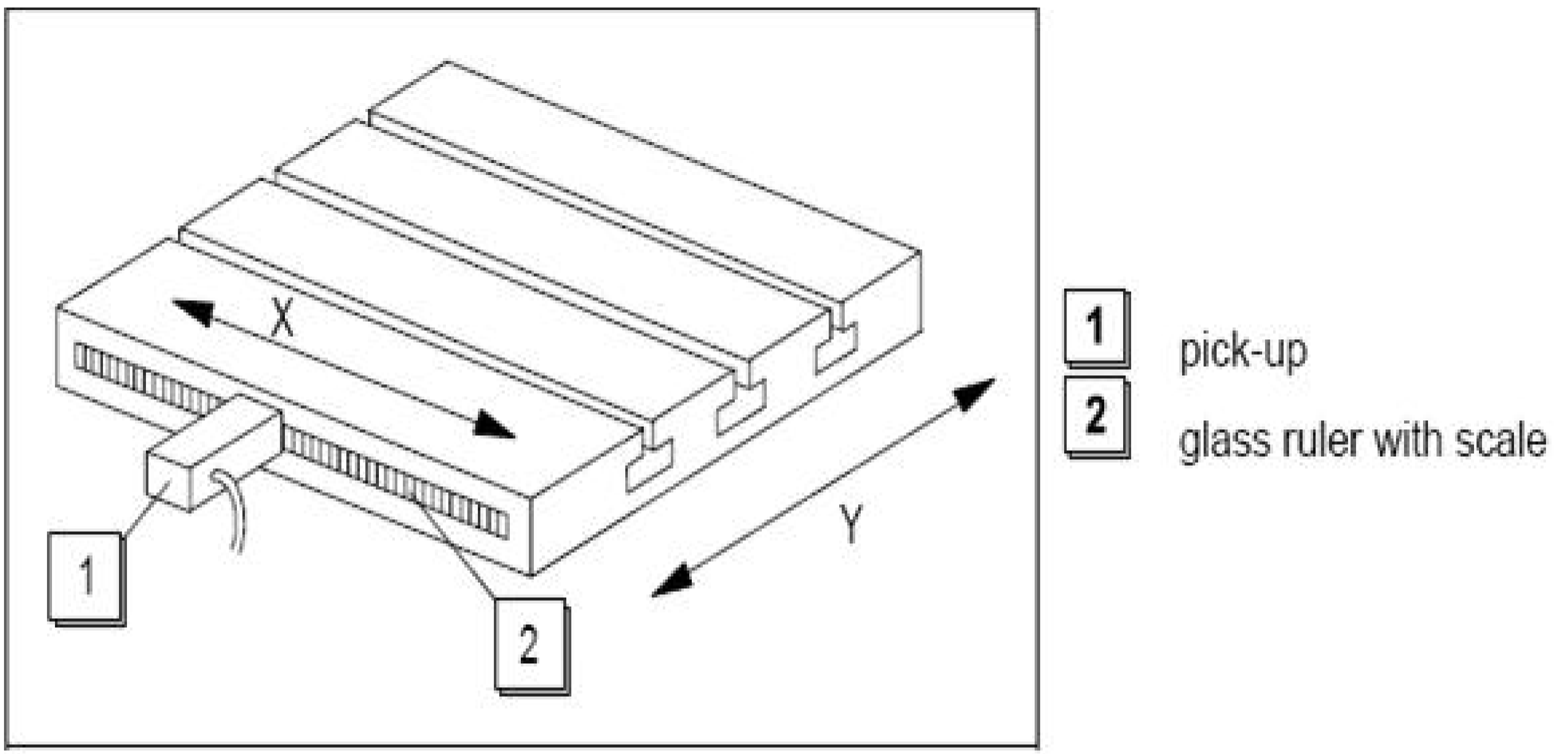


1 ball screw nut
2 Clamping ring
3 balls
4 Drive spindle



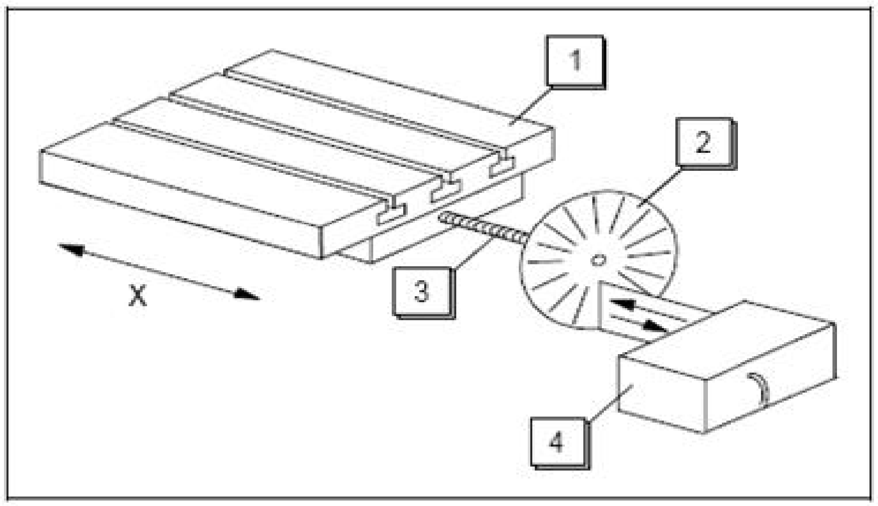
Measuring System

✓ Direct Position measuring



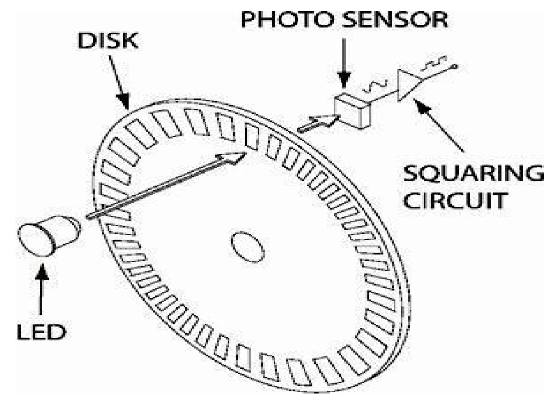
Contd.,

✓ Indirect Position measuring

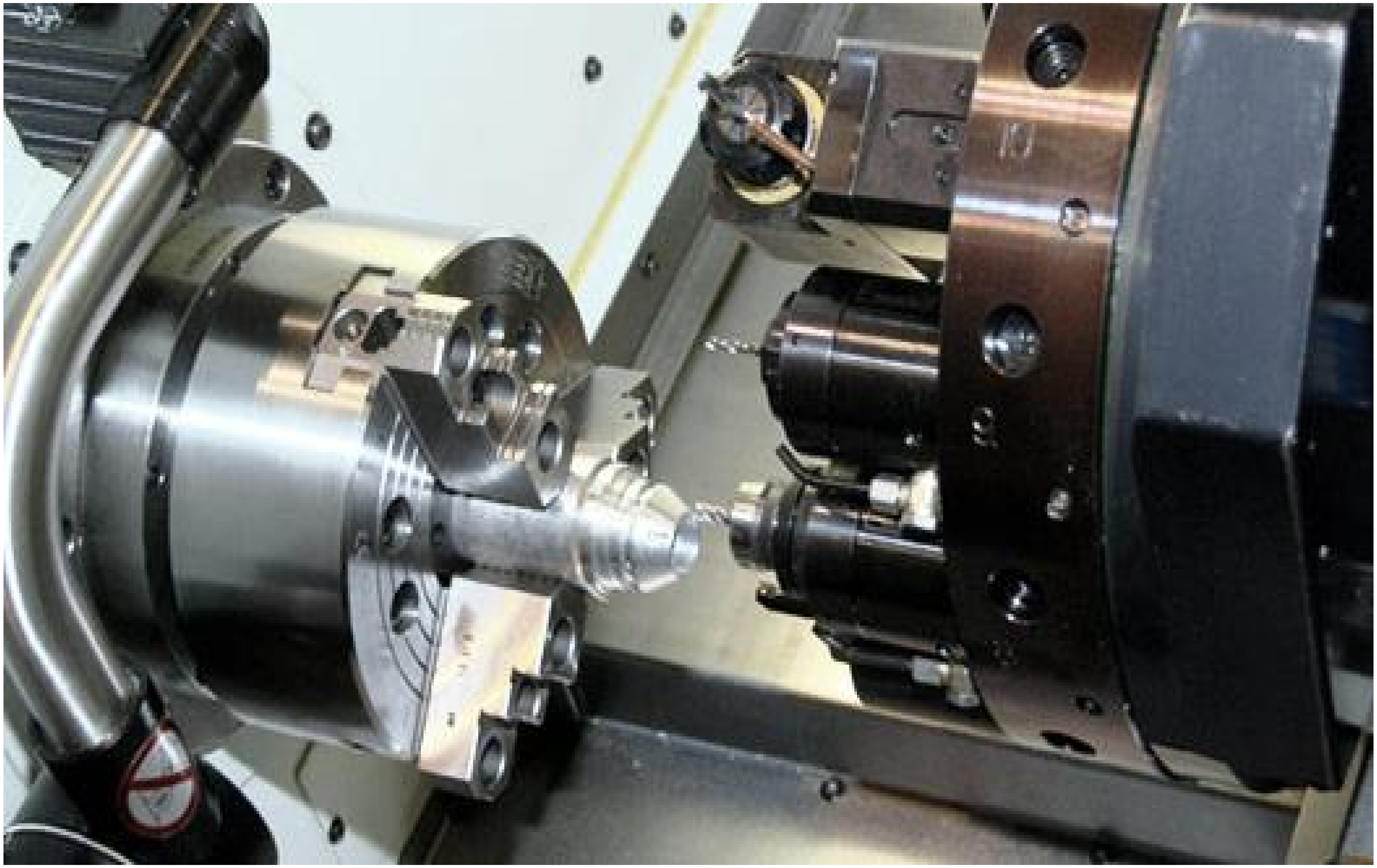


1 carrier
2 pulse disk as a measuring scale
3 spindle
4 signal generator

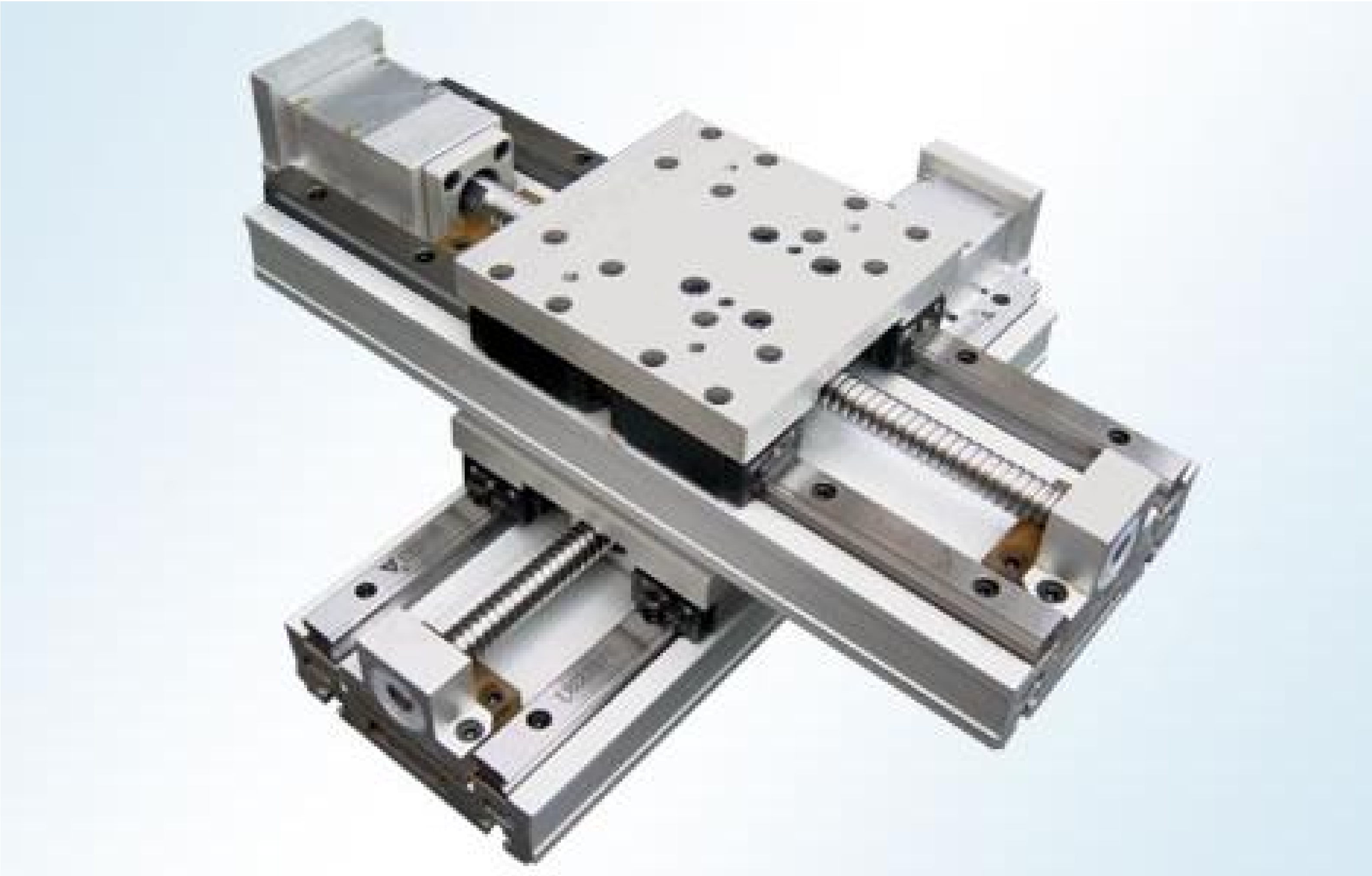
- **Rotary encoder**



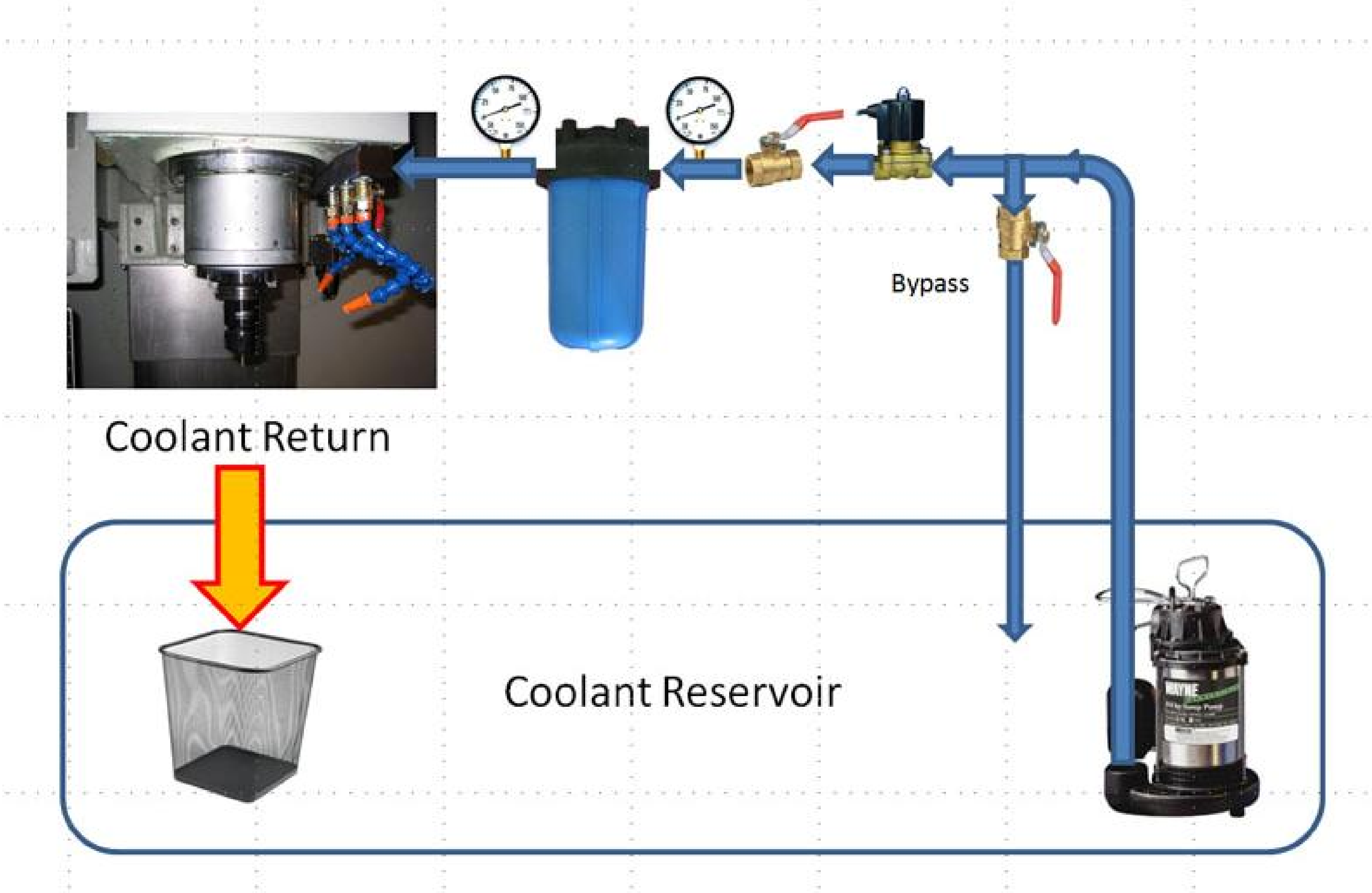
Work Spindle



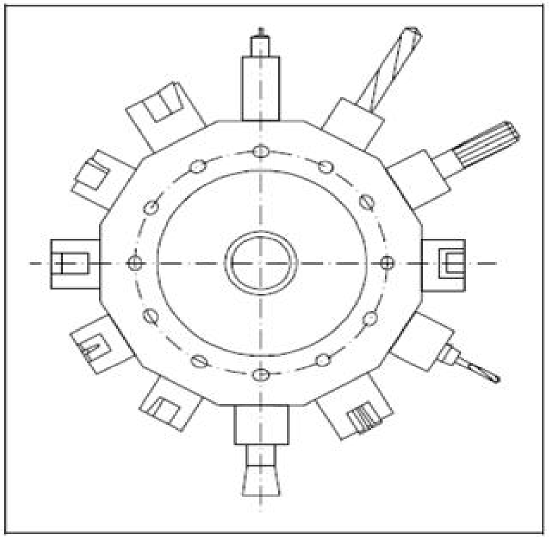
Work table



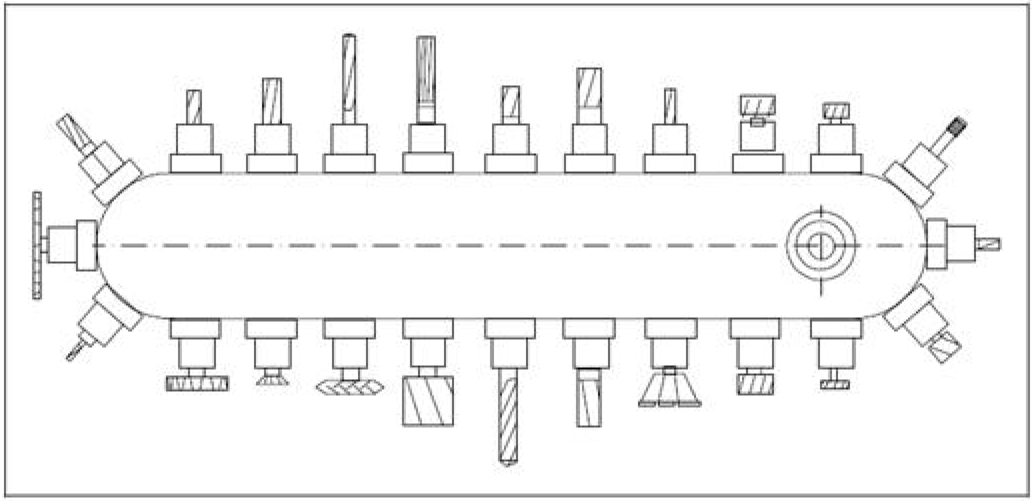
Cooling System



Tool Magazine



- **Single Turret**



- **Chain Turret**

CNC Part Programming

- ✓ Part program is a sequential set of instructions
- ✓ Storage of more than one part program
- ✓ Various forms of program input
- ✓ Program editing at the machine tool (on board computer)
- ✓ Fixed cycles and programming subroutines
- ✓ Communications interface

Benefits of CNC

- ✓ Cycle time reduction
- ✓ Nonproductive time reduction
- ✓ Greater accuracy and repeatation
- ✓ Lower scrap rates
- ✓ Reduced parts inventory and floor space
- ✓ Operator skill-level reduced

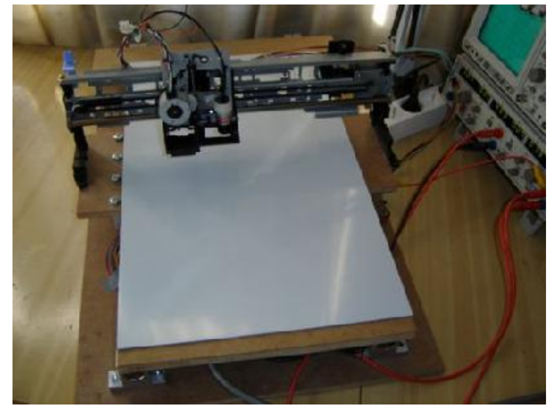
Limitations of CNC

- ✓ High machine cost
- ✓ Complicated maintenance
- ✓ Skill & training are required for programming and maintenance
- ✓ High tooling cost
- ✓ Temperature, humidity & dust must be controlled

CNC Application Characteristics

- ✓ Batch and High Volume production
- ✓ Repeat and/or Repetitive orders
- ✓ Geometry and dimensional accuracy
- ✓ Good surface finish
- ✓ Complex part geometries
- ✓ Many separate operations on one part

Applications of CNC



CNC PART PROGRAMMING

Introduction

- ✓ Part Programme is a set of instructions which instructs the machine tool about the processing steps to be performed for the manufacturing a component
- ✓ Part programme is an important part of the CNC system
- ✓ The shape of manufactured components will depends on how correctly the programme has been prepared
- ✓ The part programme is transferred to as one of the input medium which instruct the CNC machine

Types of programming

- ✓ NC coding
- ✓ Manual part programming
- ✓ Automatic Part Programming
- ✓ Languages:
  - APT
  - UNIAPT
  - COMPACT II
  - FORTRAN

Manual Part Programming

- ✓ Binary code decimal system (BCDS)
- ✓ Bit - 0 or 1 => absence or presence of hole in the tape which is in MCU
- ✓ Character - row of bits across the tape (smallest unit / digit)
- ✓ Word - sequence of characters (combination of alpha-numerical characters)
- ✓ Block - collection of words to form one complete instruction (logical order of processing)
- ✓ Part program - sequence of instructions

Block Format

- ✓ The organization of words within a block in NC part program
- ✓ Word address format - used on all modern CNC controllers
  - – Uses a letter prefix to identify each type of word
  - – Spaces to separate words within the block
  - – Allows any order of words in a block
  - – Words can be omitted if their values do not change from the previous block
  - – A new blocks can be inserted between two existing blocks

Example Format

"N" refers to the Block Numbers

O 1234;	Program Number: Letter O followed by 4 digits.
N10 ;	
N20 ;	
N30 ;	
N40 ;	
N50 ;	
N60 ;	
N70 ;	
N80 ;	
N90 ;	

All blocks ends with a ";" or often referred to as **EOB** or End Of Block

Instruction Codes

A part program consists the following words:

N, G, X, Y, Z, I, J, K, F, S, T, R, M

N - sequence number prefix (to identify the block)

G - preparatory functions (to prepare the controller)

– Example: G00 - rapid transverse move

X, Y, Z - prefixes for *x*, *y*, and *z*-axes (position & motion)

I, J, K – Interpolation parameters

F - feed rate (to specify the feed)

S - spindle speed (to specify the seed)

T - tool selection (for ATC only)

R – arc radius (to specify the tool radius in drill cycles)

M - miscellaneous functions (to specify auxiliary functions)

– Example: M08 = turn cutting fluid on

Elements of Part program

- ✓ Preparatory functions: which unit, which interpolator, absolute or incremental programming, which circular interpolation plane, cutter compensation, etc.,
- ✓ Coordinates: 3 translational and 3 rotational axis
- ✓ Machining parameters: feed, and speed
- ✓ Tool control: next tool number, tool change
- ✓ Cycle functions: canned cycle, drill cycle, ream cycle, bore cycle, mill cycle
- ✓ Miscellaneous functions: coolant on/off, spindle on/off, programme rewind, spindle rotation direction, etc.,

G-Codes

G00 Rapid Transverse
G01 Linear Interpolation
G02 Circular Interpolation, CW
G03 Circular Interpolation, CCW
G17 XY Plane,G18 XZ Plane,G19 YZ Plane
G20 Inch units
G21 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

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

Machine and Tool offset

- ✓ Difficult to place a vise in the exact same position on the machine each time
- ✓ The distance from home to the WCS is usually not known until the vise is set and aligned with the machine
- ✓ Different tools extend out from the machine spindle with different lengths
- ✓ If the tool wears or breaks, tool must be replaced, it is almost impossible to set it the exact length out of the tool holder each time
- ✓ Therefore, there must be some way to relate the (MCS) machine coordinate system to the WCS and take into account varying tool lengths
- ✓ This is done using reference points

Reference Points

✓ Machine Origin

- The machine origin is a fixed point set by the machine tool builder
- Usually it cannot be changed
- Any tool movement is measured from this point
- The controller always remembers tool distance from the machine origin

Contd.,

✓ Program Origin

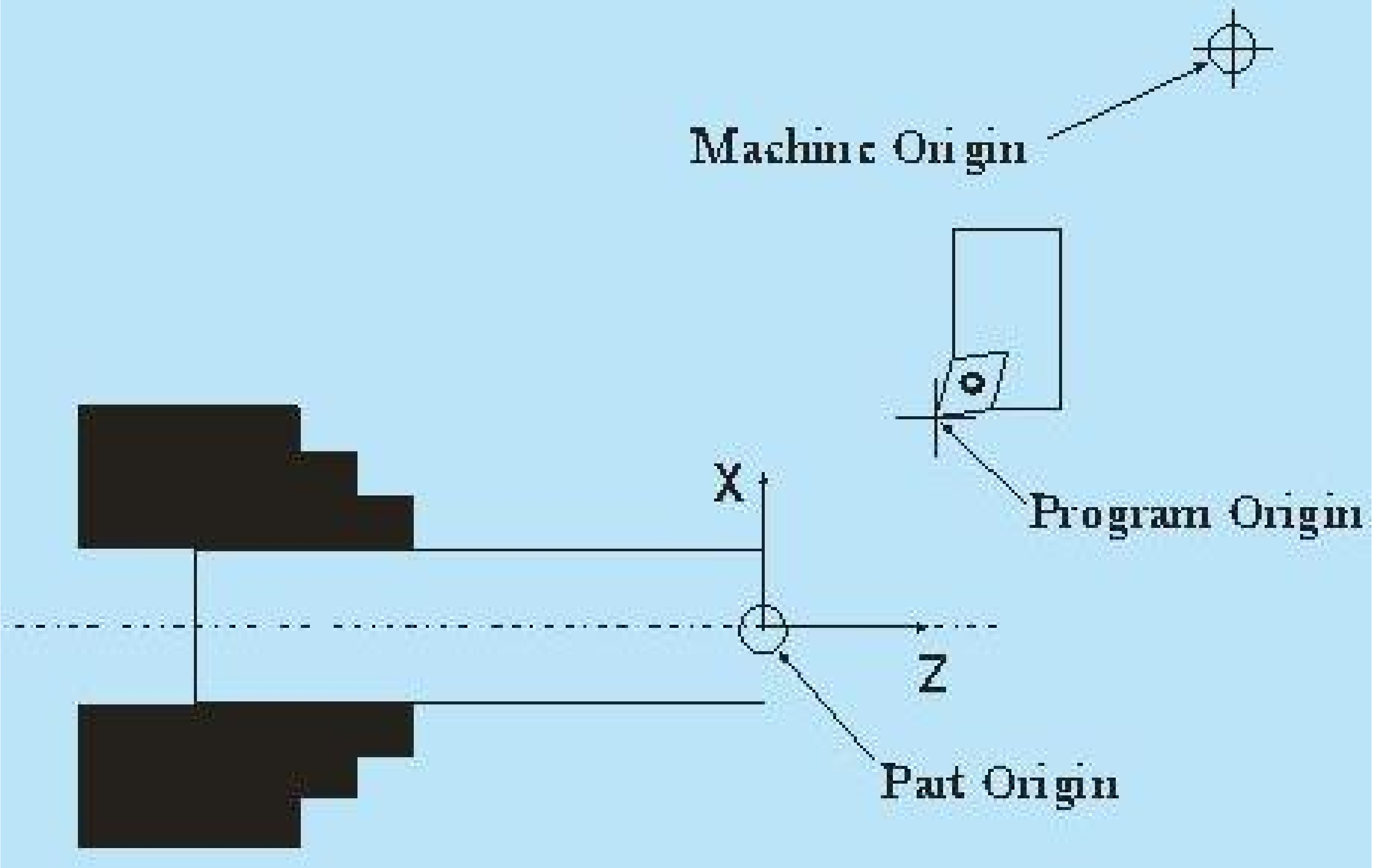
- It is also called home position of the tool
- Program origin is point from where the tool starts for its motion while executing a program and returns back at the end of the cycle
- This can be any point within the workspace of the tool which is sufficiently away from the part

Contd.,

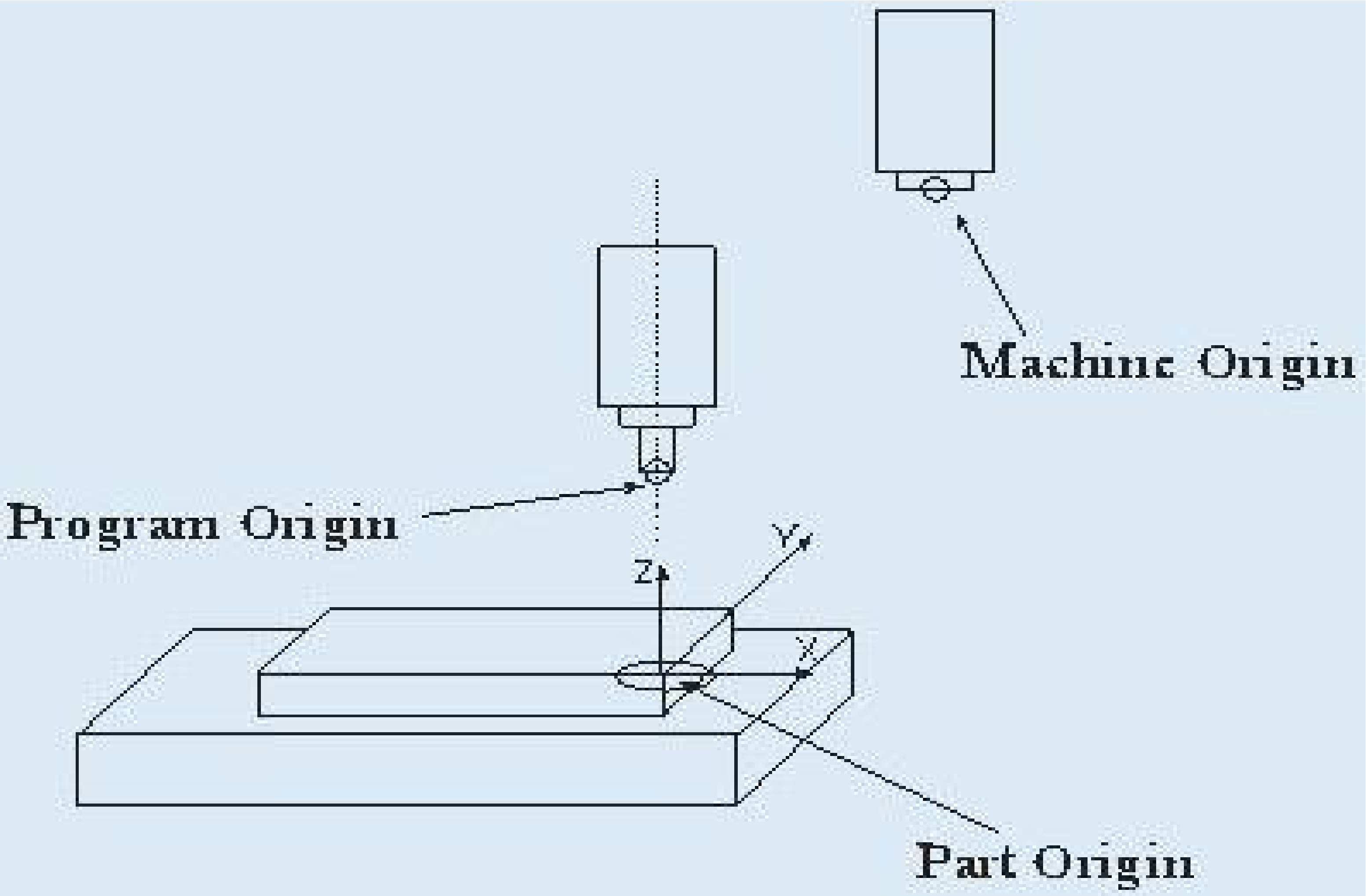
✓ Part Origin

- The part origin can be set at any point inside the machine's electronic grid system
- Establishing the part origin is also known as zero shift, work shift, floating zero or datum
- Usually part origin needs to be defined for each new setup
- Zero shifting allows the relocation of the part
- Sometimes the part accuracy is affected by the location of the part origin

Contd.,



Contd.,



Tool Pre-setter

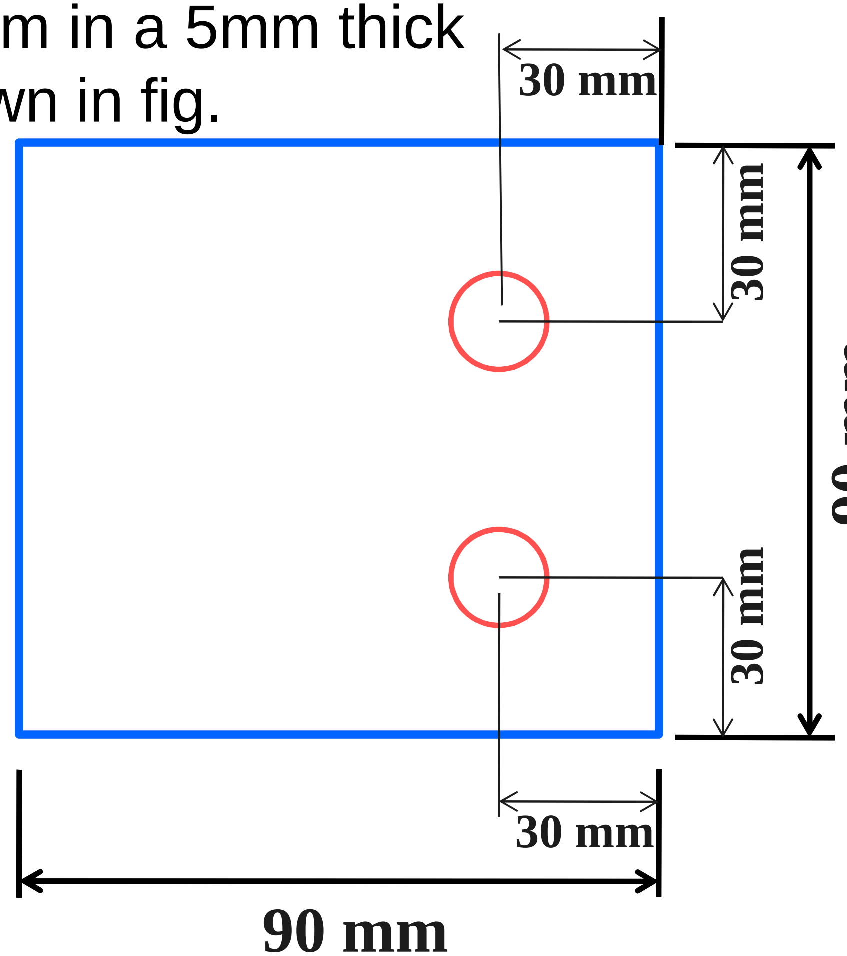
Structure of program

```
%                                                    //% symbol
O0012                                                //Program number and ID
(Sample program structure for demonstration)         //Program description
N10  G21                                             //Units setting
N20  G40  G80  G49                                   //Initial commands
N30  T01                                             //Tool T01 in waiting position
N40  M06                                             //replace present tool at spindle by T01
.
.
.
N100 G01  X20.0     Y34.0                            //Linear interpolation
N110 Y100.0                                          //Linear interpolation
N120 G00  X100.0                                     //Linear interpolation: rapid mode
N130 G01  Y20.0                                      //Linear interpolation at given feed rate
.
.
.
N200 G80  Z40.0     M09                              //Cycle cancel
N210 G28  Z40.0     M05                              //Home in z only
N220 G28  X...      Y...                             //Home in XY only
N240 M30                                             //End of program
%                                                    //Stop code
```

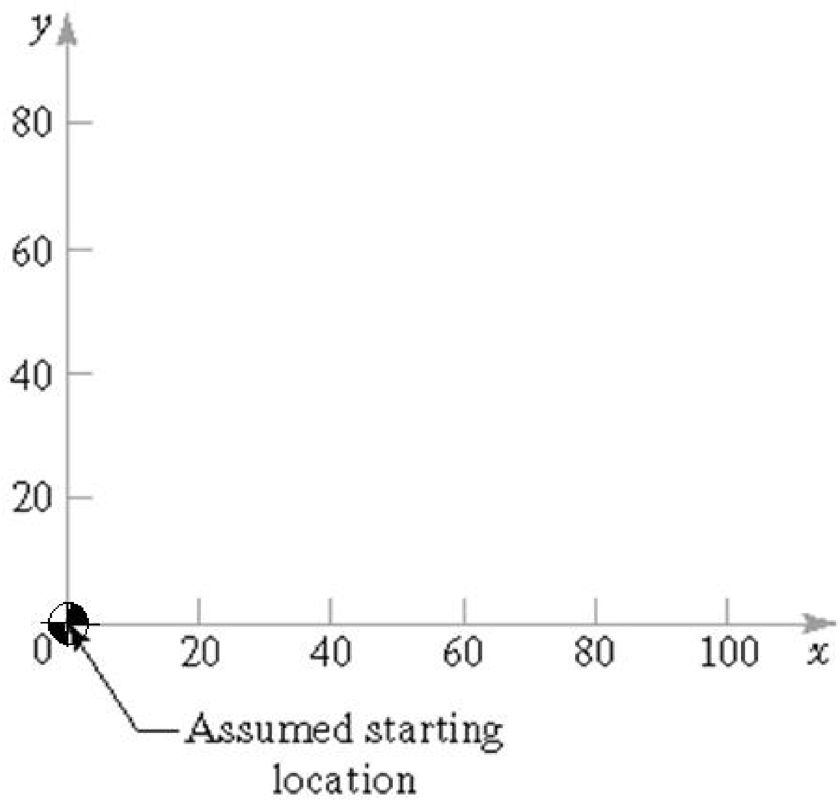
Exercise: 1 (Drilling)

- ✓ Write a CNC part program to drill two holes of diameter 10 mm in a 5mm thick mild steel plate as shown in fig.

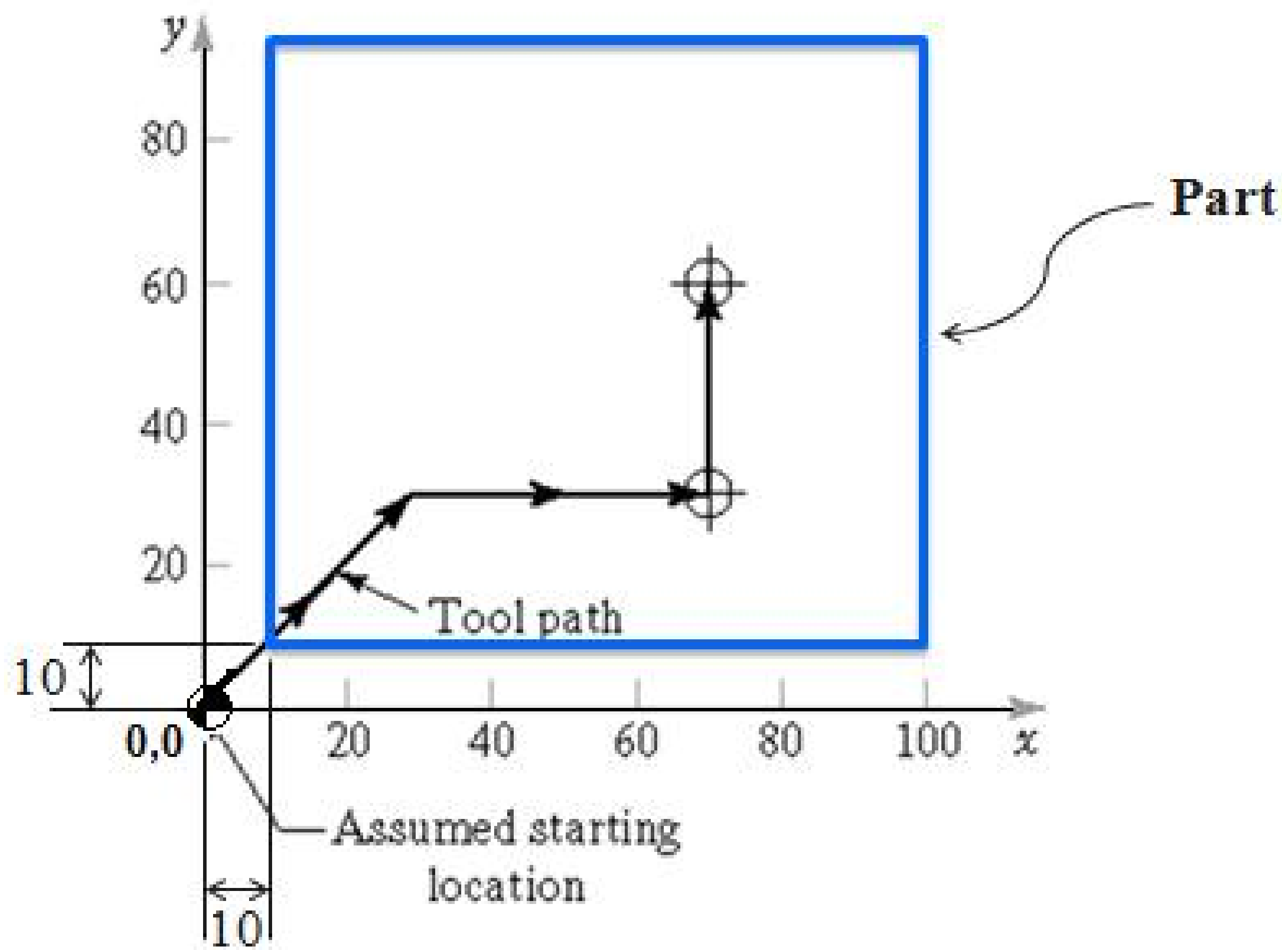
(Diagram not to scale)



Location of Reference point



Process plan



Part Program

Block 1: O0001;

Block 2: N01 G17 G21 G80 G90 G94;

Block 3: N02 G00 X0, Y0, Z10;

Block 4: N03 X30, Y30;

Block 5: N04 G01 X70 Y30 S800 M03;

Block 6: N05 Z-7 F0.3 M08;

Block 7: N06 Z2;

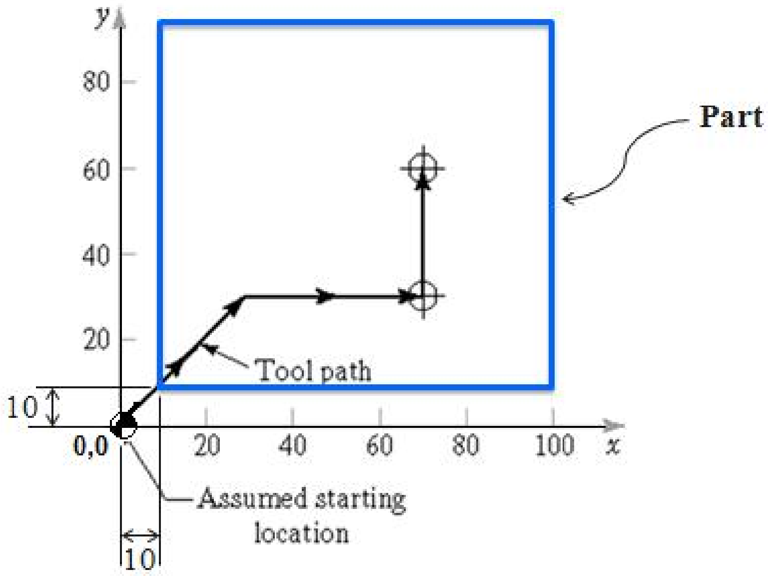
Block 8: N07 G00 Y60;

Block 9: N08 G01 Z-7 F0.3;

Block 10: N09 Z2;

Block 11: N10 G00 X0 Y0 Z10 M05 M09;

Block 12: N10 M02;

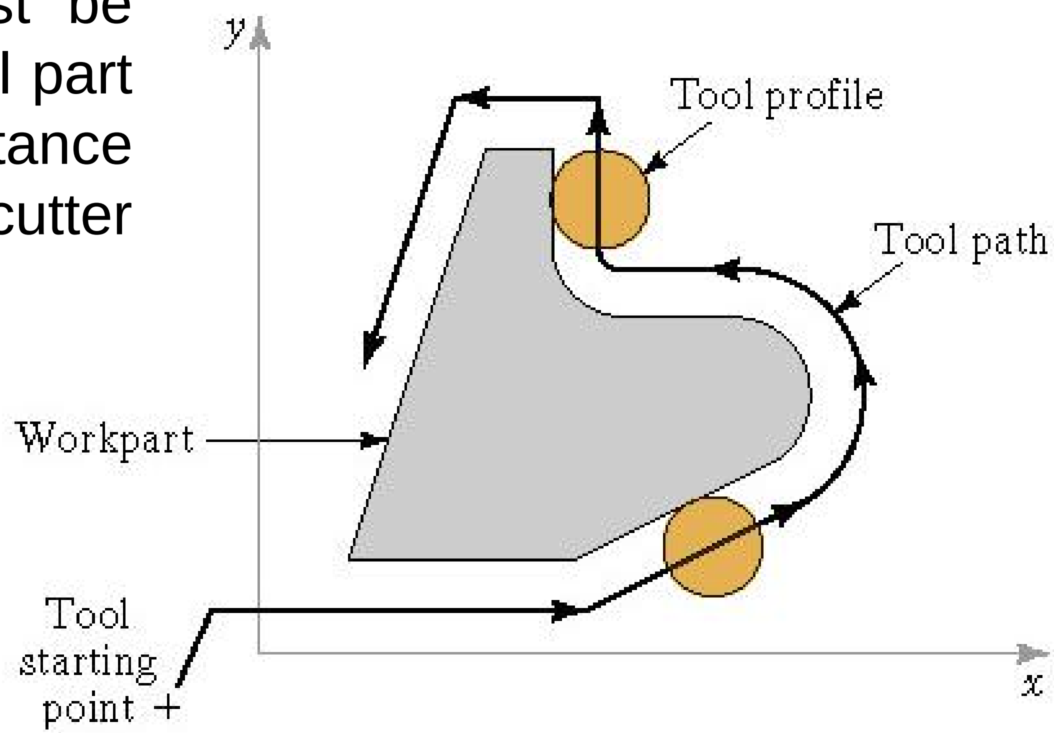


Canned Cycle

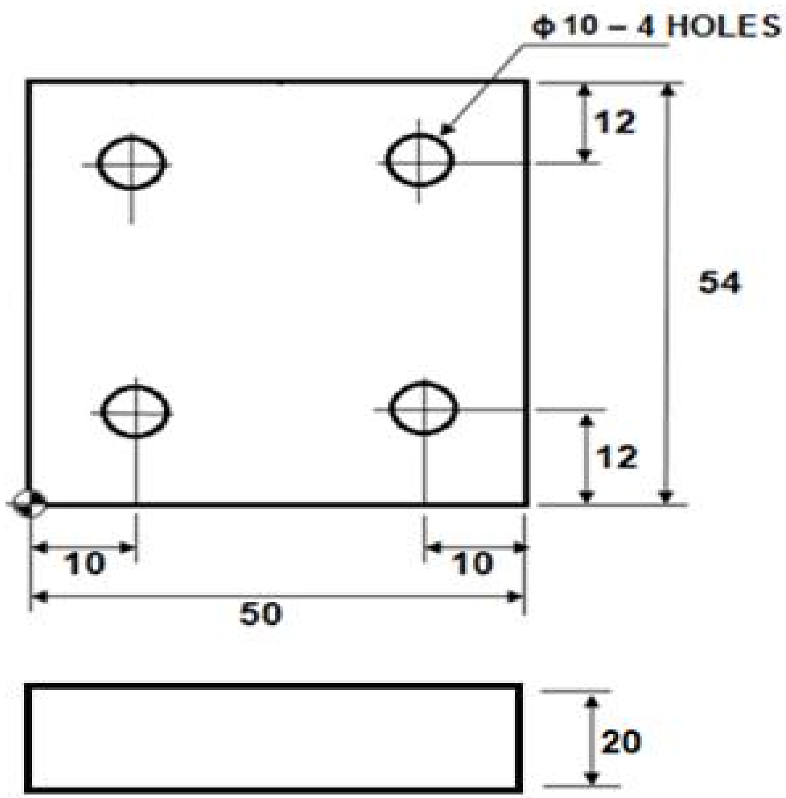
Canned Cycle	Equivalent Motion: Expanded Code
N70 G98 G83 X1. Y1. Z-1.04 R0.06 Q0.15 P0 F9. N75 G80	N70 Z0.06 N75 Z0.04 N80 G01 Z-0.19 F9. N85 G00 Z0.06 N90 Z-0.11 N95 G01 Z-0.34 N100 G00 Z0.06 N105 Z-0.26 N110 G01 Z-0.49. N115 G00 Z0.06 N120 Z-0.41 N125 G01 Z-0.64. N130 G00 Z0.06 N135 Z-0.56 N140 G01 Z-0.79 N145 G00 Z0.06 N150 Z-0.71 N155 G01 Z-0.94. N160 G00 Z0.06 N165 Z-0.86 N170 G01 Z-1.04. N175 G00 Z0.25

Cutter Offset

- ✓ Cutter path must be offset from actual part outline by a distance equal to the cutter radius



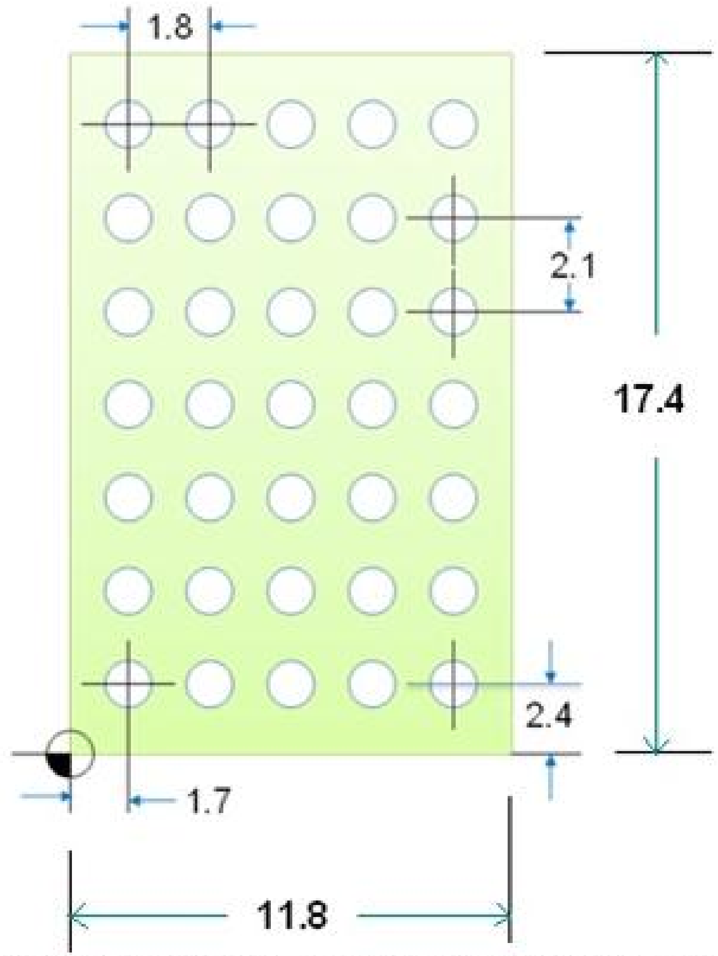
Write the part program for drilling holes in the mild steel plate of thickness 20 mm as shown in figure 1 (diagram not to scale) and assume the parameters such as speed, feed, depth of cut etc.



All Dimensions are in 'mm'

Exercise 2

- ✓ Write an efficient CNC part program to drill 35 holes of diameter of 0.5 inch each in a machine component as shown in the figure.
- ✓ The raw material to be employed is mild steel plate of 0.4 inch thickness.



MILD STEEL PLATE. ALL DIMENSIONS ARE IN INCHES

Part Program

Block 1		%
2		O0001
3	N10	G20
4	N20	G17 G40 G80 G49 G90
5	N30	G92 X… Y… Z…
6	N40	M06 T01
7	N50	G00 X1.7 Y2.4 S900 M03
8	N60	G43 Z1.0 H01 M08
9	N70	G99 G81 R0.1 Z-0.4 F3.0
10	N80	G91 Y2.1 K6 (L6)
11	N90	X1.8
12	N100	Y-2.1 K6 (L6)
13	N110	X1.8
14	N120	Y2.1 K6 (L6)
15	N130	X1.8
16	N140	Y-2.1 K6 (L6)
17	N150	X1.8
18	N160	Y2.1 K6 (L6)
19	N170	G90 G80 M09
20	N180	G28 Z10 M05
21	N190	G28 X0 Y0
22	N200	M30
23		%

Explanation

Block 1:
It indicates the start of the program.

Block 2:
It specifies the program number and ID. It is usually a alpha-numerical code and always start with an alphabet ‘O’.

Block 3:
It sets the entry of dimensional units in Imperial format.

Block 4:
G17: It selects the plane of operation as X-Y plane
G40, G80, G49 are used to cancel all usual cycle that might have left in on-mode during the execution of last CNC code.
G90 selects the method of specifying dimensions between features as ‘absolute’.

Contd.,

Block 5:

- ✓ It sets the program zero on the work part.
- ✓ There are three major environments in programming that require an established mathematical relationship.

  Machine: machine tool and control system

  Part: Workpiece + Drawing + material

  Tool: Holder + Cutting tool

- ✓ The location coordinates of the program zero with respect to the machine reference zero must be communicated with the MCU so that the MCU will convert the part program in to required signals to control the machine tool.
- ✓ This can be achieved by using a Preparatory code ‘G92’. The syntax of G92 is as follows:

G92 X… Y… Z…

Contd.,

- **Block 6:**
  Replace the existing cutting tool with tool number 1.

- **Block 7:**
  Rapid travel of tool from home position to a reference position: hole with coordinates X1.7   Y2.4.
  Switch on the spindle rotation with speed of about 900 rpm.

- **Block 8:**
  Approach to a safe position at Z = 1.0 rapidly. Meanwhile the tool length compensation is activated by using G43. It is used to communicate the length of tool registered in register number H01 to the MCU. Switch on the coolant flow.

Contd.,

Block 9:

- In the given task, number of holes is to be drilled.
- For this purpose a special function or cycle is used. It is called as drilling canned cycle.
- Its syntax and meaning are shown below.
- The number of motions/action elements of drilling operations is specified only at once.
- Later only the locations of holes to be drilled are given to the MCU.

Contd.,

- **Block 10:**
  It suggests the distance of next location of the hole. It is also suggested to carry out the same drilling operation 6 times along the Y-axis with an increment of 2.1.

- **Block 11:**
  Drill the hole at increment of 1.8 along X-direction.

- **Block 12:**
  Carry out the drilling operation 6 times along the Y-axis with decrement of 2.1.

- **Block 13:**
  Drill the hole at increment of 1.8 along X-direction.

Contd.,

Block 14:

Carry out the drilling operation 6 times along the Y-axis with increment of 2.1.

Block 15:

Drill the hole at increment of 1.8 along X-direction.

Block 16:

Carry out the drilling operation 6 times along the Y-axis with decrement of 2.1.

Block 17:

Drill the hole at increment of 1.8 along X-direction.

Block 18:

Carry out the drilling operation 6 times along the Y-axis with increment of 2.1.

Block 19:

Cancel the canned cycle and switch off the coolant flow.

Contd.,

Block 20:

Stop the spindle and go to safe position along Z direction at 0.0.

Block 21:

Go to home position via X= 0 and Y=0.

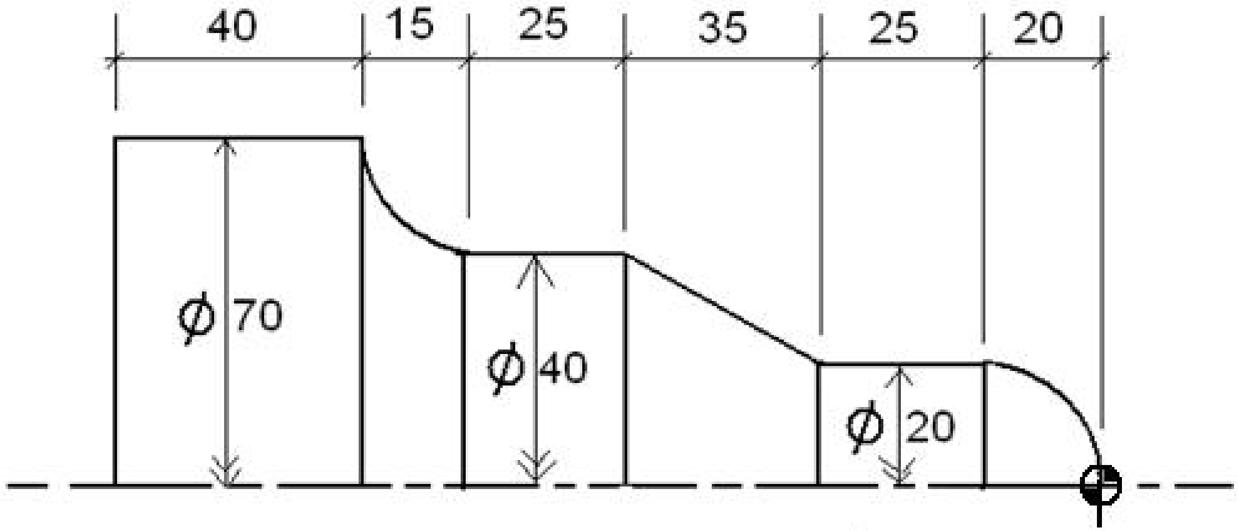
Block 22:

Stop the program from execution.

Block 23:

End the program.

Exercise 3 (Turning)



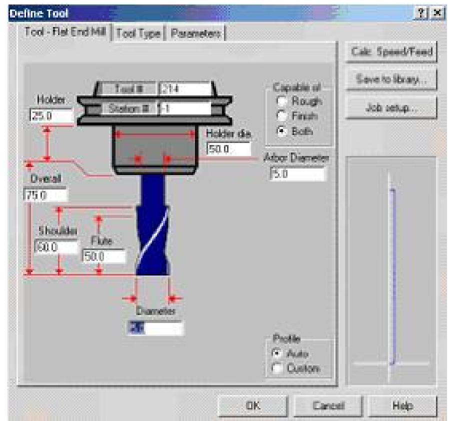
RAW MATERIAL: MS BAR OF DIAMETER 70 MM
AND LENGTH 160 MM
DIAGRAM NOT TO SCALE
ALL DIMENSIONS ARE IN MM

Part program

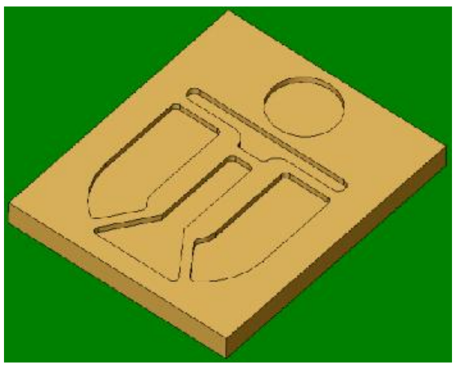
Block		
1		%
2		O0004
3	N10	G21
4	N20	G40 G90
5	N30	G54 X… Z…
6	N40	T0100 M42
7	N50	G96 S450 M03
8	N60	G00 G41 X72 Z0 T0101 M08
9	N70	G01 X0
10	N80	G00 Z5
11	N90	G42 X72
12	N100	G71 U1 R3
13	N110	G71 P120 Q190 U1 W1 F0.05
14	N120	G00 X0
15	N130	G01 Z0
16	N140	G03 X20 Z-20
17	N150	G01 Z-45
18	N160	X40 Z-80
19	N170	Z-105
20	N180	G02 X70 Z-120
21	N190	G01 X75
22	N200	G00 X100 Z20
23	N210	G70 P120 Q190 F0.03
24	N220	G00 G40 X100 Z20 T0100
25	N230	M09
26	N240	M30
27		%

Automatic part programming

Define Tool

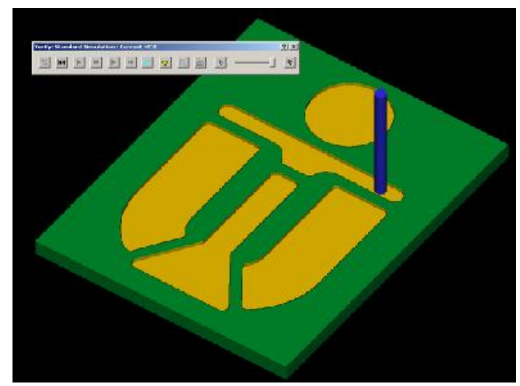


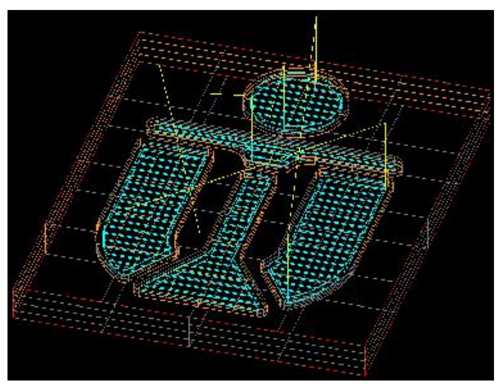
G0 X-.5 Y-.75
G0 Z.1
G1 Z-.5 F10.0
G1 X0. Y0.
G1 X.5 Y1.75
G1 X1.75
G1 Y2.5
G1 X2.569 Y 3.074



Make 3D model

CNC data





Simulate cutting

*** All the best ***