

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

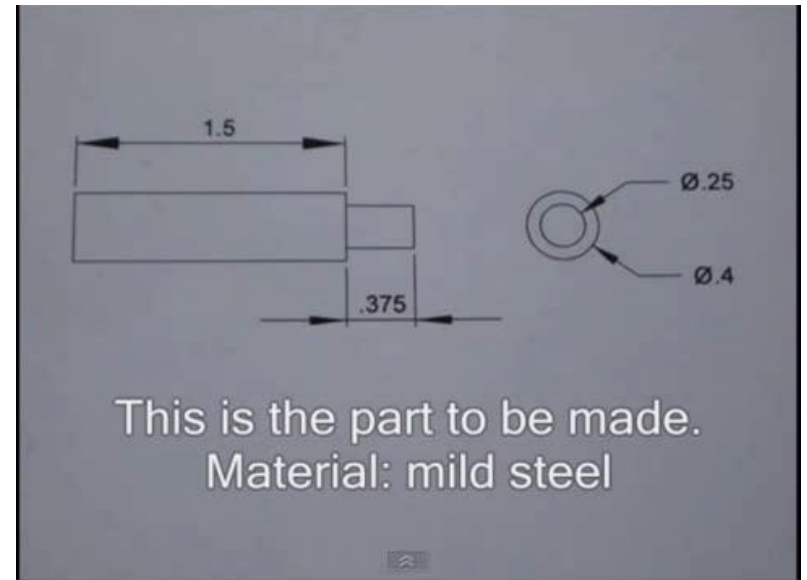
Commercial Control Units

- FANUC CONTROL
- SIEMENS
- GSK
- MECH3

CNC Vs Manual Operation



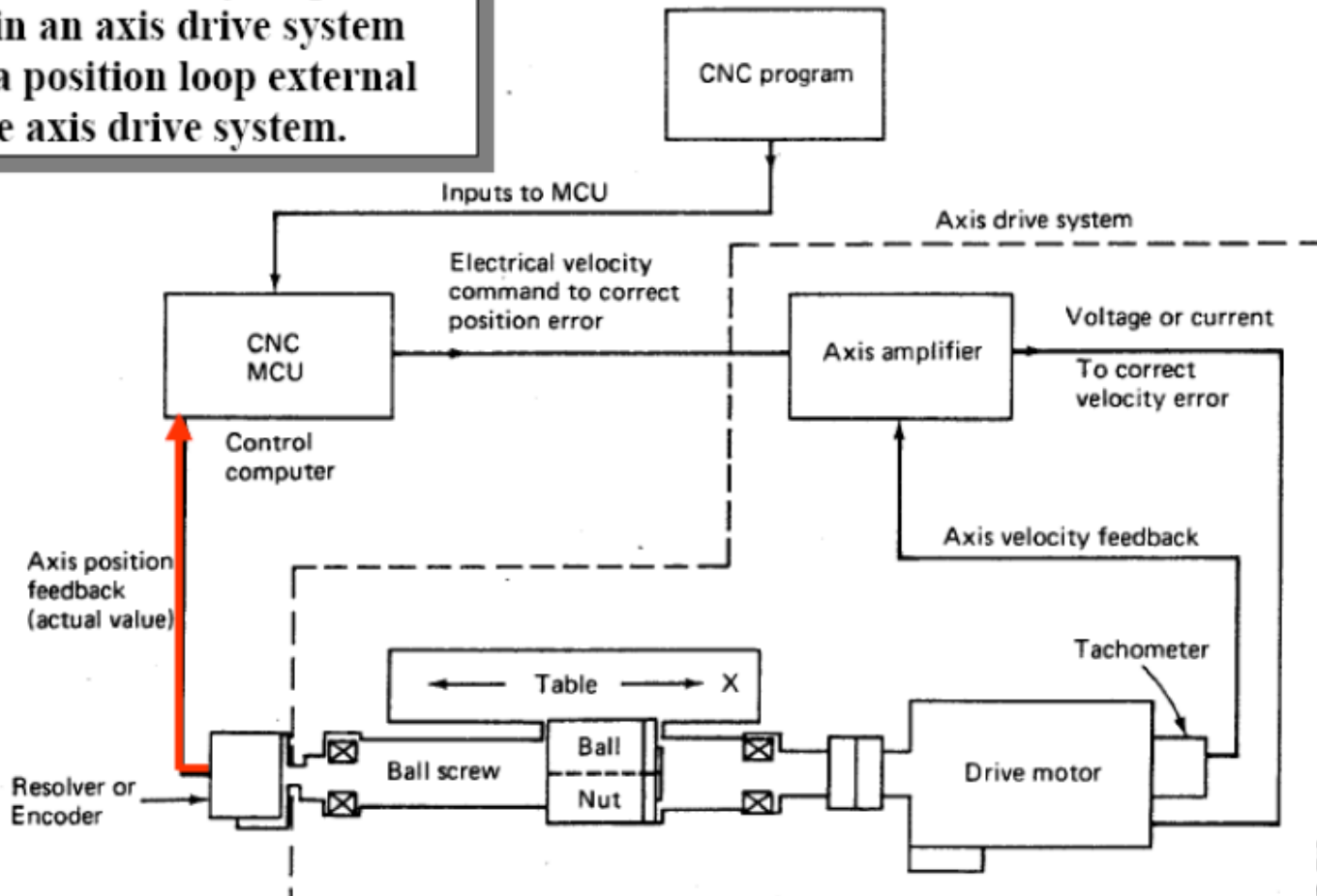
CNC Operation



Manual Operation

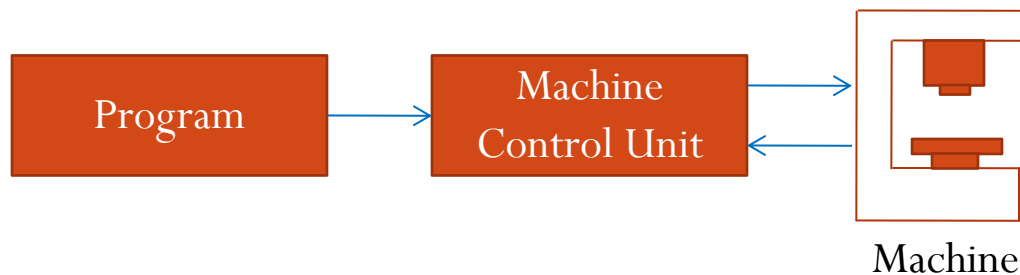
CNC Block Diagram

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.

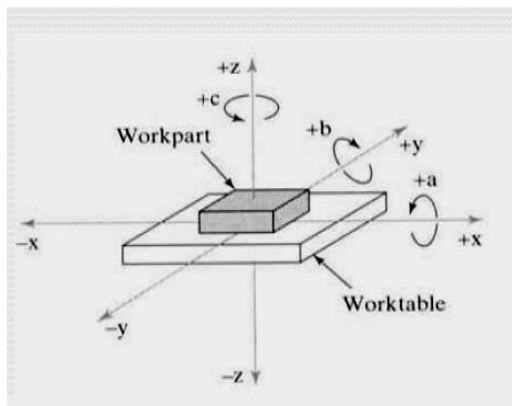


CNC Overview

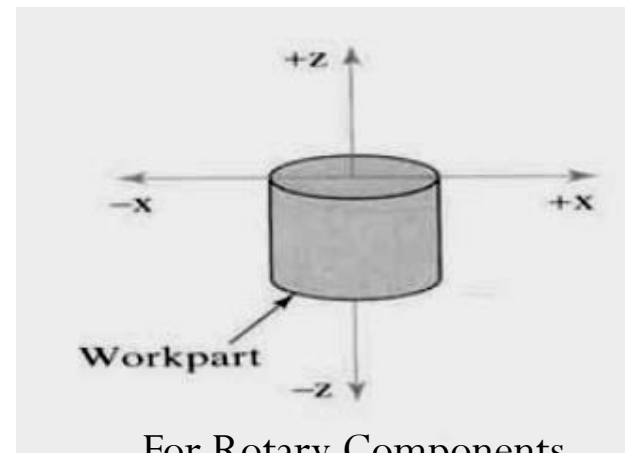
- By integrating a computer processor, computer numerical control (CNC) is obtained or “CNC”
 - allows part machining programs to be edited and stored in the computer memory
 - permit diagnostics and quality control functions during actual machining.
- All CNC machining begins with a part program,
 - 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

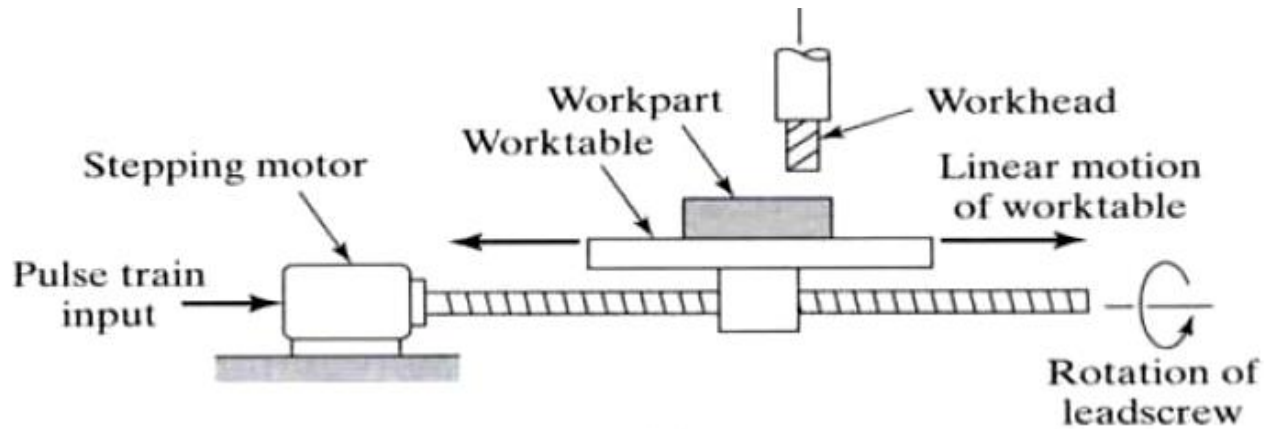


For Prismatic Components

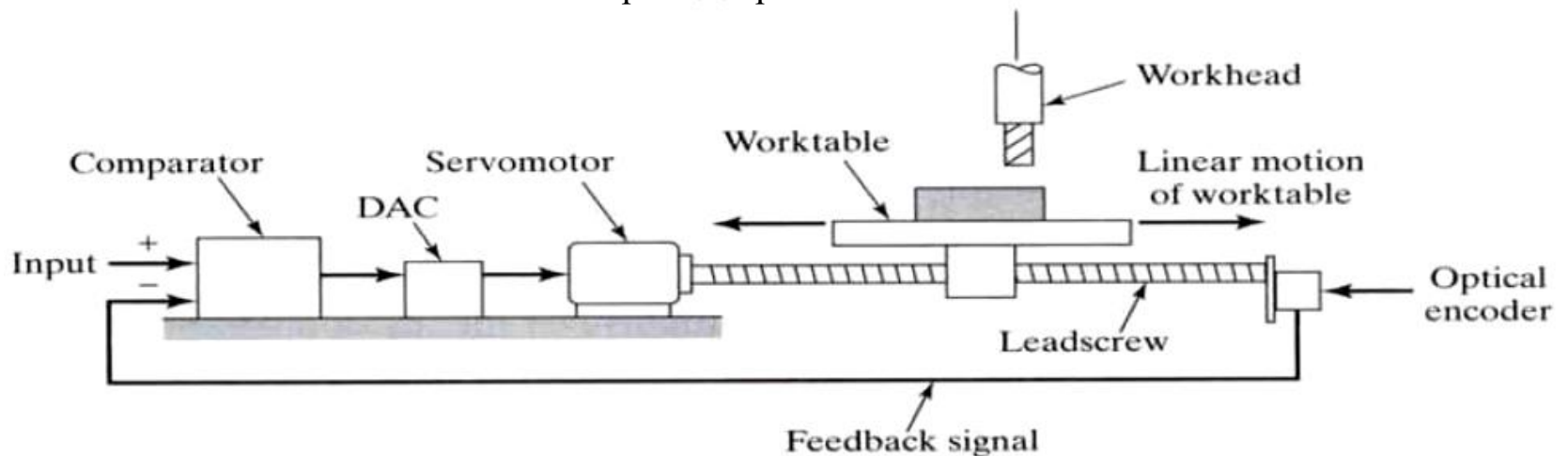


For Rotary Components

Open Loop and Closed Loop Control

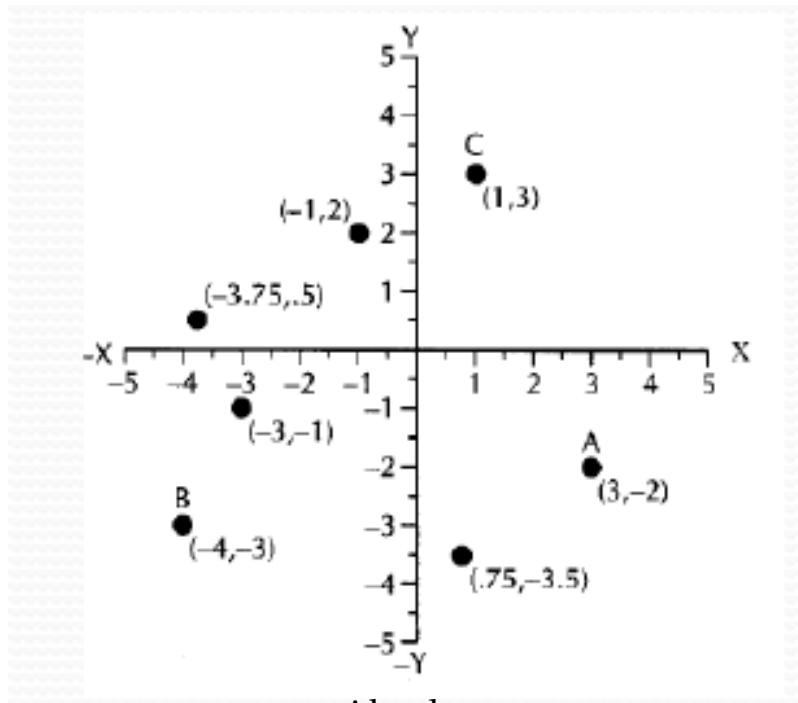


Open Loop Control

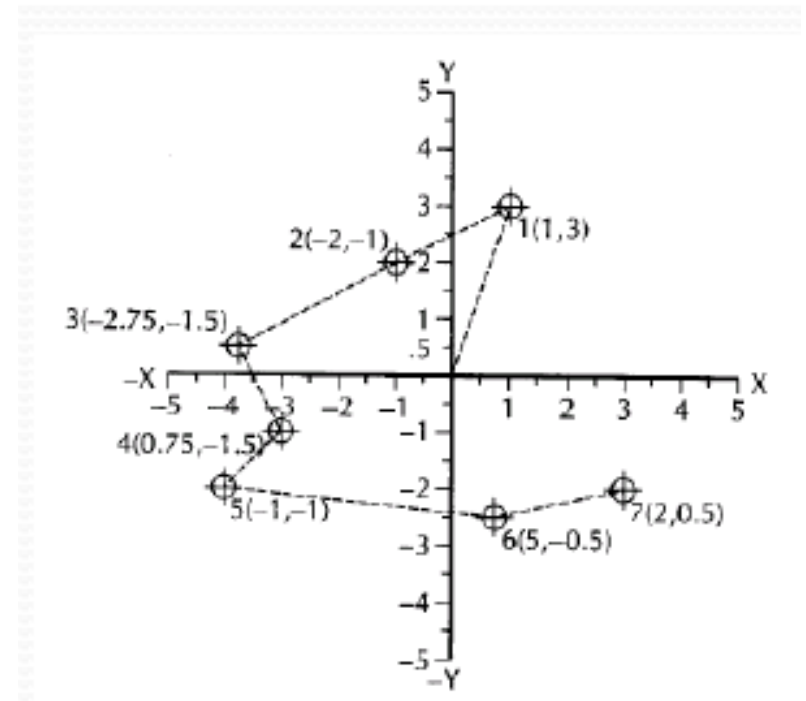


Closed Loop Control

Absolute Vs Incremental Coordinate



Absolute



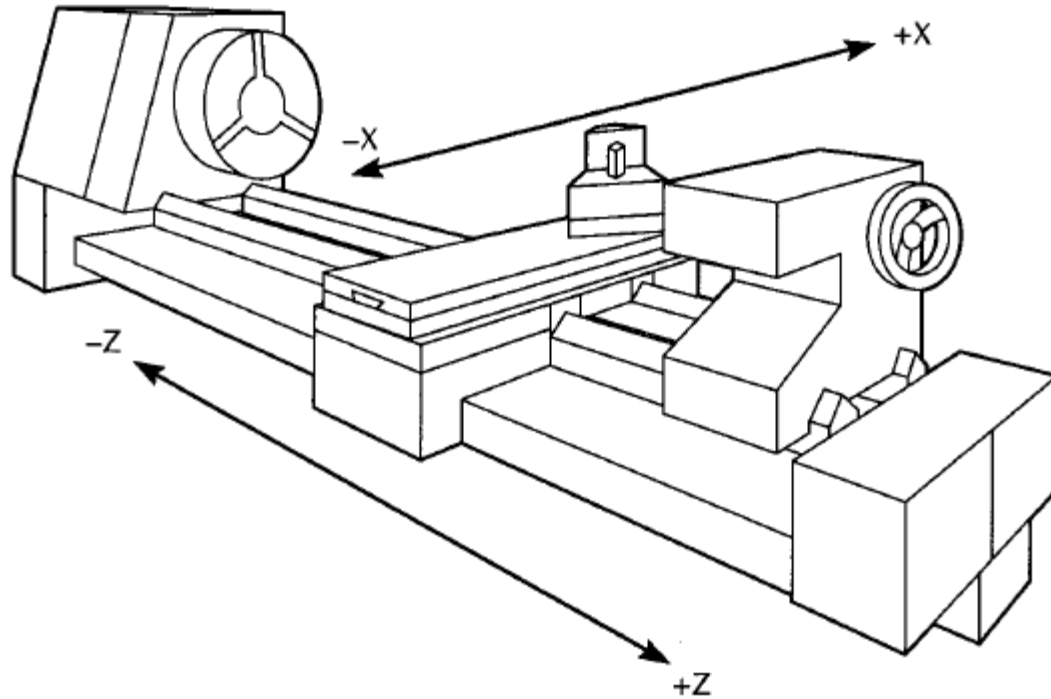
Incremental

CNC Code - Information Required

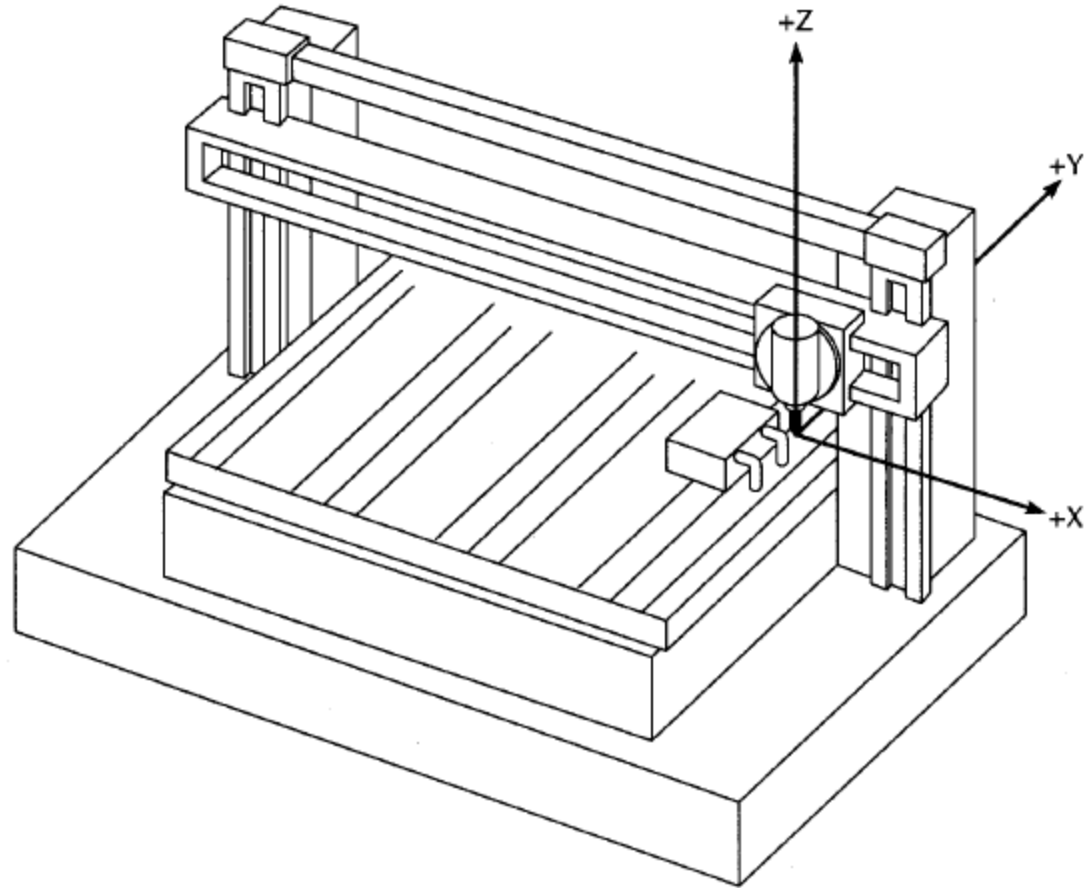
- Preparatory Information: units, incremental or absolute positioning
- Coordinates: X,Y,Z, RX,RY,RZ
- Machining Parameters: Feed rate and spindle speed
- Coolant Control: On/Off, Flood, Mist
- Tool Control: Tool and tool parameters
- Cycle Functions: Type of action required
- Miscellaneous Control: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

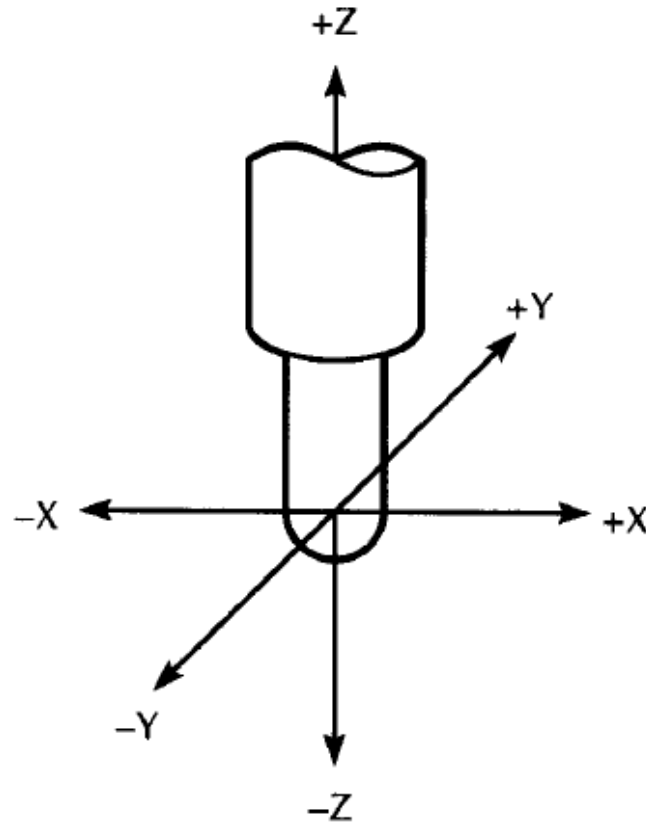
Lathe Coordinate System



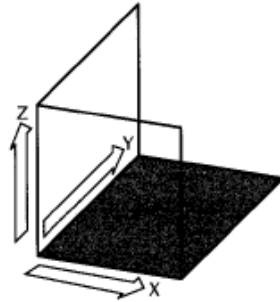
Milling Machine Coordinate System



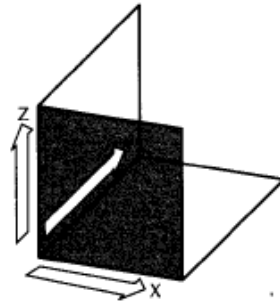
Mill Tool Coordinate System



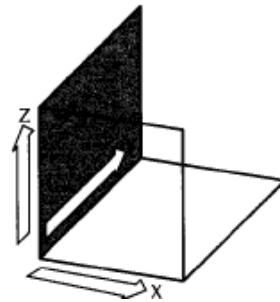
Planes



X-Y Plane

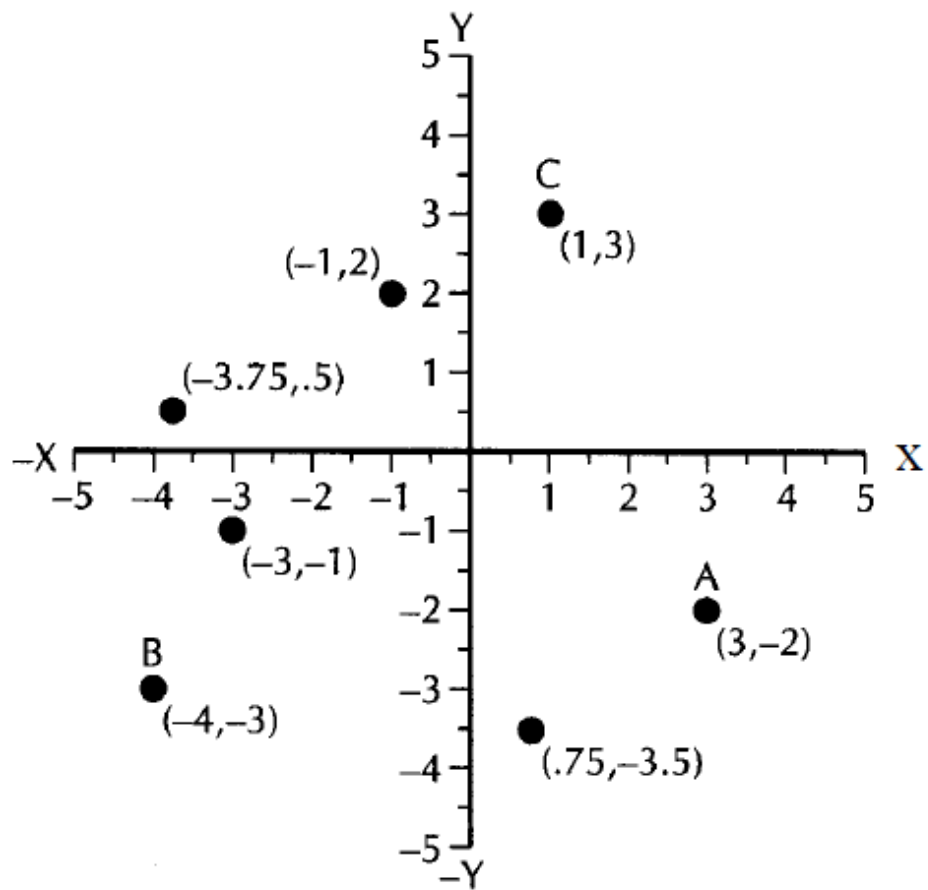


X-Z Plane

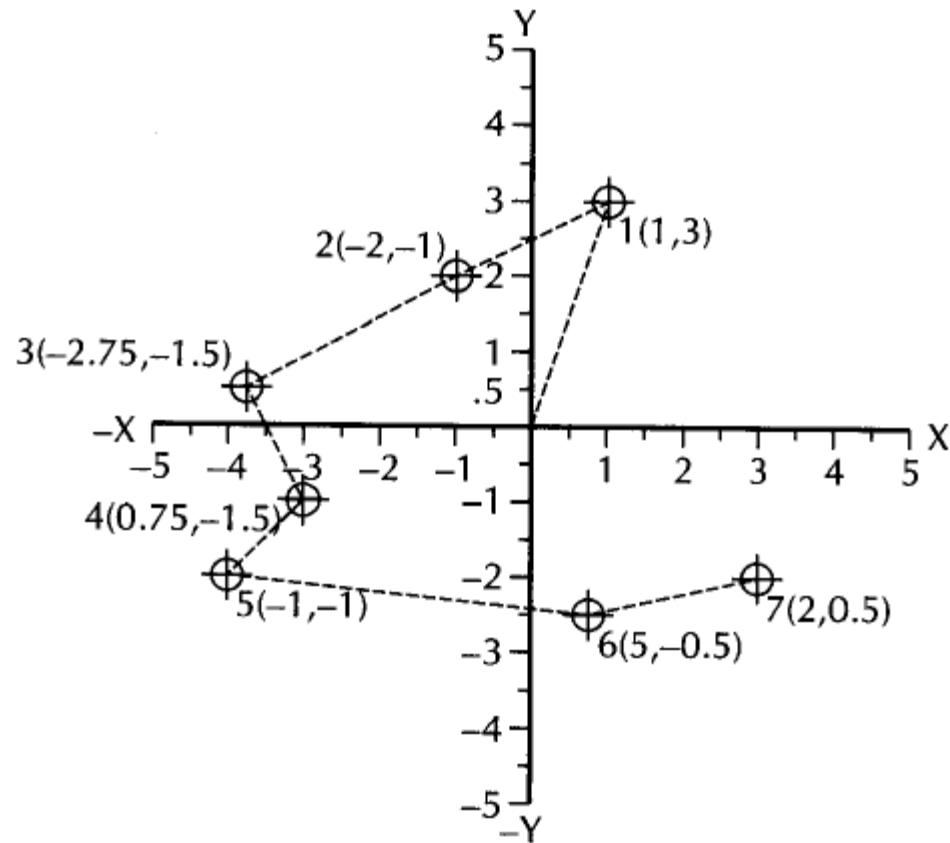


Y-Z Plane

Absolute Coordinate

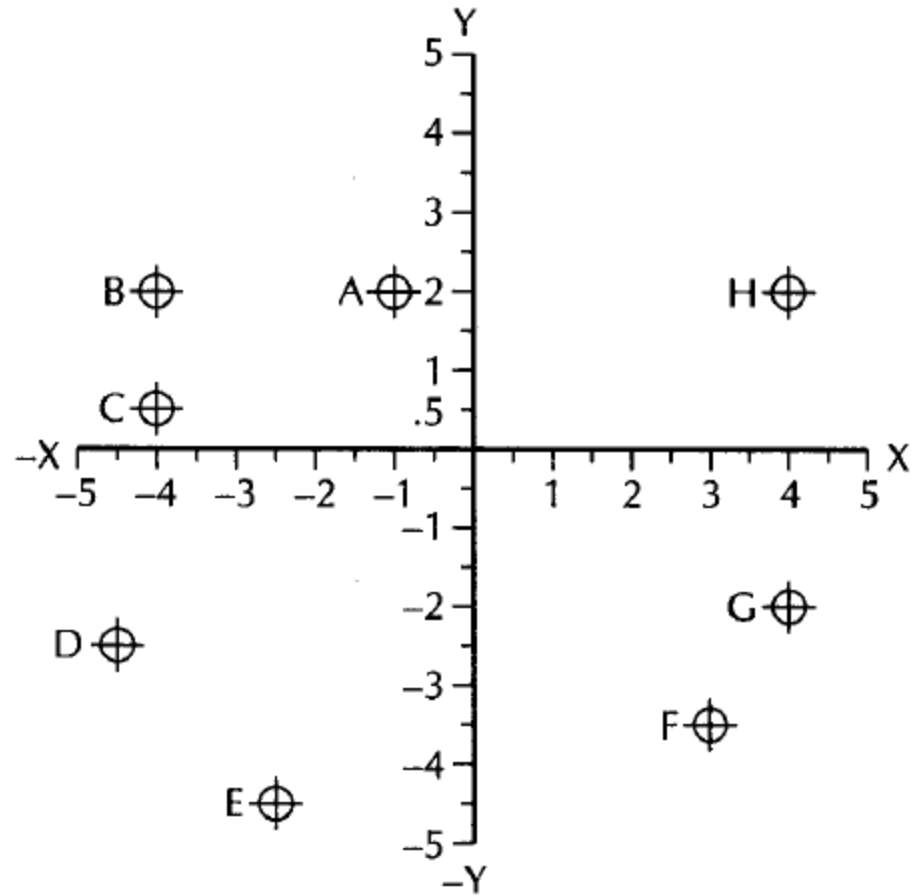


Incremental Coordinate System



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

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

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
```

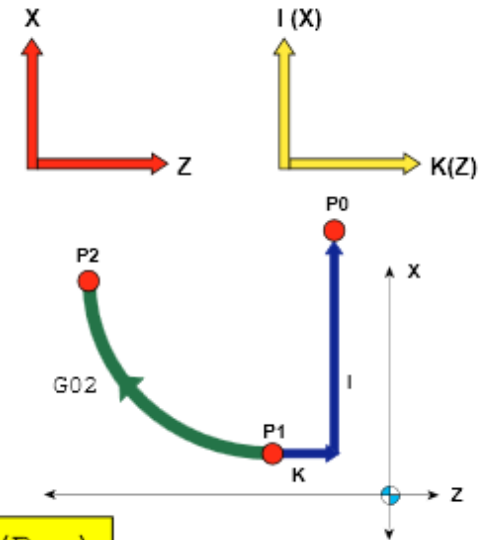
In the example, G00 and G01 are modal.

Arc

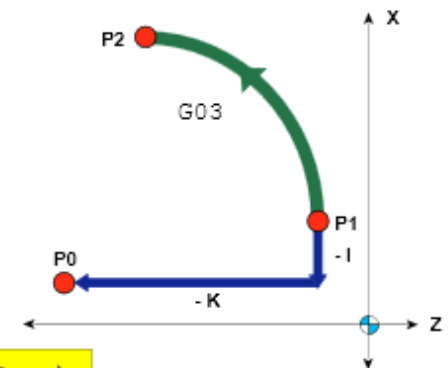
G02

G03

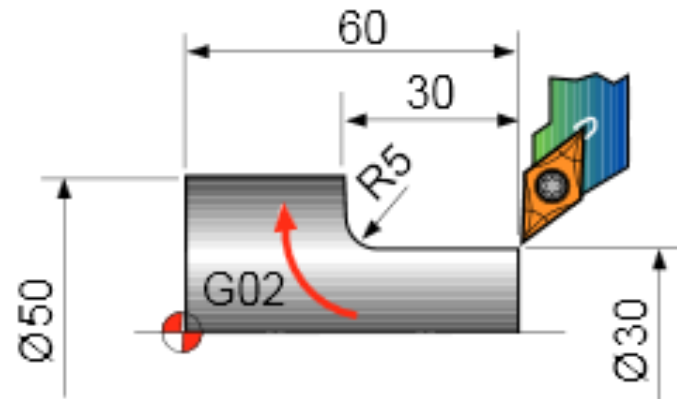
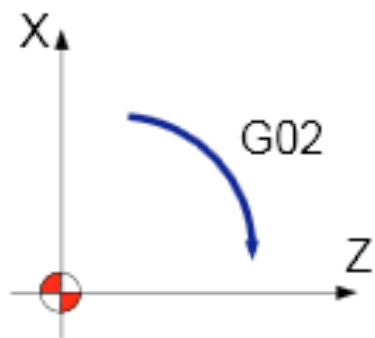
Conditions		Instruction	Meaning	
			Right hand coordinate	Left hand coordinate
1	Rotation direction	G02	CW	CCW
		G03	CCW	CW
2	Location of end point	X,Z	Location X,Z of commanded point from coordinate	
	Distance to the end point	U,W	Distance from start point to commanded point	
3	Distance between start 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..)

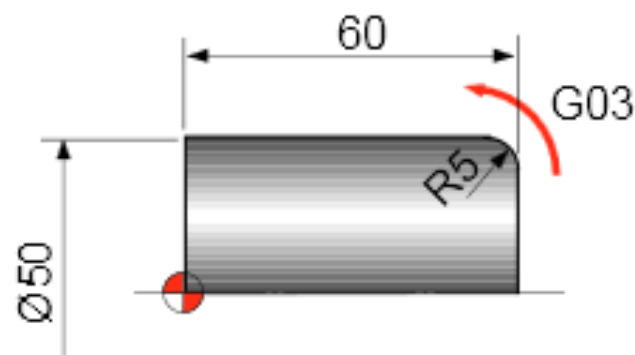
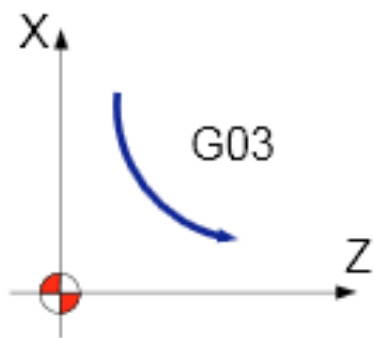


N1234 G03 X.. Z.. (R..)

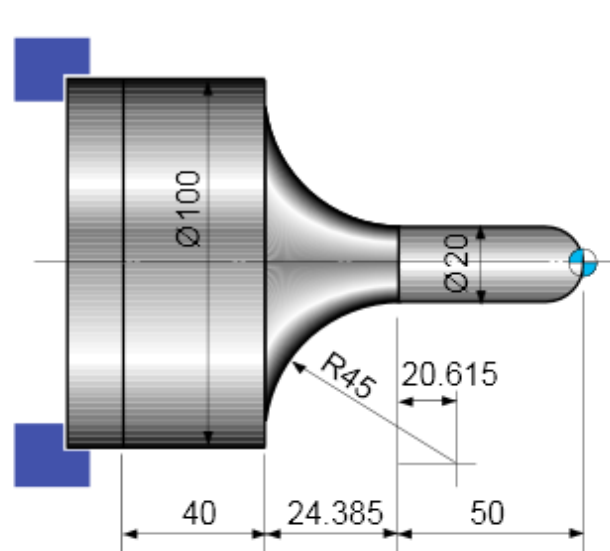


G01 X30.0 Z60.0 F0.3 :
 Z35.0 :
 G02 X40.0 Z30.0 I5.0 :
 (G02 U10.0 W-5.0 I5.0)
 G01 X50.0 :
 Z0 :

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)

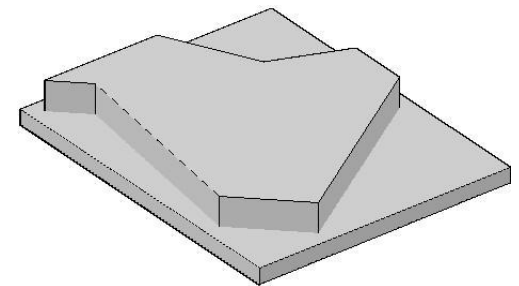
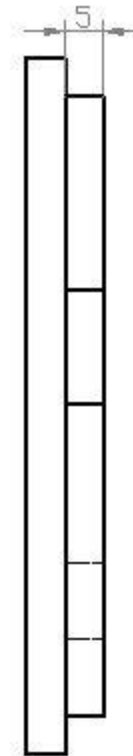
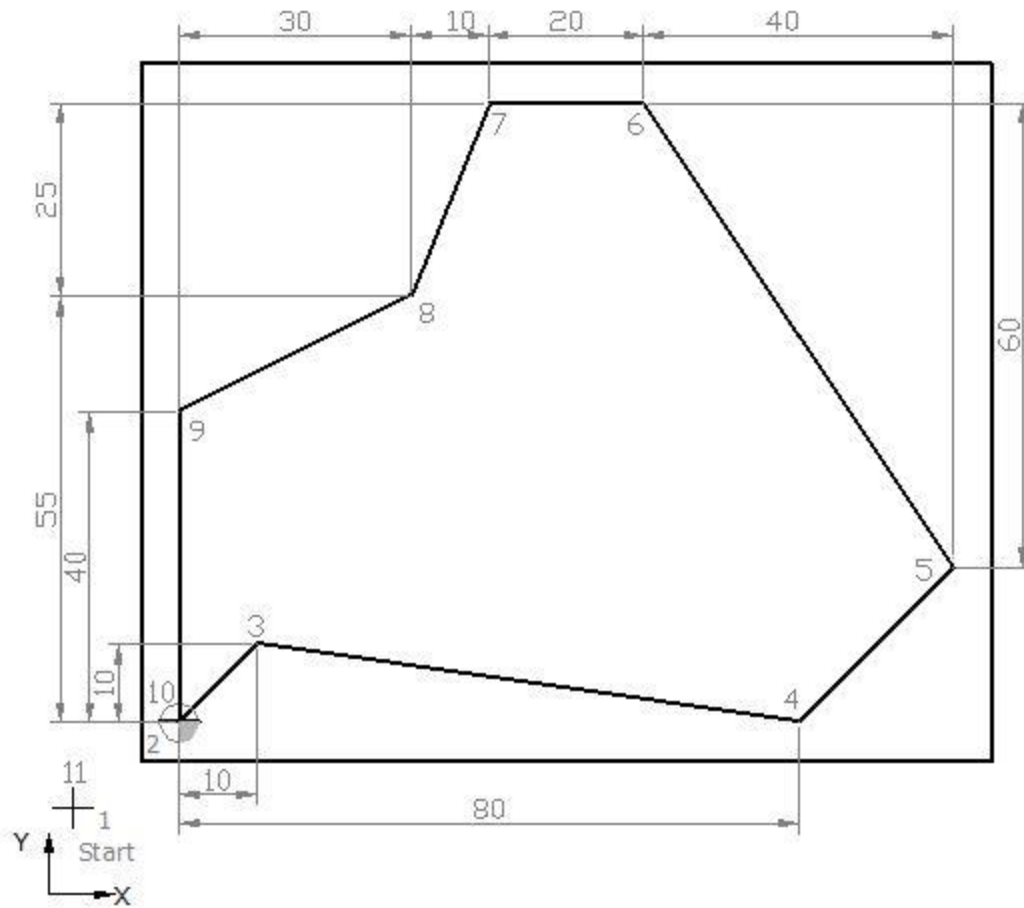
G01 Z-125.0 :

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