CAD/CAM
Numerical Controlled (NC) machines and
NC programming language

Numerical Control (NC)

A method of automatically operating a manufacturing machine based on a code of letters, numbers and special characters

Program

A complete set of coded instruction for executing an operation

Numerical Control (NC)

Numerical Control (NC) can be defined as an operation of machine tools by the means of specifically coded instructions to the machine control system.

- The instructions are combinations of the letters of alphabet, digits and selected symbols, for example: a decimal point ".", the percent sign "%" or the parenthesis symbols "()".
- All instructions are written in a logical order and a predetermined form.
- The collection of all instructions necessary to machine a part is called an NC Program, CNC Program, or a Part Program.
- Such a program can be stored for a future use and used repeatedly to achieve identical machining results at any time.

Numerical Control (NC)

NC and CNC Technology

The difference in the meaning of the NC and CNC.

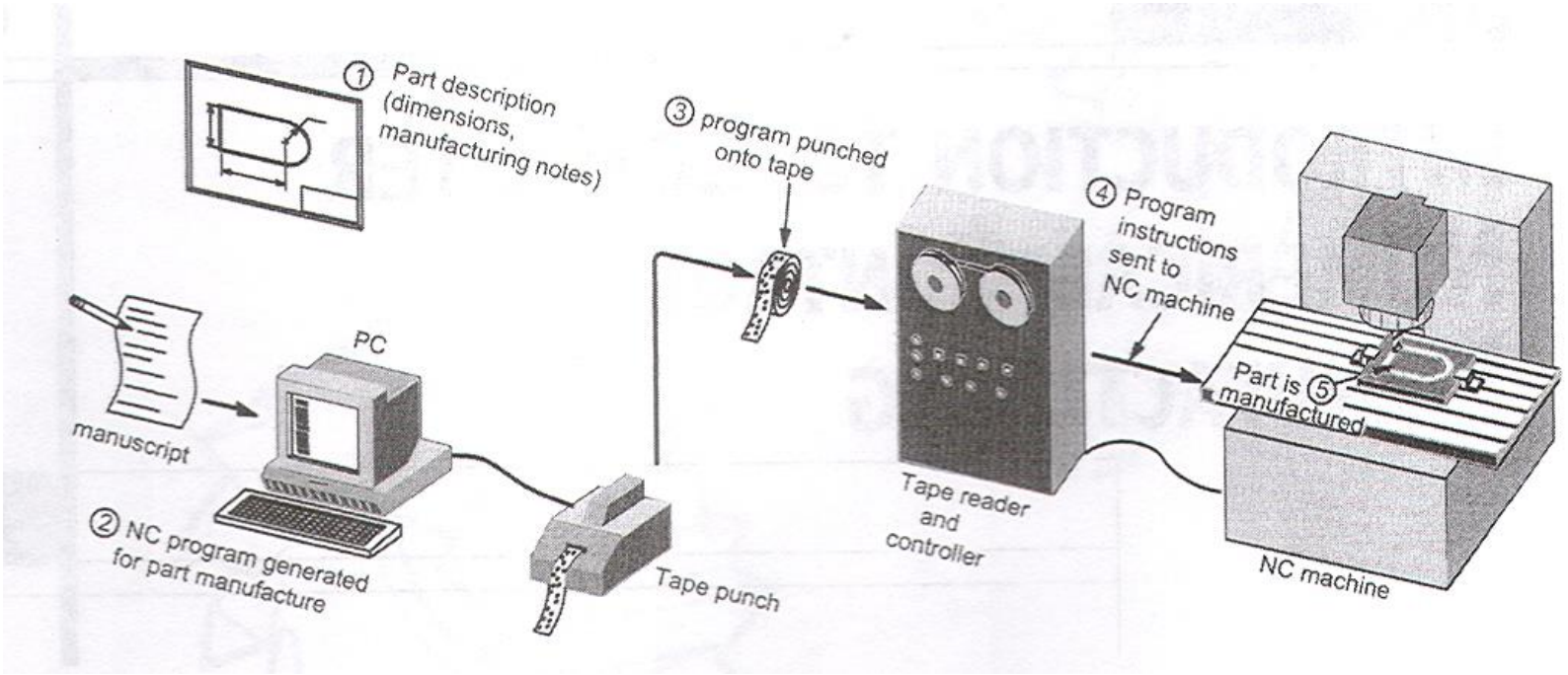
The NC stands for the older and original **Numerical Control** technology, whereby the **CNC** stands for the newer **Computerized Numerical Control** technology**.**

Both systems perform the same tasks, namely manipulation of data for the purpose of machining a part.

- In both cases, the internal design of the control system contains the logical instructions that process the data.
- The NC system uses a fixed logical functions, those that are built-in and permanently wired within the control unit. These functions cannot be changed by the programmer or the machine operator.
- The CNC system uses an internal microprocessor that contains a memory by which a variety of routines that are capable of manipulating logical functions are registered and stored.

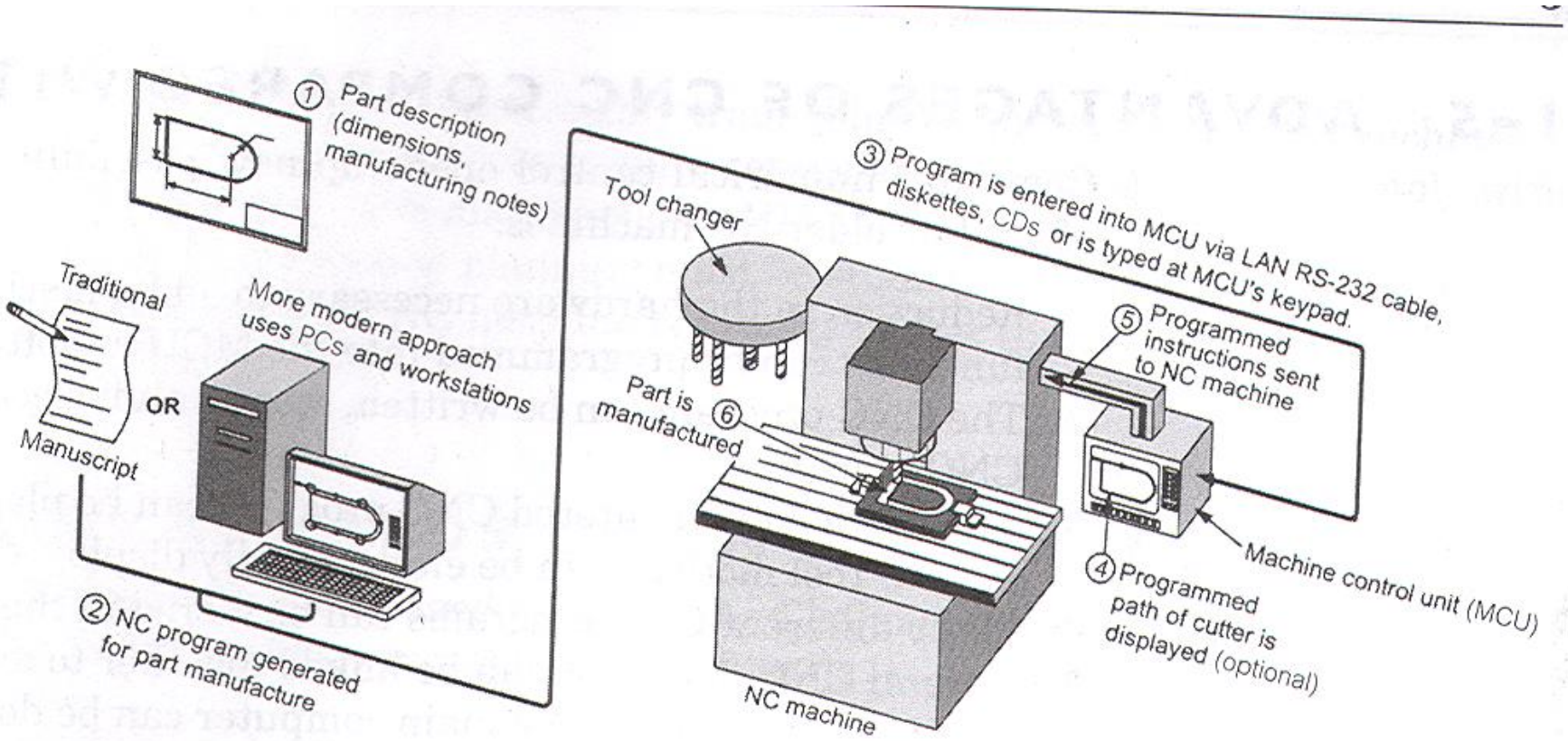
Numerical Control (NC)

A Numerical Control machine has two parts: 1. machine control unit and 2. machine tool

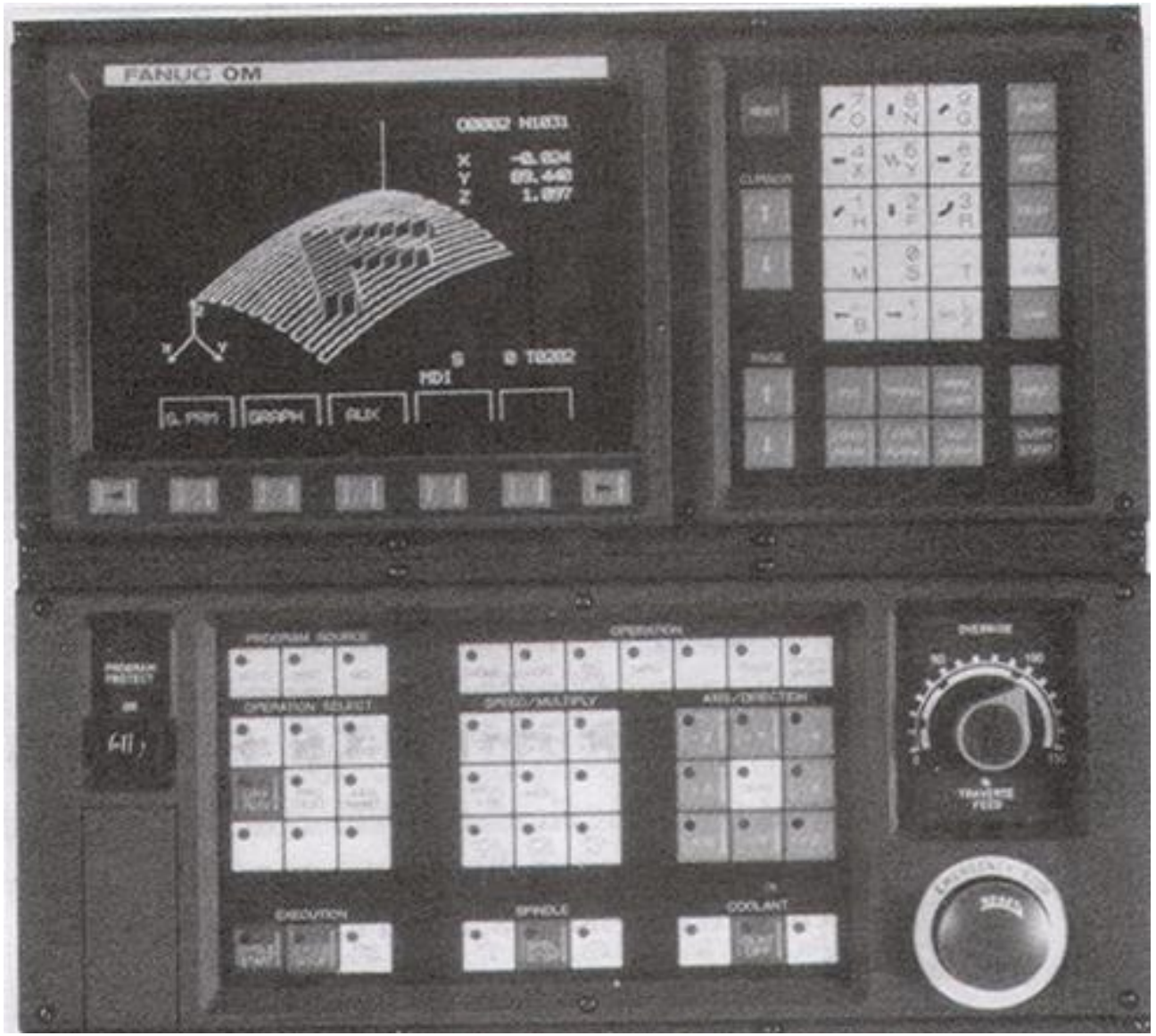


Computer Numerical Control (CNC)

- A NC machine with the added feature of an onboard computer is called a computer Numerical Control machine (CNC)



CNC control unit



CNC Machining Centers and Turning Centers

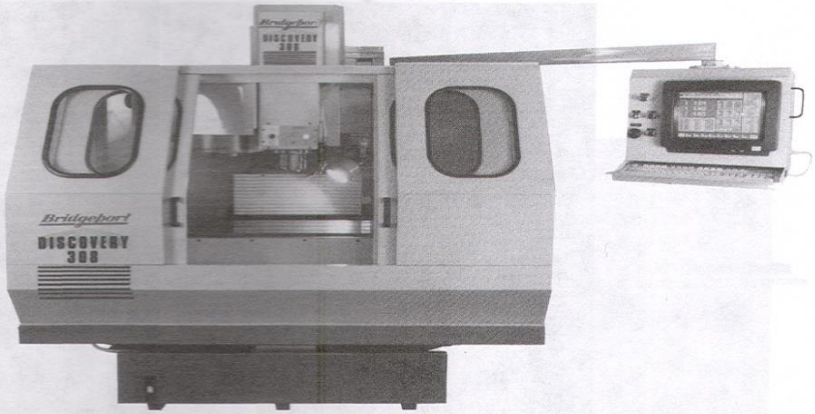


FIGURE 1–6 A vertical spindle machining center. (Photo courtesy of Bridgeport Machines, Inc.)

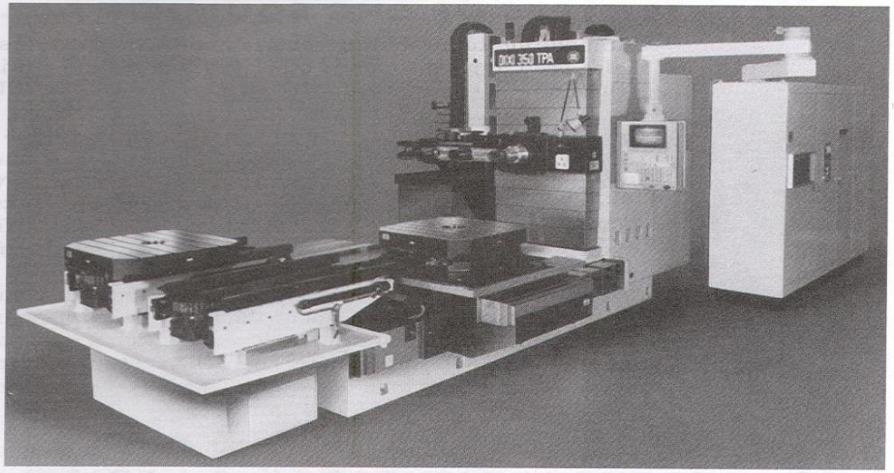


FIGURE 1–7 A horizontal spindle machining center with an automatic tool changer and two pallet work changers. (Photo courtesy of American SIP.)

Advantages of CNC over manual machining

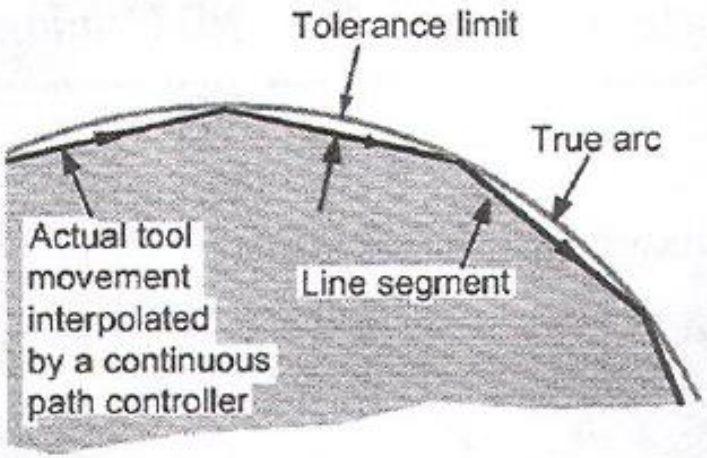
- The approach to machine a part is the same for both methods but CNC machining improves the following:
  1. Setup time reduction
  2. Lead time reduction
  3. Accuracy and repeatability
  4. Contouring of complex shapes
  5. Simplified tooling and work holding
  6. Consistent cutting time
  7. General productivity increases

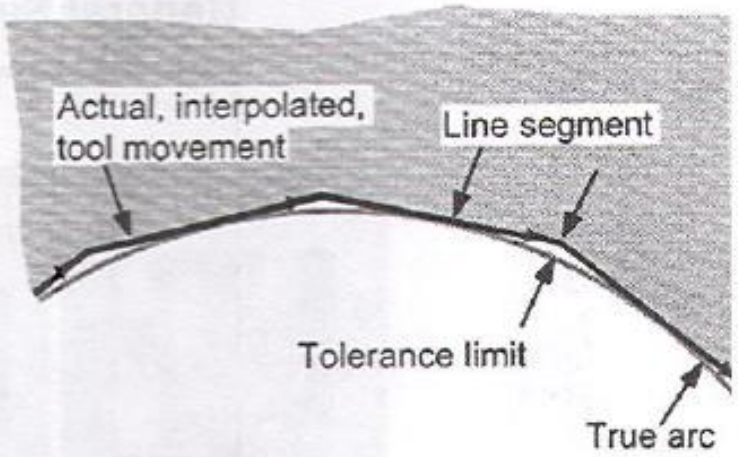
CNC Machines

Types of CNC machine tools

- Mills and Machining centers
- Lathes and Turning Centers
- Drilling machines
- Boring mills and Profilers
- EDM machines
- Punch presses and Shears
- Flame cutting machines
- Routers
- Water jet and Laser profilers
- Cylindrical grinders
- Welding machines
- Benders, Winding and Spinning machines

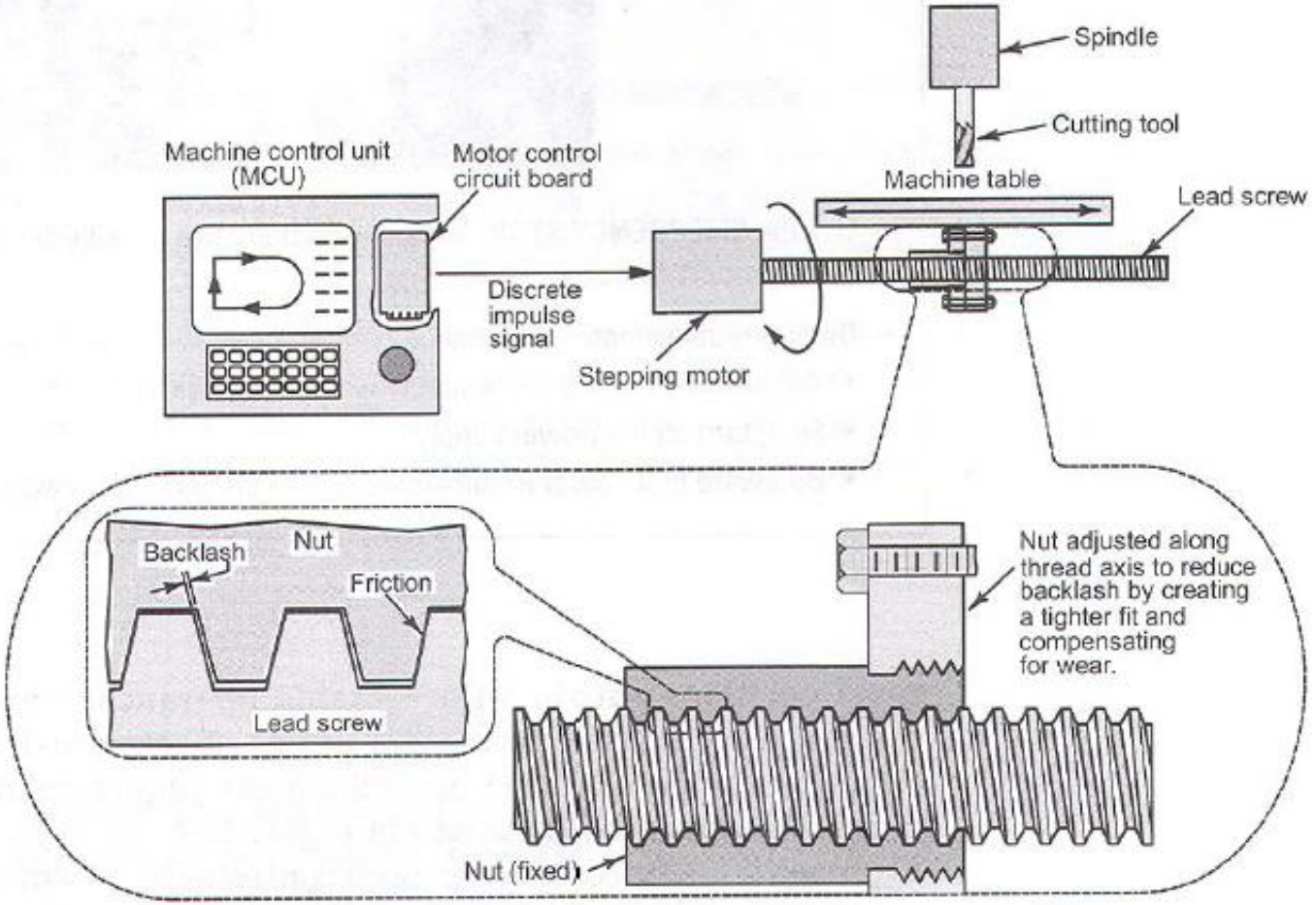
System Control





Loop systems for controlling tool movemen

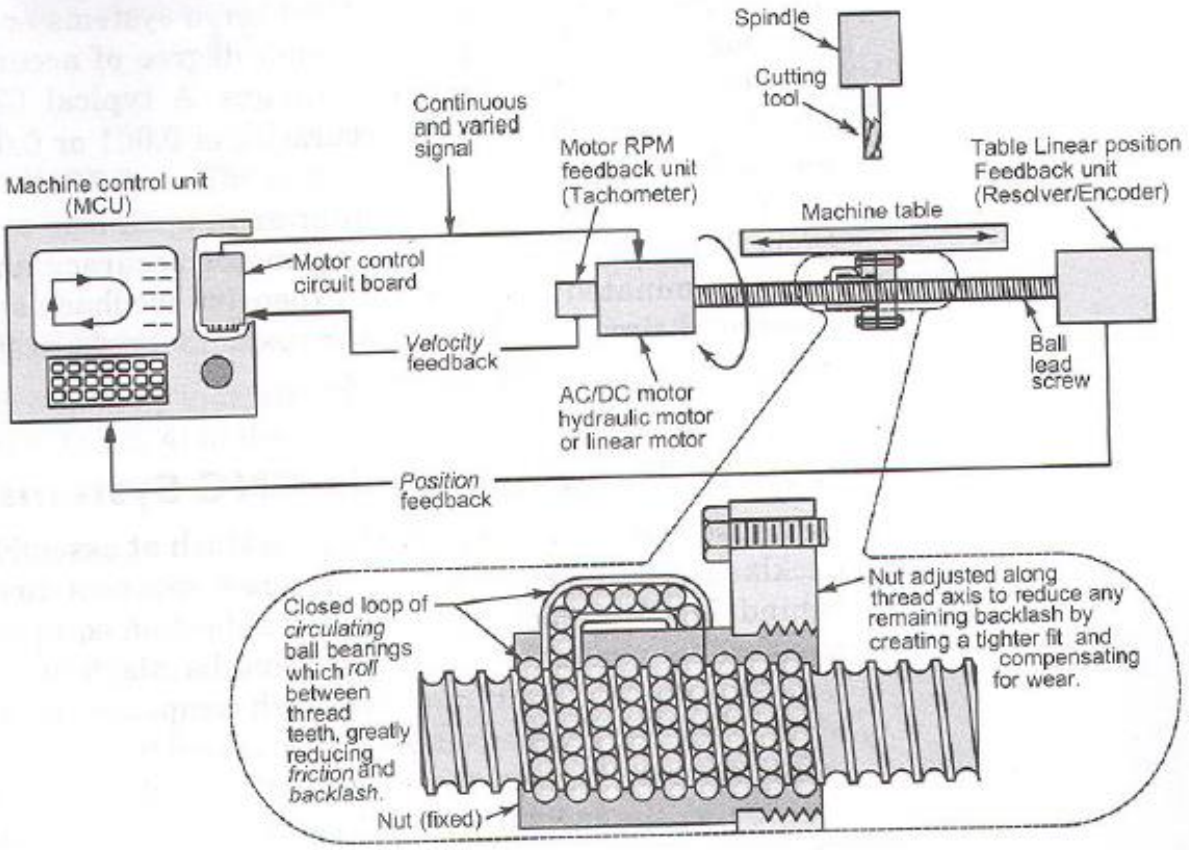
Open Loop Systems



Example of a machine with open loop using Conventional acme lead screw

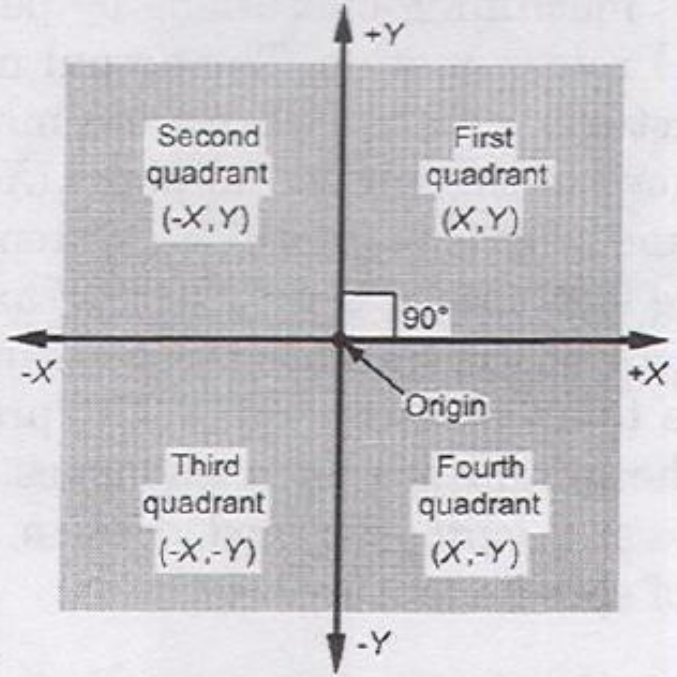
Loop systems for controlling tool movement

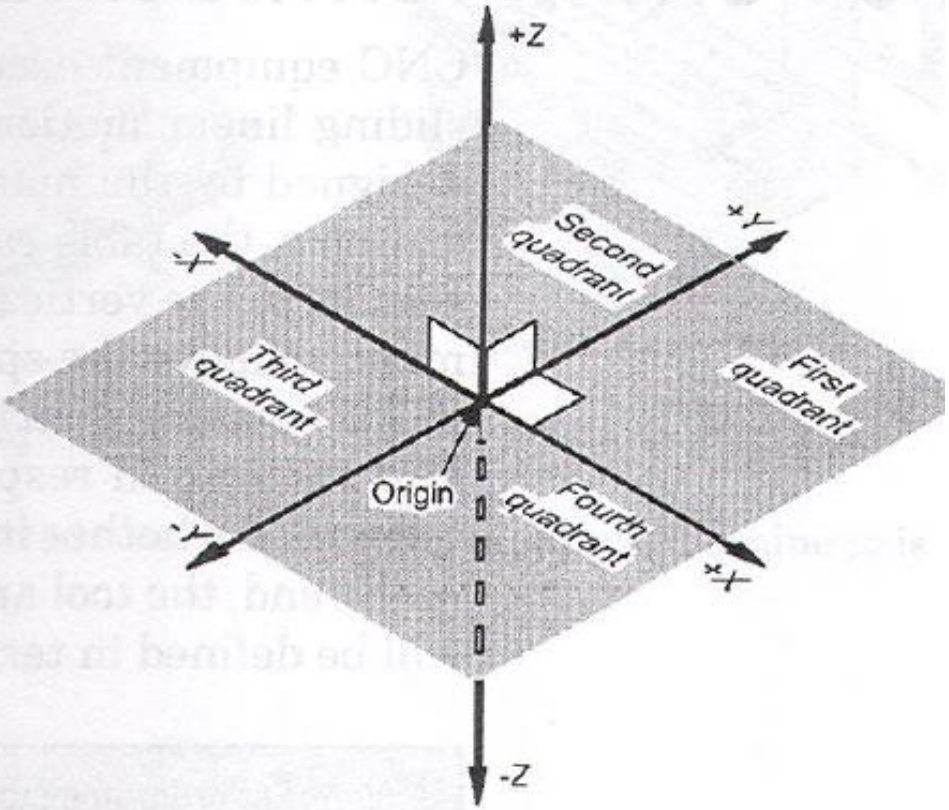
closed Loop Systems



Example of closed system using ball lead screw

Cartesian coordinate system





Machine axes

- Machine axes Milling machines and machining centers have at least three axes X, Y and Z as shown in Figure 7.1. The machines become more flexible if they have the fourth axis, usually an indexing or a rotary axis (the A axis for vertical models or the B axis for horizontal models). Even higher level of flexibility can be found on machines with five or more axes

Machine axes

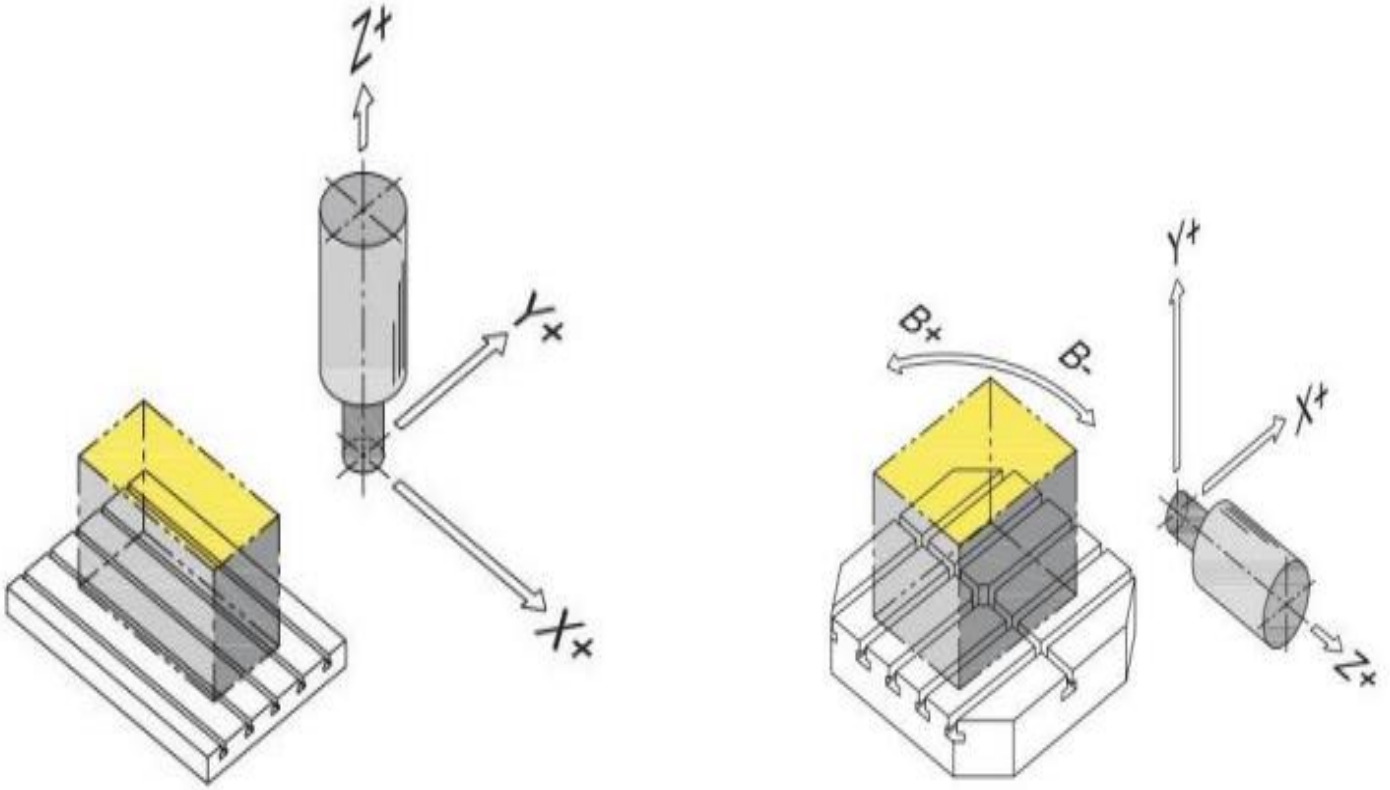


Figure 7.1: Schematic illustration of (a) CNC vertical machining center and (b) CNC horizontal machining center

Machine axes

A typical 3-axis machine uses three controlled axes of motion. They are defined as the X axis, the Y axis, and the Z axis as shown in Figure

- The X axis is parallel to the longest dimension of the machine table
- The Y axis is parallel to the shortest dimension of the table
- The Z axis is the spindle movement

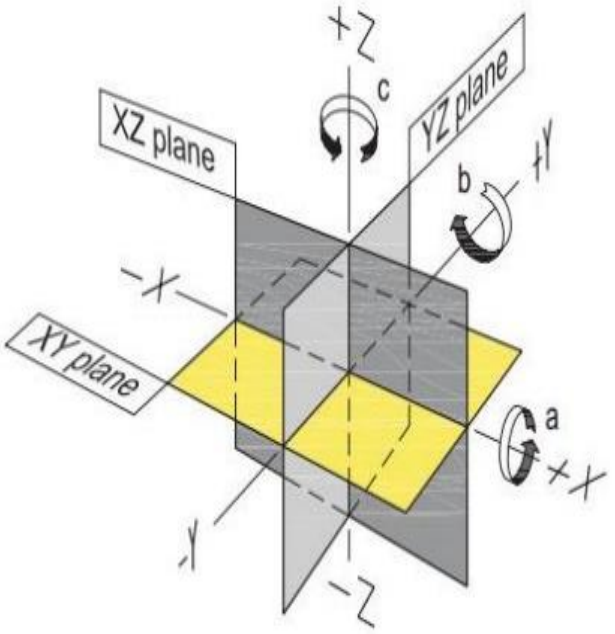


Figure 7.2: Standard orientation of planes and CNC machine tool axes

Machine axes

Additional Axes

- A CNC machine of any type can be designed with one or more additional axes, normally designated as the secondary axes using the U, V and W letters. These axes are normally parallel to the primary X, Y and Z axes respectively.
- For a rotary or an indexing applications, the additional axes are defined as A, B and C axes, as being rotated about the X, Y and Z axes, again in their respective order. Positive direction of a rotary (or an indexing) axis is the direction required to advance a right handed screw in the positive X, Y or Z axis.
- Arc center modifiers (sometimes called the arc center vectors) are not true axes, yet they are also related to the primary axes XYZ.

Machine axes

- A typical CNC lathe is designed with two standard axes one axis is the X axis, the other axis is the Z axis. Both axes are perpendicular to each other and represent the typical two-axis lathe motions.

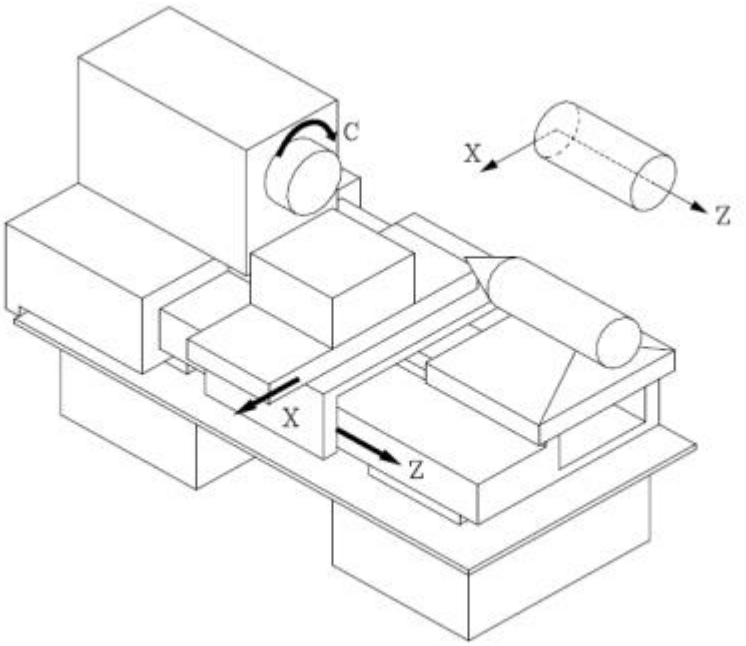
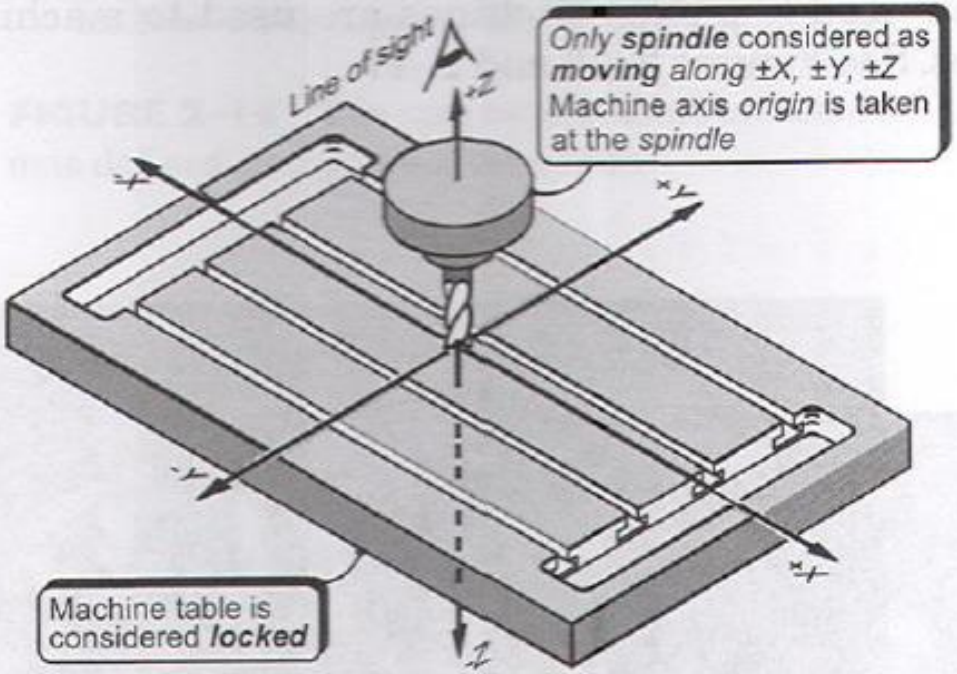
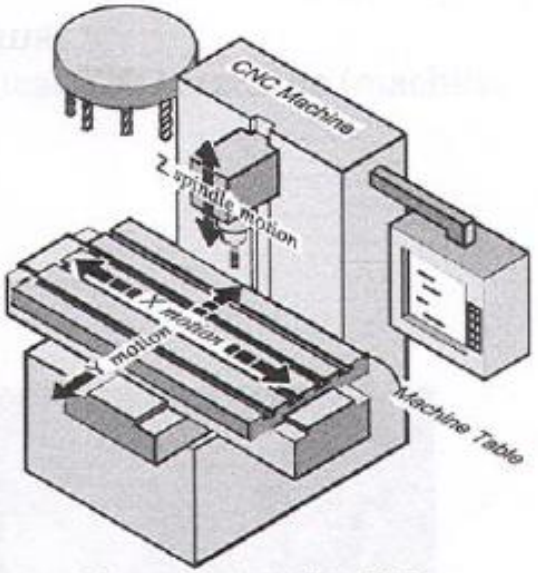


Figure 7.3: Illustration of the two axis CNC lathe machine

CNC machines axis of Motion

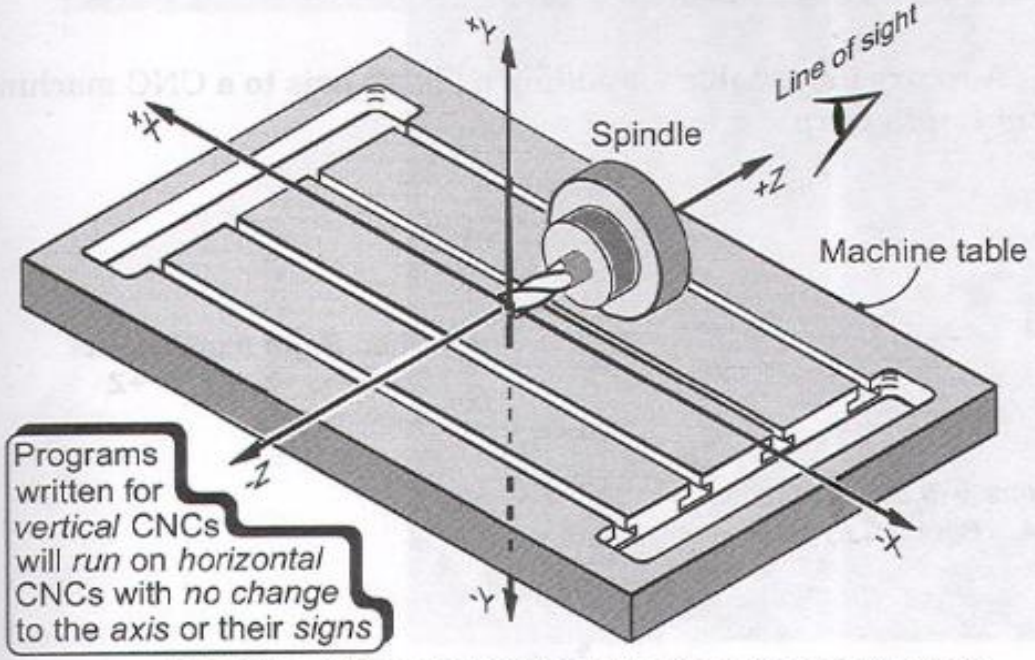


Machine Axis as Used for ***Programming*** Purposes

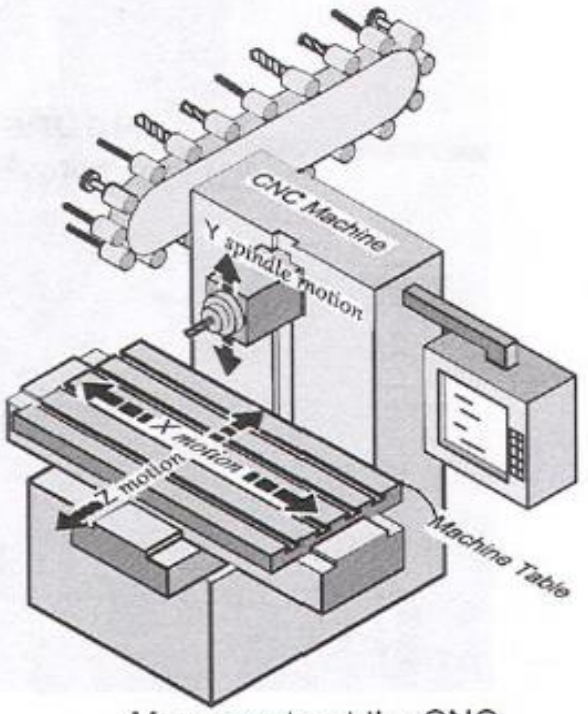


Movements at the CNC

Machine axis for a three-axis vertical CNC machine (machine axes defined as spindle movement



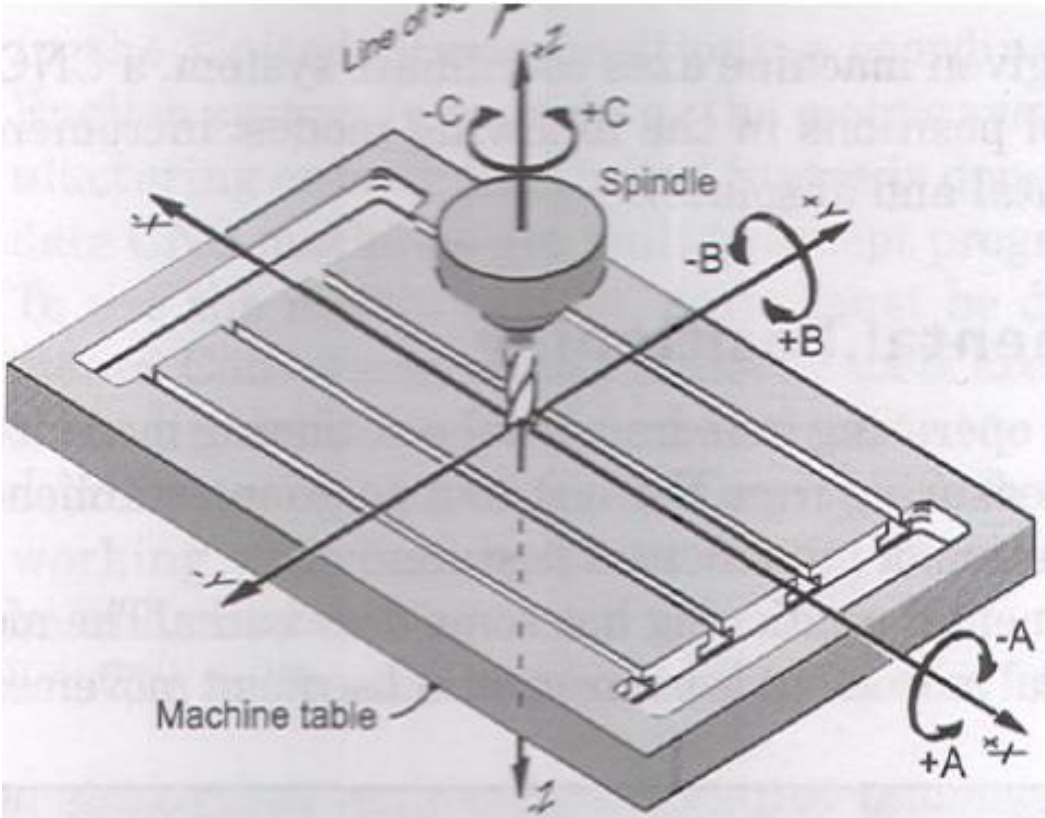
Machine Axis as Used for ***Programming*** Purposes



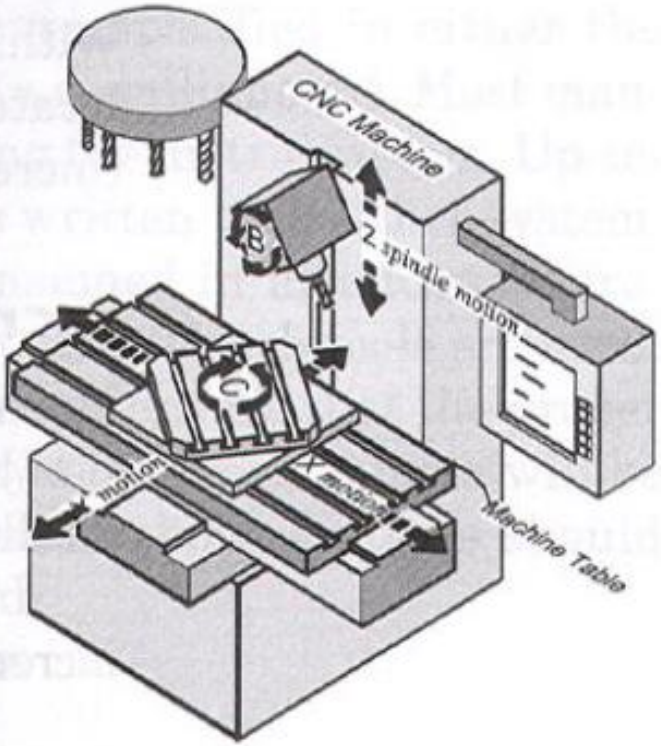
Movements at the CNC

Machine axis for a three-axis vertical CNC machine (machine axes defined as spindle movement

CNC machines axis of Motion

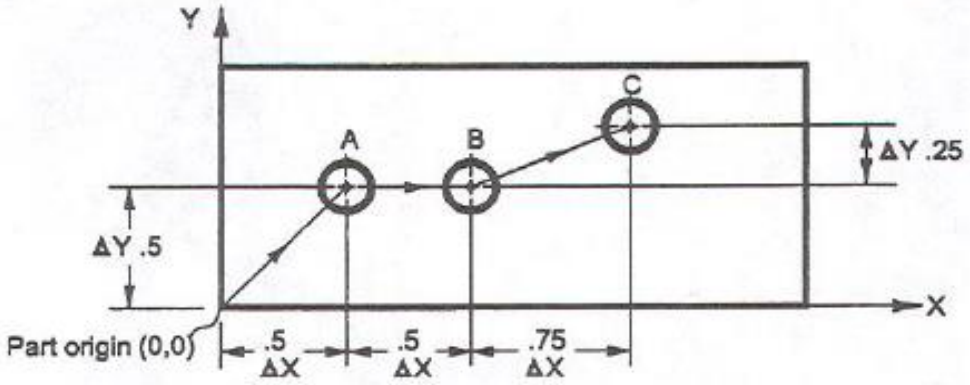


Machine Axis as Used for ***Programming*** Purposes



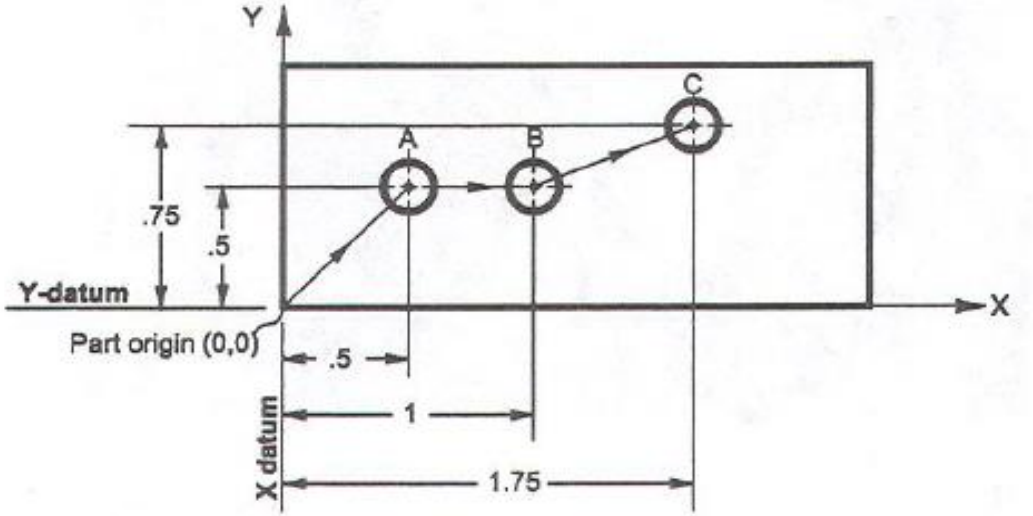
Movements at the CNC
(5-Axis Vertical Spindle CNC)

Incremental and Absolute positioning



Tool position
A
B
C

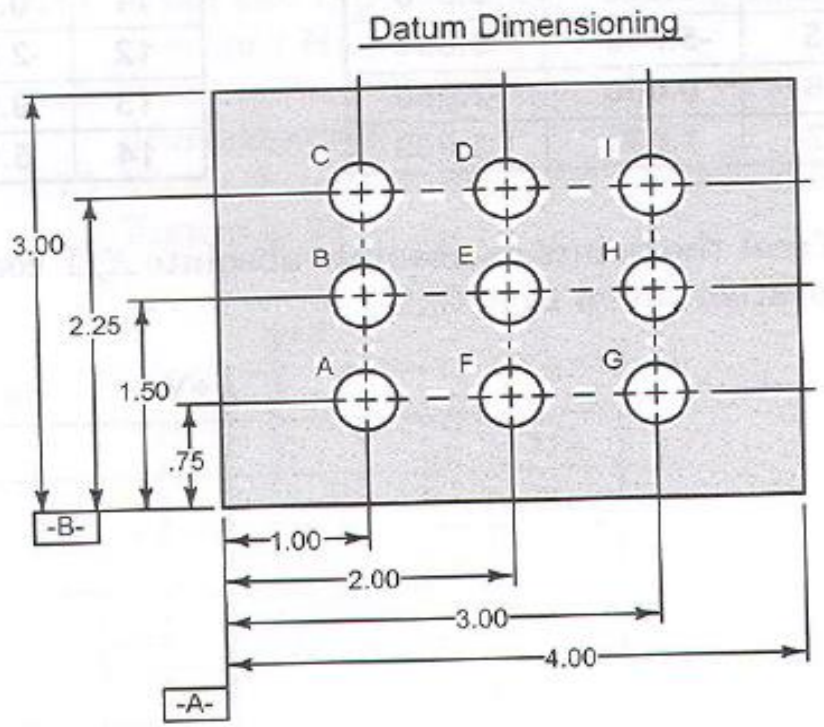
Datum positioning for incremental positioning



Tool position
A
B
C

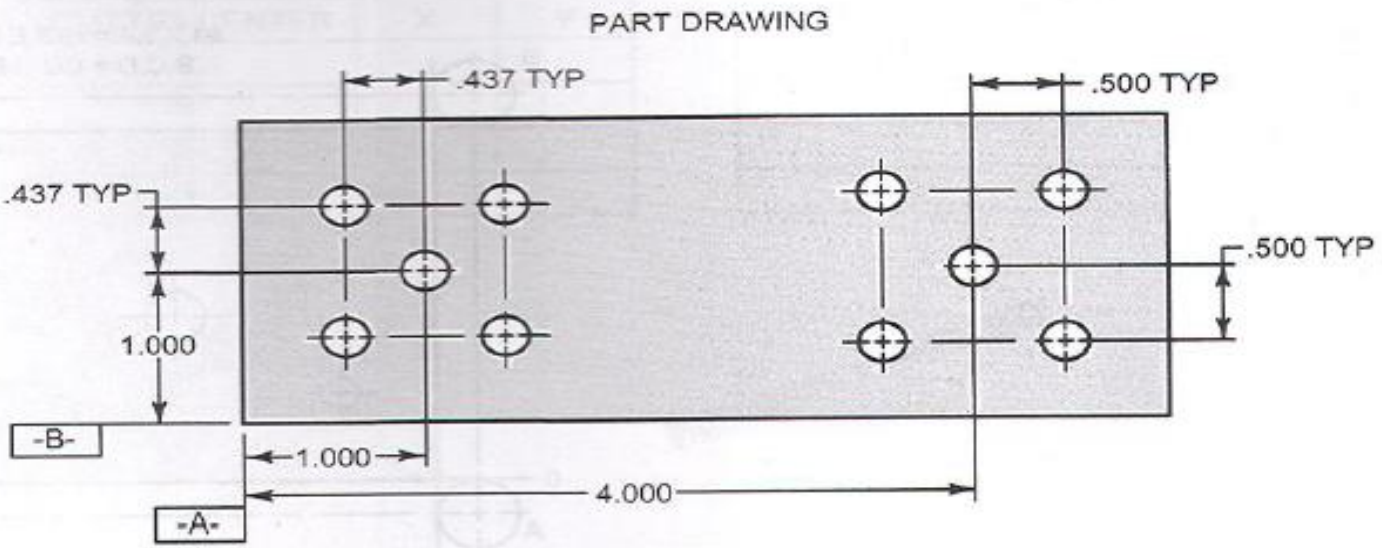
Datum positioning for absolute positioning

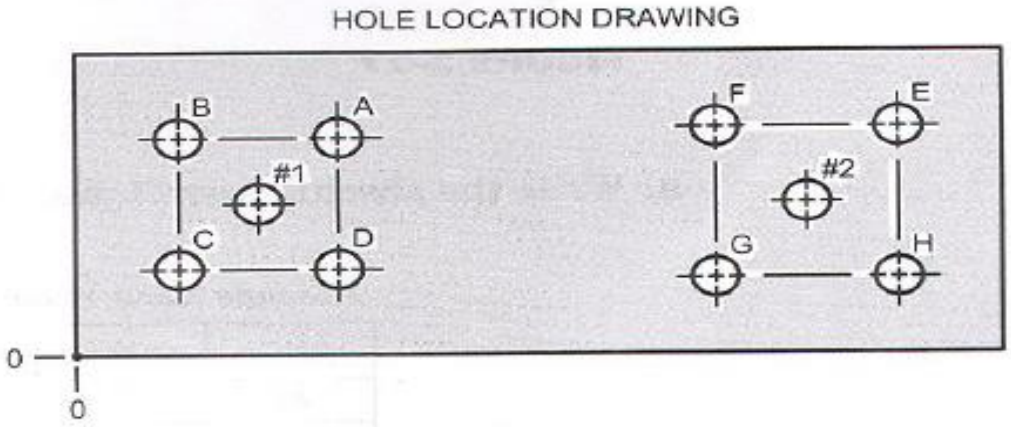
Incremental and Absolute positioning



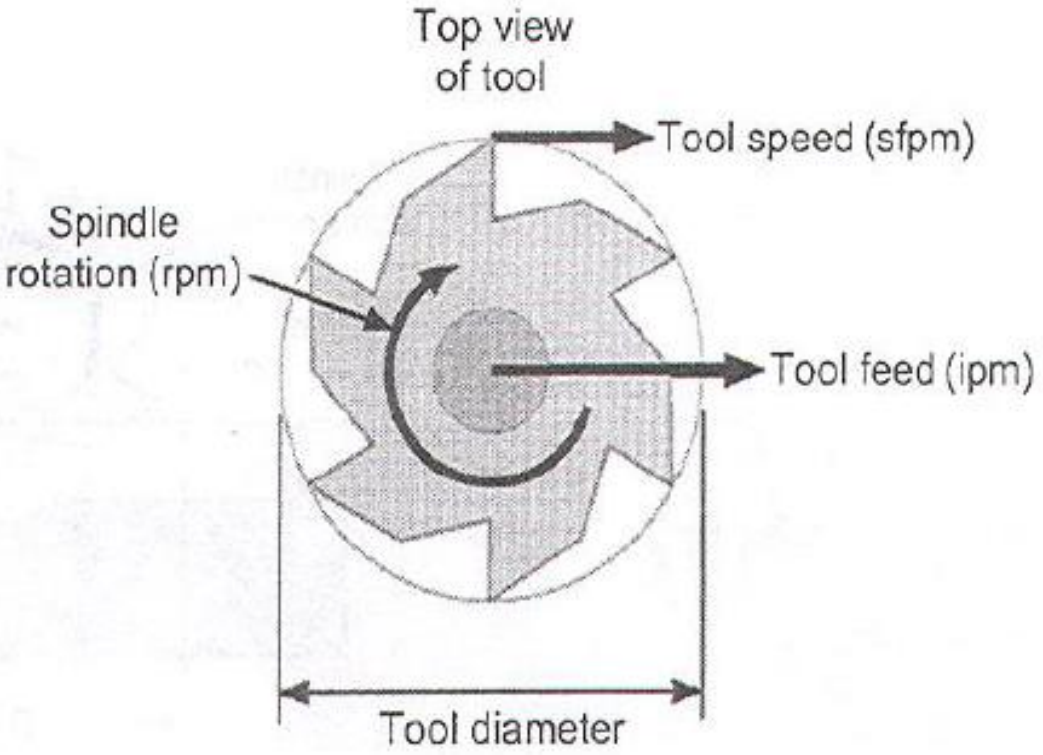
HOLE	X	Y
A		
B		
C		
D		
E		
F		
G		
H		
I		

Incremental and Absolute positioning



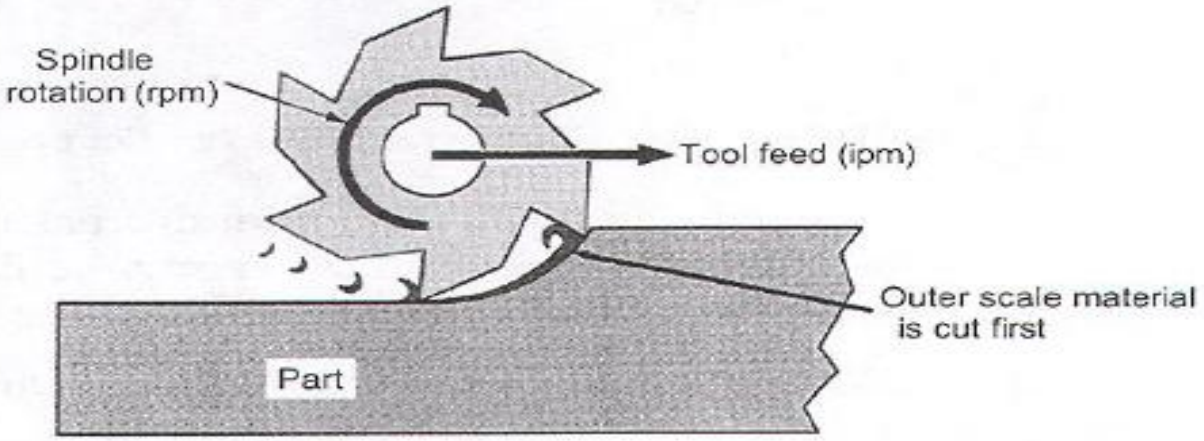


Tool speeds and feeds for milling

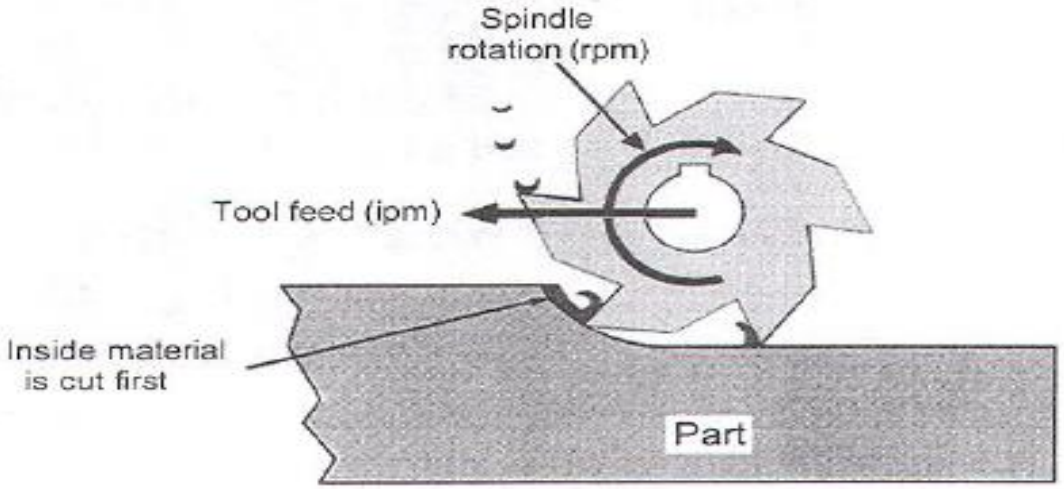


Tool feed.

Climb and conventional Milling



Climb milling.



Conventional milling.

Typical programming procedure

- The following items form a fairly common and logical sequence of tasks done in CNC programming. The items are only in a suggested order, offered for further evaluation. This order may be changed to reflect special conditions or working habits. Some items may be missing or redundant:

1. Study of initial information (drawing and methods)
2. Material stock (blank) evaluation
3. Machine tool specifications
4. Control system features
5. Sequence of machining operations

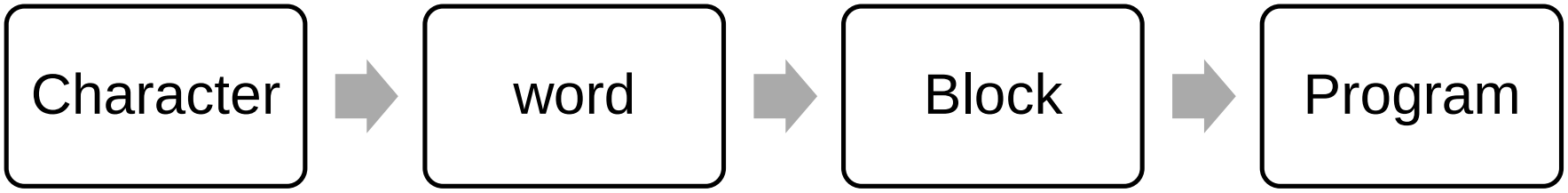
Typical programming procedure

Cont..

6. Tooling selection and arrangement of cutting tools
7. Setup of the part
8. Technological data (speeds, feed rates, etc.)
9. Determination of the tool path
10. Working sketches and mathematical calculations
11. Program writing and preparation for transfer to CNC
    Program testing and debugging
12. Program documentation

Basic programming terms

- Four basic terms used in CNC programming :



Character

- Character: smallest unit in CNC programming, it can be one of the three forms combined to make a meaningful word. The combination of characters is called alpha-numerical program input
  - Digit: ten digits, from 0-9 (not location of x,y,z)
  - Letter: capital letter (A, B, C, …,Z)
  - Symbol: decimal point, minus, and positive signs, percent sign, parenthesis

Word

- A combination of alpha-numerical characters creating a single instruction to the control system
- Usually each word starts with a capital letter followed by a number
- Examples: program number, axis position, feed rate, speed, miscellaneous functions.

Block

- Block is used as a multiple instruction
- Consists of individual lines of instructions sequenced in a logical order
- Each block is composed of one or several words and each word is composed of two or more characters

A block programmed in ISO language can consist of:

- Sequence block (N)
- Preparatory (G) functions
- Miscellaneous functions (M)
- Axis coordinates (X...C)
- Spindle speed (S)
- Feed rate (F)
- Tool number (T)

This order should be maintained within each block, although it is not necessary for every block to contain the information

CNC Program

- Begins with a program number
- Followed by blocks of instructions
- Program ends with stop code

The system of programming CNC machines is called “ word address format” which is based on a combination of a one letter and one or more digit

CNC Program

- A CNC program usually begins with a program number or similar identification, followed by the blocks of instructions in a logical order.

- The program ends with a stop code or a program termination symbol, such as the percent sign “%”.

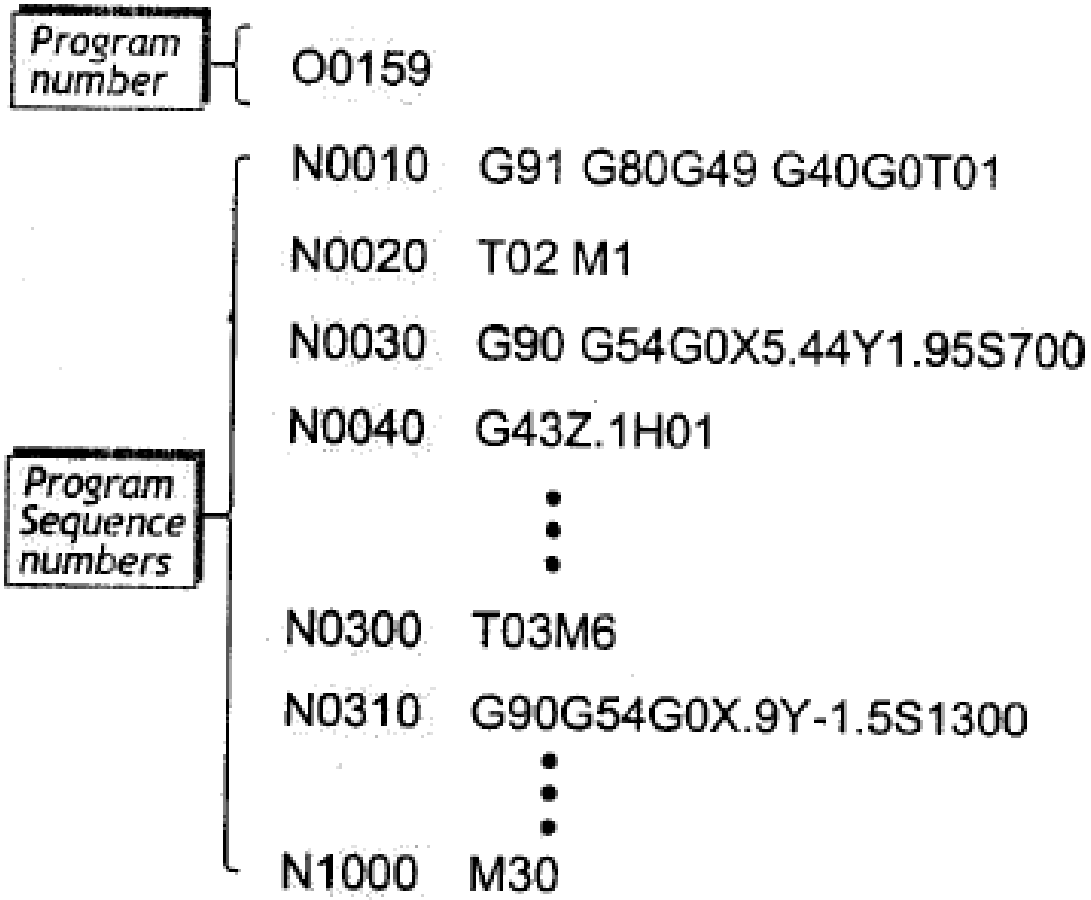
- The series of blocks arranged in a logical order that is required to machine a complete part or a complete operation is the part program also known as a CNC program

Program words

Program number (O): the machine recognizes programs according to a numeric code, and most machines can store several different programs, program numbers range from O 01 to O 09999

Sequence Numbers (N): a sequence of number is an optional tag that can be coded at the beginning of a block if need, the computer will execute the program blocks in the order in which they appear regardless of the sequence number

The are only used to enable the operator locate the specific line of a program when entering or revising a program (checking)
Number range from N 01 to N 9999


Program number
O0159
Program Sequence numbers
N0010 G91 G80G49 G40G0T01
N0020 T02 M1
N0030 G90 G54G0X5.44Y1.95S700
N0040 G43Z.1H01
N0300 T03M6
N0310 G90G54G0X.9Y-1.5S1300
N1000 M30

Symbols in programming

- Symbols in programming In addition to the basic symbols, Fanuc can accept other symbols for different applications. Table 8.1 describes all symbols available on the Fanuc controls.

Table 7.1: Symbols meanings on the Fanuc controllers

Symbol	Description	Comment
.	Decimal point	Fractional part of a number
+	Plus sign	Positive value or *addition* sign in Fanuc macros
-	Minus sign	Negative value or *subtraction* sign in Fanuc macros
*	Multiplication sign	*Multiplication* sign in Fanuc macros
/	Slash (front slash)	Block skip function symbol or *division* sign in Fanuc macros
( )	Parenthesis	Program comments & messages
%	Percent sign	Stop code (end of program file)
:	Colon	Program number designation (rare)
,	Comma	Used only within comments
[ ]	Brackets	Calculations in Fanuc macros
;	Semicolon	Non programmable End-Of-Block symbol (screen display only)
#	Sharp sign	Variable definition or call in Fanuc macros
=	Equal sign	Equality symbol in Fanuc macros

Preparatory command (G-code)

- The program address G identifies a preparatory command, often called the G code.
- This address has one and only objective that is to preset or to prepare the control system to a certain desired condition, or to a certain mode or a state of operation.
- The term preparatory command indicates its meaning. A G code will prepare the control to accept the programming instructions in a specific way.

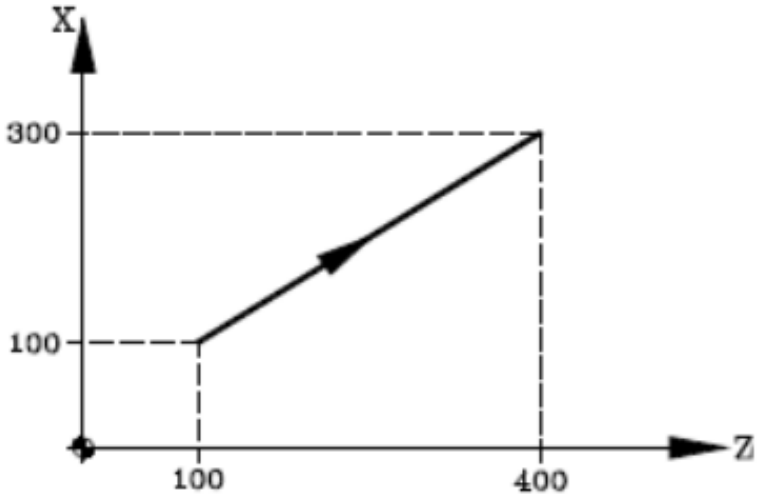
Preparatory command (G-code)

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 -9	Fixed cycles
G19	Y-Z Plane	G90	Absolute dimension programming
		G91	Incremental deimension programming

Preparatory command (G-code)

- Rapid traverse (G00) The movements programmed after G00 are executed at the rapid feedrate indicated in the axis machine parameter "G00FEED".

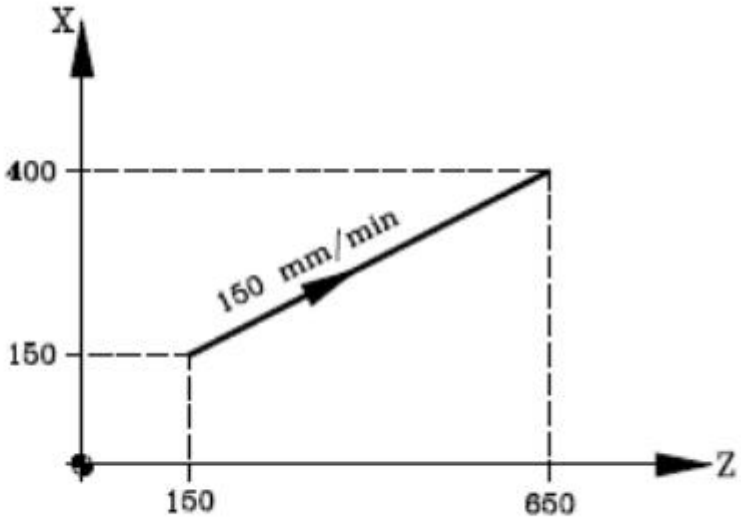
Independently of the number of axis which move, the resulting path is always a straight line between the starting point and the final point.




```
X100 Z100                ; Starting point
G00 G90 X300 Z400        ; Programmed tool Path
```

Preparatory command (G-code)

- Linear interpolation (G01) The movements programmed after G01 are executed according to a straight line and at the programmed feedrate "F"



G01 G90 X800 Z650 F150

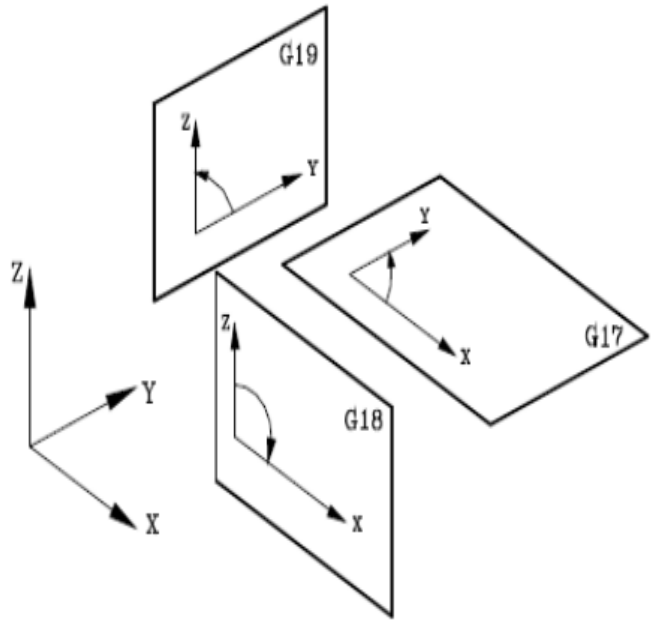
Plane selection (G16, G17, G18, G19)

G16 Polar coordinate system

G17 Selects the XY plane

G18 Selects the ZX plane

G19 Selects the YZ plane



Part dimensioning. Millimeters (G21) or inches (G20)

G20 Programming in inches

G21 Programming in millimeters

Always use only one unit of dimensioning in a part program and never mix Metric and English units in the same program

NC WORDS (continue)

X, Y, Z, A, B, C Codes. coordinate positions of the tool.

The coordinates may be specified in decimal number (Decimal Programming), or integer number (Basic Length Unit (BLU Programming).

Example:

move from location (2,3,-6) will look like in the word address program like this:

X 2.0 Y 3.0 Z -6.0

Absolute/incremental programming (G90, G91)

G90 Absolute coordinates

G91 Incremental coordinates

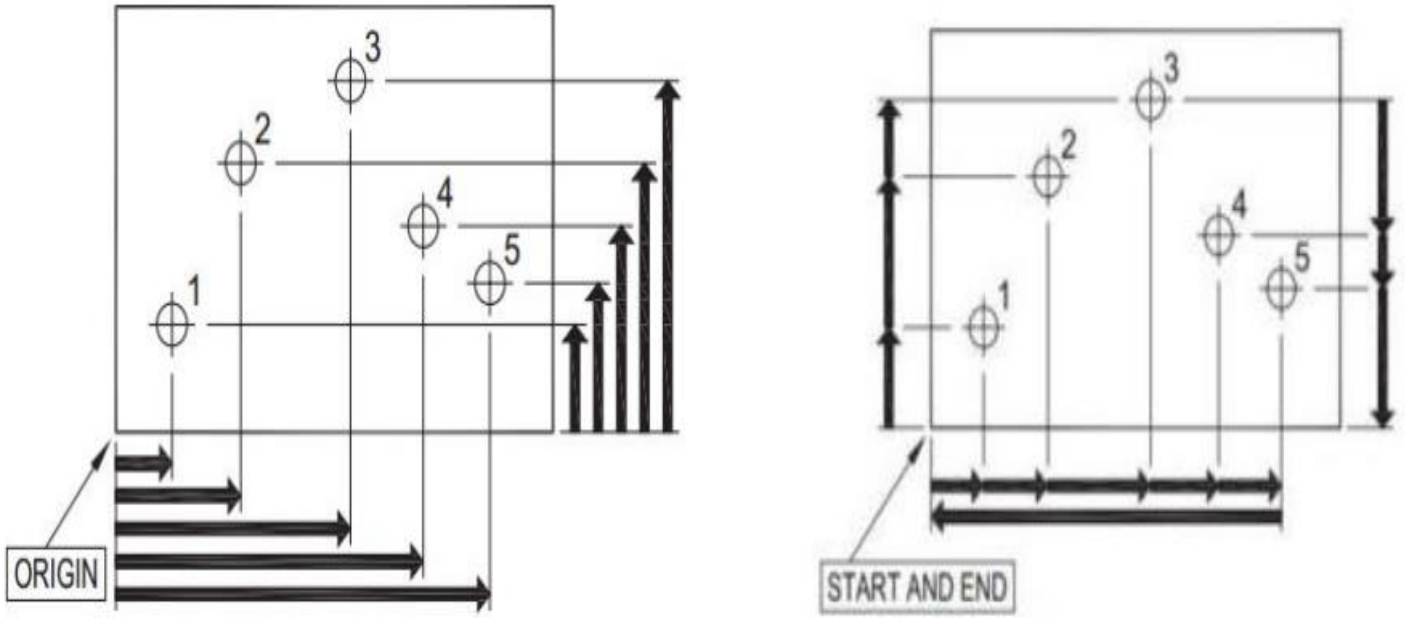


Figure 8.6: (a) Absolute dimensioning - measured from part origin G90 command will be used in the program and (b) Incremental dimensioning - measured from the current tool location G91 command will be used in the program

FEED RATE **(F CODE)**

The feed rate is the rate at which the cutting tool moves along a programming axis, and is specified by the numerical value following the address F. In the English system, the feed rate is expressed in inches per minute, and in the metric system in millimeters per minute.

Feed rate "F" The machining feed rate can be selected from the program. It remains active until another feed rate is programmed. It is represented by the letter F and depending on whether it is working in G94 or G95, it is programmed in mm/minute (inches/minute) or in mm/revolution (inches/revolution).

The programmed feed rate F is effective working in linear (G01) or circular (G02, G03) interpolation.
If function F is not programmed, the CNC assumes the feed rate to be F0.

FEED RATE **(F CODE)**

- When working in rapid travel (G00), the machine will move at the rapid feed rate indicated by the axis machine parameter "G00FEED", apart from the F programmed.

- Feed rate in mm/min or inches/min (G94) From the moment the code G94 is programmed, the control takes that the feed rates programmed through F5.5 are in mm/min or inches/mm. Feed rate in mm/rev or inches/rev (G95) From the moment when the code G95 is programmed, the control assumes that the feed rates programmed through F5.5 are in mm/rev or inches/mm

- **SPINDLE SPEED (S CODE)**

- The address S controls the speed at which the spindle rotates in rpm. A numerical value, up to four digits maximum, is entered following the address S. No decimal point is allowed with the numerical value.

- The turning speed of the spindle is programmed directly in rpm via code S5.4 (G97) or in m/min (ft/min when working in inches) when in constant surface speed mode (G96).

TOOL FUNCTION (T CODE)

- Tool number (T) and tool offset (D)
- Designates the tool number to be used
- With the "T" function, it is possible to select the tool and with the "D" function it is possible to select the offset associated with it. When defining both parameters, the programming order is T D.
- For example: T6 D17

Miscellaneous functions

- The address M in a CNC program identifies a miscellaneous function,
- Also called machine functions
- Programmers need to activate certain aspects of the machine operations or controlling the program flow
- They are very important for the program to be effective

M code	Description
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset, no rewind)
M03	Spindle rotation normal
M04	Spindle rotation reverse
M05	Spindle stop
M06	Automatic tool change (ATC)
M07	Coolant mist ON
M08	Coolant ON (coolant pump motor ON)
M09	Coolant OFF (coolant pump motor OFF)
M19	Spindle orientation
M30	Program end (always with reset and rewind)
M48	Feedrate override cancel OFF *(deactivated)*
M49	Feedrate override cancel ON *(activated)*
M60	Automatic pallet change (APC)
M78	B axis clamp *(nonstandard)*
M79	B axis unclamp *(nonstandard)*
M98	Subprogram call
M99	Subprogram end

Sequence of words to make a full block

General Syntax

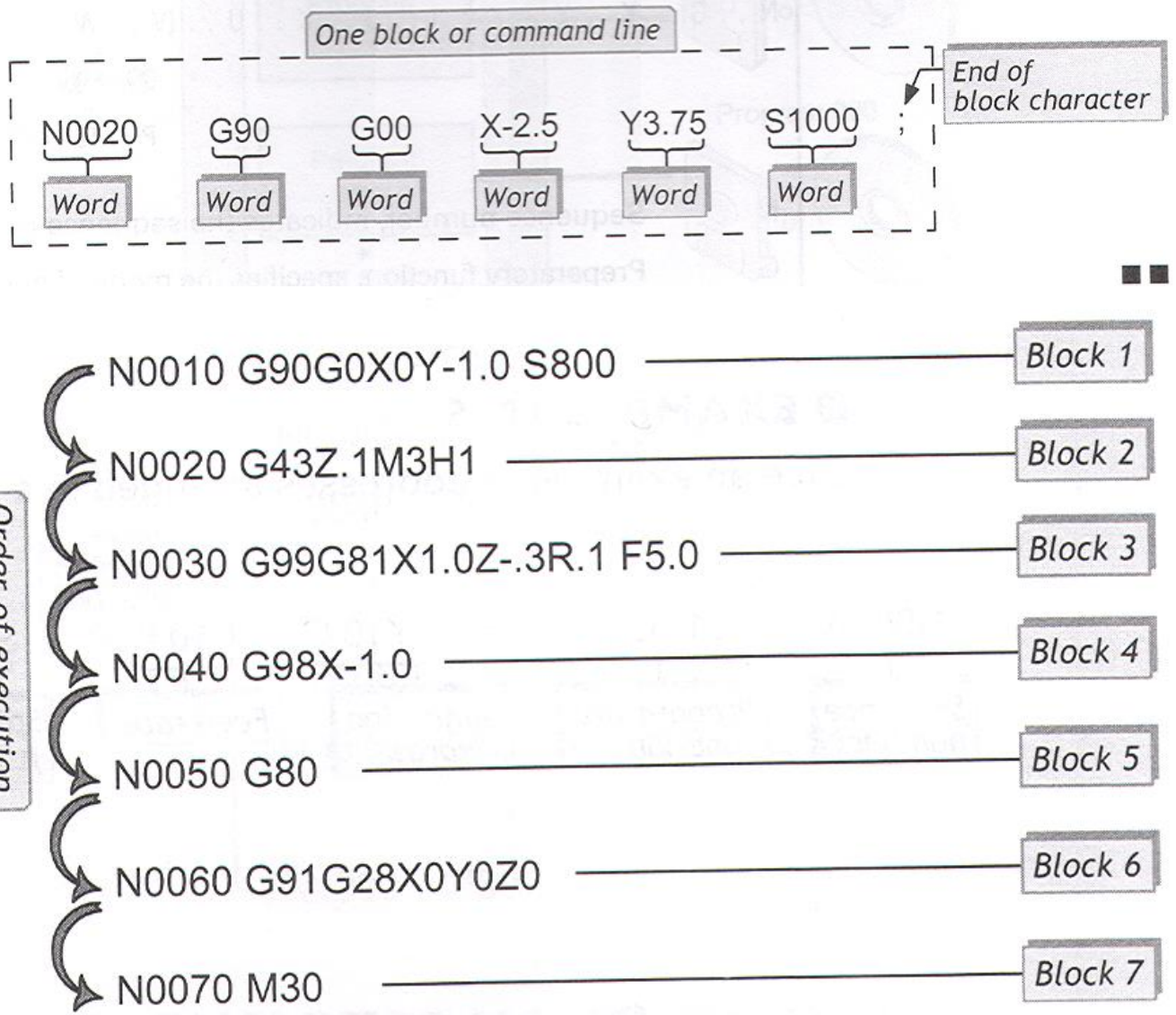
①→②→③→④→⑤→⑥→⑦→⑧→⑨→⑩→⑪→⑫→⑬→⑭

N . . G . . X . . Y . . Z . . I . . J . . K . . U . . (V . . W . . A . . B . . C)

⑮→⑯→⑰→⑱→⑲→⑳→㉑→㉒

P . . Q . . R . . F . . S . . T . . M . . H

N	Sequence number, indicates the sequence number of the block.
G	Preparatory function, specifies the mode of operation in which a command is to be executed.
X, Y, Z	Dimension words, designate the amounts of axis movements.
I, J, K	
U, V, W	
A, B, C	
P, Q, R	
F	Feed rate, designates the relative speed of the cutting tools with respect to the work.
S	Spindle function, designates the spindle speed in revolutions per minute (rpm).
T	Tool function, designates the number of the tools to be used.
M	Miscellaneous function, designates a machine function such as spindle on/off or coolant on/off.
H, D	Auxiliary input function, specifies tool length offset number, number of repetitions of a fixed cycle, and so on.


One block or command line
End of block character
N0020
G90
G00
X-2.5
Y3.75
S1000
;
Word
Word
Word
Word
Word
Word
Order of execution
N0010 G90G0X0Y-1.0 S800
N0020 G43Z.1M3H1
N0030 G99G81X1.0Z-.3R.1 F5.0
N0040 G98X-1.0
N0050 G80
N0060 G91G28X0Y0Z0
N0070 M30
Block 1
Block 2
Block 3
Block 4
Block 5
Block 6
Block 7

BASIC REQUIREMENT OF NC MACHINE CONTROL

1. Preparatory functions: which unit, which interpolator, absolute or incremental programming, which circular interpolation plane, cutter compensation, etc.
2. Coordinates: three translational, and three rotational axes.
3. Machining parameters: feed, and speed.
4. Tool control: tool diameter, next tool number, tool change.
5. Cycle functions: drill cycle, ream cycle, bore cycle, mill cycle, clearance plane.
6. Miscellaneous control: spindle on/off, tape rewind, spindle rotation direction, pallet change, clamps control, Coolant control, etc.
7. Interpolators: linear, circular interpolation