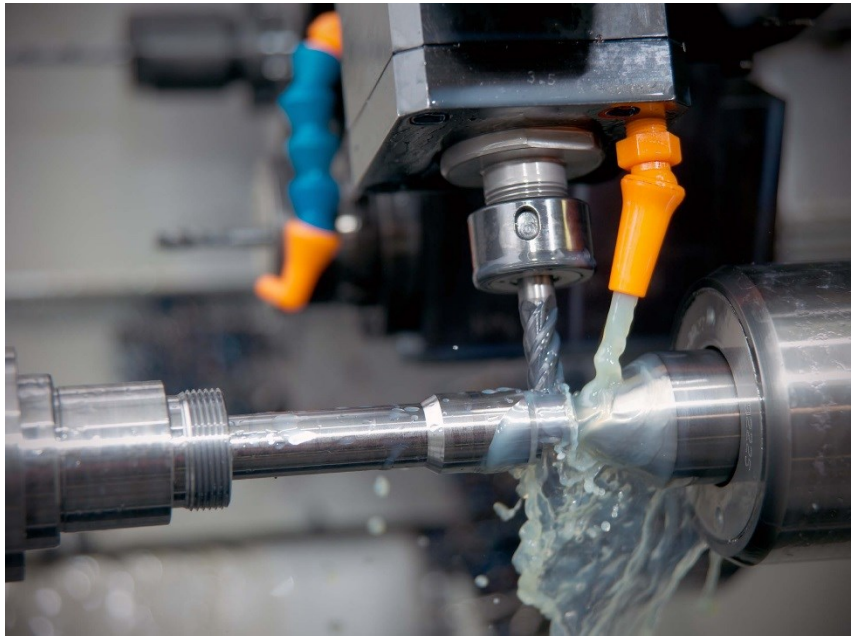
Computer Numerical Control

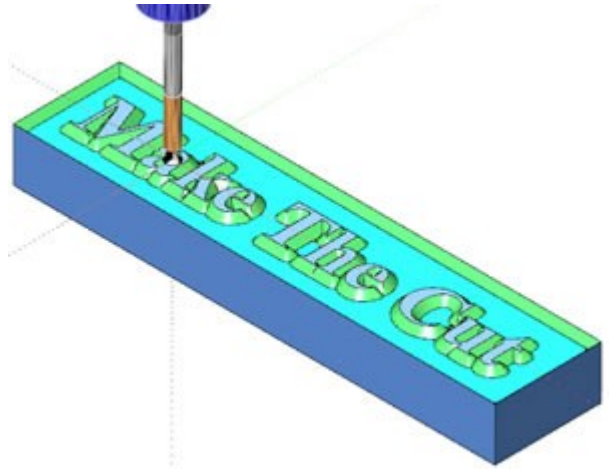
Chapter 8

Table of Content

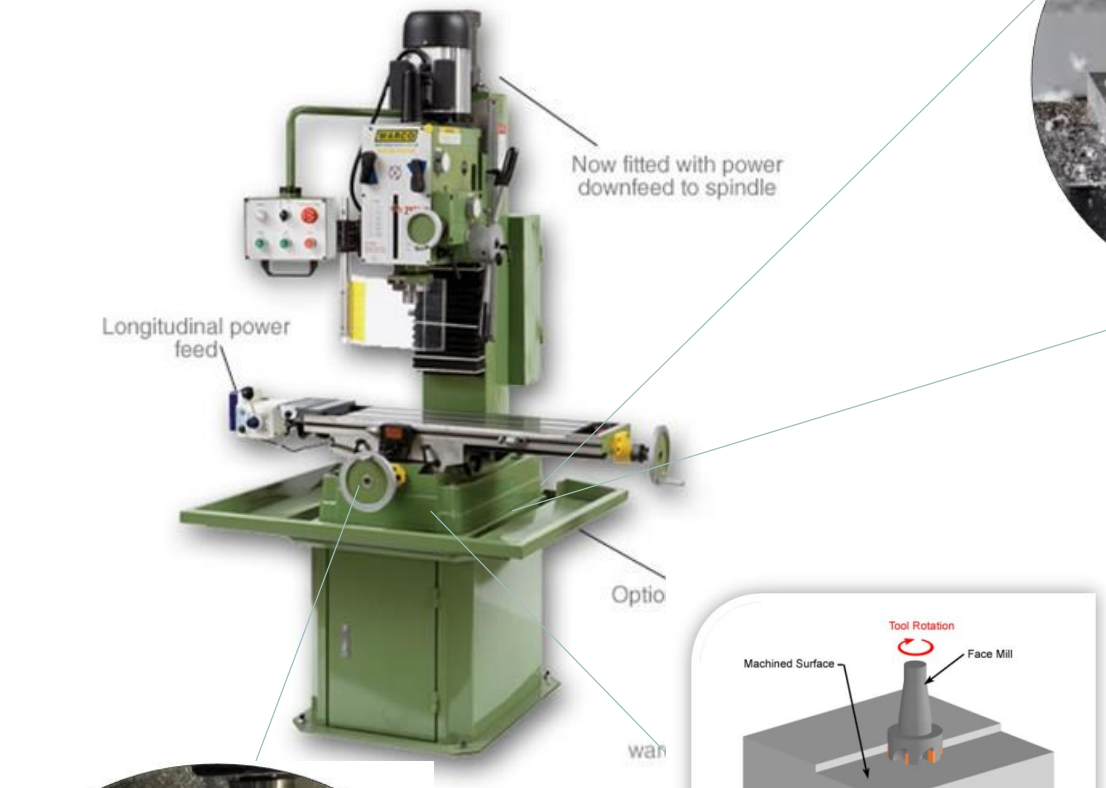
Introduction

- A machine tool is a machine for shaping or machining metal or other rigid materials, usually by cutting, boring, grinding, shearing, or other forms of deformation.
- Toolpath: The path through space that the tip of a tool follows on its way to producing the desired workpiece.
- Machine tools employ some sort of tool that does the cutting or shaping. All machine tools have some means of constraining the workpiece and provide a guided movement of the parts of the machine.
- Examples of machine tools:
  - Milling Machine
  - Lathe Machine
  - Drilling Machine
  - Grinding Machine

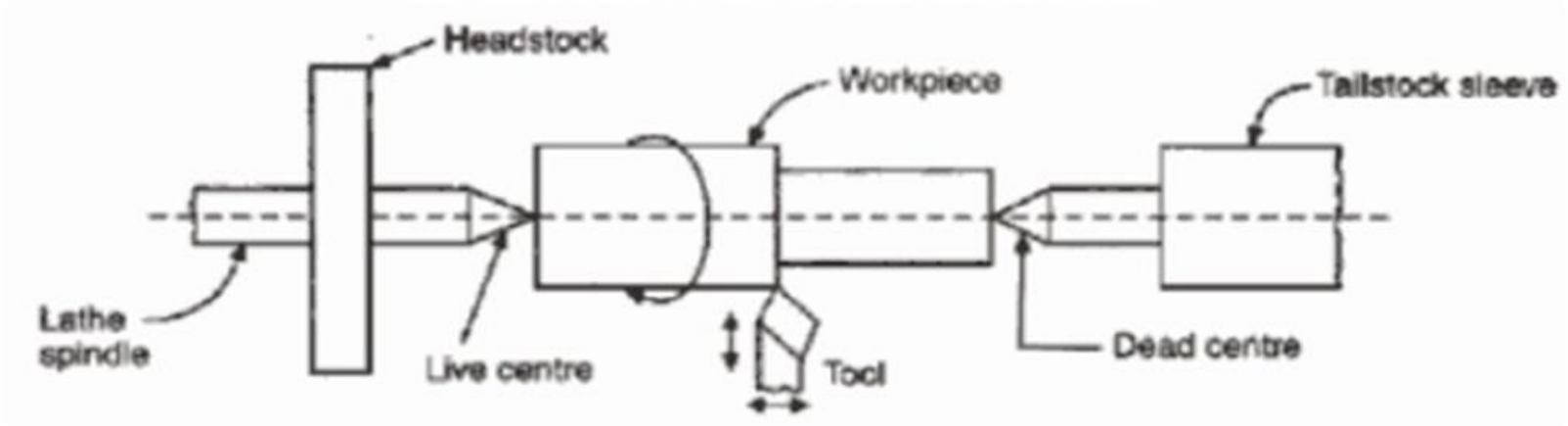




Milling Machine



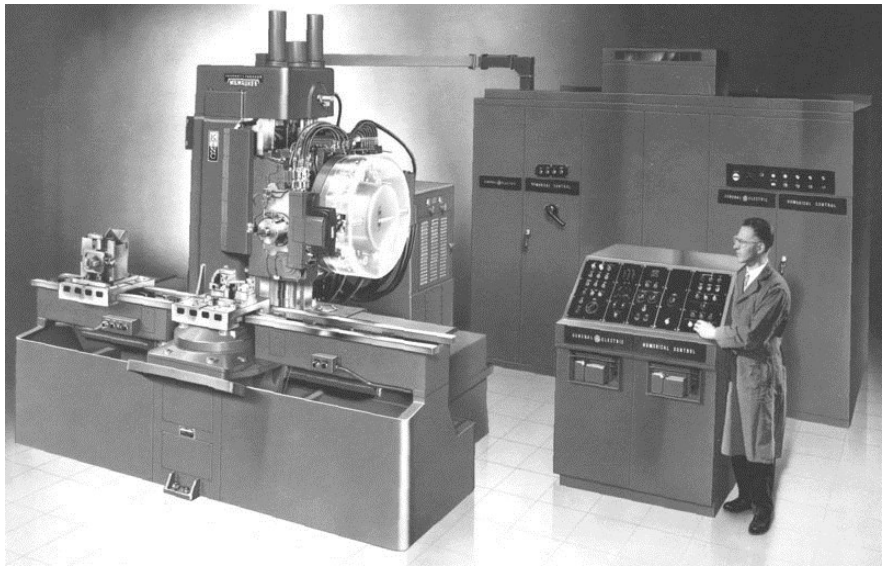
Lathe Machine

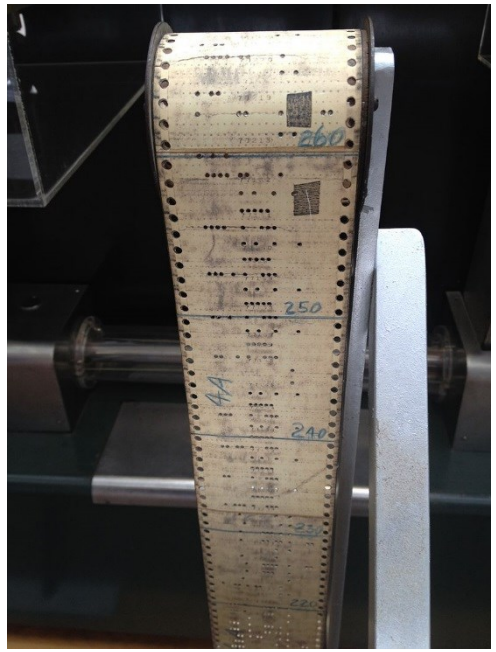


Grinding and Drilling Machines

History

- 15th century - machining metal.
- 18th century - industrialization, production-type machine tools.
- 1955 - John Parsons and US Air Force define a need to develop a machine tool capable of machining complex and close tolerance aircraft parts with the same quality time after time (repeatability). MIT is the subcontractor and builds the machine for the project.
- 1959 - MIT announces Automatic Programmed Tools (APT) programming language
- 1960 - Direct Numerical Control (DNC). This eliminates paper tape punch programs and allows programmers to send files directly to machine tools.
- 1970's - CNC machine tools & Distributed Numerical Control
- 1980's - Graphics based CAM systems introduced.





NC Overview

A numerical control, or “NC”, system controls many machine functions and movements which were traditionally performed by skilled operators.

Numerical control developed out of the need to meet the requirements of high production rates, uniformity and consistent part quality.

Programmed instructions are converted into output signals which in turn control machine operations such as spindle speeds, tool selection, tool movement, and cutting fluid flow.

By integrating a computer processor, computer numerical control, or “CNC” as it is now known, allows part machining programs to be edited and stored in the computer memory as well as permitting diagnostics and quality control functions during the actual machining.

Part program is a series of coded instructions required to produce a part, controls the movement of the machine tool and on/off control of auxiliary functions such as spindle rotation and coolant. The coded instructions are composed of letters, numbers and symbols.

All CNC machining begins with a part program, which is a sequential instructions or coded commands that direct the specific machine functions. The part program may be manually generated or, more commonly, generated by computer aided part programming systems

NC MACHINES Specifications

Accuracy

Spindle and axis motor horsepower

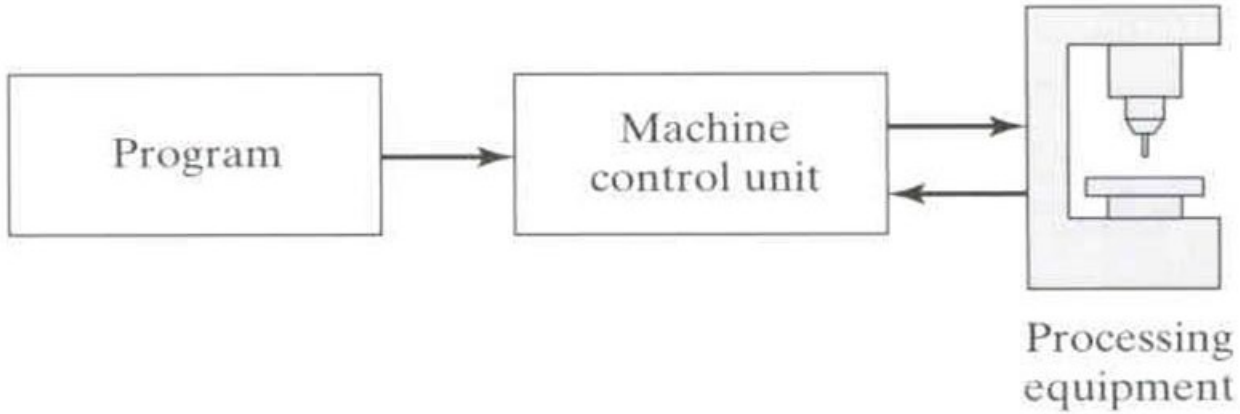
Number of controlled axes

Dimension of workspace

Number of tool changers

Features of the machine and the controller.

- On-machine programming
- Data communication
- Graphical interface



NC Advantages and Disadvantages

ADVANTAGES

- Easier to program
- Easy storage of existing programs
- Easy to change a program
- Avoids human errors
- NC machines are safer to operate
- Complex geometry is produced as cheaply as simple ones
- Usually generates closer tolerances than manual machines

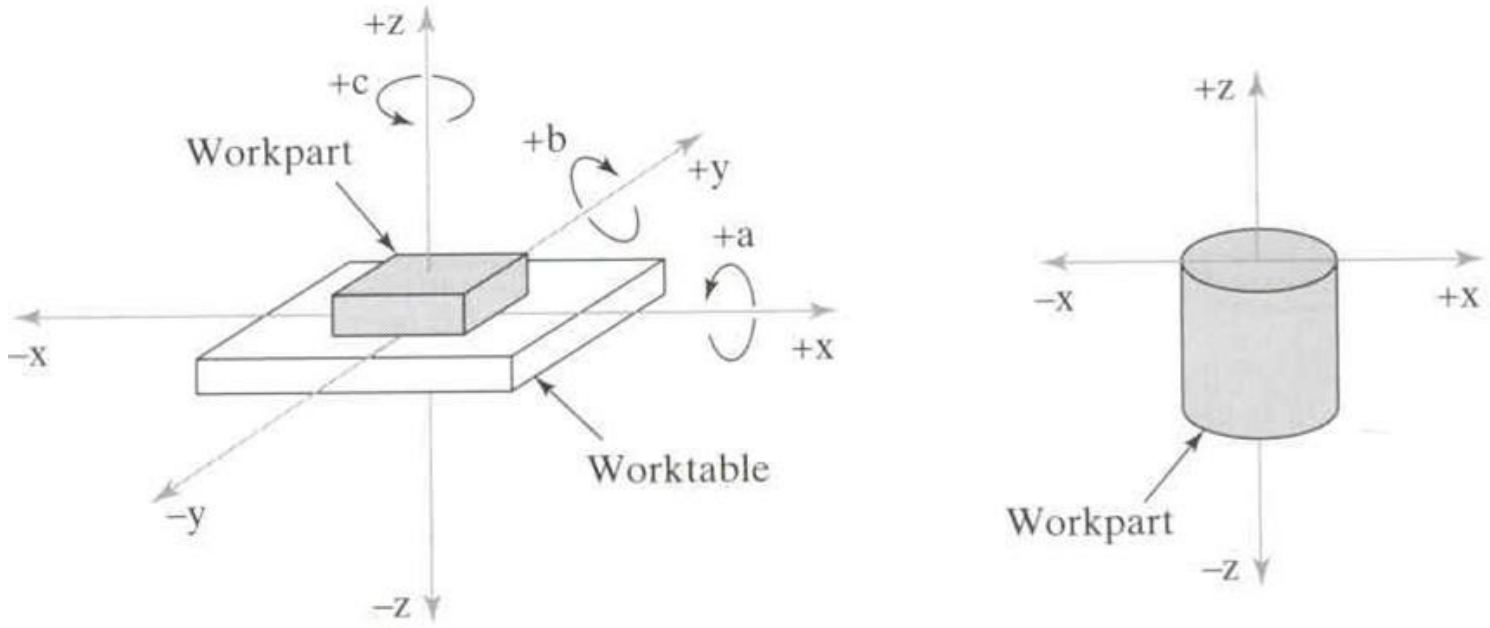
DISADVANTAGES

- Increase in electrical maintenance.
- Increase in initial investment.
- Training of existing workforce.
- Not cost-effective for low-level production on simple parts

Basic CNC Principles

All computer controlled machines are able to accurately and repeatedly control motion in various directions. Each of these directions of motion is called an axis.

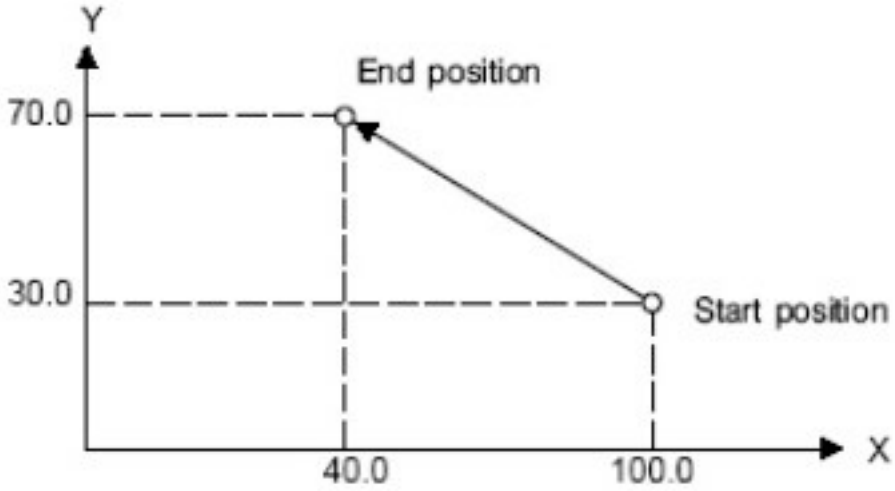
CNC axis may be either a linear axis in which movement is in a straight line, or a rotary axis with motion following a circular path.



Basic CNC Principles

Position Programming Systems

Two types of programming modes, the incremental system and the absolute system, are used for CNC. Both systems have applications in CNC programming, and no system is either right or wrong all the time.



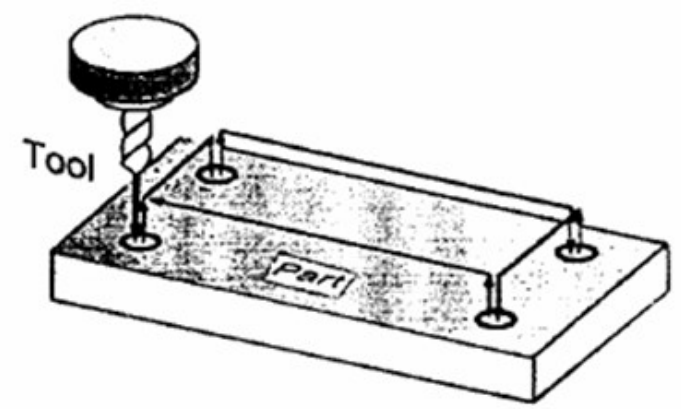
- Incremental program locations are always given as the distance and direction from the immediately preceding point (-60,40)

- Absolute program locations are always given from a single fixed zero or origin point (40,70)

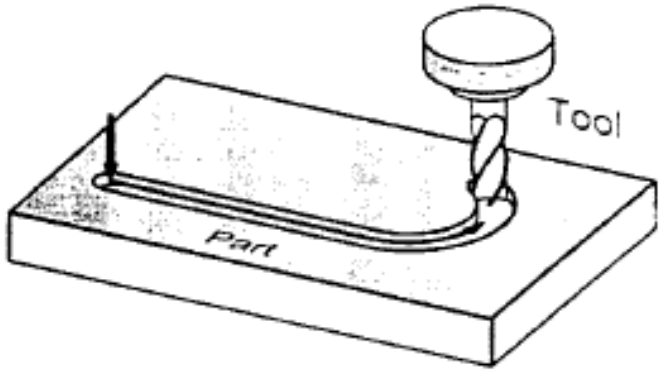
Basic CNC Principles

CNC Motion control Programming falls into two categories point-to point and continuous path machining.

- POINT TO POINT:
  - Moving at maximum rate from point to point.
  - Accuracy of the destination is important but not the path.
  - The tool is not in continuous contact with the part while it is moving.
  - Drilling is a good example.



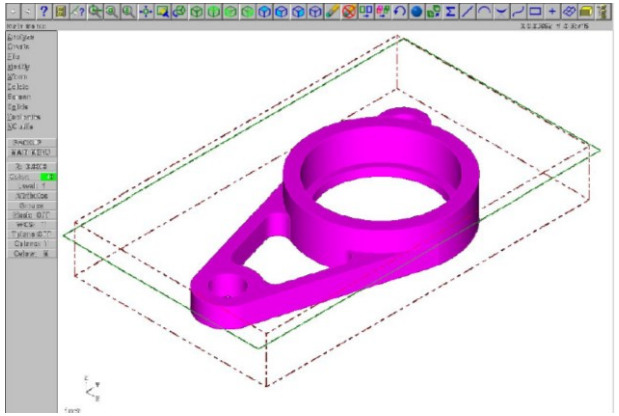
- CONTINUOUS PATH
  - Controls both the displacement and the velocity.
  - Precise control during the whole operation.
  - The tool maintain continuous contact with the part while it is moving.
  - Milling is good example.

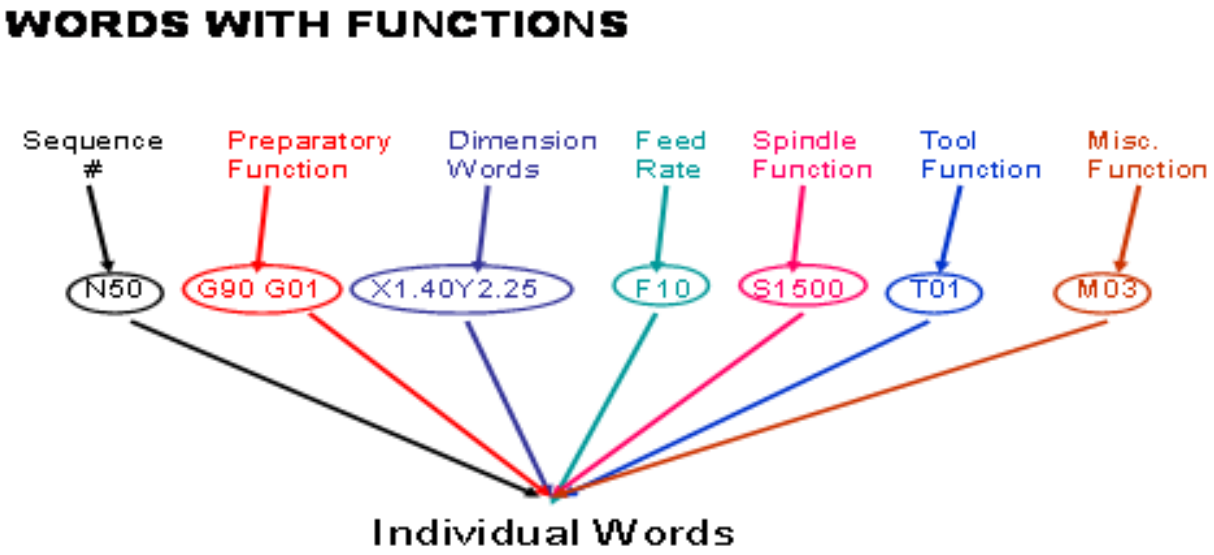


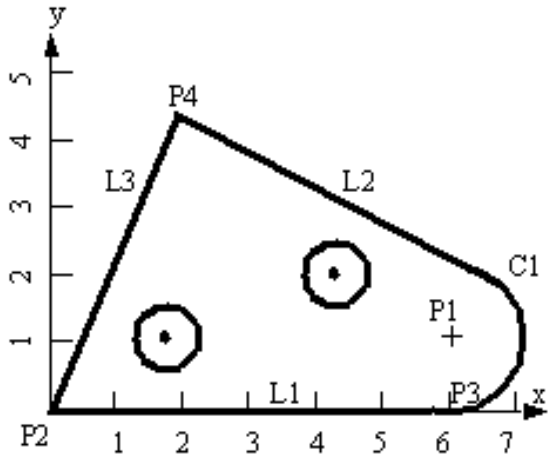
Basic CNC Principles

CNC Programming Methods:

- Computer Aided Manufacturing (CAM) Systems.
- Automatically Programmed Tools (APT).
- Word address (It is the most common programming format used for CNC programming systems)







Programming Methods (CAM)

Computer Aided Manufacturing (CAM) Systems.

- Graphic representation of the part
- PC based
- Integrated CAD/CAM functionality
- “Tool & material libraries
- Tool path simulation
- Cut time calculations for cost estimating

Programming Methods (ATP)

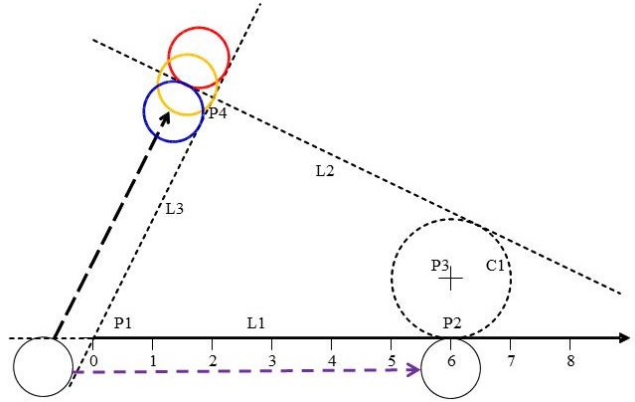
- Automatically Programmed Tools (APT)
  - A text based system in which a programmer defines a series of lines, arcs, and points which define the overall part geometry locations. These features are then used to generate a cutter location (CL) file.
  - Developed as a joint effort between the aerospace industry, MIT, and the US Airforce
  - Still used today and accounts for about 5 -10% of all programming in the defense and aerospace industries
  - Requires excellent 3D visualization skills
  - Capable of generating machine code for complicated part programs
  - Four types of APT statements:
    - Geometry statements

      Define the geometric elements that define the part
    - Motion statements

      Describe the path taken by the cutting tool
    - Post-processor statements

      Apply to the specific machine tool, such as feeds, speeds, feature actuation (coolant on, etc)
    - Auxiliary statements

      Miscellaneous statements that identify the part, tool, tolerances, etc.

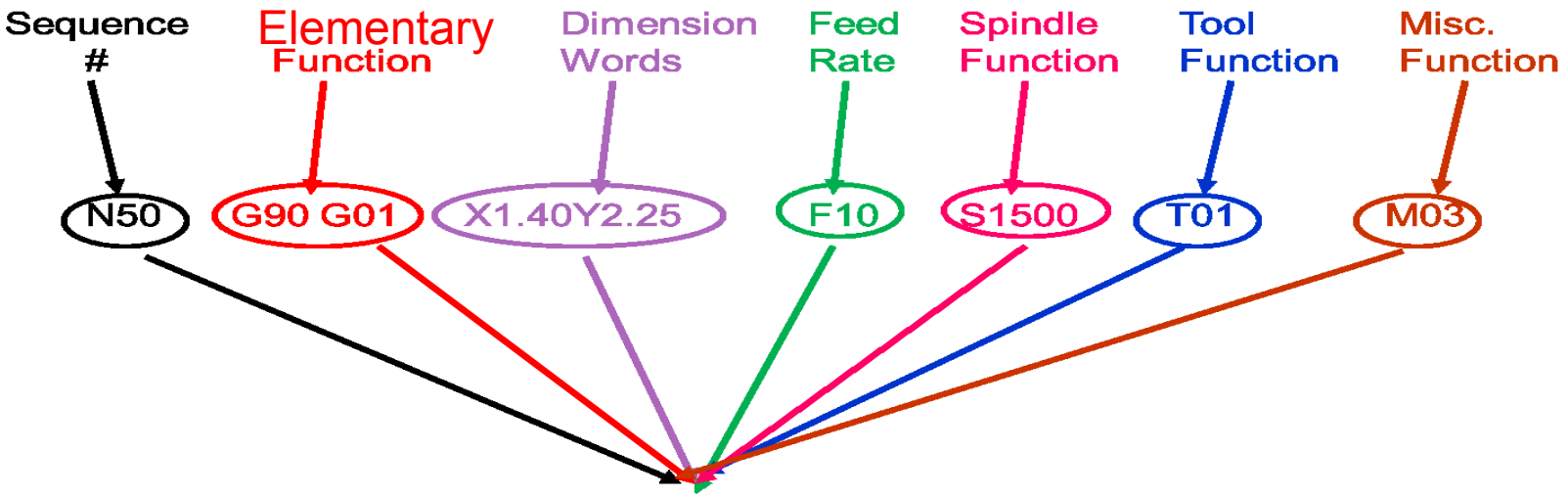


Geometry	P1=Point/12,20,0 C1=Circle/Center,P1,Radius,3 LN1=Line/P1,P2
Motion	GOFWD/ L3, TO, L2

Programming Methods (Word address)

Numerical Control (NC) Language
A series of commands which “direct” the cutter motion and support systems of the machine tool.

This format contains a large number of different codes that transfers program information from the part print to machine servos, relays, micro-switches, etc., to manufacture a part. These codes, which conform to EIA (Electronic Industries Association) standards RS-273, are in a logical sequence called a block of information



Common Format of a Block

Word Address 1

- N – Sequence or line number

A tag that identifies the beginning of a block of code. It is used by operators to locate specific lines of a program when entering data or verifying the program operation.

- G – Elementary function

G words specify the mode in which the machine is to move along its programmed axes.

Word Address 2

- Dimension Words

  X – Distance or position in X direction

  Y – Distance or position in Y direction

  Z – Distance or position in Z direction

- M – Miscellaneous functions

M words specify CNC machine functions not related to dimensions or axial movements.

Word Address 3

- F – Feed rate (inches per minute or millimeters per minute)

Rate at which cutting tool moves along an axis.

- S – Spindle speed (rpm – revolutions per minute)

Controls spindle rotation speed.

- T – Tool number

Specifies tool to be selected.

Word Address 4

- I – Circular cutting reference for x axis
- J – Circular cutting reference for y axis
- K – Circular cutting reference for z axis

Common G Codes

- G00 – Rapid positioning mode

Tool is moved along the shortest route to programmed X,Y,Z position. Usually NOT used for cutting.

- G01 – Linear Interpolation mode

Tool is moved along a straight-line path at programmed rate of speed.

- G02 – Circular motion clockwise (cw)
  - G02 X3.5 Y1 R0.5

  Or
  - G02 X3 Y3.5 I-0.5 J0

- G03 – Circular motion counter clockwise (ccw)
  - G03 X3.5 Y1 R0.5

  Or
  - G03 X3 Y3.5 I-0.5 J0

Common G Codes

- G17 – XY plane
- G18 – XZ plane
- G19 – YZ plane
- G20 – Inch Mode
- G21 – Metric Mode

G Codes: G90, G91

G90 – Absolute Coordinate Reference

References the next position from an absolute zero point which is set once for the entire program.

G91 – Incremental Coordinate Reference

References the next position from the previous position.

G Codes: Cutter Compensation

- G40 – Cancel cutter diameter compensation.
- G41 – Cutter compensation left.
- G42 – Cutter compensation right.

Common M words

- M00 – Programmed pause

Automatically stops machine until operator pushes a button to resume program.

- M01 – Optional stop

A stop acted upon by the machine when operator has signaled this command by pushing a button.

- M02 – End of program

Stops program when all lines of code are completed. Must be last command in program.

Common M words

- M03 – Turn spindle on

In clockwise direction

- M04 – Turn spindle on

In counter clockwise direction

- M05 – Stop spindle

Usually used prior to tool change or at end of program.

- M06 – Tool change

Stops program and calls for a tool change, either automatically or manually.

Example 1

N05 G90 G20 G17

N10 M06 T2

N15 M03 S1200

N20 G00 X2 Y0.5 z0.125

N25 G01 Z-0.125 F5

N30 X3 F15

N35 G03 X3.5 Y1 R0.5

N40 G01 Y3

N45 G03 X3 Y3.5 I-0.5 J0

N50 G01 X2

N55 G03 X2 Y1.5 I0 J-1

N60 G01 Y0.5

N65 G00 Z0.1

N70 X1.5 Y2.5

N75 G01 Z-0.25 F5

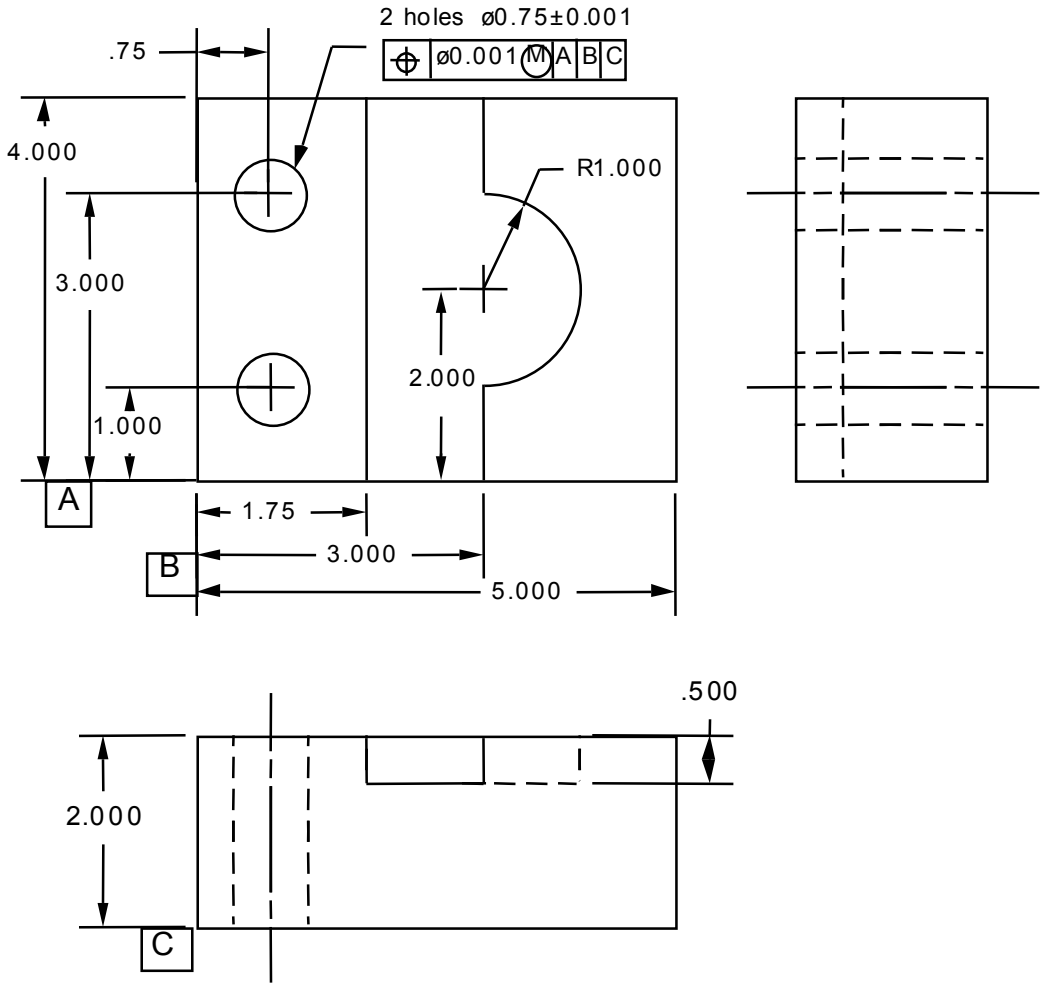
N80 G03 X1.5 Y2.5 I0.5 J0

N85 G00 Z1

N90 X0 Y0

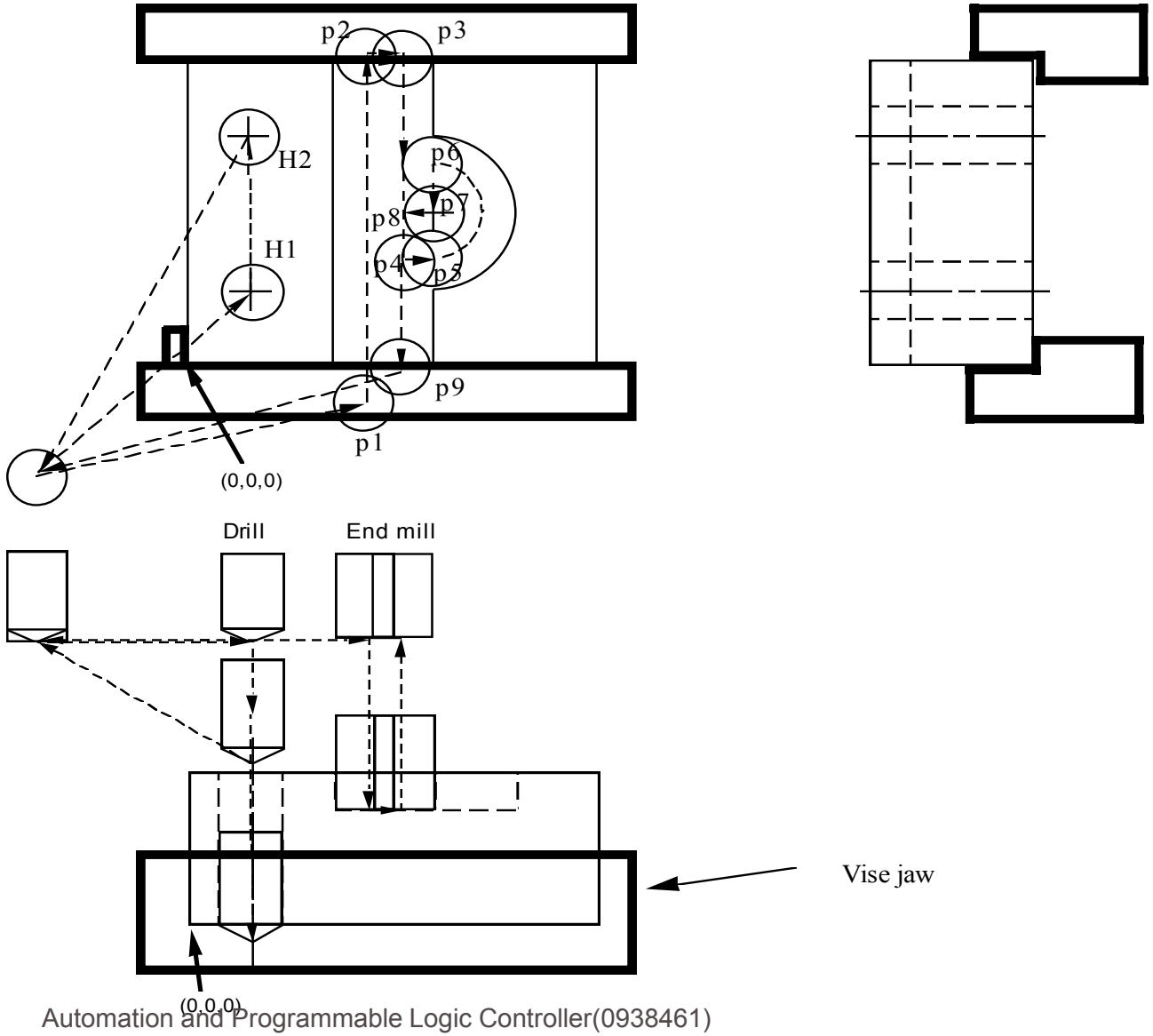
N95 M05

Example 2



All dimension in inches. All tolerance ±0.001"

SETUP AND CUTTER PATH



Example 3

N00 G90 ; ABSOLUTE COORDINTAES

N01 M06T1 ; LOAD TOOL 1

N02 M03 S3000 ; TURN SPINDLE ON TO 3000 RPM

N03 G00 Z .1; RAPID TO .1 ABOVE PART

N04 G00 X.5Y.5 ; RAPID TO POINT A

N05 G01 Z-.0625F9 ; PLUNGE 1/16 AT 9 IN/MIN.

N06 G01 X.5Y1.5; STRAIGHT LINE INTERP TO B

N07 G01 X.875Y1; STRAIGNT LINE INTERP TO C

N08 G01 X1.25Y1.5; STRAIGHT LINE INTERP TO D

N09 G01 X1.25Y.5; STRAIGHT LINE INTERP TO E

N10 G01 Z.1; RETRACT CUTTING TOOL

N11 G00 X1.75Y1.5; RAPID TO POINT F

N12 G01 Z-.0625F9; PLUNG AT 9 IN/MIN.

N13 G01 X1.75Y.5; STRAING LINE INTERP TO H

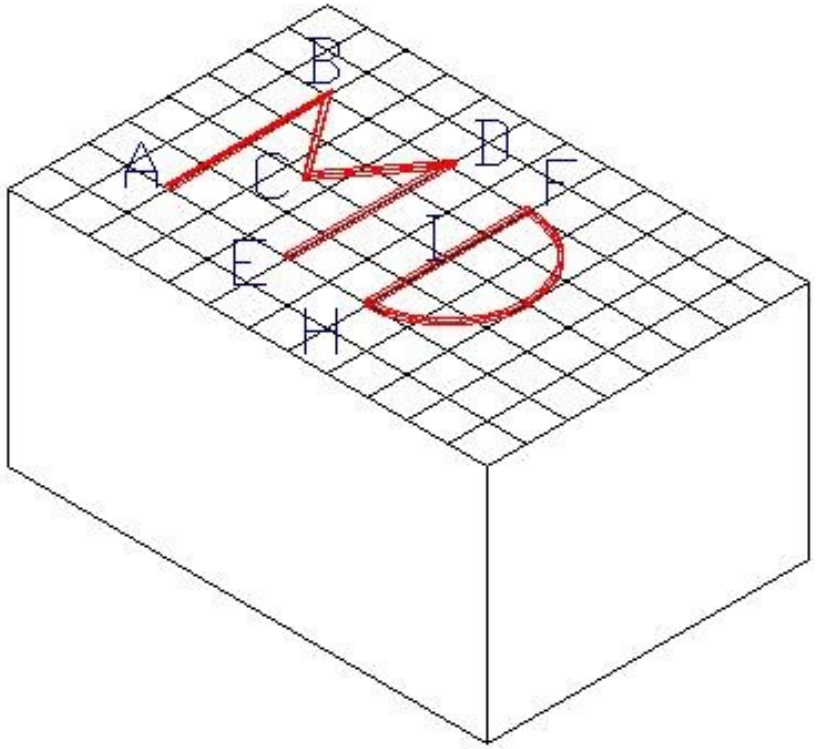
N14 G03 X1.75Y1.5I1.75J1; CCW CIRCLE INTERP.

N15 G01Z.1; RETRACT CUTTING TOOL

N16 M05; TURN OFF SPINDLE

N17 M06T00; UNLOAD CUTTING TOOL

N18 M02; END OF PROGRAM



Example 4