The University of Tennessee at Martin

MARTIN School of Engineering

CNC Programming

Lecture 25

Engineering 475

Automated Production Systems

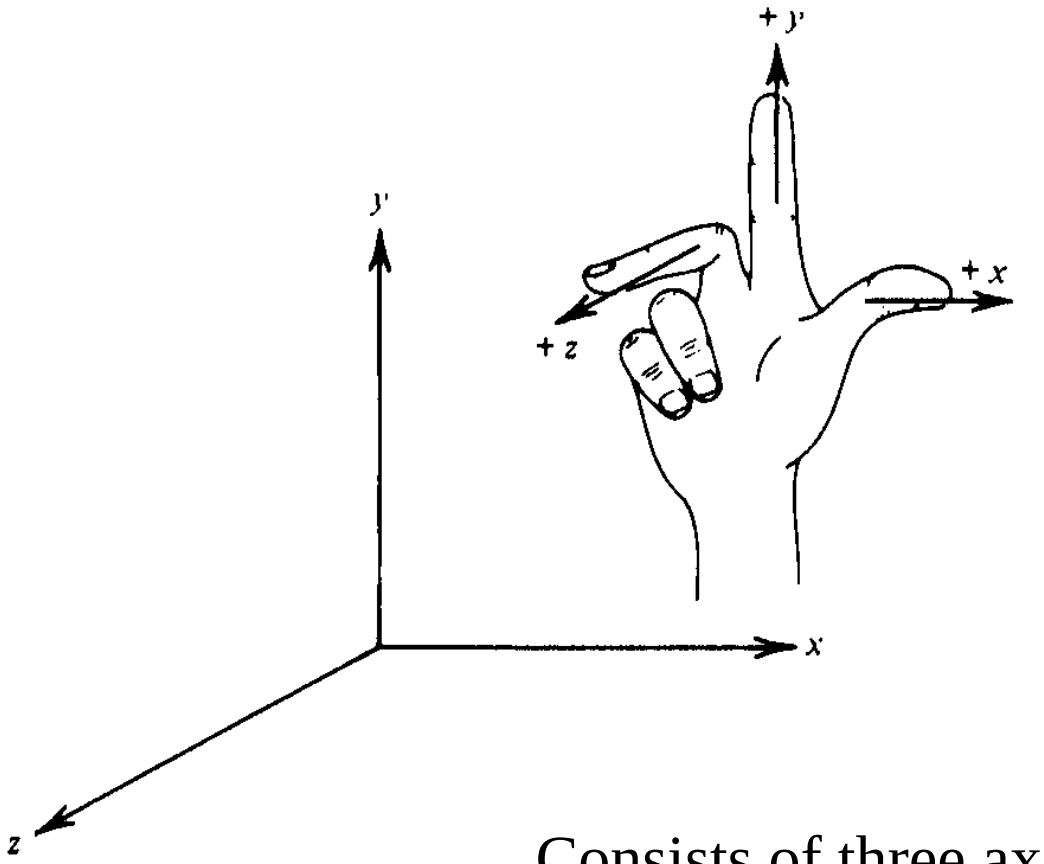
MARTIN

Information Needed by a CNC Machine

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

This information is conveyed to the machine through a set of instructions arranged in a desired sequence – **Program**.

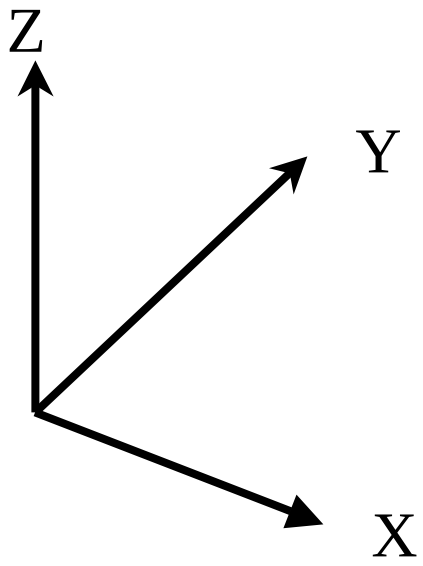
Cartesian Coordinate Systems

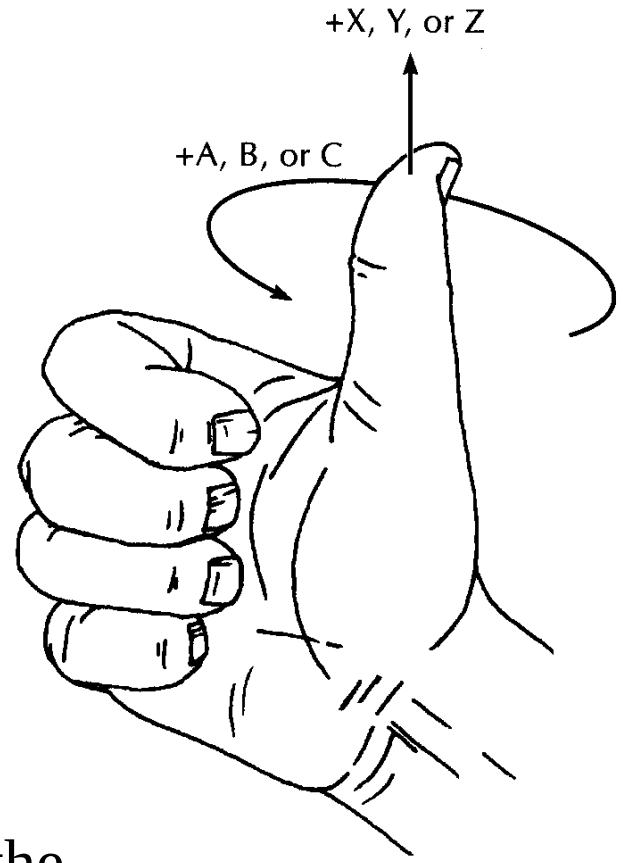


Consists of three axes positioned 90 degrees from each other.

Chang, Fig. 10.1

Right Hand Rule

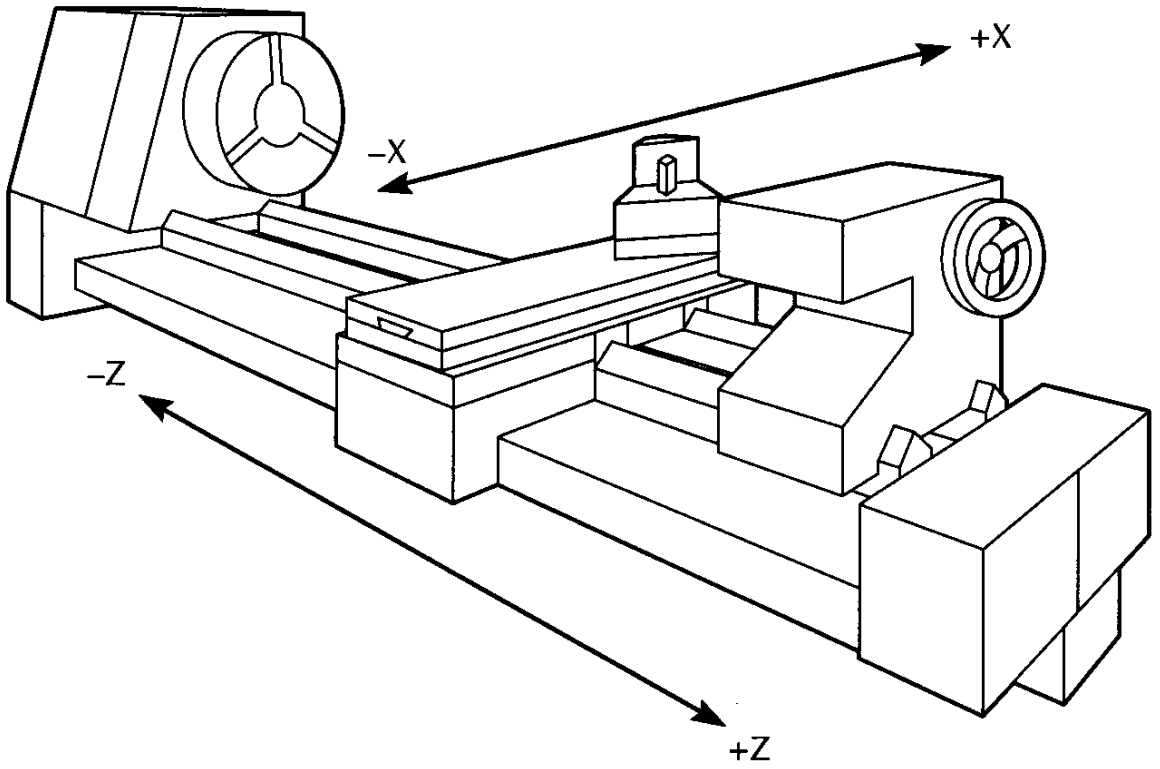




The right hand rule is used to define the positive direction of the coordinate axes.

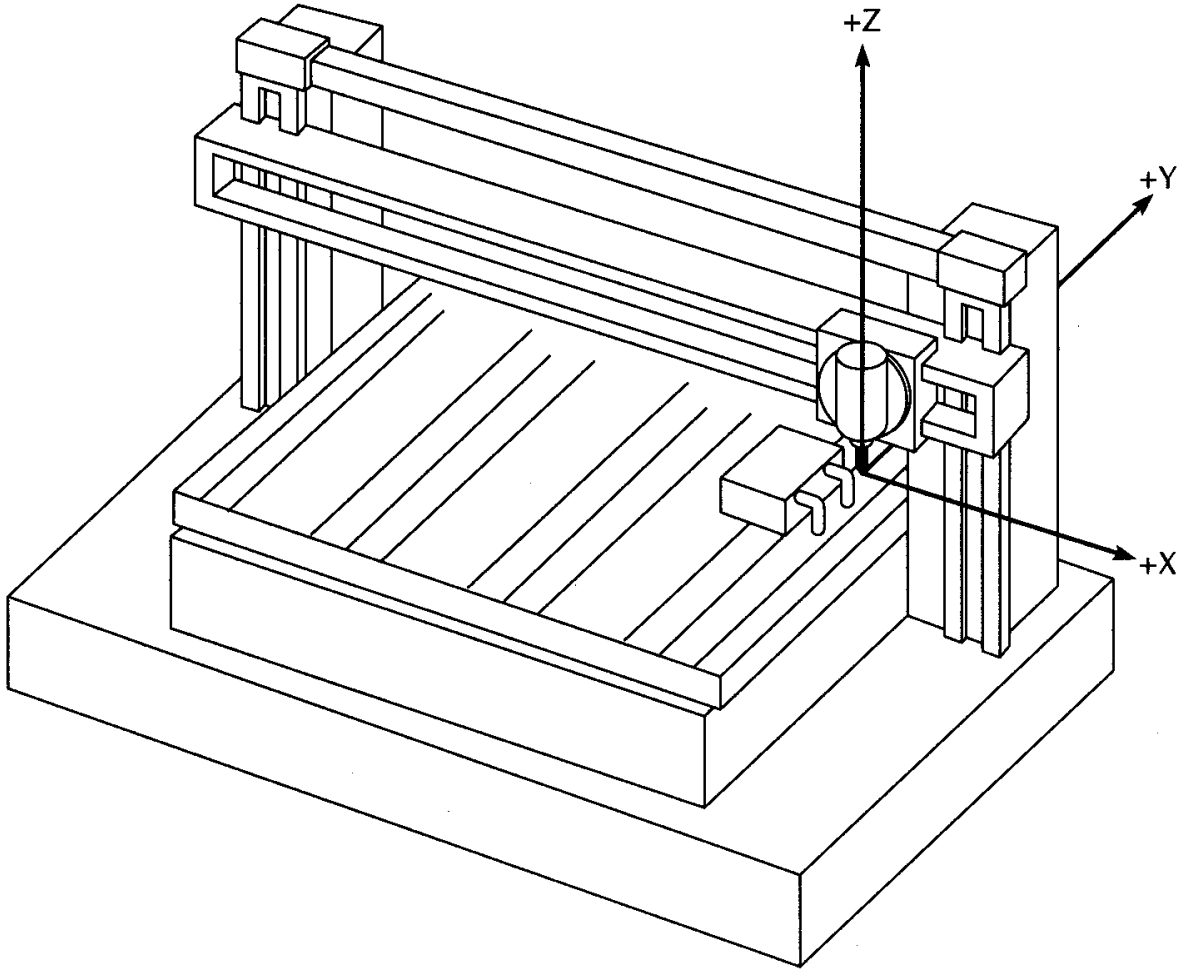
Nanfara, Fig. 2.1

Standard Lathe Coordinate System



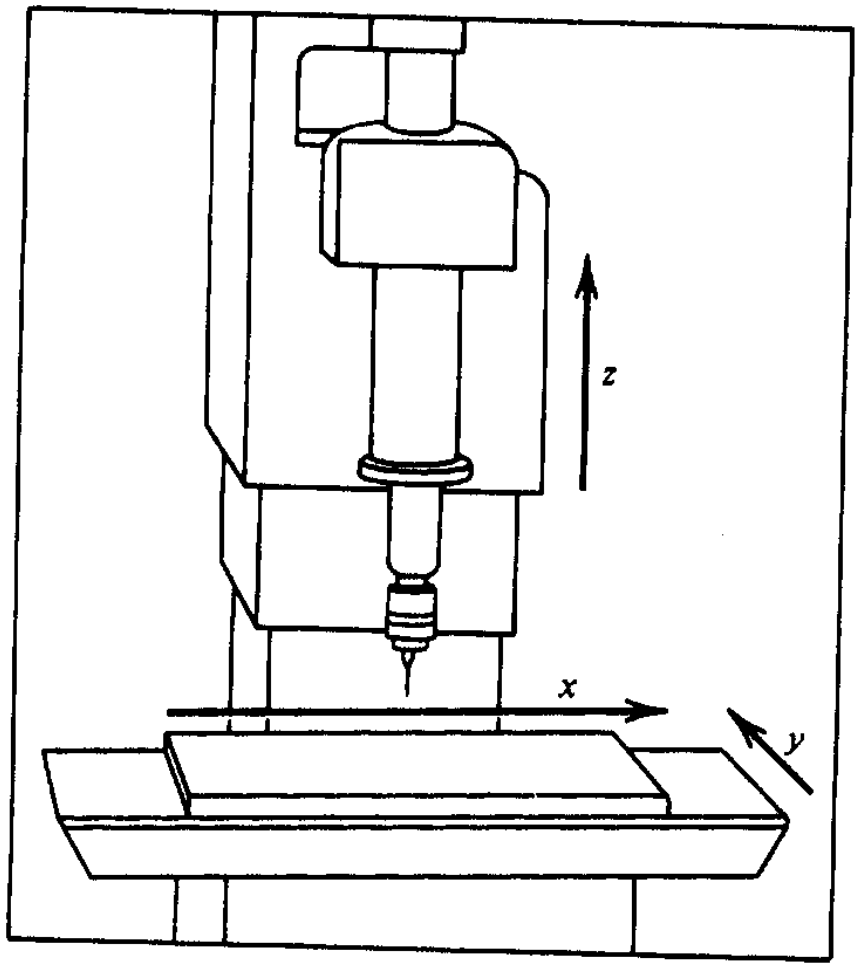
Nanfara, Fig. 2-4

Gantry Mill Coordinate System



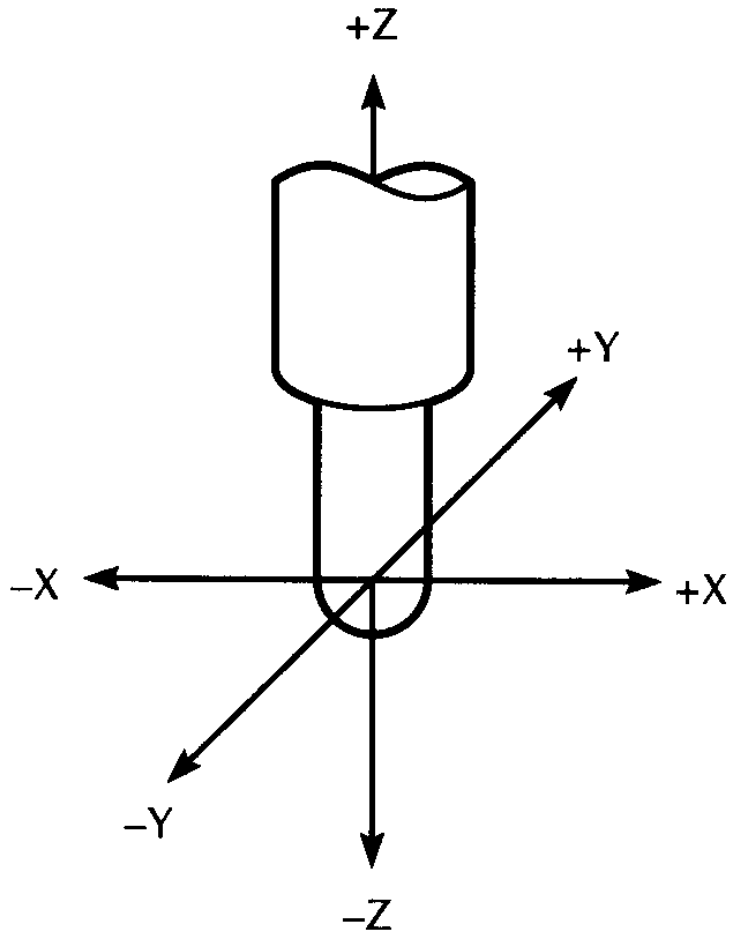
Nanfara, Fig. 2-3

Vertical Mill or Drill Coordinate System



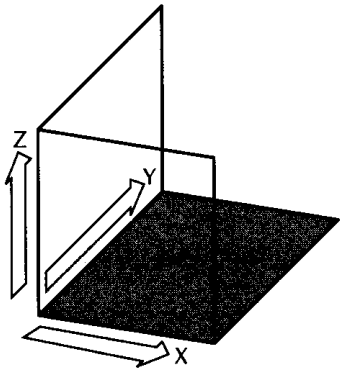
Chang, Fig. 10.3

Milling Tool Coordinate System

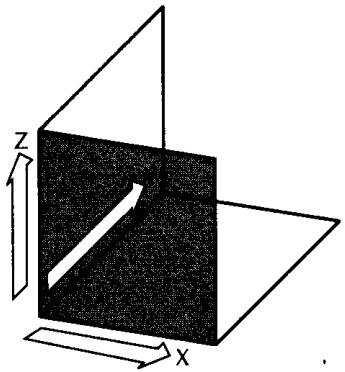


Nanfaro, Fig. 2-6(b)

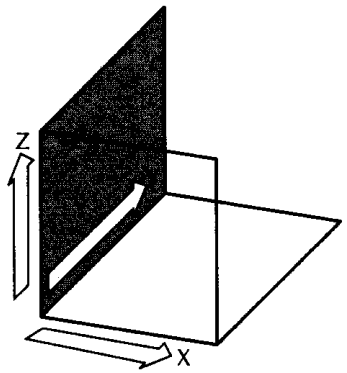
Cartesian Coordinate System Planes



X-Y Plane



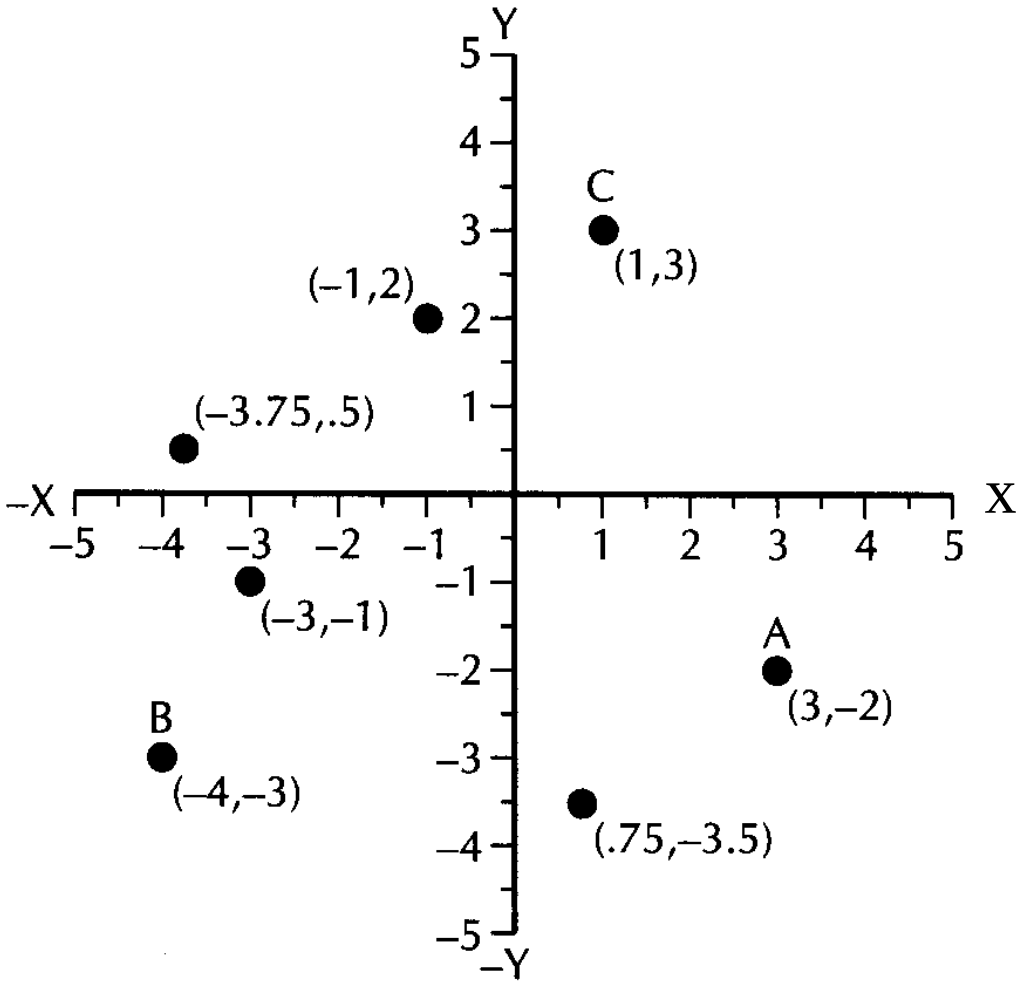
X-Z Plane



Y-Z Plane

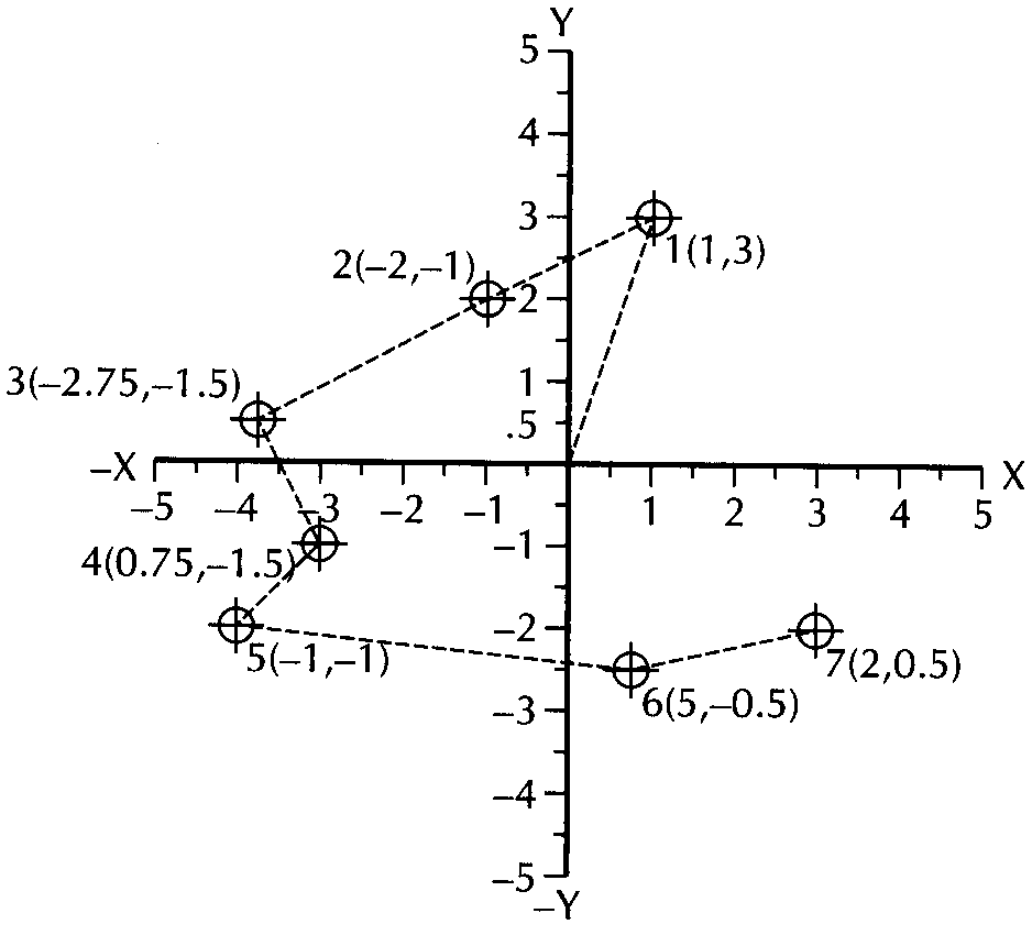
Nanfaro, Fig. 2-7

Absolute Coordinates



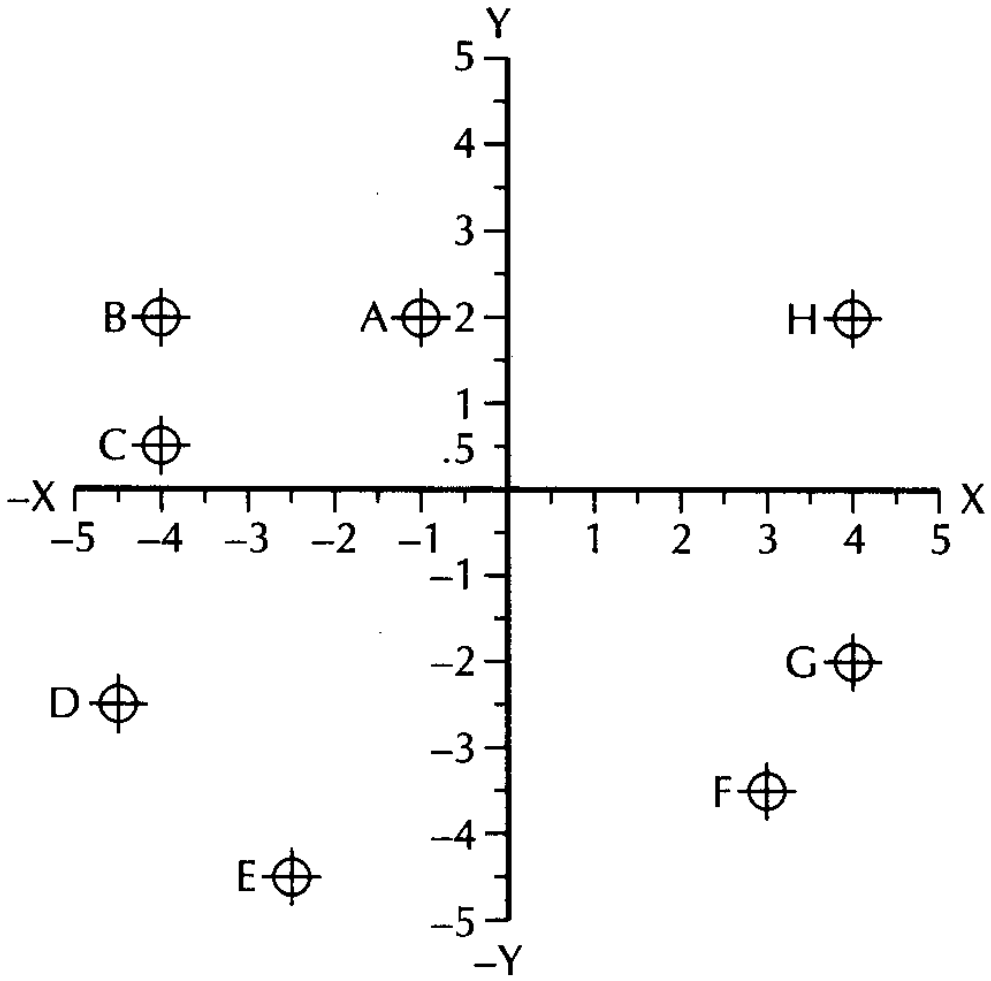
Nanfaro, Fig. 2-10

Incremental Coordinates



Nanfaro, Fig. 2-11

Coordinate System Exercise



Letter Codes

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.

Letters used in Codes

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

Letter Codes
(G and M Codes)

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
```

- **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
G74	Full-circle programming off
G75	Full-circle programming on
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)

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
```

- **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.

Letter Codes
(X,Y, and Z Codes)

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
```

- **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)

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
```

- **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).

Letter Codes
(F, S, and T Codes)

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
```

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 and P Codes)

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
```

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

Example CNC Program

```
%
:1045
N5 G90 G20
N10 M06 T3
N15 M03 S1250
N20 G00 X1 Y1
N25 Z0.1
N30 G01 Z-0.125 F5
```

% 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.

Three Major Phases of a CNC Program

Phase 1 - Program Setup

```
%                   (Program start flag)
:1001               (Four digit program number)
N5 G90 G20          (Absolute units, inches)
N10 M06 T2          (Stop for tool change, use tool # 2)
N15 M03 S1200       (Turn the spindle on CW to 1200 rpm)
```

Three Major Phases of a CNC Program

Phase 2 – Material Removal

```
N20 G00 X1 Y1          (Rapid to X1,Y1 from origin point)
N25 Z0.125             (Rapid down to Z0.125)
N30 G01 Z-0.125 F5     (Feed down to Z-0.125 at 5 ipm)
N35 G01 X2 Y2          (Feed diagonally to X2,Y2)
N40 G00 Z1             (Rapid up to Z1)
N45 X0 Y0              (Rapid to X0,Y0)
```

Three Major Phases of a CNC Program

Phase 3 – System Shutdown

N50 M05	(Turn the spindle off)
N55 M30	(End of program)

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

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
```

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”.

In the example, G00 and G01 are 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)

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

Assignment

Review the material presented.

Next class we look at an example, and will develop G-code programs in class.