CNC Programming

Motivation and uses

To manufacture complex curved geometries in 2D or 3D was extremely expensive by mechanical means (which usually would require complex jigs to control the cutter motions)

Machining components with repeatable accuracy

Unmanned machining operations

Advantages of CNC

- Easier to program;

- Easy storage of existing programs;

- Easy to change a program

- Avoids human errors

- safer to operate

- Complex geometry is produced as cheaply as simple ones

- Usually generates closer tolerances than manual machines

Conventional milling machines

Vertical Milling machine architecture
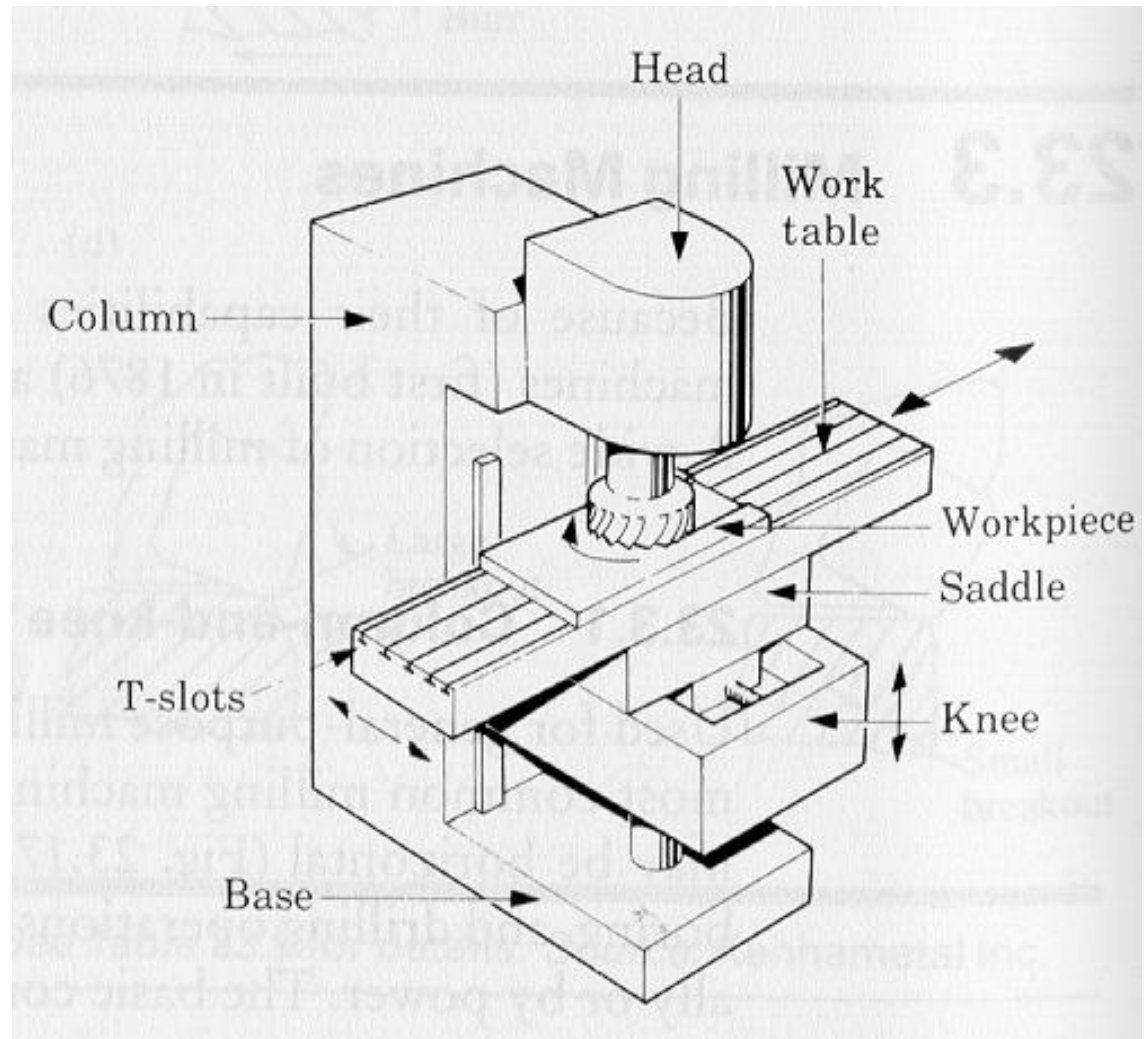
Types of CNC machines

Based on Motion Type:

       Point-to-Point   or   Continuous path

Based on Control Loops:

       Open loop   or   Closed loop

Based on Power Supply:

       Electric   or   Hydraulic   or   Pneumatic

Based on Positioning System

       Incremental   or   Absolute

Basic CNC Principles

- Using a vertical mill machining center as an example, there are typically three linear axes of motion. Each is given an alphabetic designation or address. The machine table motion side to side is called the "X" axis. Table movement in and out is the "Y" axis, while head movement up and down the column is the "Z" axis.

STEPPING MOTOR

SPINDLE HEAD

Z

Z CONTROL

MACHINE TABLE

STEPPING MOTOR

X

STEPPING MOTOR

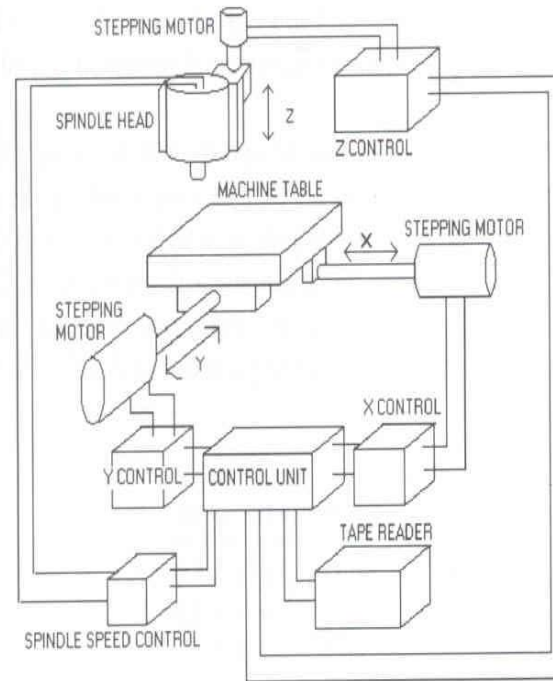Y

X CONTROL

Y CONTROL — CONTROL UNIT
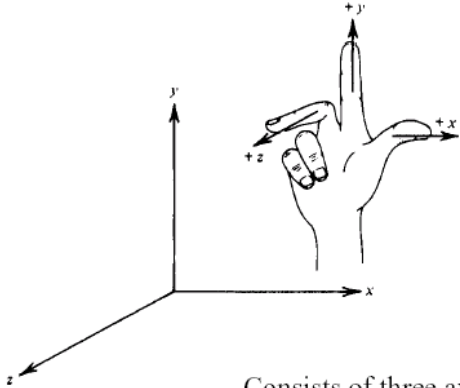
TAPE READER

SPINDLE SPEED CONTROL

Figure 8.7 Servo mechanism.
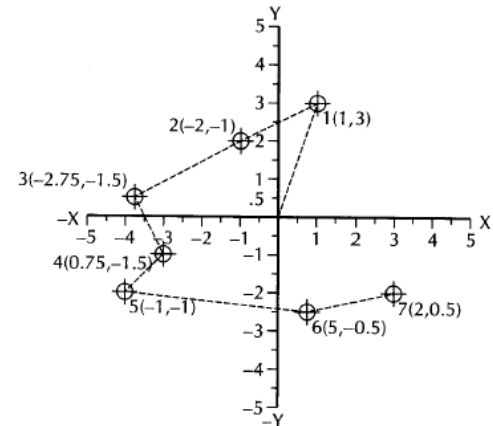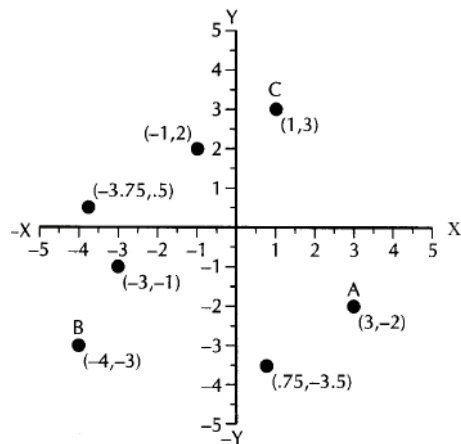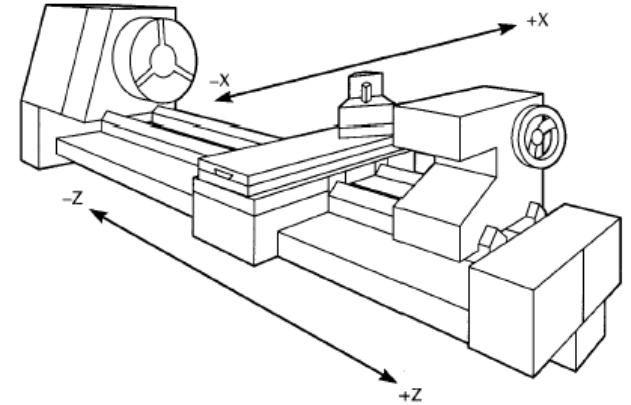
Servo drives are

- tolerant to tool cutting forces
- insensitive to load mass inertias (stiffness)
- insensitive to running friction forces
- linear to a high degree

Basic CNC Principles
Coordinates System

Consists of three axes positioned 90 degrees from each other.

Absolute Coordinate System

Incremental Coordinate System

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. Depending on the machine type there are commonly two to five axes.

Additionally, a 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.
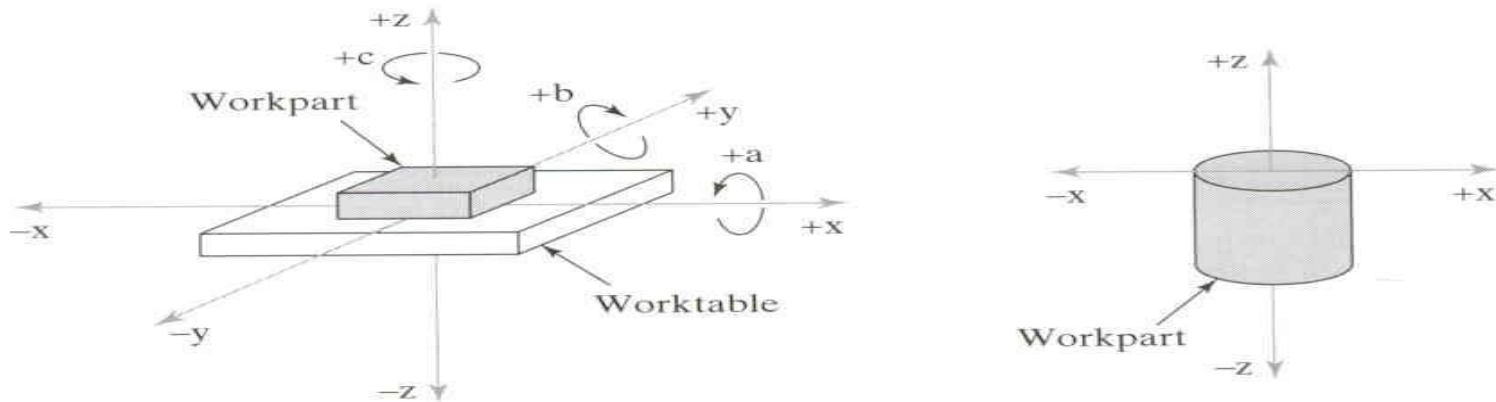

Figure 6.2 Coordinate systems used in NC: (a) for flat and prismatic work and (b) for rotational work. (On most turning machines, the z-axis is horizontal rather than vertical as we have shown it.)

Basic CNC Principles

If a rotary table is added to the machine table, then the fourth axis is designated the "b" axis.
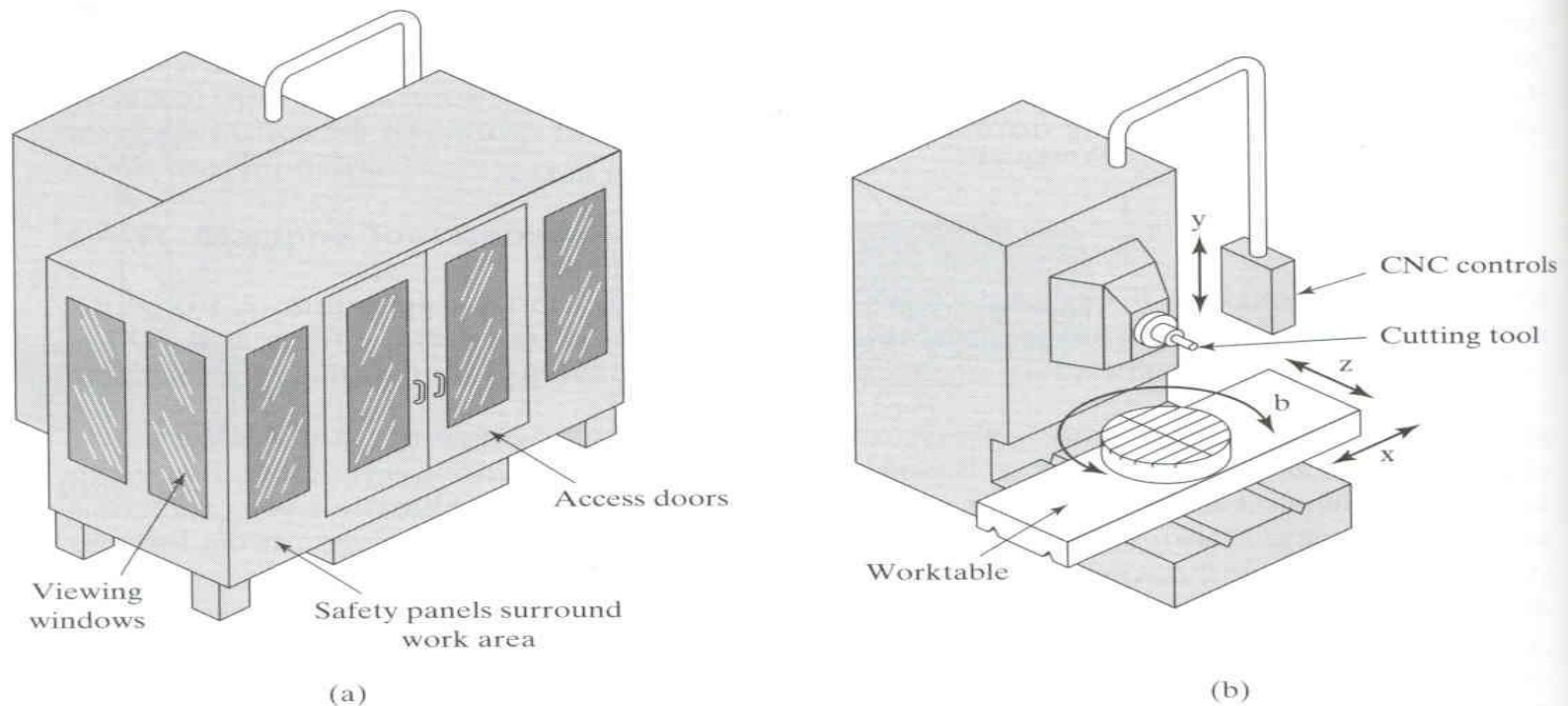

Figure 6.11 (a) Four-axis CNC horizontal milling machine with safety panels installed and (b) with safety panels removed to show typical axis configuration for the horizontal spindle.

How CNC Works

- Controlled by G and M codes.

- These are number values and co-ordinates.

- Each number or code is assigned to a particular operation.

- Typed in manually to CAD by machine operators.

- G&M codes are automatically generated by the computer software.

Features of CNC Machinery

- The tool or material moves.

- Tools can operate in 1-5 axes.

- Larger machines have a machine control unit (MCU) which manages operations.

- Movement is controlled by a motors (actuators).

- Feedback is provided by sensors (transducers)

- Tool magazines are used to change tools automatically.

Tools

- Most are made from
  high speed steel (HSS),
  tungsten carbide or ceramics.

- Tools are designed to direct waste away from the material.

- Some tools need coolant such as oil to protect the tool and work.

Tool Paths, Cutting and Plotting Motions

- Tool paths describes the route the cutting tool takes.
- Motion can be described as point to point, straight cutting or contouring.
- Speeds are the rate at which the tool operates e.g. rpm.
- Feeds are the rate at which the cutting tool and work piece move in relation to each other.
- Feeds and speeds are determined by cutting depth, material and quality of finish needed. e.g. harder materials need slower feeds and speeds.
- Rouging cuts remove larger amounts of material than finishing cuts.
- Rapid traversing allows the tool or work piece to move rapidly when no machining is taking place.

Manual NC programming

Part program: A computer program to specify

- Which tool should be loaded on the machine spindle;

- What are the cutting conditions (speed, feed, coolant ON/OFF etc)

- The start point and end point of a motion segment

- how to move the tool with respect to the machine.

Standard Part programming language: RS 274-D (Gerber, GN-code)

History of CNC

The RS274-D is a **word** **address format**

Each line of program == 1 **block**

   Each block is composed of several instructions, or (words)

Sequence and format of words:

N3 G2 X+1.4 Y+1.4 Z+1.4 I1.4 J1.4 K1.4 F3.2 S4 T4 M2

sequence no
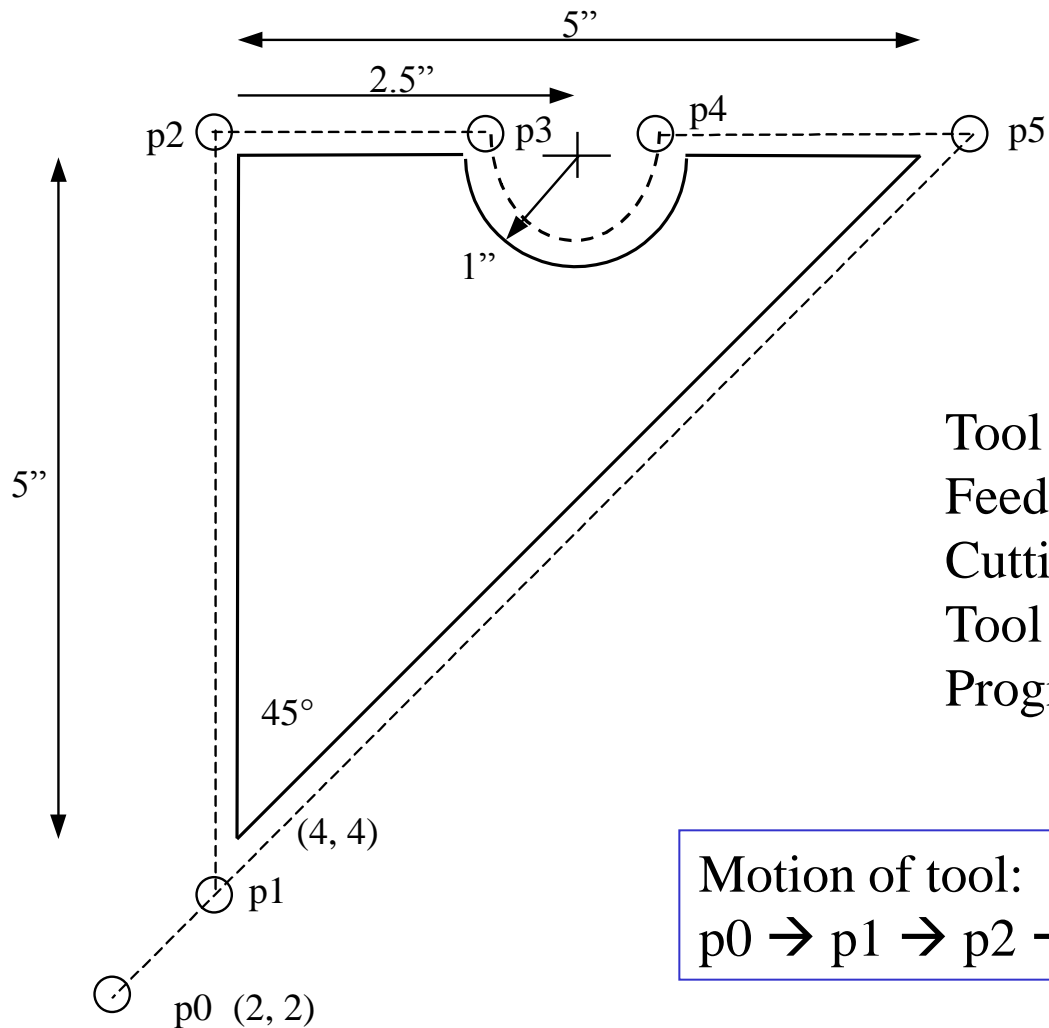
destination coordinates dist to center of circle tool

preparatory function

feed rate spindle speed

Other function

Manual Part Programming Example

5"

2.5"

p2   p3   p4   p5

1"

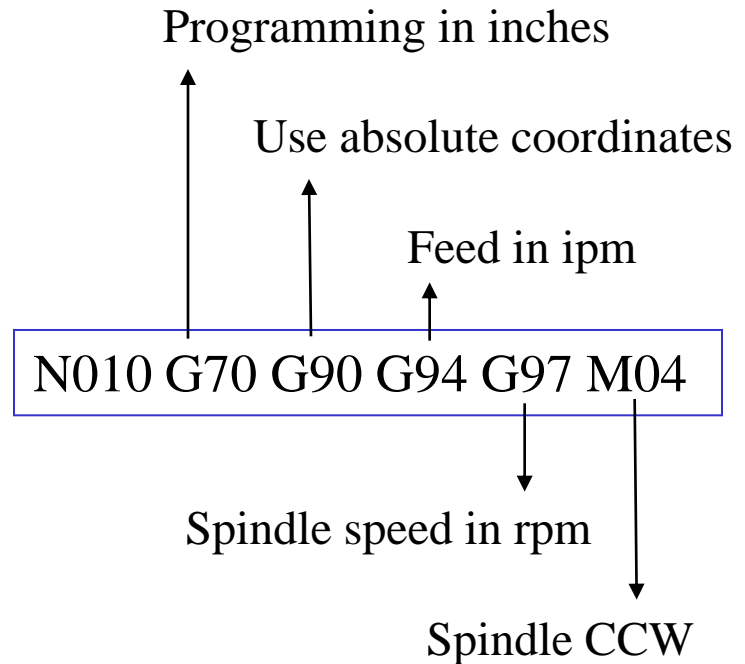5"

45°

(4, 4)

p1

p0 (2, 2)

Tool size = 0.25 inch,
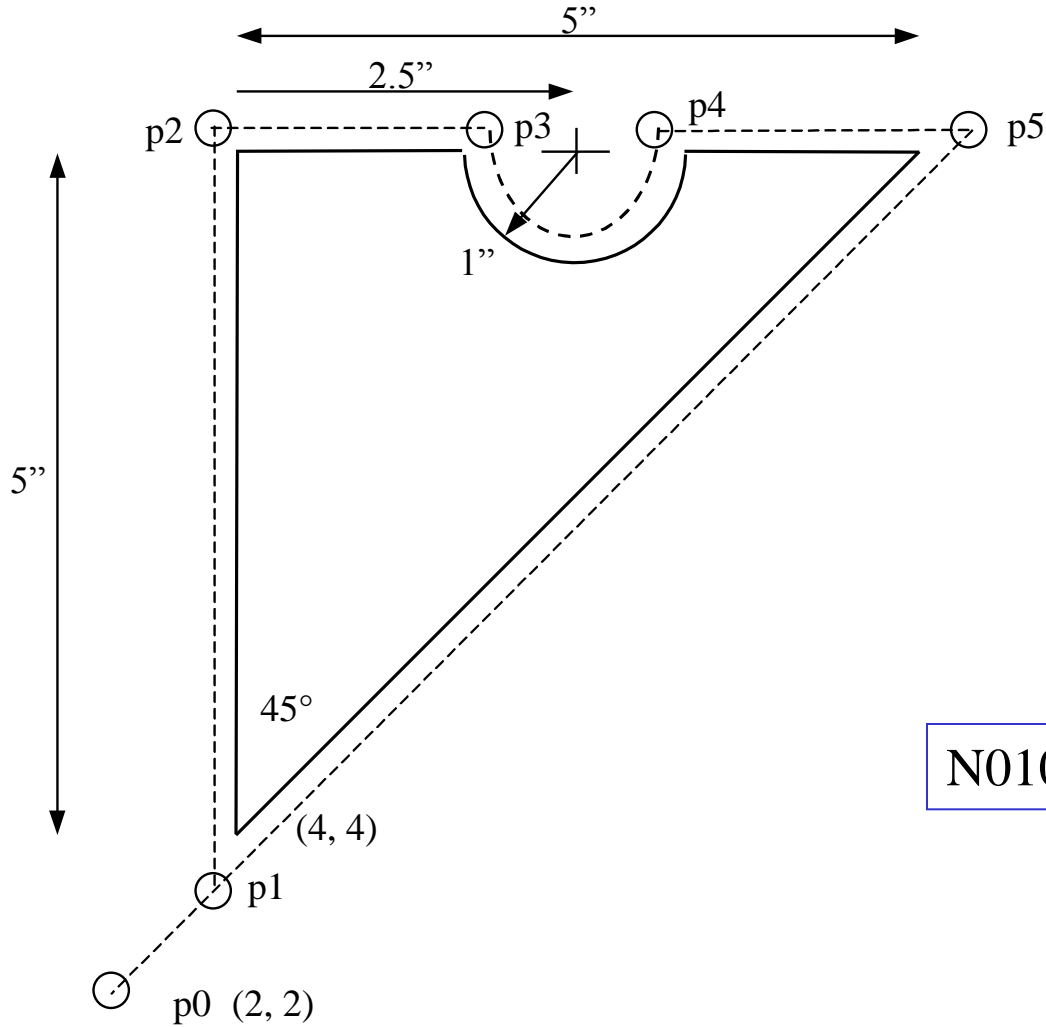Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

Motion of tool:
p0 → p1 → p2 → p3 → p4 → p5 → p1 → p0

1. Set up the programming parameters

5"

2.5"

p2   p3   p4   p5

1"

5"

45°

(4, 4)

p1

p0 (2, 2)

Programming in inches

Use absolute coordinates

Feed in ipm

N010 G70 G90 G94 G97 M04

Spindle speed in rpm

Spindle CCW

2. Set up the machining conditions

Machine moves in XY-plane

Use full-circle interpolation

Feed rate

Spindle speed

N020 G17 G75 F6.0 S300 T1001 M08

Tool no.

Flood coolant ON

5"

2.5"

p2   p3   p4   p5

Linear interpolation

1"

target coordinates

N030 G01 X3.875 Y3.698

5"

45°

45°

(4, 4)

(4, 4)

p1

p1

p1:

p0 (2, 2)

x = 4 - 0.125 = 3.875
y = 4 - 0.125/tan 22.5 = 3.698
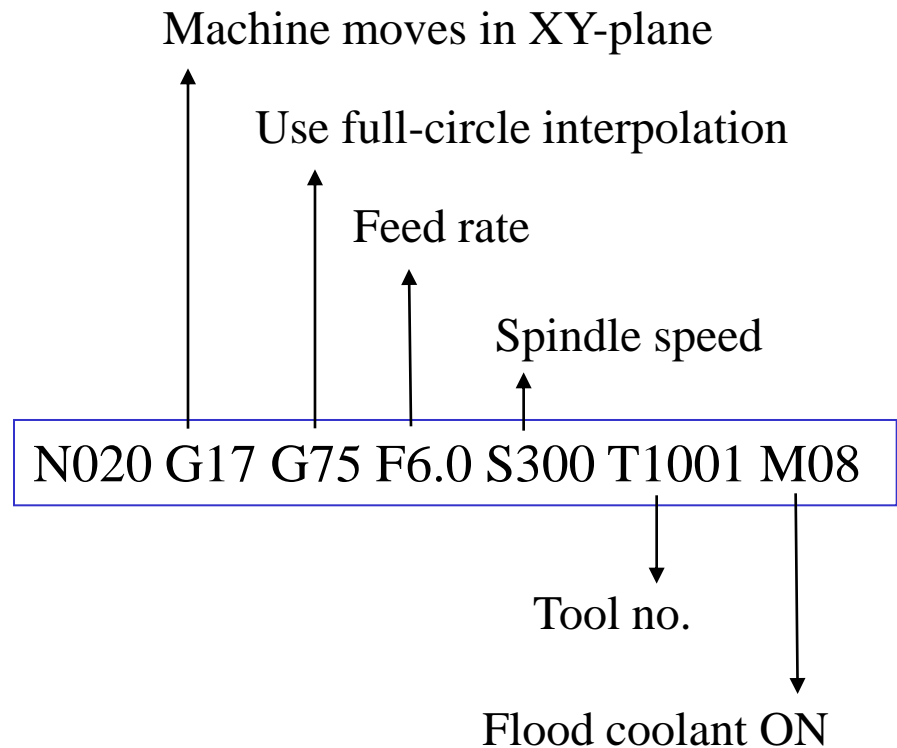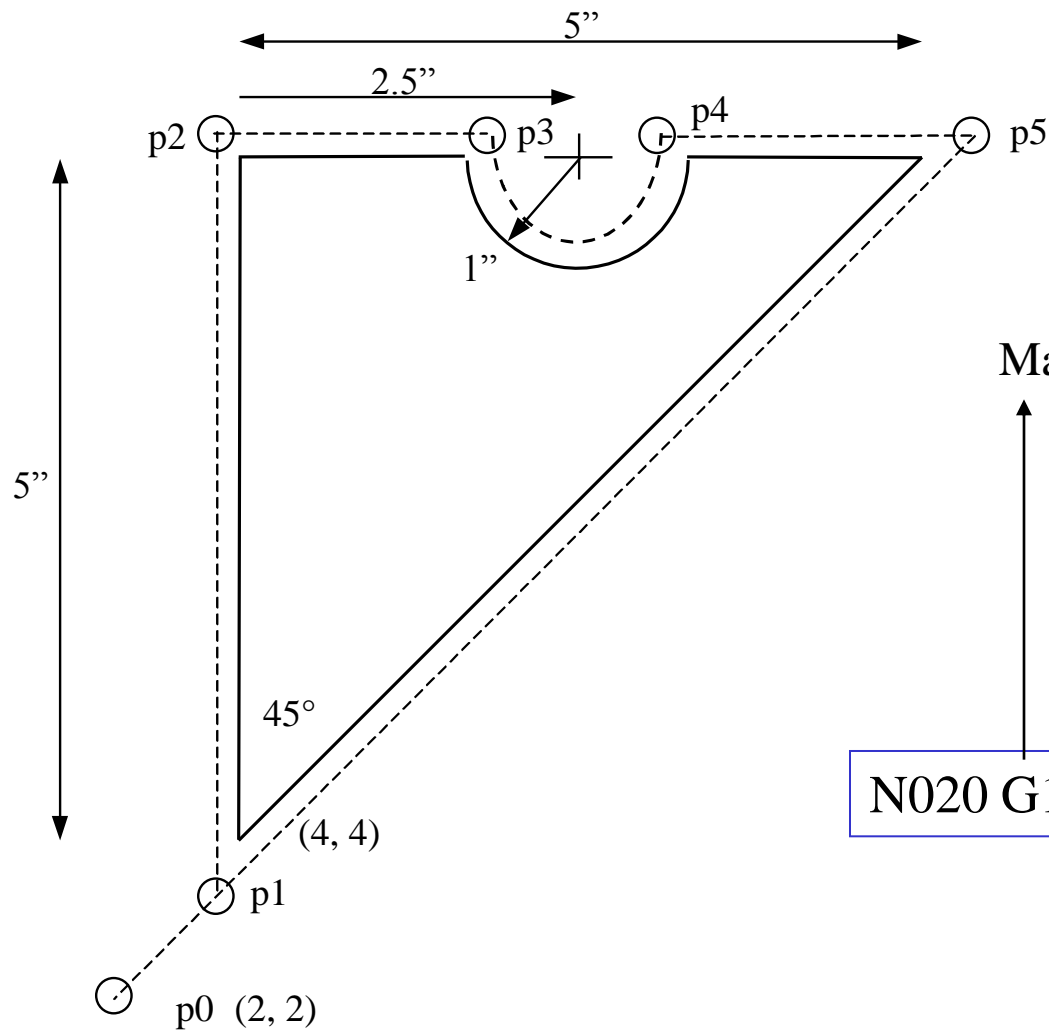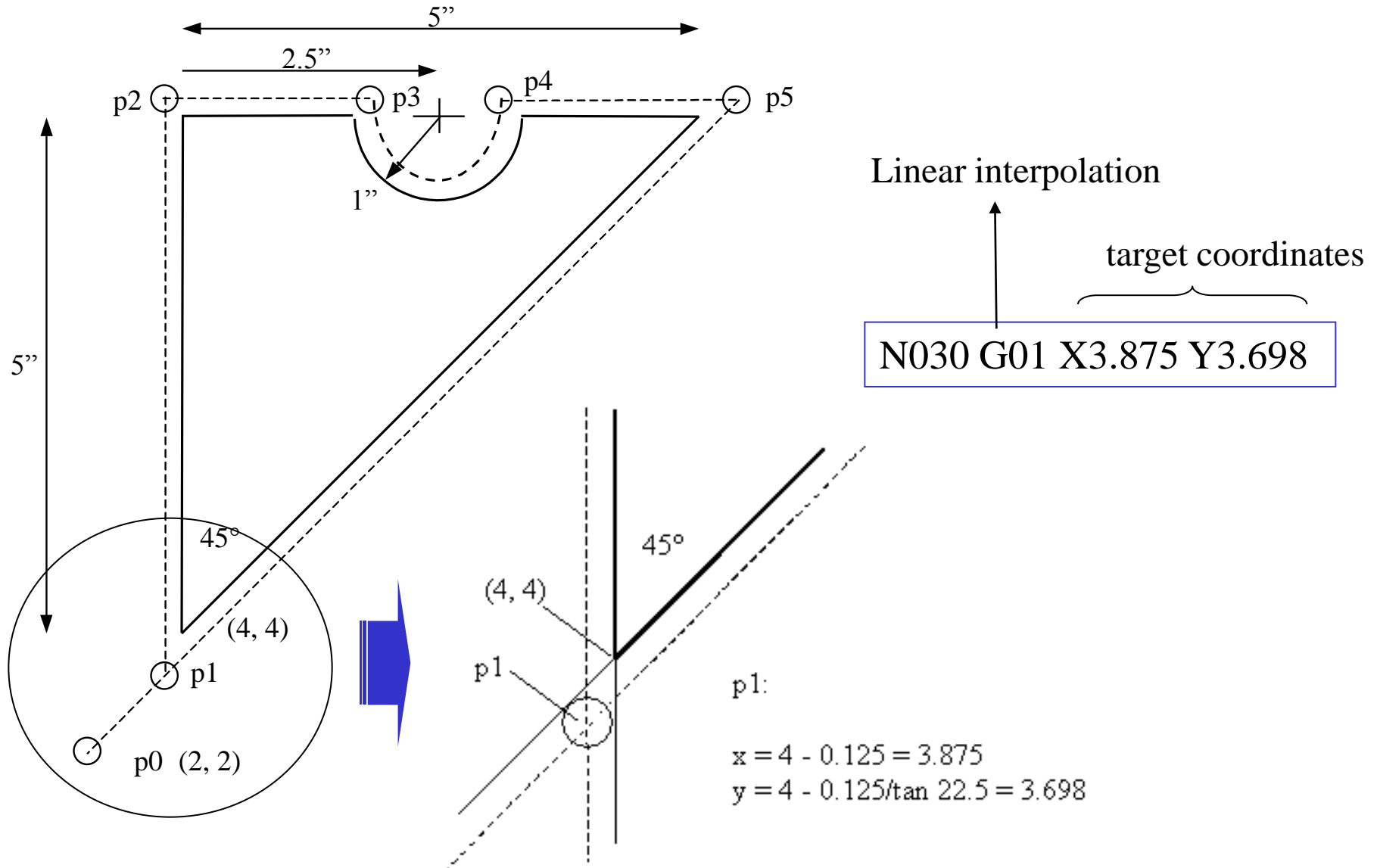
4. Cut profile from p1 to p2

5"

2.5"

p2 p3 p4 p5

1"

5"

45°

(4, 4)

p1

p0 (2, 2)
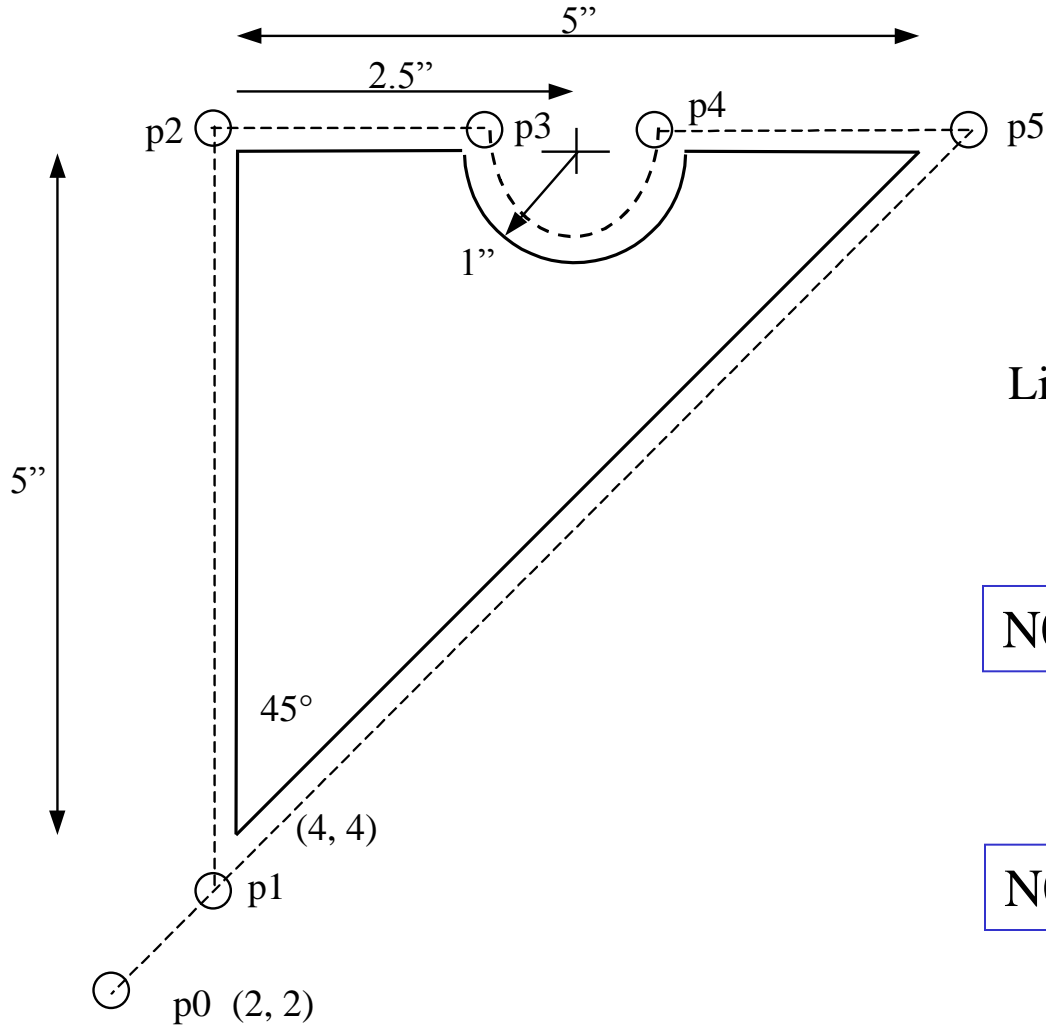
Linear interpolation

target coordinates

N040 G01 X3.875 Y9.125

or

N040 G01 Y9.125

X-coordinate does not change ➜ no need to program it

5. Cut profile from p2 to p3

5"

2.5"

p2

p3

p4

p5

Linear interpolation

target coordinates

N050 G01 X5.634 Y9.125

1"

5"

45°

(4, 4)

p1

p0 (2, 2)

$y = 9 + 0.125 = 9.125$

$(6.5 - x)^2 + 0.125^2 = (1 - 0.125)^2$

$x = 5.634$

p3

(x, y)

.125

(6.5, 9)

1"
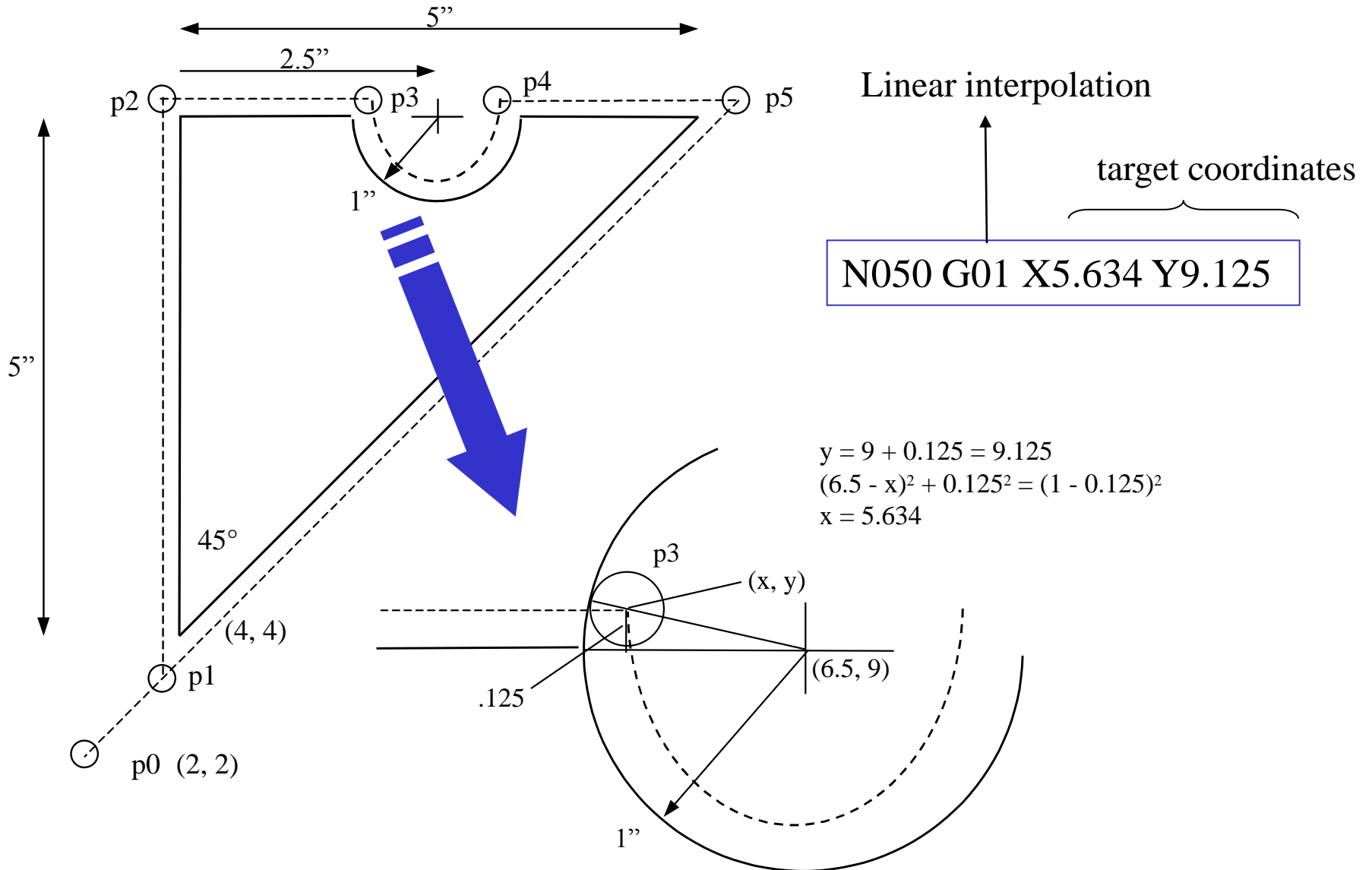
6. Cut along circle from p3 to p4
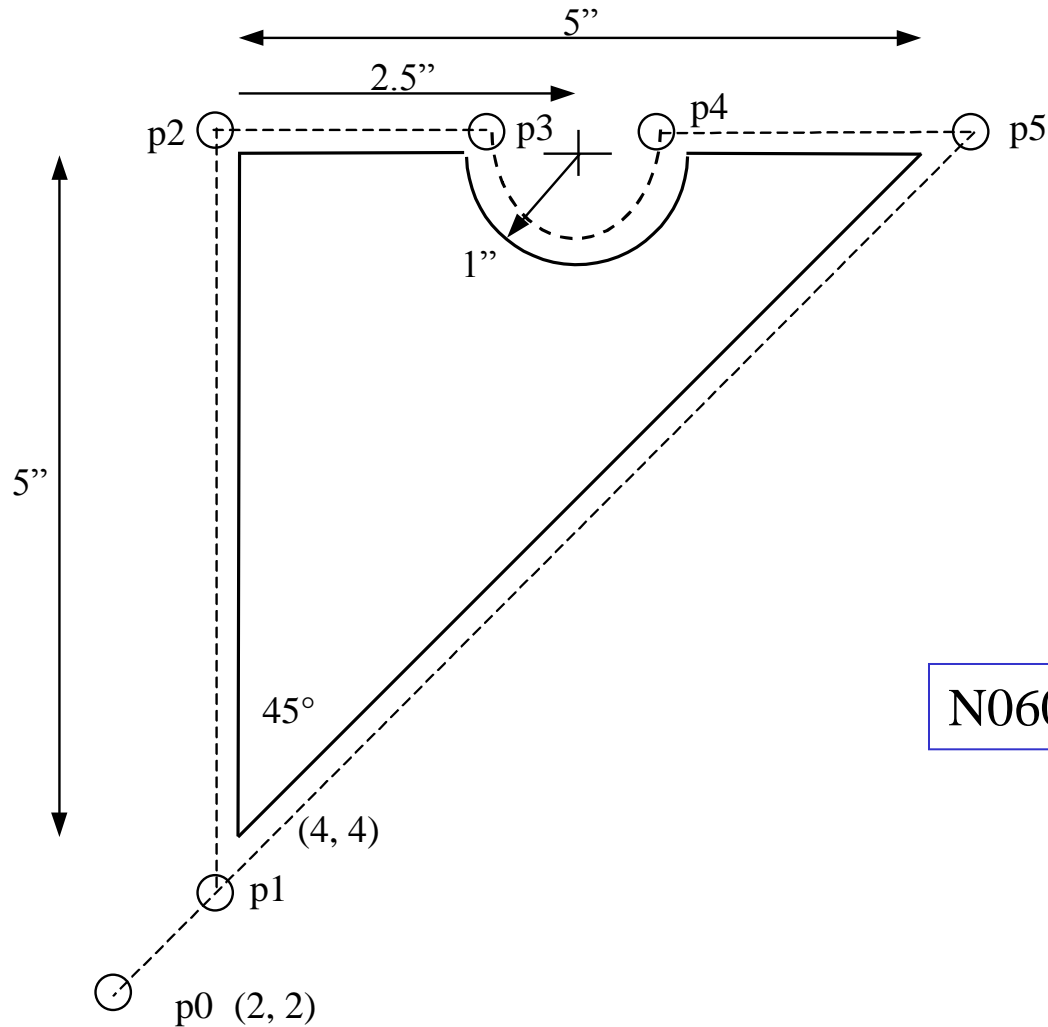
circular interpolation, CCW motion

target coordinates

N060 G03 X7.366 Y9.125 I6.5 J9.0

coordinates of center of circle

5"

2.5"

p2   p3   p4   p5

1"

5"

Linear interpolation

45°

target coordinates (Y is unchanged)

(4, 4)

p1

N070 G01 X9.302

p0 (2, 2)

5"

2.5"

p2   p3   p4   p5

1"

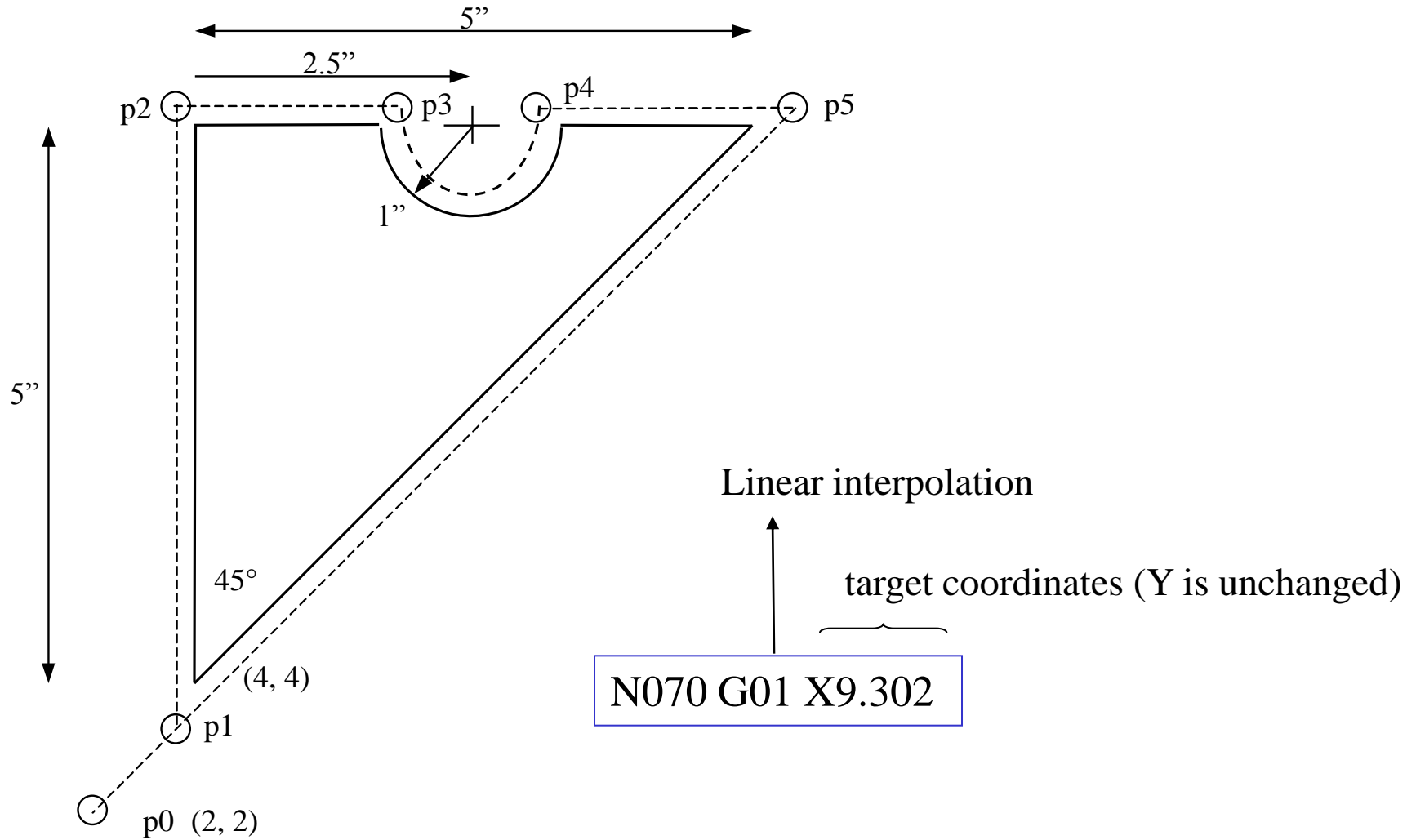5"

Linear interpolation

target coordinates (see step 3)

45°

N080 G01 X3.875 Y3.698

(4, 4)

p1

p0 (2, 2)
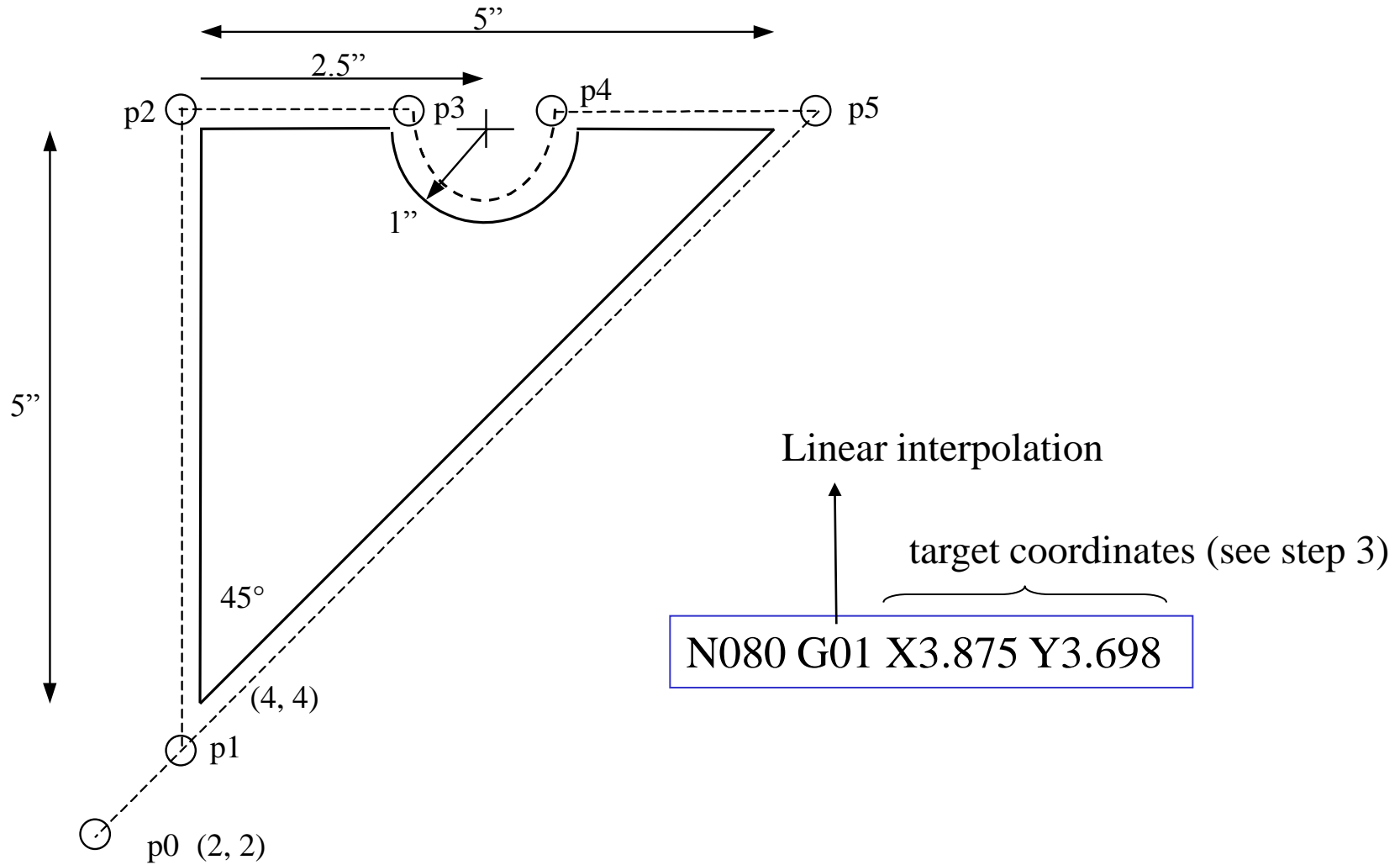
9. Return to home position, stop program

5"

2.5"

p2 p3 p4 p5

1"

5"

45°

(4, 4)

p1

p0 (2, 2)

Linear interpolation

target coordinates (see step 3)

N090 G01 X2.0 Y2.0 M30

end of data

N100 M00

program stop
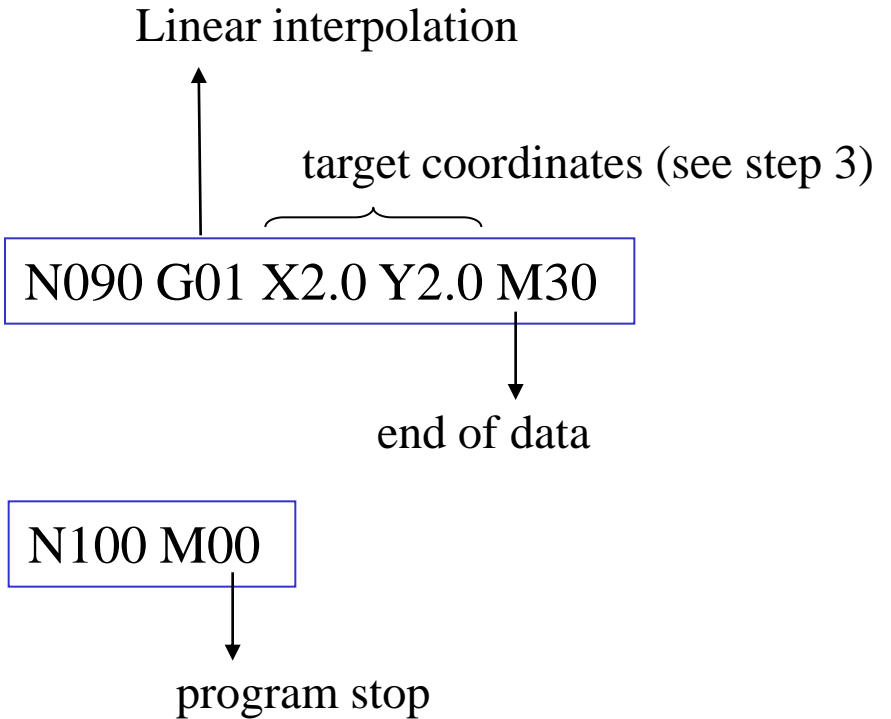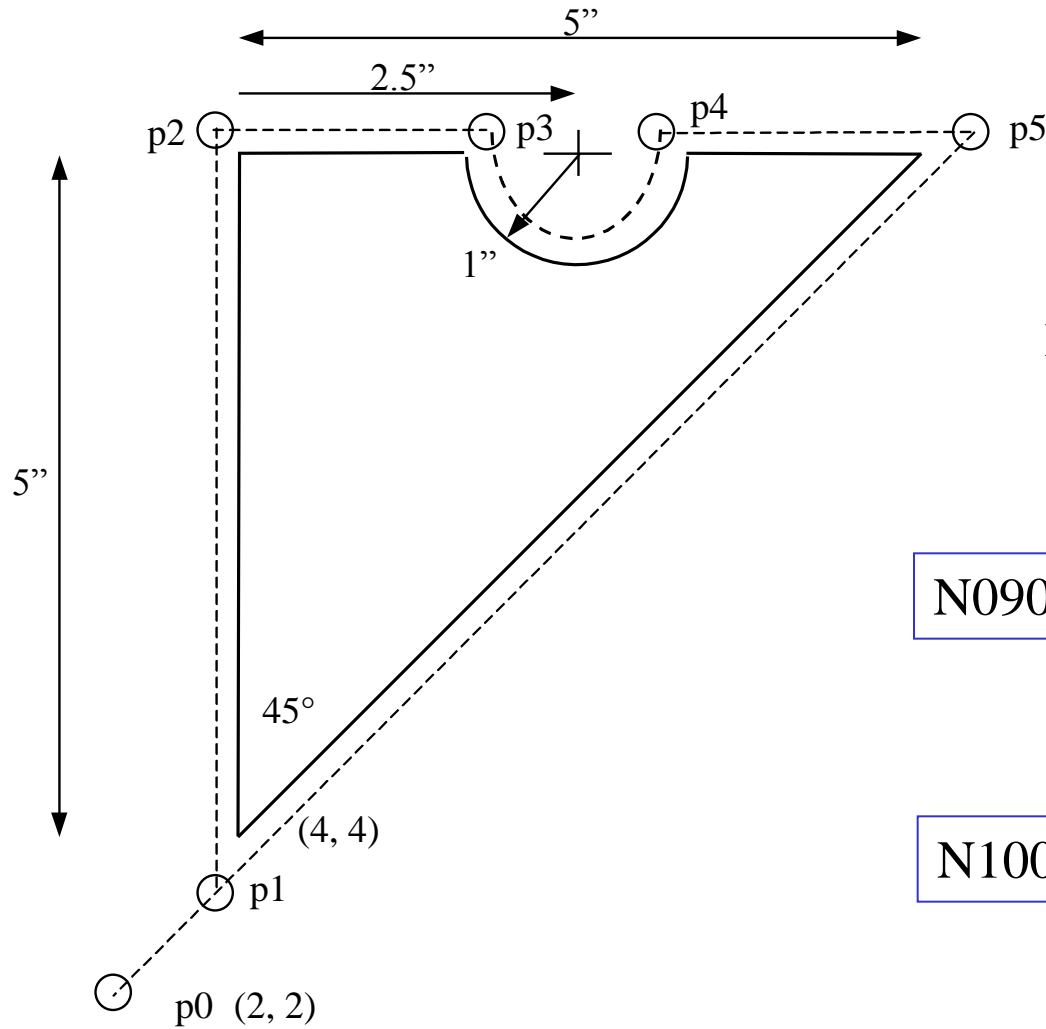
CNC Programming Basics

- CNC instructions are called part program commands.

- When running, a part program is interpreted one command line at a time until all lines are completed.

- Commands, which are also referred to as blocks, are made up of words which each begin with a letter address and end with a numerical value.

CNC Programming Basics

- Each letter address relates to a specific machine function. "G" and "M" letter addresses are two of the most common. A "G" letter specifies certain machine preparations such as inch or metric modes, or absolutes versus incremental modes.

- A "M" letter specifies miscellaneous machine functions and work like on/off switches for coolant flow, tool changing, or spindle rotation. Other letter addresses are used to direct a wide variety of other machine commands.

CNC programming

Important things to know:

- Coordinate System

- Units, incremental or absolute positioning

- Coordinates: X,Y,Z, RX,RY,RZ

- Feed rate and spindle speed

- Coolant Control: On/Off, Flood, Mist

- Tool Control: Tool and tool parameters

CNC programming

- Programming consists of a series of instructions in form of letter codes

- Preparatory Codes:

- G codes- Initial machining setup and establishing operating conditions

- N codes- specify program line number to executed by the MCU

- Axis Codes: X,Y,Z - Used to specify motion of the slide along X, Y, Z direction

- Feed and Speed Codes: F and S- Specify feed and spindle speed

- Tool codes: T – specify tool number

- Miscellaneous codes – M codes For coolant control and other activities

Programming Key Letters

- O - Program number (Used for program identification)
- N - Sequence number (Used for line identification)
- G - Preparatory function
- X - X axis designation
- Y - Y axis designation
- Z - Z axis designation
- R - Radius designation
- F – Feed rate designation
- S - Spindle speed designation
- H - Tool length offset designation
- D - Tool radius offset designation
- T - Tool Designation
- M - Miscellaneous function

Explanation of commonly used G codes

- G00 – Preparatory code to control final position of the tool and not concerned with the path that is followed in arriving at the final destination.

- G01 – Tool is required to move in a straight line connecting current position and final position. Used for tool movement without any machining- point to point control. (linear interpolation)

- G02 – Tool path followed is along an arc specified by I, J and K codes.(circular interpolation)

Table of Important G codes

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

Table of Important 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

Program Command Parameters

Optimum machine programming requires consideration of certain machine operating parameters including:
• Positioning control
• Compensations
• Special machine features

Positioning control is the ability to program tool and machine slide movement simultaneously along two or more axes. Positioning may be for point-to-point movement or for contouring movement along a continuous path. Contouring requires tool movement along multiple axes simultaneously. This movement is referred to as "Interpolation" which is the process of calculating intermediate values between specific points along a programmed path and outputting those values as a precise motion. Interpolation may be linear having just a start and end point along a straight line, or circular which requires an end point, a center and a direction around the arc.

Rules for programming

Block Format

N135 G01 X1.0 Y1.0 Z0.125 F5

Sample Block
• 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 feed rate
• 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)
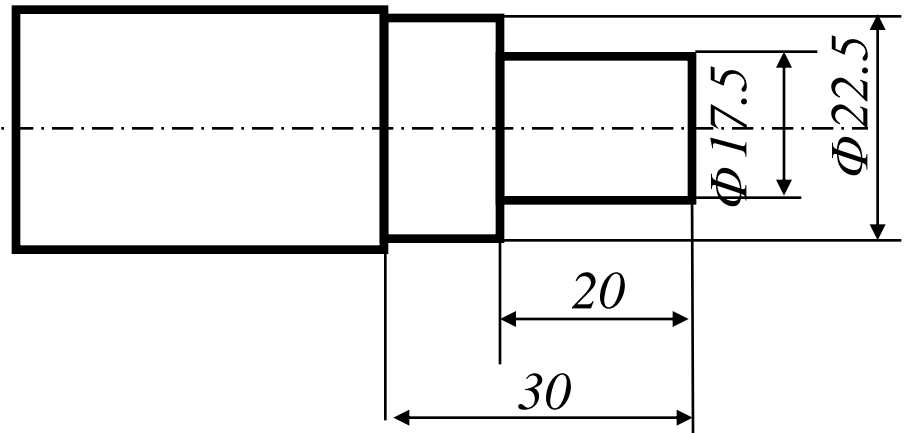• The data within a block should follow the sequence shown in the above sample block
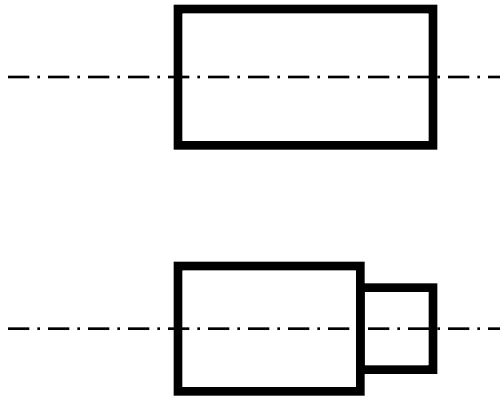
APT Programming Example
Cylindrical Part

Raw Material

Φ25

70

Finished Part

Φ17.5

Φ22.5

20

30
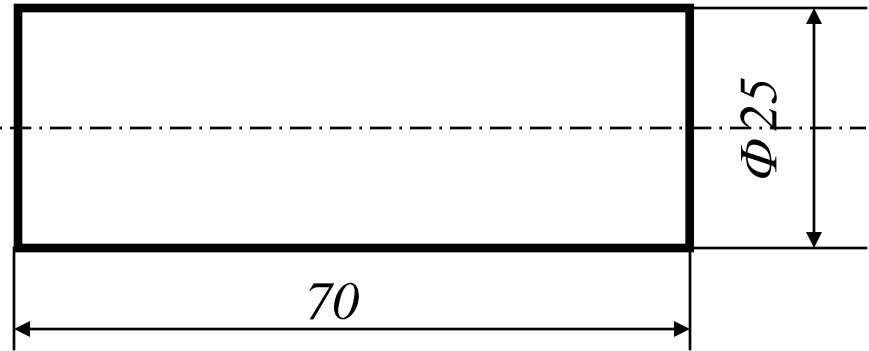
APT Programming Example (*Cylindrical Part*)

O0013

N0005 G53
N0010 T0303
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X22.50 Z2.0 S500
N0080 G01 Z-30.0 F100
N0090 G00 X23.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
N0110 G00 Z2.0
N0120 X50.0 Z50.0
N0130 M30

APT Program Interpretation

O0013

Program identification number

APT Program Interpretation

O0013

N0005 G53

To cancel any previous working zero point

APT Program Interpretation

O0013
N0005 G53
N0010 T0303

N0010 Sequence number
T0303 Select tool number 303

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.0 Z0.0 S500 M04

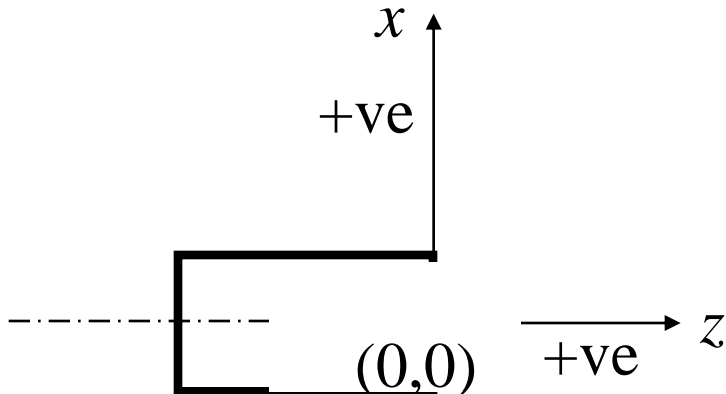**G57 To set the working zero point as saved**
G00 Rapid movement (no cutting)
X26.0 *X* **location (as a diameter; 13 form zero)**
Z0.0 *Z* **location**
S500 Spindle speed is 500 rpm
M04 Rotate spindle counterclockwise

x

+ve

+ve   *z*

(0,0)

APT Program Interpretation

O0013

N0005 G53
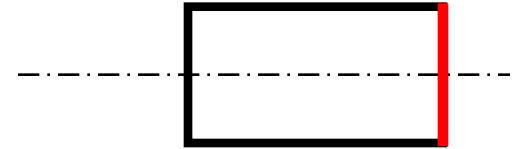
N0010 T0404

N0020 G57 G00 X26.00 Z0.0 S500 M04

N0030 G01 X-0.20 F100

G01 Linear interpolation (cutting)

X-0.20 Move only in x direction until you pass

       the center by 0.1 mm (facing)

F100 Set feed rate to 100 mm/min.

APT Program Interpretation

O0013
N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0

G00 Move rapidly away from work piece (no cutting)
Z2.0 the movement is 2 mm away from the face.

APT Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0

Go to a safe location away from the workpiece [x = 50 (25 from zero), z = 50] to change the tool.

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404

T0404 Select tool number 404

APT Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X22.50 Z2.0 S500

G57 PS0
G00 Rapid movement (no cutting)
X22.50 *X* **location (as a diameter; 11.25 form zero)**
Z2.0 *Z* **location**
S500 Spindle speed is 500 rpm

APT Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 Z-30.0 F100

G01 Linear interpolation (cutting)
Z-30 Move only in z direction (external turning)
F100 Set feed rate to 100 mm/min.

APT Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
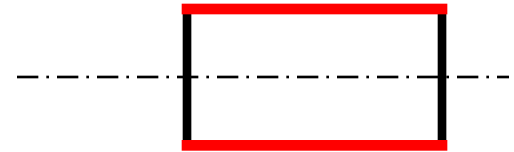N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X23.0 Z2.0 S500

G00 Move rapidly away from work piece (no cutting) to location x= 23.0 (11.50 from zero) and z = 2.0.

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D0=200 D2=200 D3=650

G84 Turning cycle for machining the step
X17.5 final diameter
Z-20 length of step is 20 mm
D0=200 Finish allowance in X direction (0.2 mm) D2=200 Finish allowance in Z direction (0.2 mm)
D3=650 Depth of cut in each pass (0.65 mm)

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
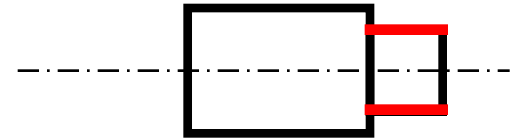N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
N0110 G00 Z2.0

G00 Move rapidly away from workpiece (no cutting)
Z2.0 the movement is 2 mm away from the face.

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
N0110 G00 Z2.0
N0120 X50.0 Z50.0

X50.0 Z50.0 Move to the tool changing location

APT Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
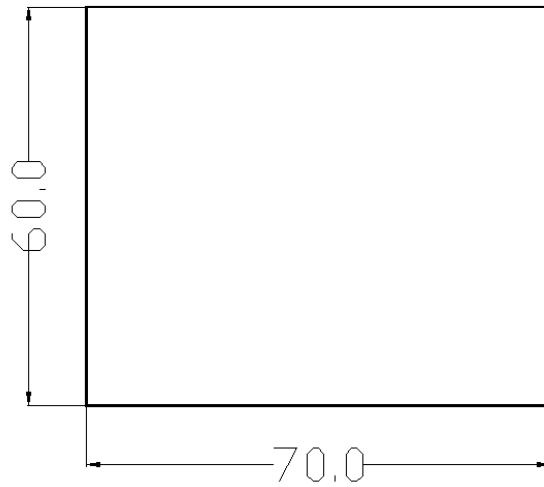N0110 G00 Z2.0
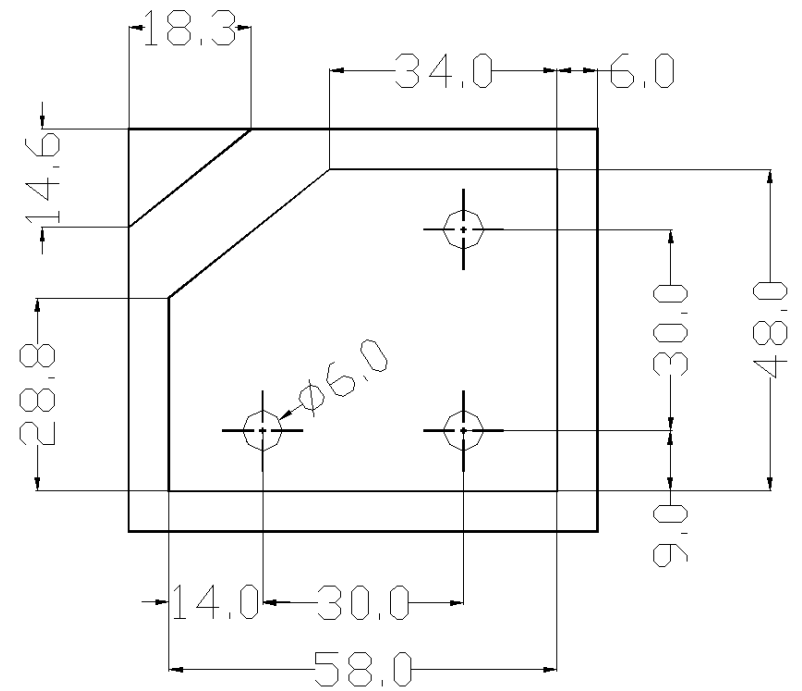N0120 X50.0 Z50.0 T00
N0130 M30

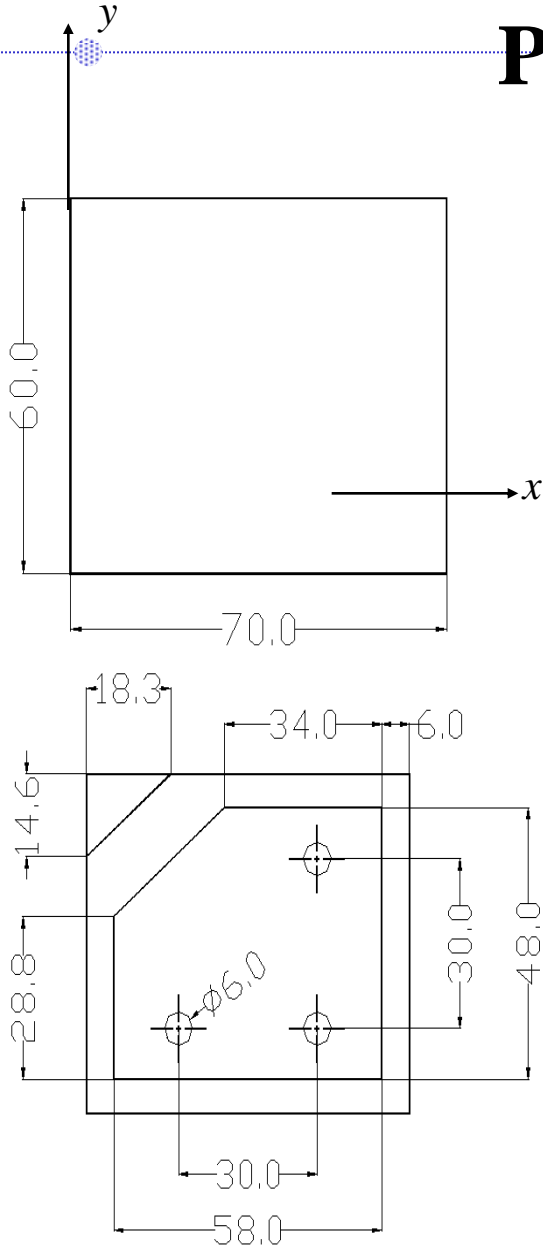M30 Program End

Programming Example

Raw Material

Finished Part

Programming Example

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
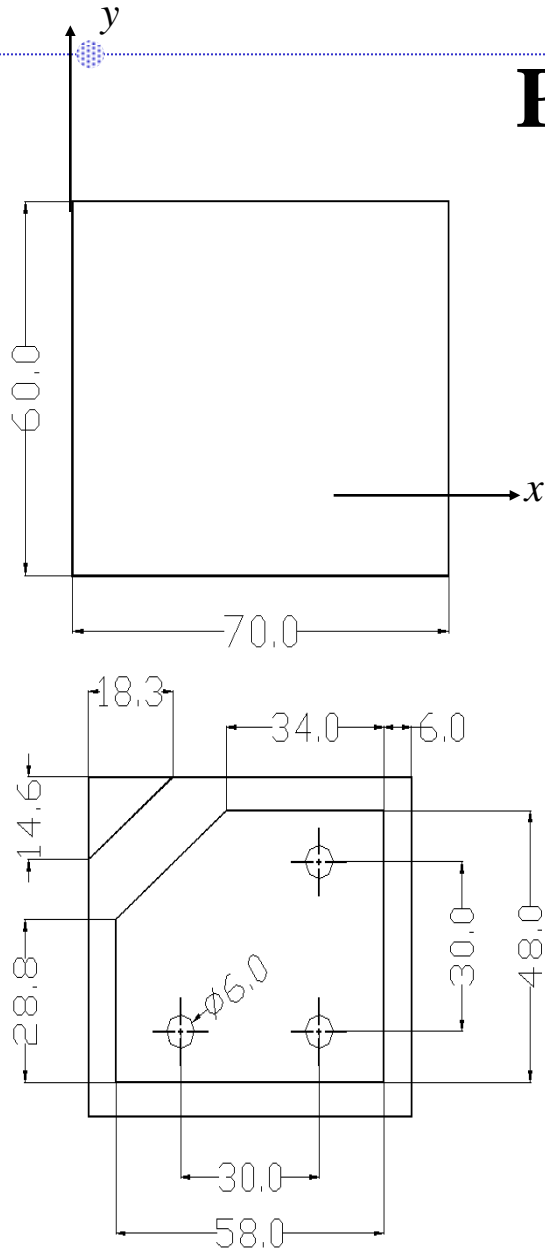N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02

Programming Example

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
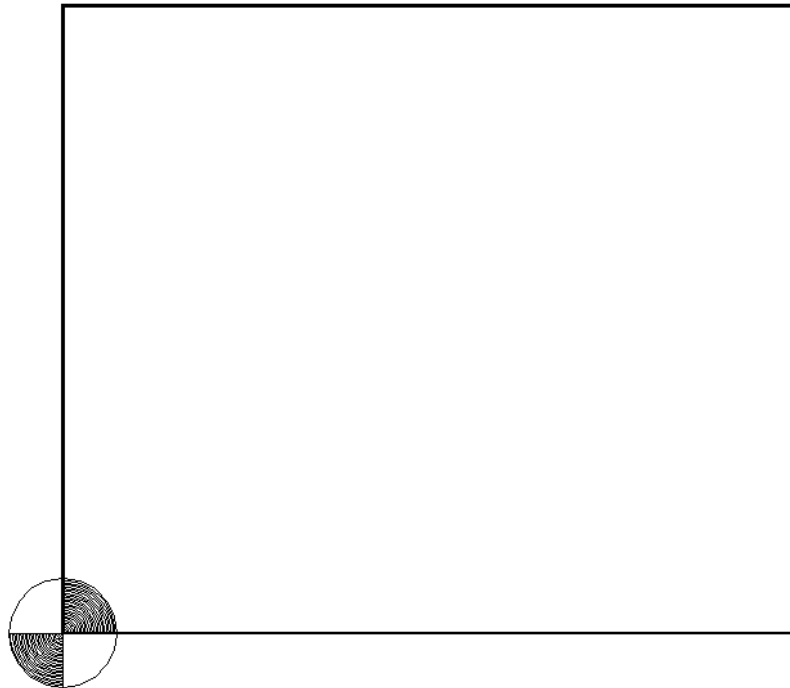**N012 G01 X50 Y45 Z10 ZFeed 150**
N013 M05
N014 M02

Program Interpretation

G55 X200 Y80

Setting the datum to the lower left corner of the work piece

Program Interpretation

G55 X200 Y80
Program 1

Program Identification Number

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1

N001 Sequence Number
M06   Tool Change (End Mill with
Diameter=12mm
T1   Tool Number

Program Interpretation

G55 X200 **Y80**
Program 1
N001 M06 T1
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

G55 X200 Y80

Program 1

N001 M06 T1

N002 M03 rpm 400

N003 G01 X-8 Y0 Z0 XYFeed 150

N004 G01 X-8 Y0 Z-0.5 ZFeed 150

Lower the end mill to determine the depth of cut

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75

Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min

Program Interpretation

G55 X200 Y80
Program 1
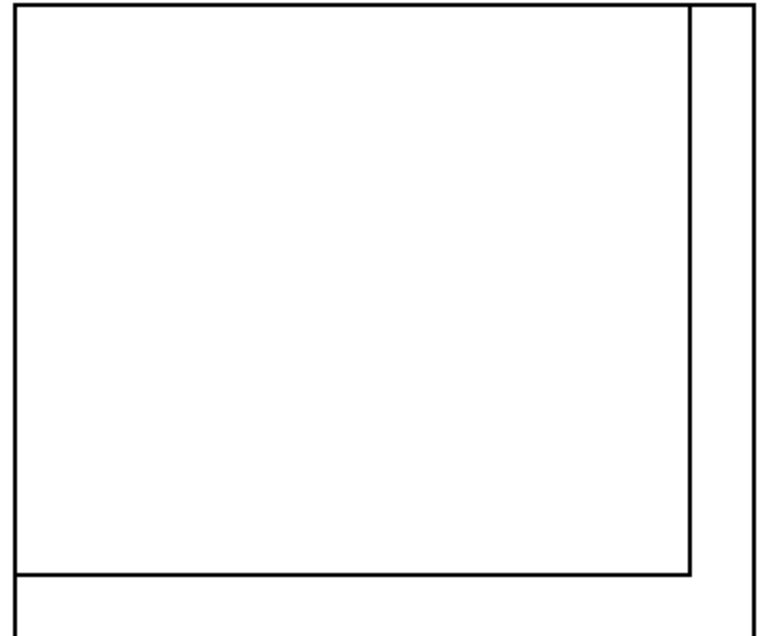N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75

Move from the lower left corner of the
work piece to the right lower one
cutting with feed=75mm/min

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
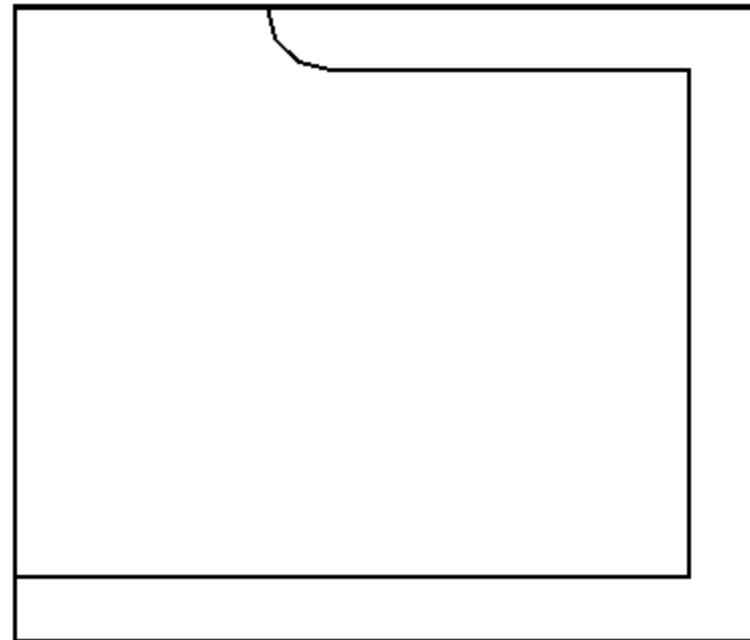N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75

Cutting the horizontally up to X=30

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
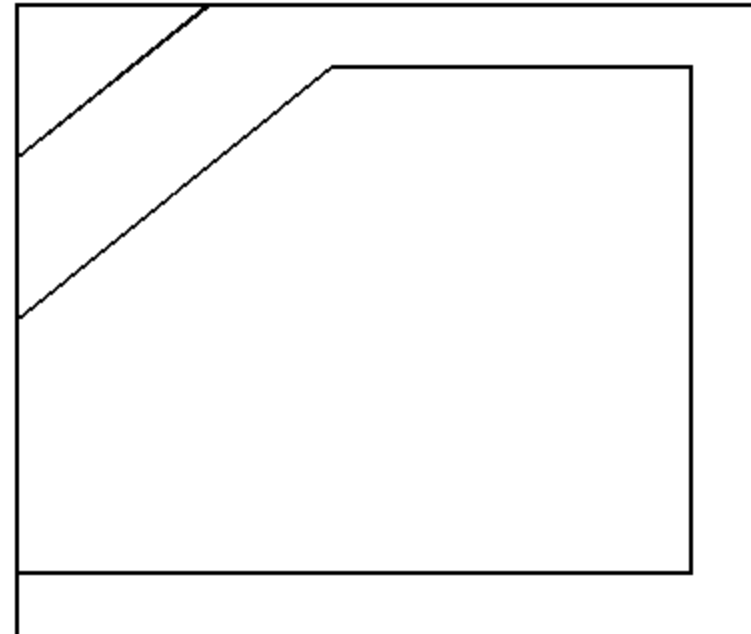N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75

Cutting to X=0 & Y=40

Program Interpretation

G55 X200 Y80

Program 1

N001 M06 T1

N002 M03 rpm 400

N003 G01 X-8 Y0 Z0 XYFeed 150

N004 G01 X-8 Y0 Z-0.5 ZFeed 150

N005 G01 X70 Y0 Z-0.5 XYFeed 75

N006 G01 X70 Y60 Z-0.5 XYFeed 75

N007 G01 X30 Y60 Z-0.5 XYFeed 75

N008 G01 X0 Y40 Z-0.5 XYFeed 75

N009 G01 X0 Y0 Z-0.5 XYFeed 75

Complete the countering

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5

Repeat 7 times blocks from N003 to N009
with incremental offset of Z=-0.5

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
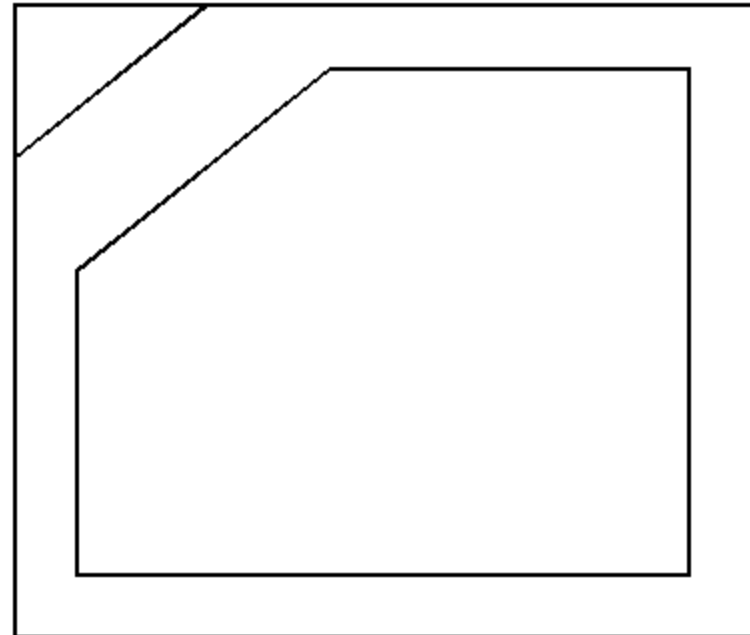N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05

Spindle Off

Program Interpretation

G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02

End Program

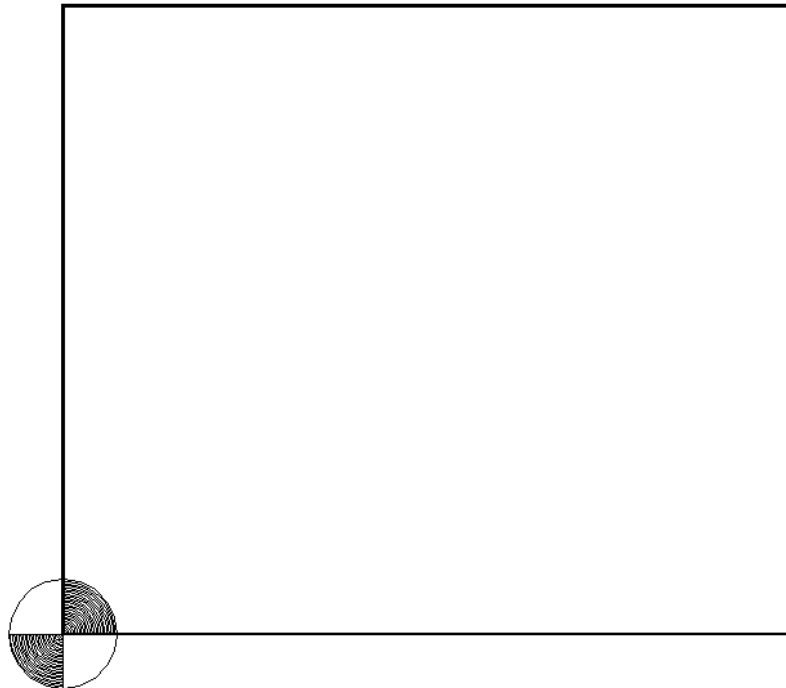# Program Interpretation

Tool Change

Changing the tool

Program Interpretation

Tool Change

G55 X200 Y80

Setting the datum to the lower left corner of the work piece

Program Interpretation

Tool Change
G55 X200 Y80
Program 2

Program Identification Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2

N001 Sequence Number
M06   Tool Change (Drill with
Diameter=6mm
T2   Tool Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

Stop above the center of the first hole

Program Interpretation

Tool Change

G55 X200 Y80

Program 2

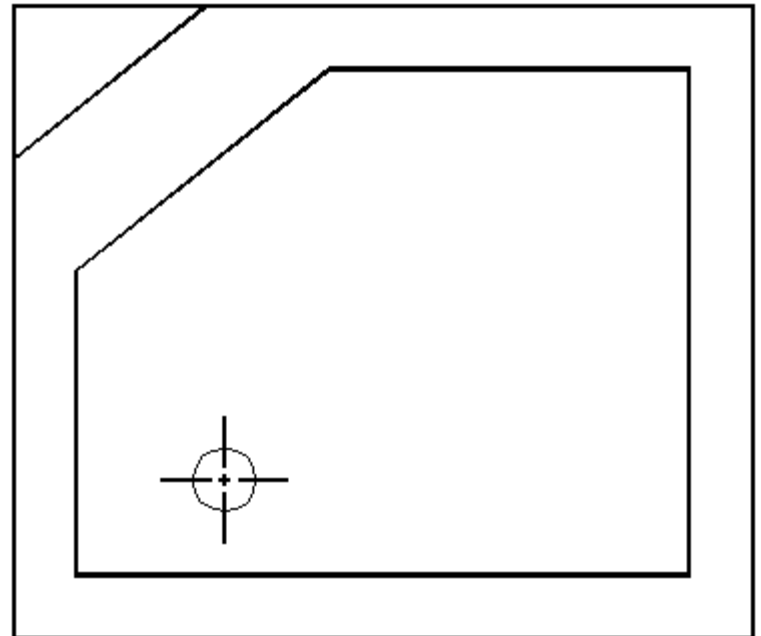**N001 M06 T2**

N002 M03 rpm 400

N003 G01 X-8 Y0 Z0 XYFeed 150

N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

N005 G01 X20 Y15 Z-10 ZFeed 75

Start Drill the first hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150

Retract to a position above the hole

Program Interpretation

Tool Change

G55 X200 Y80

Program 2

N001 M06 T2

N002 M03 rpm 400

N003 G01 X-8 Y0 Z0 XYFeed 150

N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

N005 G01 X20 Y15 Z-10 ZFeed 75

N006 G01 X20 Y15 Z10 ZFeed 150

N007 G01 X50 Y15 Z10 ZFeed 150

Stop above the center of the second hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75

Drill the second hole

Program Interpretation

Tool Change

G55 X200 Y80

Program 2

N001 M06 T2

N002 M03 rpm 400

N003 G01 X-8 Y0 Z0 XYFeed 150
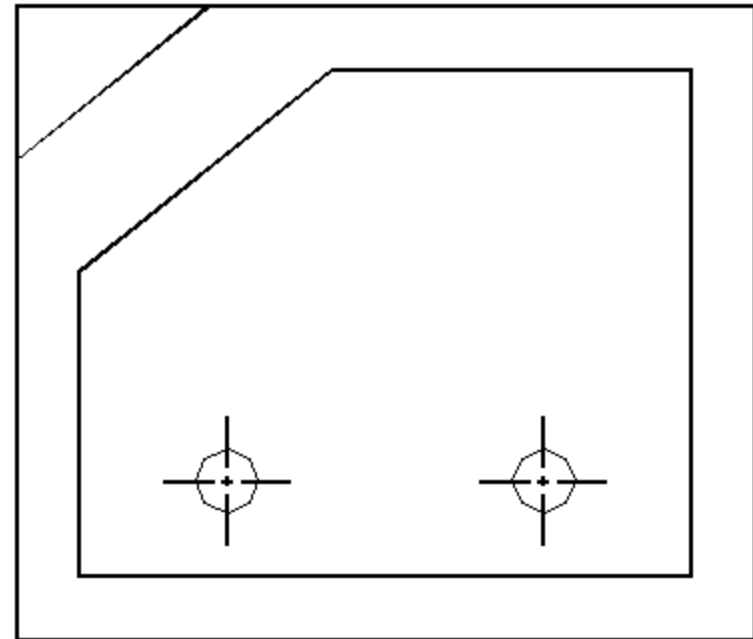
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

N005 G01 X20 Y15 Z-10 ZFeed 75

N006 G01 X20 Y15 Z10 ZFeed 150

N007 G01 X50 Y15 Z10 ZFeed 150

N008 G01 X50 Y15 Z-10 ZFeed 75

N009 G01 X50 Y15 Z10 ZFeed 150

Retract to a position above the second hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150

Stop above the center of the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
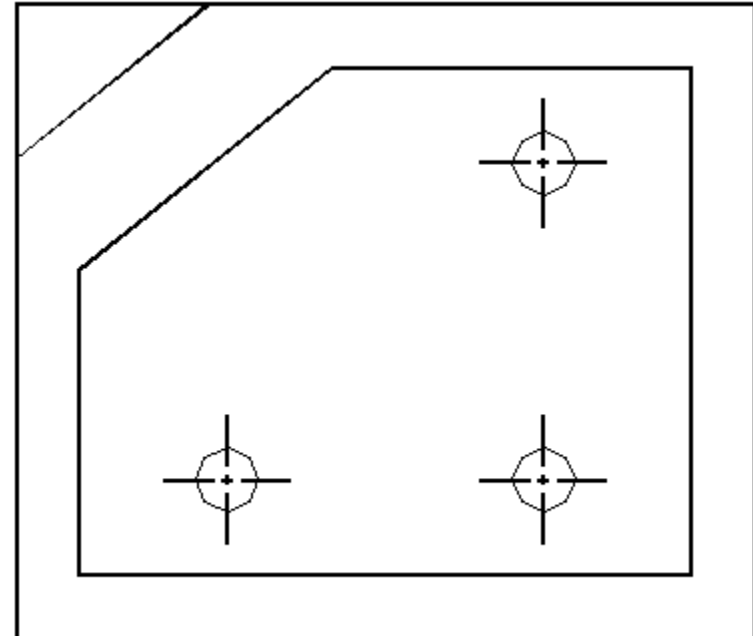**N006 G01 X20 Y15 Z10 ZFeed 150**
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75

Drill the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150

Retract to a position above the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
Spindle off

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02
End Program

Program Interpretation

Tool Change

G55 X200 Y80

Program 2

N001 M06 T2

N002 M03 rpm 400

N003 G01 X-8 Y0 Z0 XYFeed 150

N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

N005 G01 X20 Y15 Z-10 ZFeed 75

N006 G01 X20 Y15 Z10 ZFeed 150

N007 G01 X50 Y15 Z10 ZFeed 150

N008 G01 X50 Y15 Z-10 ZFeed 75

N009 G01 X50 Y15 Z10 ZFeed 150

N010 G01 X50 Y45 Z10 ZFeed 150

N011 G01 X50 Y45 Z-10 ZFeed 75

N012 G01 X50 Y45 Z10 ZFeed 150

N013 M05

N014 M02

End Program

CAD/CAM

Two computer-based systems which impact the use of CNC technology are computer aided design and computer aided manufacturing.

A computer aided design, or CAD, system uses computers to graphically create product designs and models. These designs can be reviewed, revised, and refined for optimum end use and application. Once finalized, the CAD design is then exported to a computer aided manufacturing, or CAM, system.
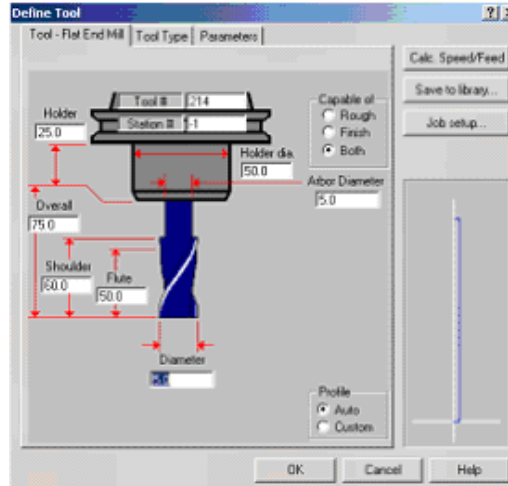
CAM systems assist in all phases of manufacturing a product, including process planning, production planning, machining, scheduling, management and quality control.
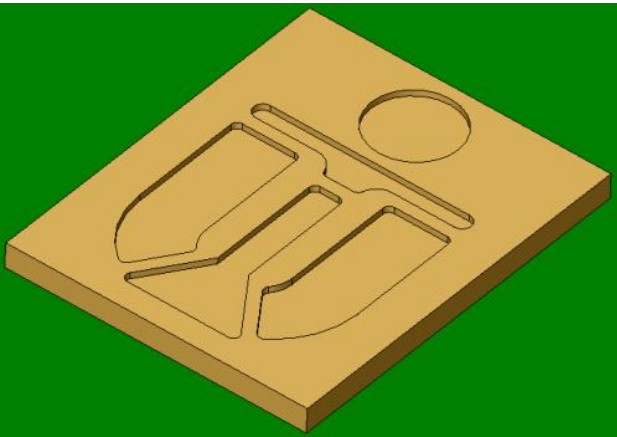
Automatic Part Programming
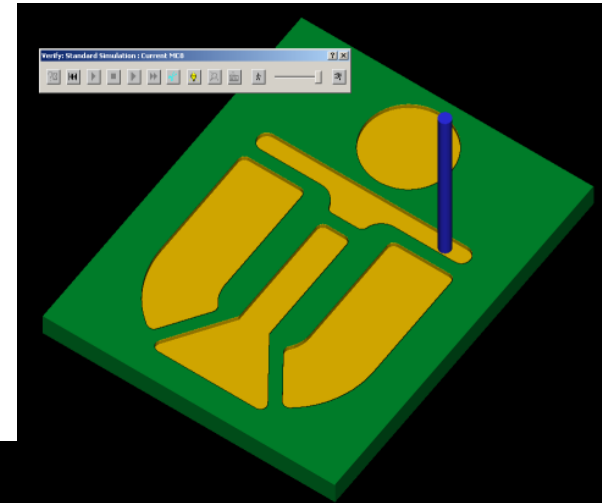
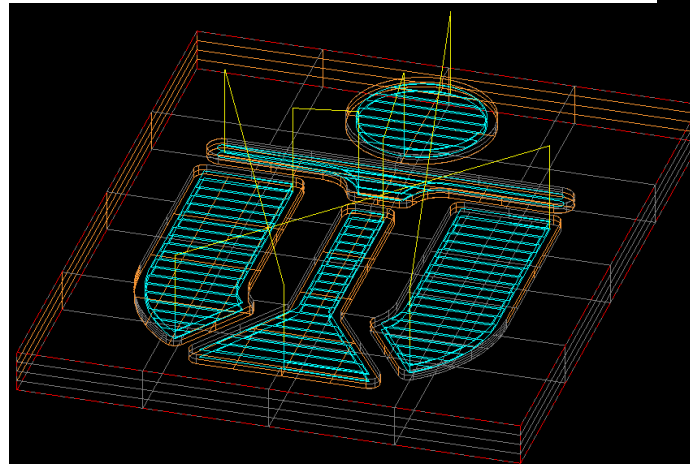Software programs can automatic generation of CNC data
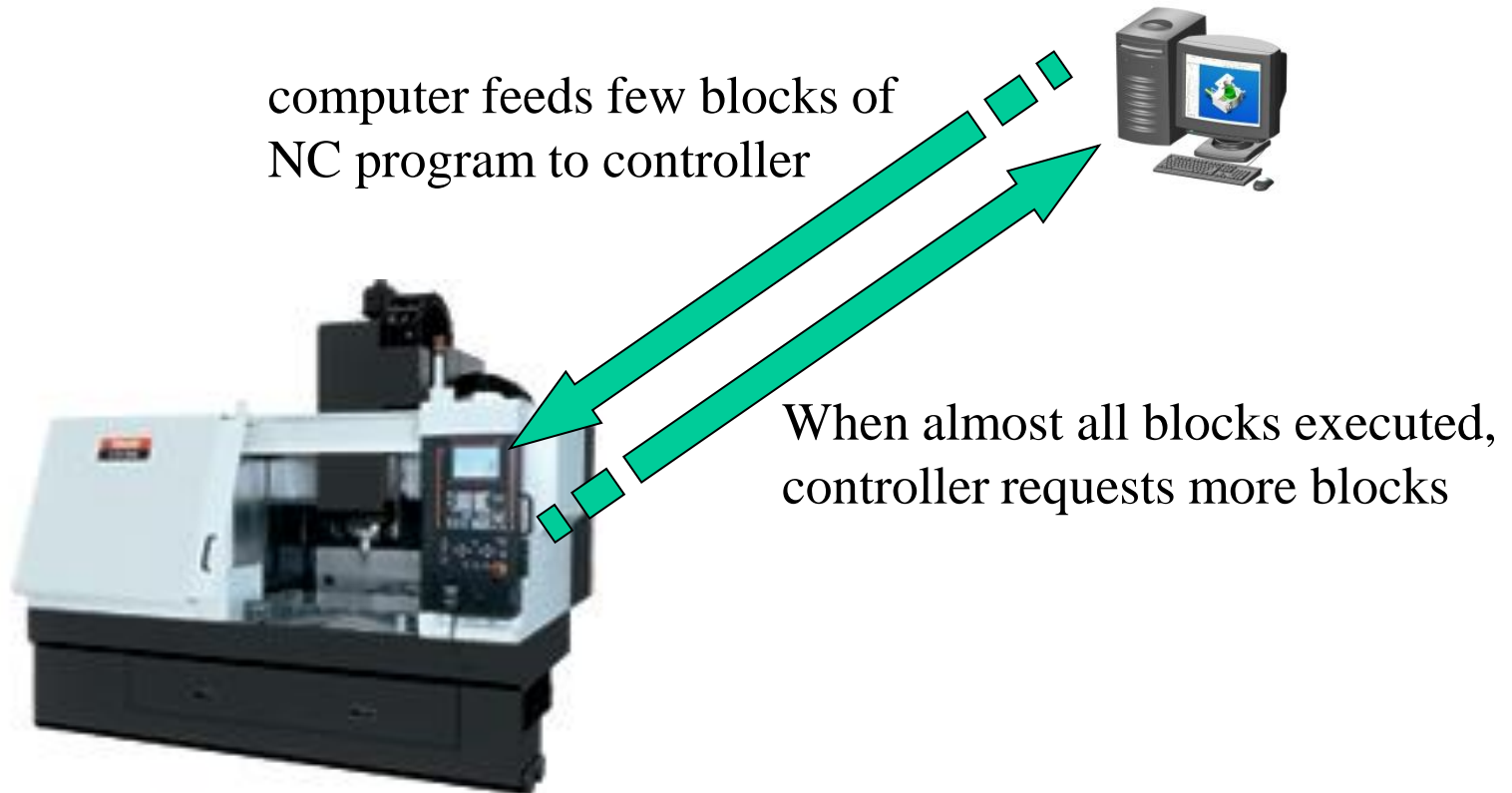
Define Tool

CNC data

Make 3D model

Simulate cutting

Automatic part programming and DNC

Very complex part shapes ➔ very large NC program

NC controller memory may not handle HUGE part program

computer feeds few blocks of
NC program to controller

When almost all blocks executed,
controller requests more blocks

Summary

CNC machines allow precise and repeatable control in machining

CNC lathes, Milling machines, etc. are all controlled by NC programs

NC programs can be generated manually, automatically

Additional references: RS274D code descriptions