

# Computer Numerical Control

Dr Atul Thakur

Assistant Professor

Department of Mechanical Engineering

# CNC Machines

- CNC : Computer Numerical Control
- In a CNC Machine functions and slide movements are controlled by motors using computer programs.
- Conventionally, a human operator decides and adjusts various machines parameters like feed, depth of cut etc depending on type of job, and controls the slide movements by hand.



# Types of Milling Machine

- Mills and Machining Centres
- Lathes and Turning Centres
- Drilling Machines
- EDM Sinker and wire cut Machines
- Flame and Laser-Cutting Machines
- Water Jet Profilers

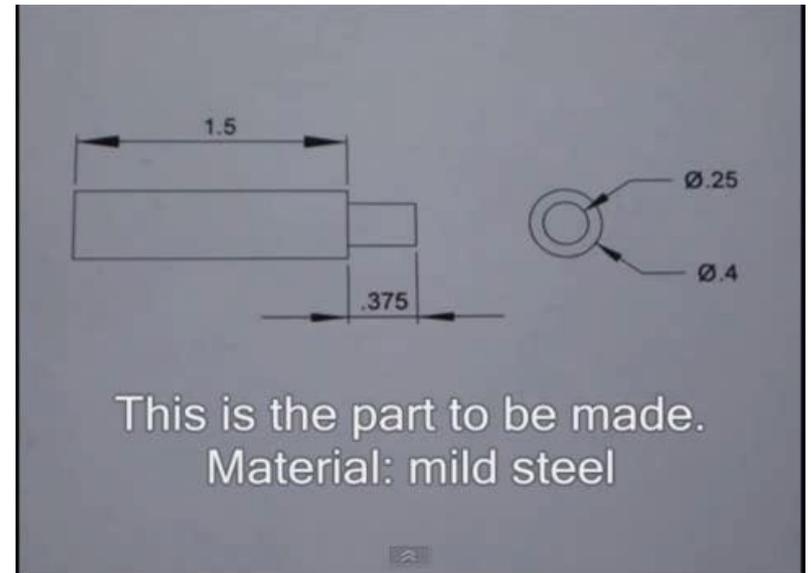
# Commercial Control Units

- FANUC CONTROL
- SIEMENS
- GSK
- MECH3

# CNC Vs Manual Operation



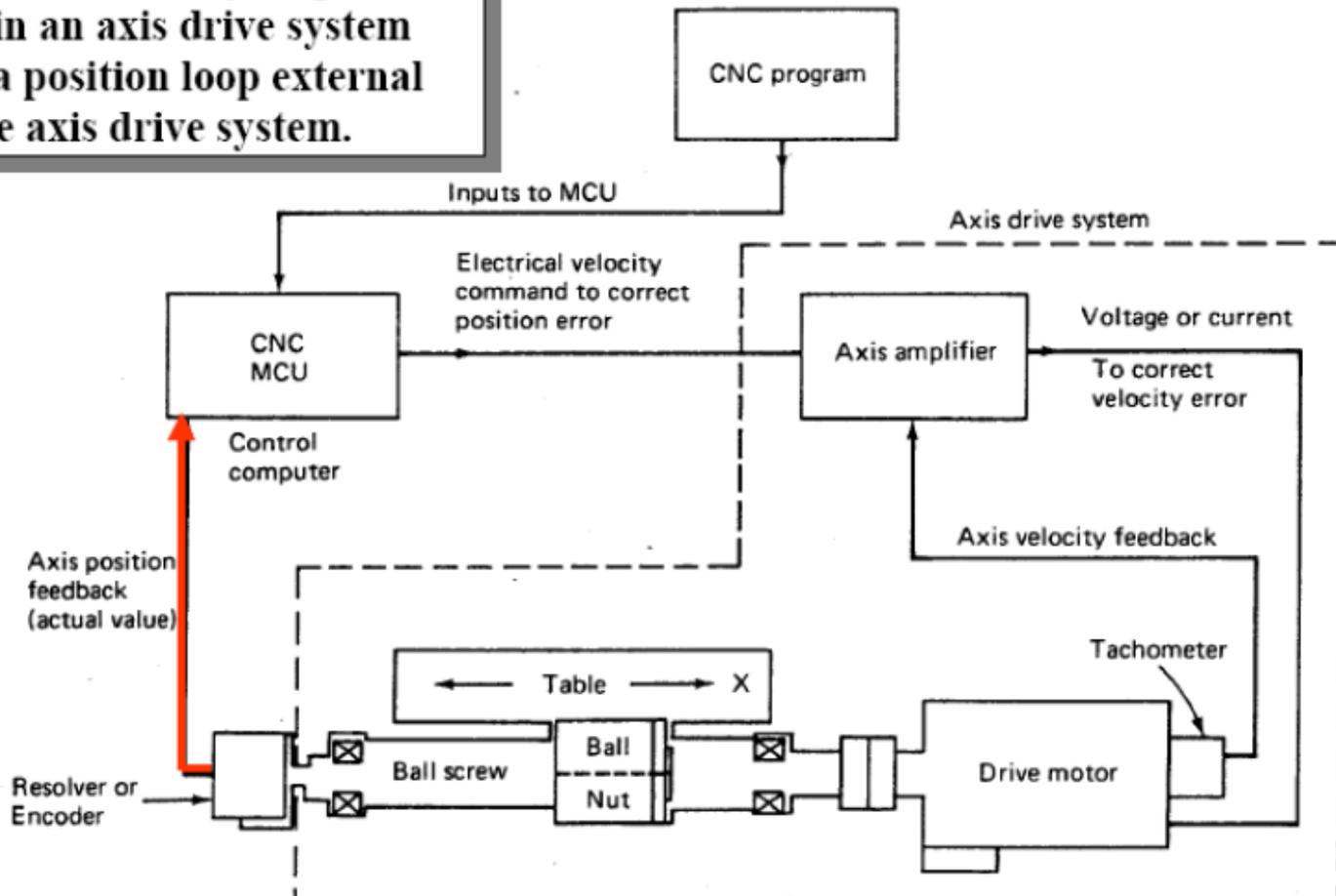
CNC Operation



Manual Operation

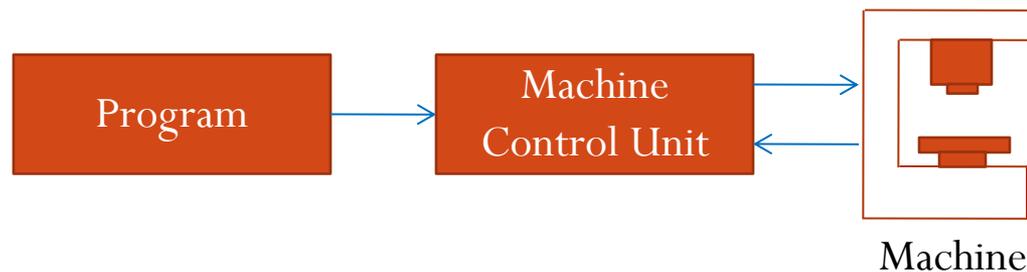
# CNC Block Diagram

**A CNC control system includes a velocity loop within an axis drive system and a position loop external to the axis drive system.**



# CNC Overview

- A numerical control, or “NC”, system automatically controls many machine functions and movements
  - traditionally performed by skilled machinists.
- Numerical control developed to meet the requirements of
  - high production rates,
  - uniformity, and
  - consistent part quality.
- Programmed instructions converted into output signals which in turn control machine operations such as spindle speeds, tool selection, tool movement, and cutting fluid flow.

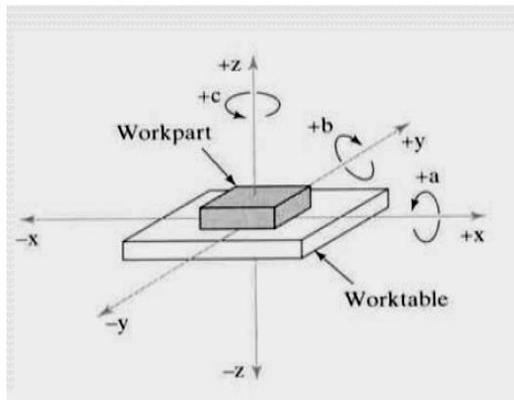


# CNC Overview

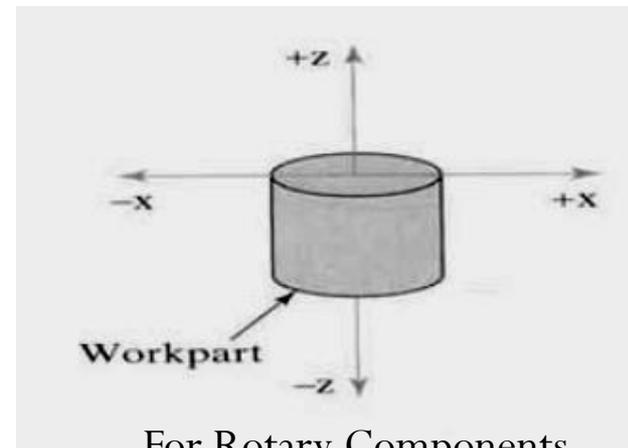
- By integrating a computer processor, computer numerical control (CNC) is obtained or “CNC”
  - allows part machining programs to be edited and stored in the computer memory
  - permit diagnostics and quality control functions during actual machining.
- All CNC machining begins with a part program,
  - a sequential instructions or coded commands that direct the specific machine functions.
- Part program may be
  - manually generated or,
  - using computer aided part programming systems.

# Basic Principle of CNC

- All computer controlled machines are able to *accurately* and *repeatedly* control motion in various directions
- Each of these directions of motion known as *axis*
- Depending on the machine type there are commonly two to five axes
- Two types of CNC axes
  - linear axis in which movement is in a straight line,
  - rotary axis with motion following a circular path

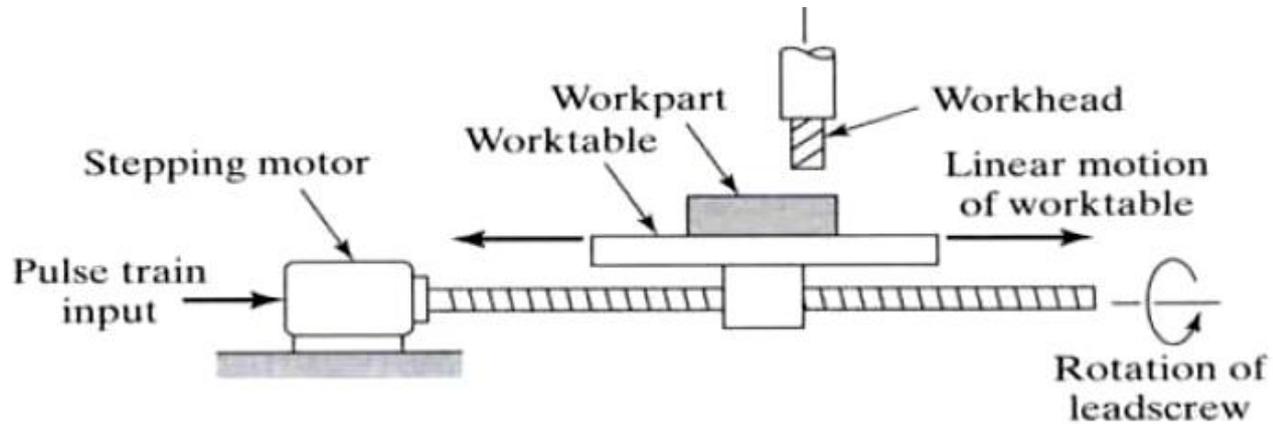


For Prismatic Components

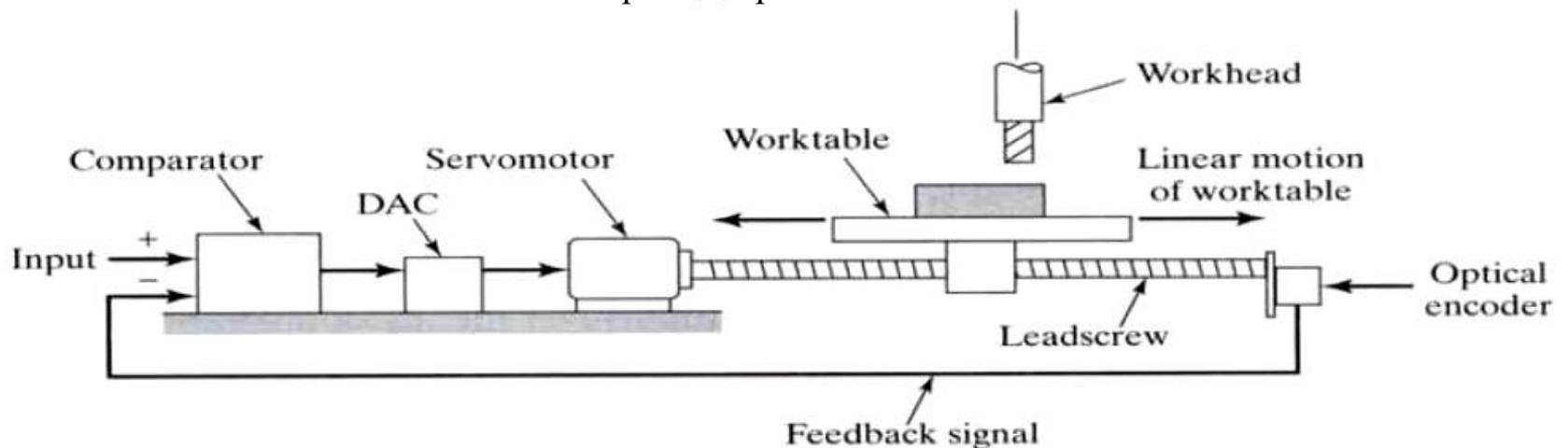


For Rotary Components

# Open Loop and Closed Loop Control

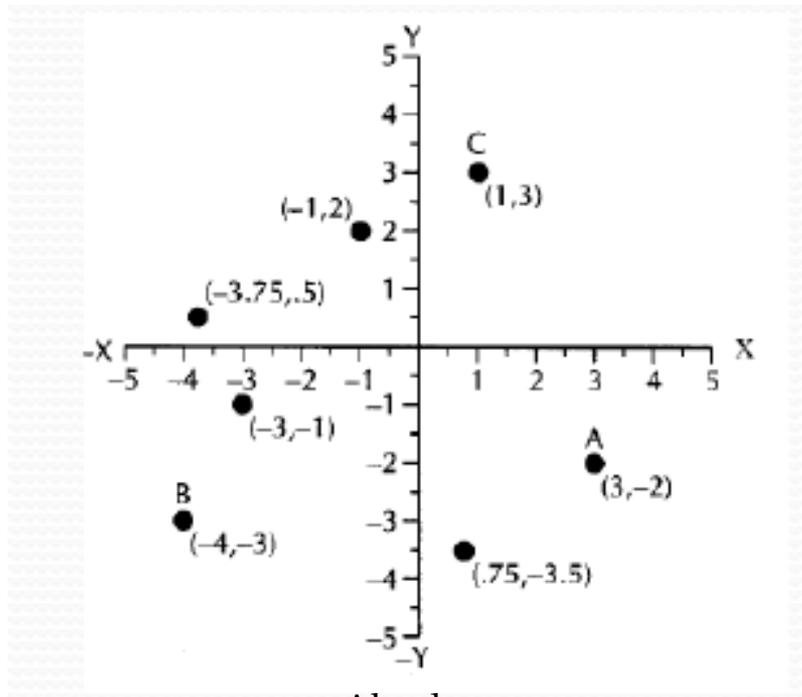


Open Loop Control

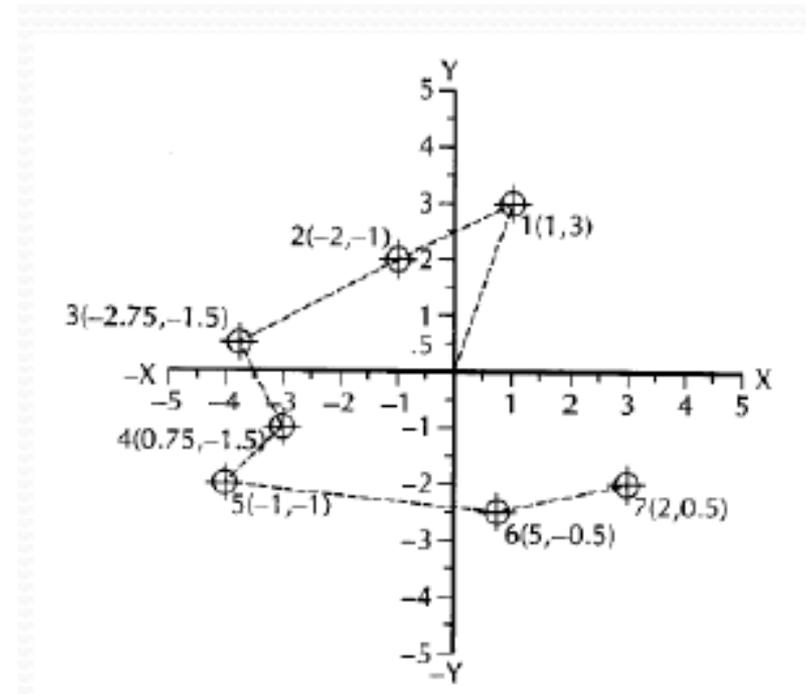


Closed Loop Control

# Absolute Vs Incremental Coordinate



Absolute



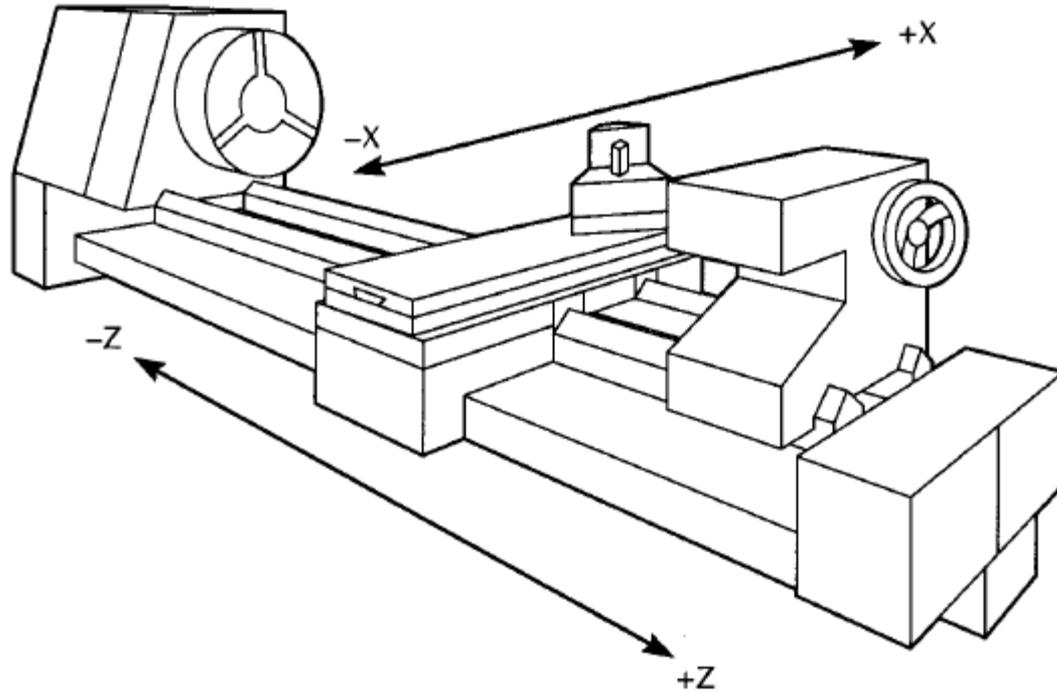
Incremental

# CNC Code - Information Required

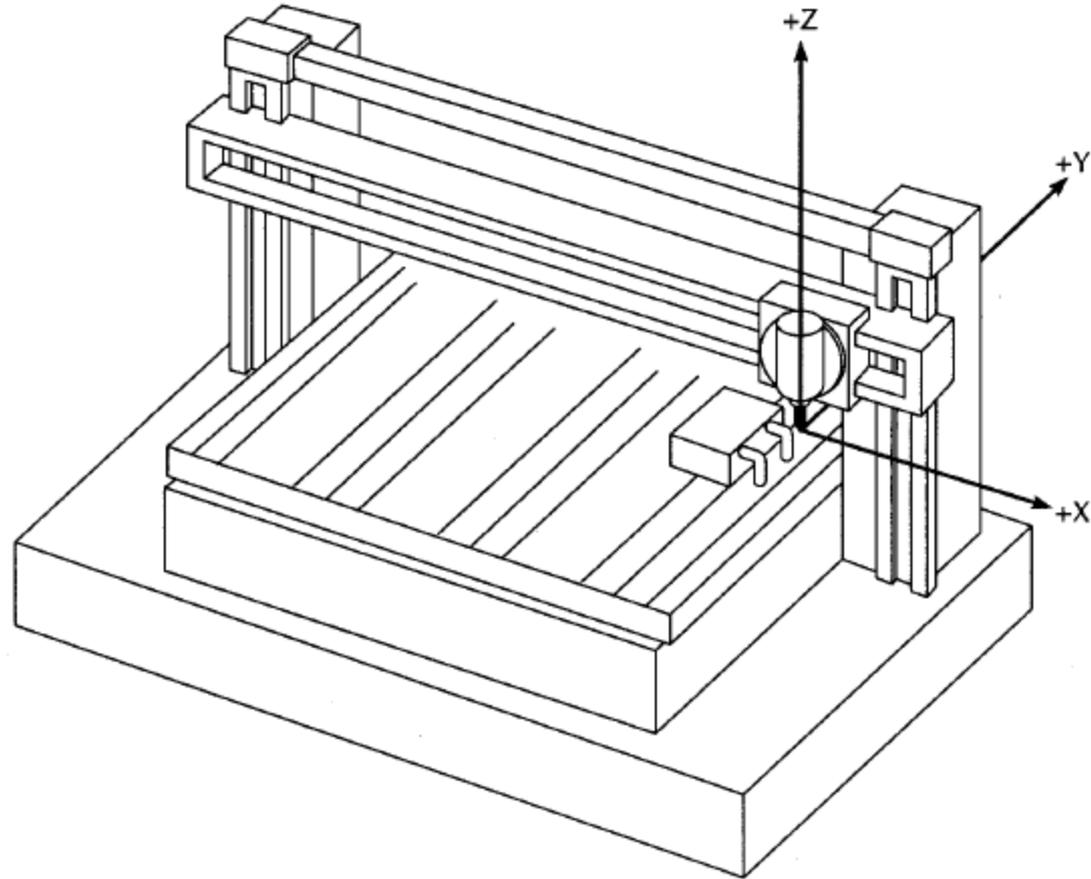
- Preparatory Information: units, incremental or absolute positioning
- Coordinates: X,Y,Z, RX,RY,RZ
- Machining Parameters: Feed rate and spindle speed
- Coolant Control: On/Off, Flood, Mist
- Tool Control: Tool and tool parameters
- Cycle Functions: Type of action required
- Miscellaneous Control: Spindle on/off, direction of rotation, stops for part movement

Information conveyed to CNC machine through a set of instructions arranged in a particular sequence – **Program**

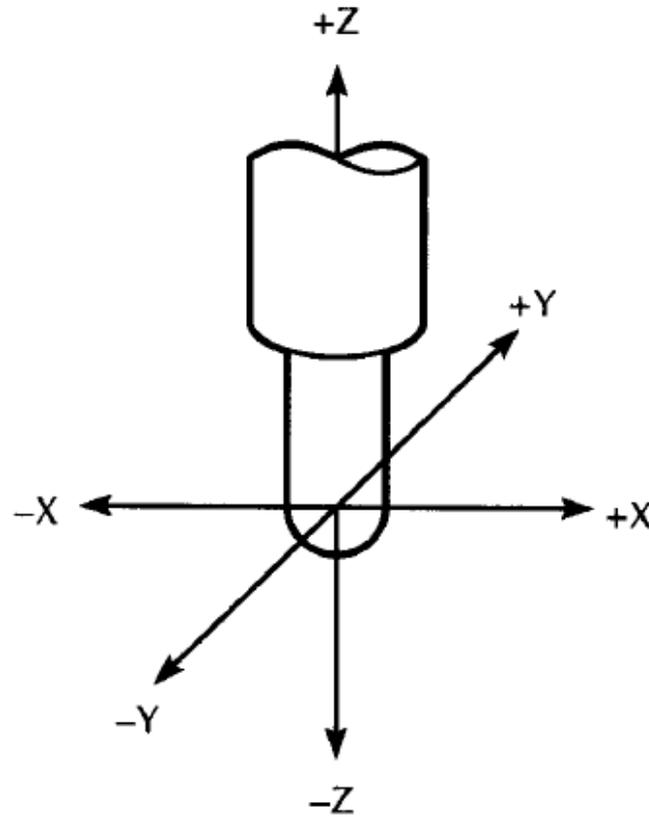
# Lathe Coordinate System



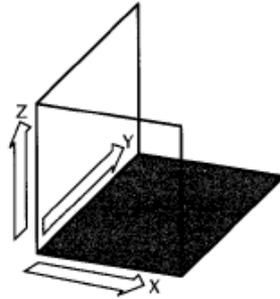
# Milling Machine Coordinate System



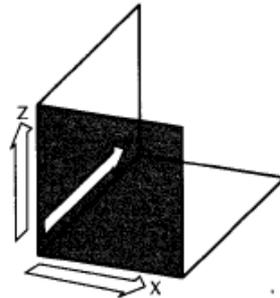
# Mill Tool Coordinate System



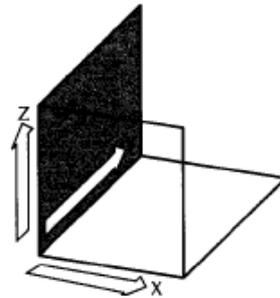
# Planes



X-Y Plane

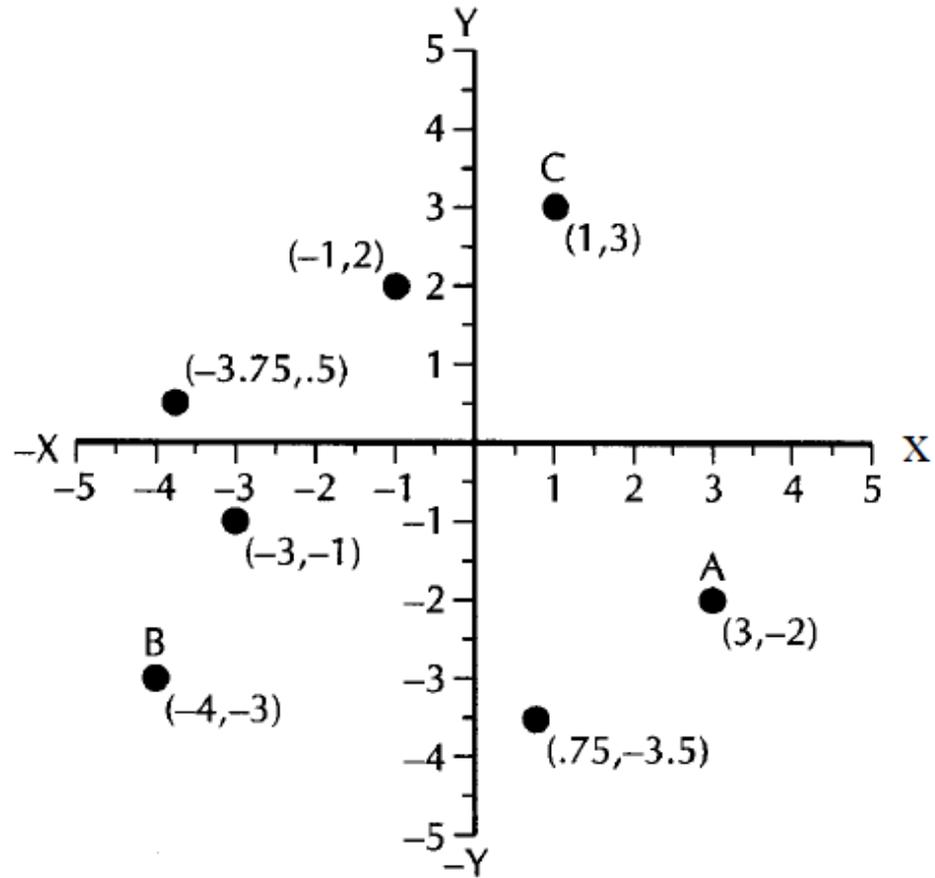


X-Z Plane

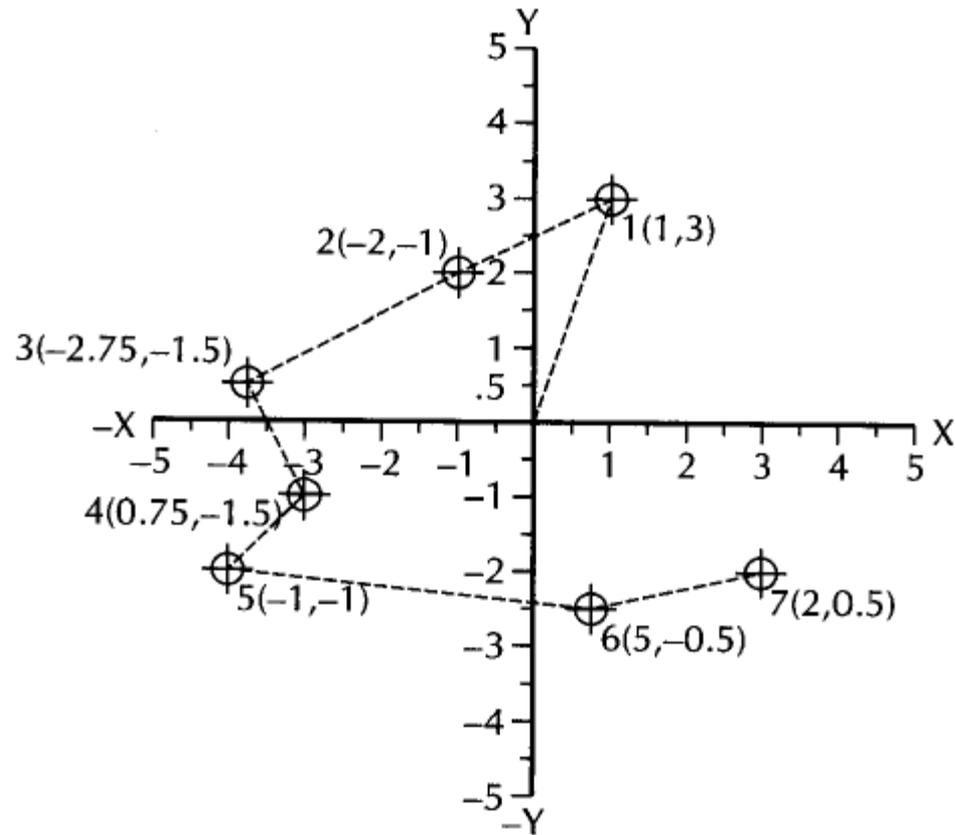


Y-Z Plane

# Absolute Coordinate

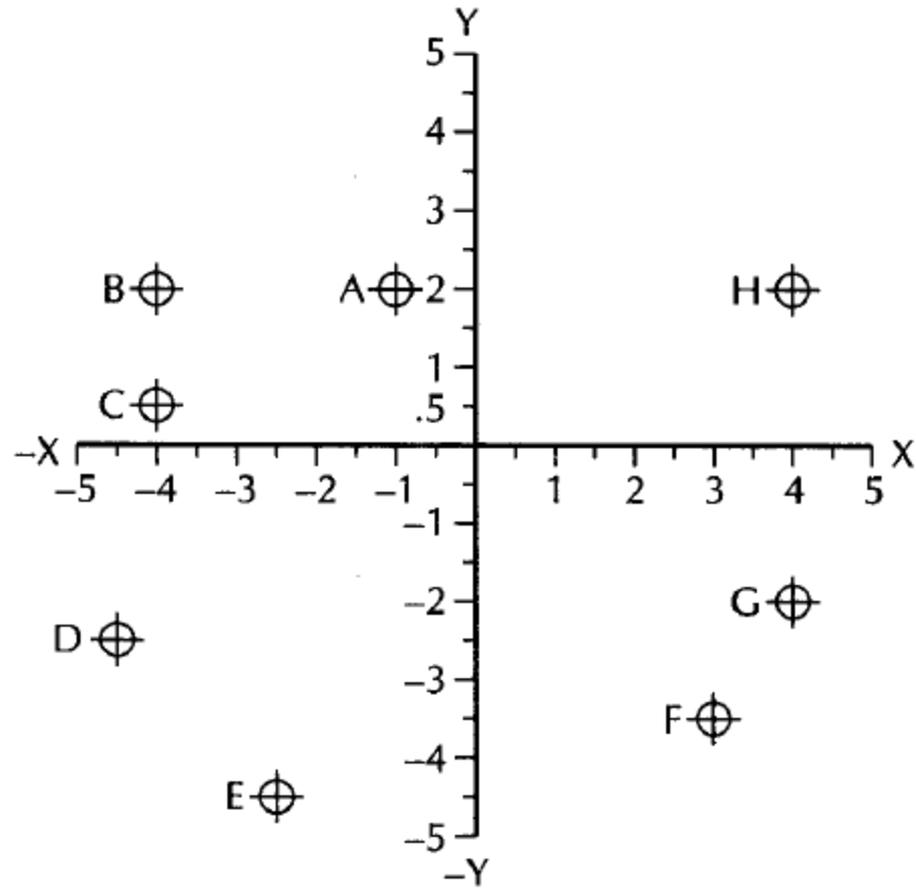


# Incremental Coordinate System



# Exercise

- A(-1,2)
- B(-3,0)
- C(0,-1.5)
- D(-0.5, -3)
- E(2,-2)
- F(5,1)
- G(1,2)
- H(0,2)



# Example CNC Program

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

- 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
- N,G,X,Y,Z,A,B,C,I,J,K,F,S,T,R,M

# Codes (G and M)

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

- **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 G - Codes

|            |                                    |                |                                     |
|------------|------------------------------------|----------------|-------------------------------------|
| <b>G00</b> | <b>Rapid traverse</b>              | <b>G40</b>     | <b>Cutter compensation – cancel</b> |
| <b>G01</b> | <b>Linear interpolation</b>        | <b>G41</b>     | <b>Cutter compensation – left</b>   |
| <b>G02</b> | <b>Circular interpolation, CW</b>  | <b>G42</b>     | <b>Cutter compensation- right</b>   |
| <b>G03</b> | <b>Circular interpolation, CCW</b> | <b>G70</b>     | <b>Inch format</b>                  |
| <b>G04</b> | <b>Dwell</b>                       | <b>G71</b>     | <b>Metric format</b>                |
| <b>G08</b> | <b>Acceleration</b>                | <b>G74</b>     | <b>Full-circle programming off</b>  |
| <b>G09</b> | <b>Deceleration</b>                | <b>G75</b>     | <b>Full-circle programming on</b>   |
| <b>G17</b> | <b>X-Y Plane</b>                   | <b>G80</b>     | <b>Fixed-cycle cancel</b>           |
| <b>G18</b> | <b>Z-X Plane</b>                   | <b>G81-G89</b> | <b>Fixed cycles</b>                 |
| <b>G19</b> | <b>Y-Z Plane</b>                   | <b>G90</b>     | <b>Absolute dimensions</b>          |
| <b>G20</b> | <b>Inch Units (G70)</b>            | <b>G91</b>     | <b>Incremental dimensions</b>       |
| <b>G21</b> | <b>Metric Units (G71)</b>          |                |                                     |

# 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 |

# Codes (N)

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

- **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.

# Codes (X, Y, Z)

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

- **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

# Codes (I, J, K)

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

- **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

# Codes (F, S, T)

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

- **F-code:** used to specify the feed rate
  - Relative translation of tool w.r.t. work piece
- **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.

# Example

```
%  
:1045  
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250
```

```
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N40 G00 Z1  
N45 X0 Y0
```

```
N50 M05  
N55 M30
```

## Setup

- **%** 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
- **G90 G20** (Absolute units, inches)
- **M06 T3** (Stop for tool change, use tool # 3)
- **M03 S1250** (Turn the spindle on CW to 1250 rpm)

## Cutting

- **G00 X1 Y1** (Rapid to X1, Y1 from origin point)
- **Z0.1** (Rapid down to Z0.1)
- **G01 Z-0.125 F5** (Feed Z-0.125 with feed 5 inch/min)
- **G00 Z1** (Retract fast to Z1)
- **X0Y0** (Retract fast to origin)

## Shutdown

- M05 (Stop spindle)
- M30 (End of program)

# Additional Information

- 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 Code

- 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”.

```
N5 G90 G20  
N10 M06 T3  
N15 M03 S1250  
N20 G00 X1 Y1  
N25 Z0.1  
N30 G01 Z-0.125 F5  
N35 X3 Y2 F10  
N40 G00 Z1  
N45 X0 Y0  
N50 M05  
N55 M30
```

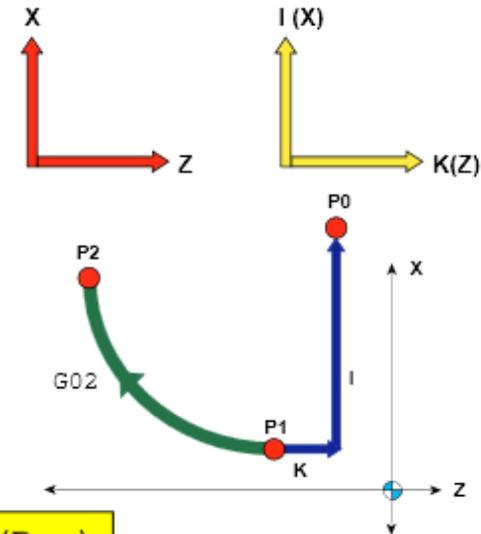
In the example, G00 and G01 are modal.

# Arc

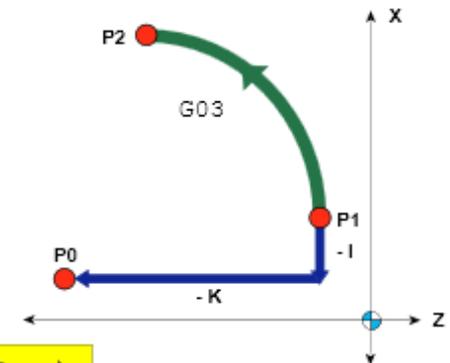
**G02**

**G03**

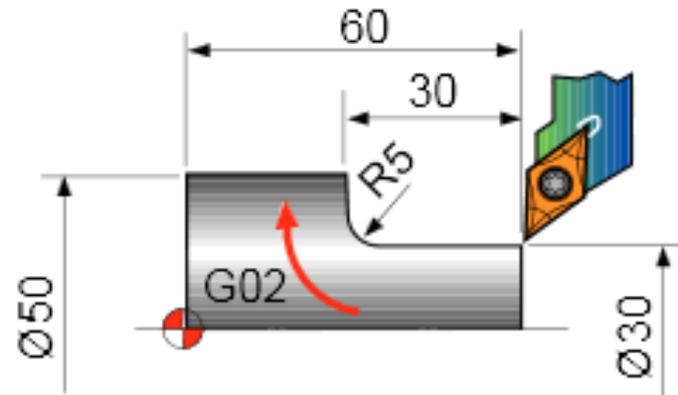
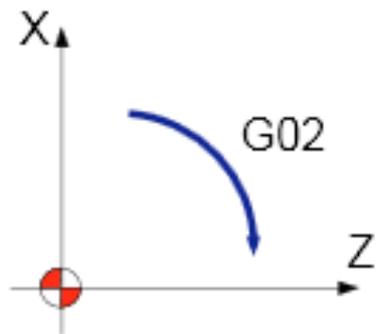
| Conditions |   | Instruction | Meaning   |                      |
|------------|---|-------------|---|----------------------|
|            |   |             | Right hand coordinate   | Left hand coordinate |
| 1          | Rotation direction                                | G02         | CW  | CCW                  |
|            |   | G03         | CCW   | CW                   |
| 2          | Location of end point                             | X,Z         | Location X,Z of commanded point from coordinate   |                      |
|            | Distance to the end point                         | U,W         | Distance from start point to commanded point  |                      |
| 3          | Distance between start point and the center point | I,K         | Distance from start point to the center of and arc with sign, radius value (I always designates the radius) |                      |
|            | Arc radius with no sign radius of circumference   | R           | Radius of circumference   |                      |



N1234 G02 X.. Z.. (R..)

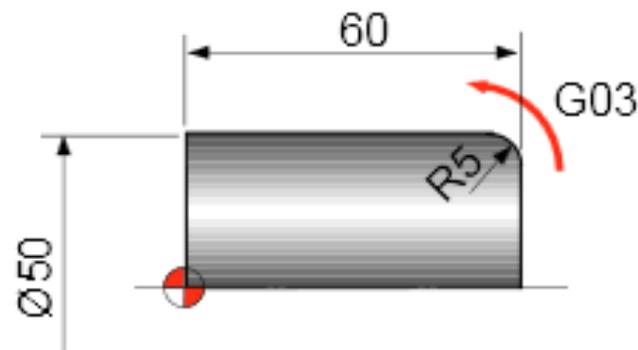
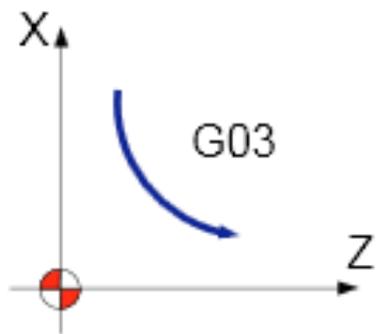


N1234 G03 X.. Z.. (R..)

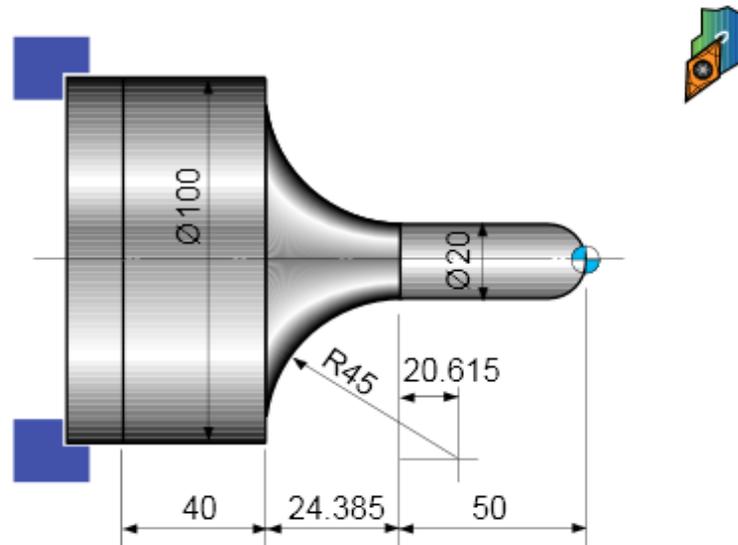


```
G01 X30.0 Z60.0 F0.3 :
Z35.0 :
G02 X40.0 Z30.0 I5.0 :
(G02 U10.0 W-5.0 I5.0)
G01 X50.0 :
Z0 :
```

G03 X(u) Z(w) R\_ F\_ :



```
G01 X40.0 Z60.0 F0.3 :
G03 X50.0 Z55.0 K-5.0 :
```



N10 :

N20 G50 S2000 T0300 :

G96 S200 M03 :

G00 X0 Z3.0 T0303 M08 :

G42 G01 Z0 F0.2 :

G03 X20.0 Z-10.0 R10.0 :

G01 Z-50.0 :

G02 X100.0 Z-74.385 I40.0 K20.615 : (G02 X100.0 Z-74.385 R45.0)

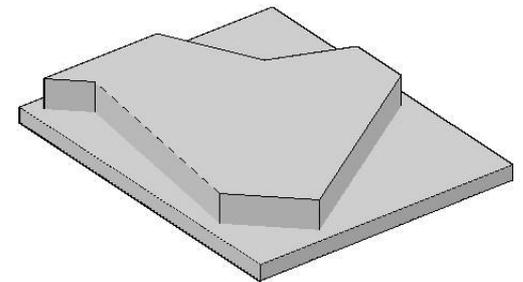
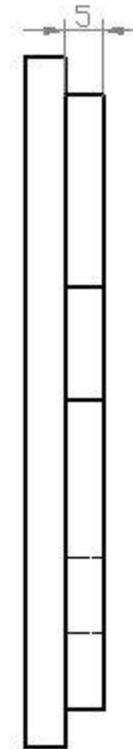
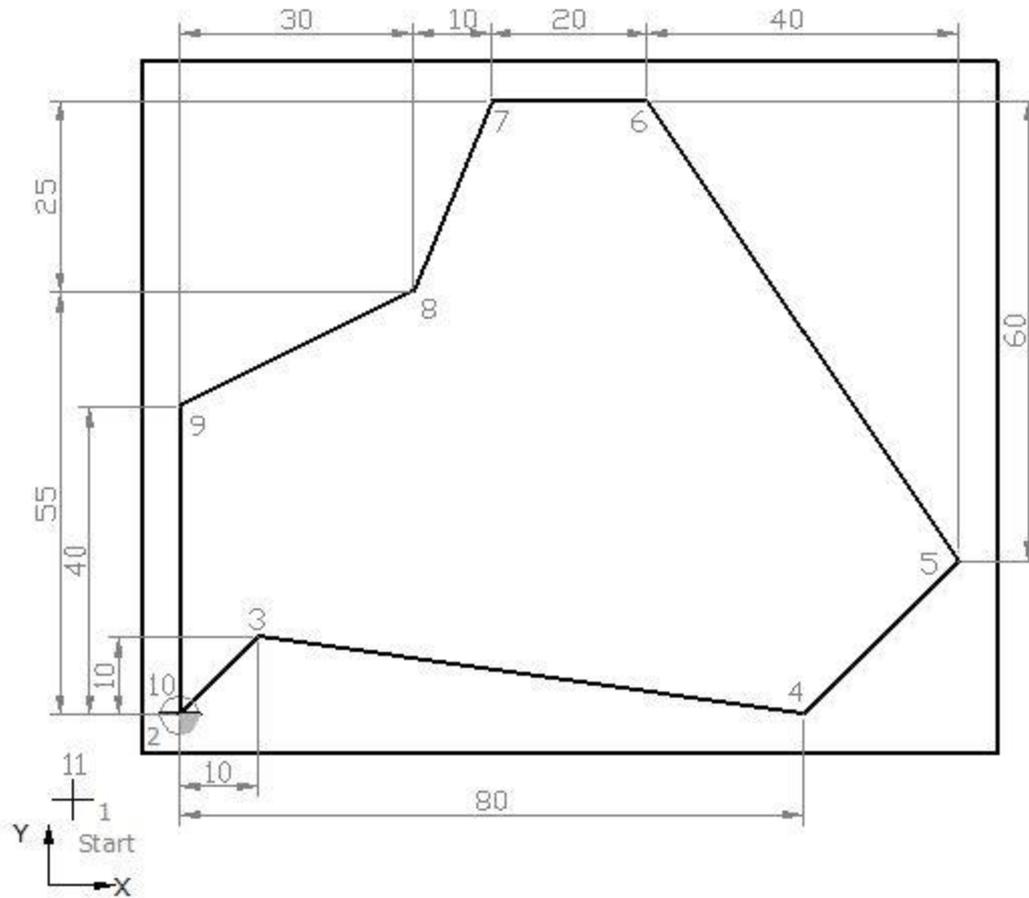
G01 Z-125.0 :

G40 U2.0 W1.0

G00 X200.0 Z200.0 M09 T0300 :

M30 :

# Example



```
N5 G00 G54 G64 G90 G17 X-20 Y-20 Z50
N10 S450 M03 F250 D01 (12.5 MM DIA)
N15 C0
N20 Z5
N25 G01 Z0
N30 Z-5
N35 G42 G91 X20 Y20
N40 X10 Y10
N45 X70 Y-10
N50 X20 Y20
N55 X-40 Y60
N60 X-20
N65 X-10 Y-25
N70 X-30 Y-15
N80 X0 Y-40
N85 G40 G90 X-20 Y-20
N90 G00 Z50
N95 Y100
N100 M30
```

### Explanation of CNC G-Code

G00 : Rapid traverse.

G54 : Zero Offset no. 1.

G64 : Continuous-path mode.

G90 : Absolute dimensioning system.

G91 : Incremental dimensioning system.

G17 : X-Y plan selection.

G42 : Cutter radius compensation activation (right hand side movement)

G40 : Cutter radius compensation de-active

S : Spindle speed

F : Axes motion feed

M : Cutter motion (3=clockwise, 4=anti-clockwise)

D : Tool no