UNIT-III
CNC CONTROL SYSTEM AND PART PROGRAMMING

By
P.Janagarathinam AP/MECH
Dept Mechanical Engineering

- **The part program** is a sequence of instructions, which describe the work, which has to be done on a part, in the form required by a computer under the control of a numerical control computer program.
- It is the task of preparing a program sheet from a drawing sheet. All data is fed into the numerical control system using a standardized format.
- Programming is where all the machining data are compiled and where the data are translated into a language which can be understood by the control system of the machine tool. The machining data is as follows :

(a) Machining sequence classification of process, tool start up point, cutting depth, tool path, etc.
(b) Cutting conditions, spindle speed, feed rate, coolant, etc.
(c) Selection of cutting tools.

- While preparing a part program, need to perform the following steps :

(a) Determine the startup procedure, which includes the extraction of dimensional data from part drawings and data regarding surface quality requirements on the machined component.
(b) Select the tool and determine the tool offset.
(c) Set up the zero position for the work piece.
(d) Select the speed and rotation of the spindle.
(e) Set up the tool motions according to the profile required.
(f) Return the cutting tool to the reference point after completion of work.
(g) End the program by stopping the spindle and coolant

- The part programming contains the list of coordinate values along the *X*, *Y* and *Z* directions of the entire tool path to finish the component. The program should also contain information, such as feed and speed.
- Each of the necessary instructions for a particular operation given in the part program is known as an NC word. A group of such NC words constitutes a complete NC instruction, known as block.
- The commonly used words are *N*, *G*, *F*, *S*, *T*, and *M*. The same is explained later on through examples.

Hence the methods of part programming can be of two types depending upon the two techniques as below :

(a) Manual part programming, and

(b) Computer aided part programming.

MANUAL PART PROGRAMMING

- The programmer first prepares the program manuscript in a standard format. Manuscripts are typed with a device known as flexo writer, which is also used to type the program instructions.
- After the program is typed, the punched tape is prepared on the flexo writer. Complex shaped components require tedious calculations.
- This type of programming is carried out for simple machining parts produced on point-to-point machine tool.

To be able to create a part program manually, need the following information:

(a) Knowledge about various manufacturing processes and machines.
(b) Sequence of operations to be performed for a given component.
(c) Knowledge of the selection of cutting parameters.
(d) Editing the part program according to the design changes.
(e) Knowledge about the codes and functions used in part programs.

Computer Aided Part Programming

- If the complex-shaped component requires calculations to produce the component are done by the programming software contained in the computer.
- The programmer communicates with this system through the system language, which is based on words.
- There are various programming languages developed in the recent past, such as APT (Automatically Programmed Tools), ADAPT, AUTOSPOT, COMPAT-II, 2CL, ROMANCE, SPLIT is used for writing a computer programme, which has English like statements. A translator known as compiler program is used to translate it in a form acceptable to MCU.

Standard G and M Codes

- The most common codes used when programming NC machines tools are G-codes (preparatory functions), and M codes (miscellaneous functions). Other codes such as *F*, *S*, *D*, and *T* are used for machine functions such as feed, speed, cutter diameter offset, tool number, etc.
- G-codes are sometimes called cycle codes because they refer to some action occurring on the *X*, *Y*, and/or *Z*-axis of a machine tool. The G-codes are grouped into categories such as Group 01, containing codes G00, G01, G02, G03, which cause some movement of the machine table or head. Group 03 includes either absolute or incremental programming.
- A G00 code rapidly positions the cutting tool while it is above the workpiece from one point to another point on a job. During the rapid traverse movement, either the *X* or *Y*-axis can be moved individually or both axes can be moved at the same time. The rate of rapid travel varies from machine to machine.

G-Codes (Preparatory Functions)

Code Function

G00 Rapid positioning

G01 Linear interpolation

G02 Circular interpolation clockwise (CW)

G03 Circular interpolation counterclockwise (CCW)

G20 Inch input (in.)

G21 Metric input (mm)

G24 Radius programming

G28 Return to reference point

G29 Return from reference point

G32 Thread cutting

G40 Cutter compensation cancel

G41 Cutter compensation left
G42 Cutter compensation right
G43 Tool length compensation positive (+) direction
G44 Tool length compensation minus (-) direction
G49 Tool length compensation cancels
G 53 Zero offset or M/c reference
G54 Settable zero offset
G84 canned turn cycle
G90 Absolute programming
G91 Incremental programming
Note : On some machines and controls, some may be differ.

M-Codes (Miscellaneous Functions)

- M or miscellaneous codes are used to either turn ON or OFF different functions, which control certain machine tool operations.
- M-codes are not grouped into categories, although several codes may control the same type of operations such as M03, M04, and M05, which control the machine tool spindle. Some of important codes are given as under with their functions:

Code Function
M00 Program stop
M02 End of program
M03 Spindle start (forward CW)
M04 Spindle start (reverse CCW)
M05 Spindle stop
M06 Tool change
M08 Coolant on
M09 Coolant off

M10 Chuck - clamping
M11 Chuck - unclamping
M12 Tailstock spindle out
M13 Tailstock spindle in
M17 Tool post rotation normal
M18 Tool post rotation reverse
M30 End of tape and rewind or main program end
M98 Transfer to subprogram
M99 End of subprogram
Note : On some machines and controls, some may be differ.

NC and CNC machines and Control Programming

Introduction to NC and CNC machines

CNC controls and RS274 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

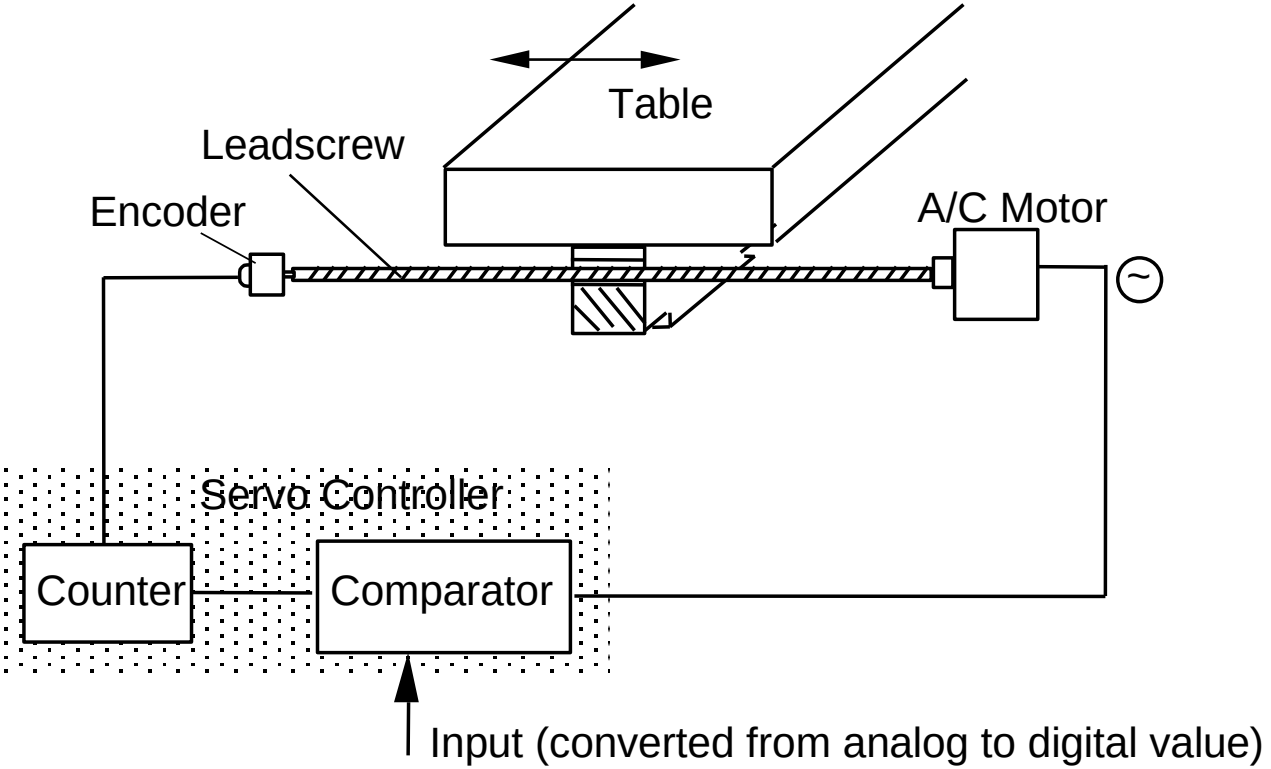
Unmanned machining operations

Advantages of CNC

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

NC machines

Motion control is done by: servo-controlled motors



CNC terminology

BLU: basic length unit ➔
smallest programmable move of each axis.

Controller: (Machine Control Unit, MCU) ➔
Electronic and computerized interface between operator and m/c

Controller components:

1. Data Processing Unit (DPU)
2. Control-Loops Unit (CLU)

Controller components

Data Processing Unit:

Input device [RS-232 port/ Tape Reader/ Punched Tape Reader]

Data Reading Circuits and Parity Checking Circuits

Decoders to distribute data to the axes controllers.

Control Loops Unit:

Interpolator to supply machine-motion commands between data points

Position control loop hardware for each axis of motion

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

Motion Control and feedback

Encoder outputs: electrical pulses (e.g. 500 pulses per revolution)

Rotation of the motor → linear motion of the table: by the **leadscrew**

The **pitch** of the leadscrew: horizontal distance between successive threads

One thread in a screw → **single start screw**: Dist moved in 1 rev = pitch

Two threads in screw → **double start screw**: Dist moved in 1 rev = 2* pitch

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)

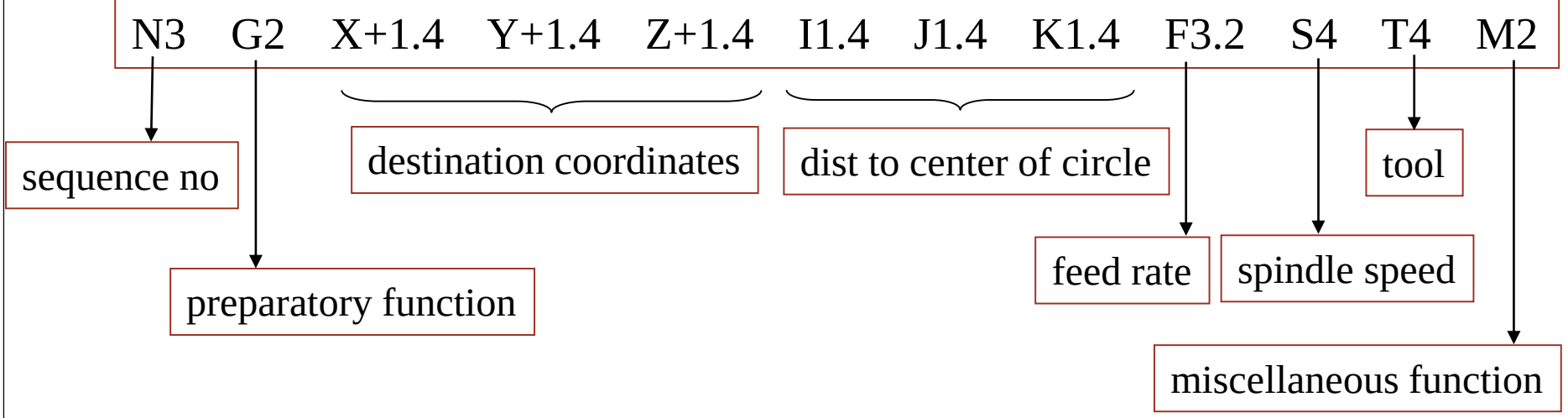
History of CNC

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

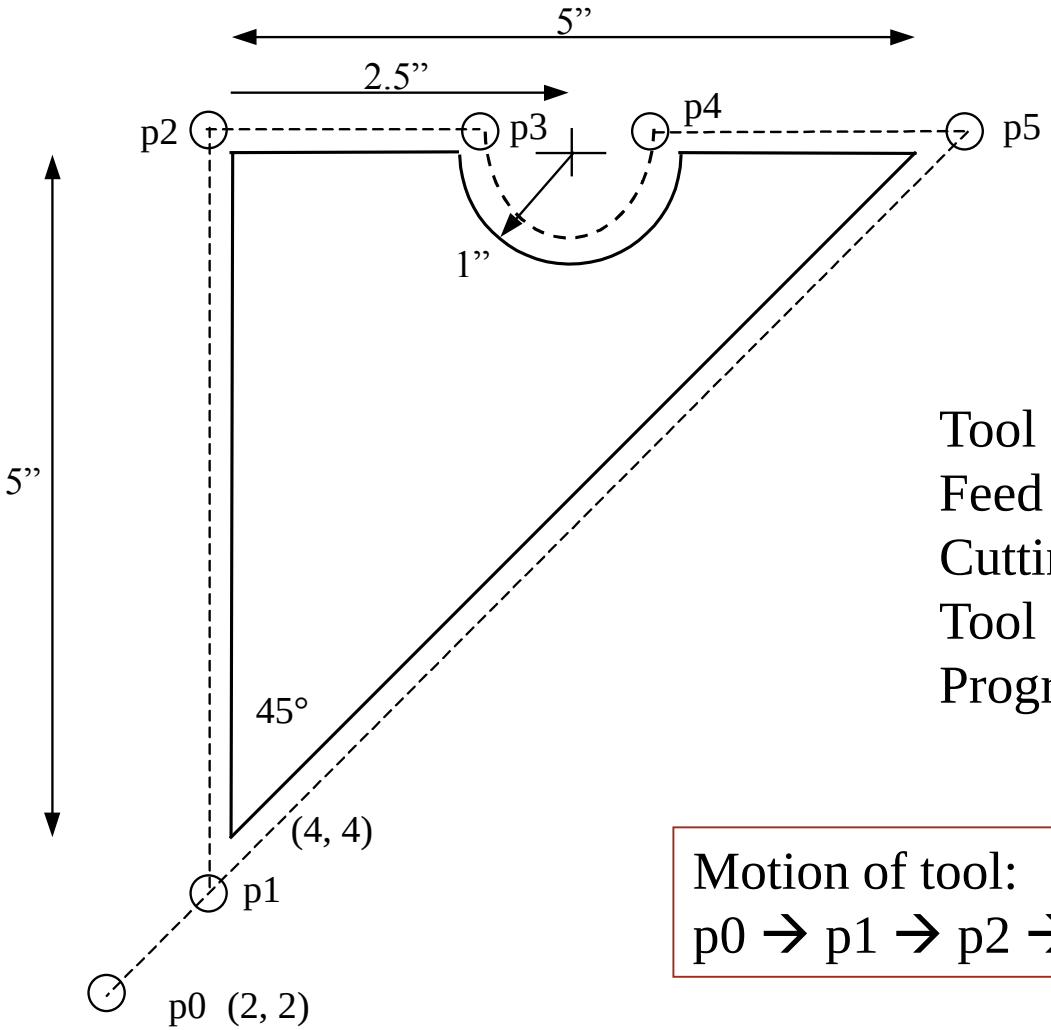
Each line of program == 1 **block**

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

Sequence and format of words:



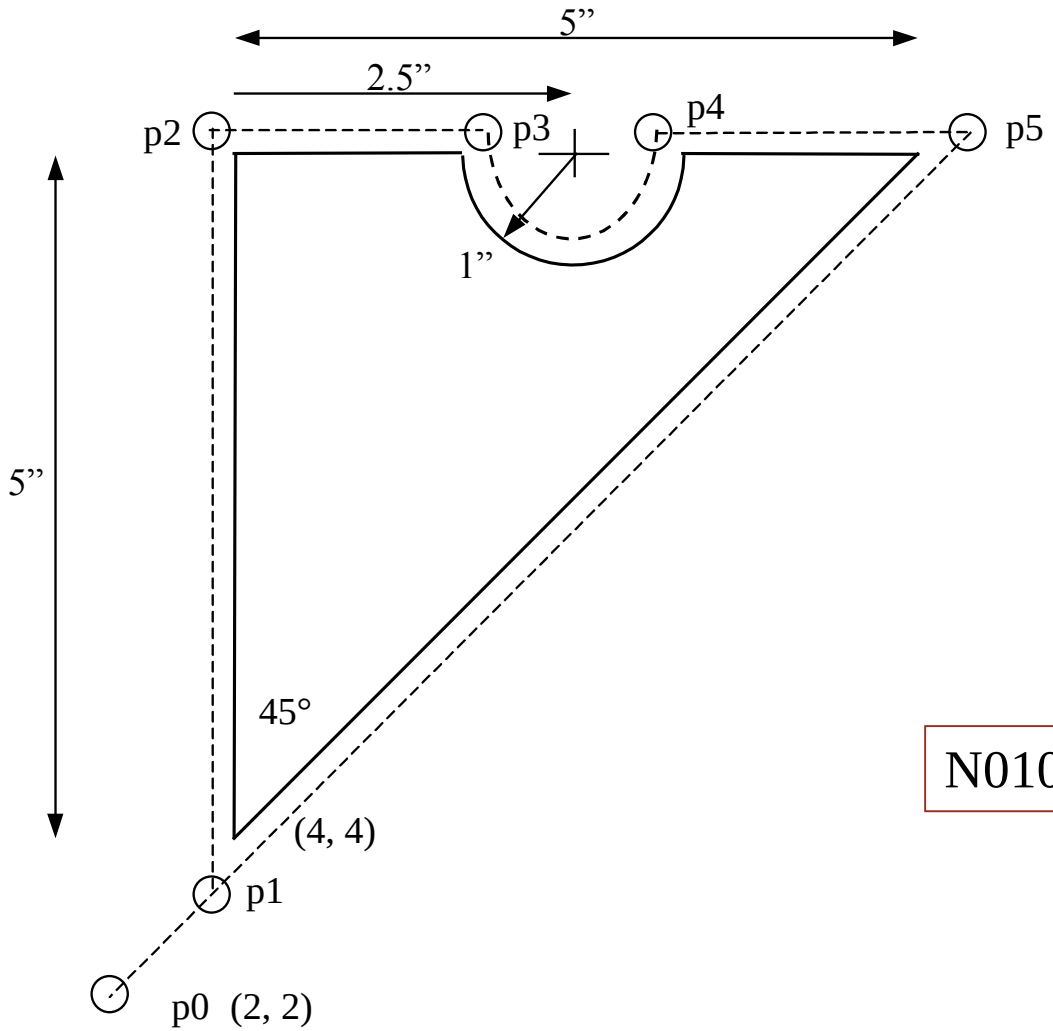
Manual Part Programming Example



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



Programming in inches

Use absolute coordinates

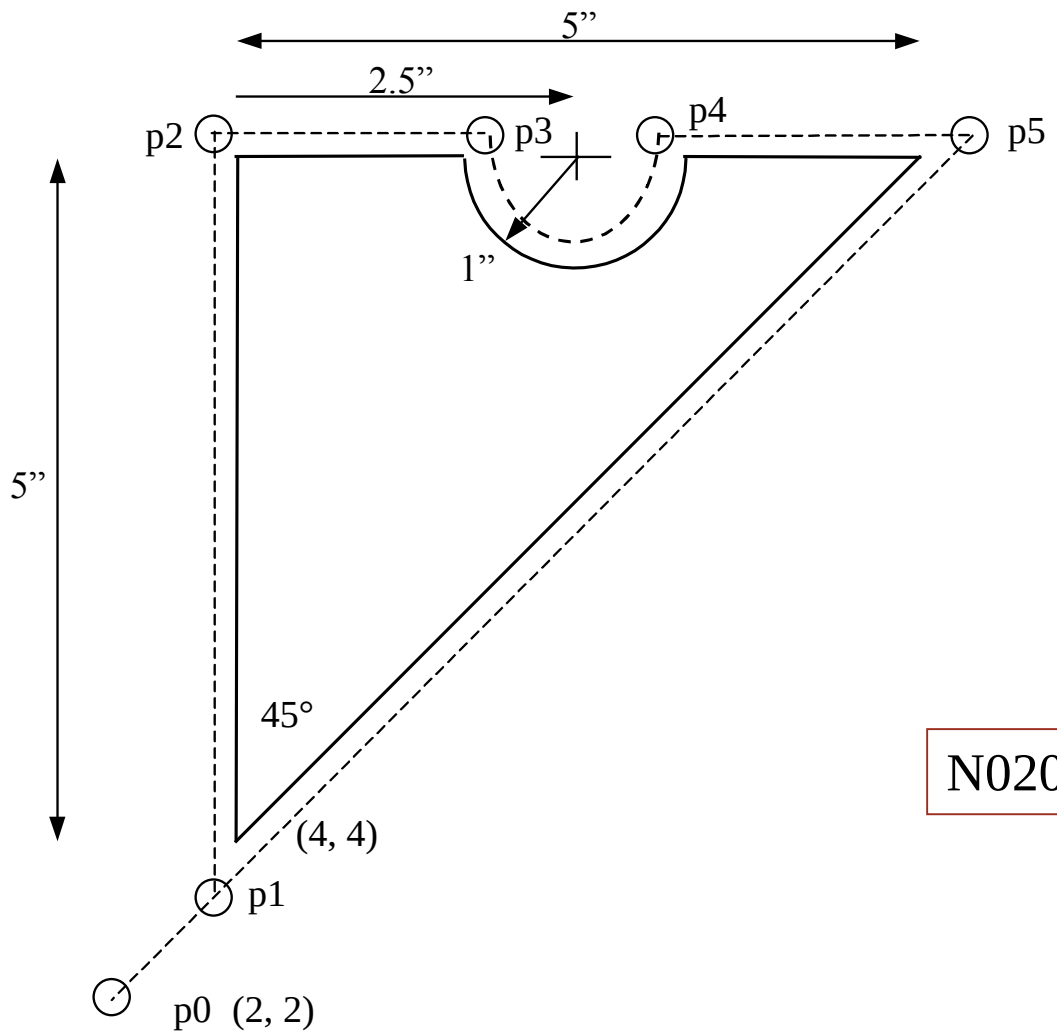
Feed in ipm

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

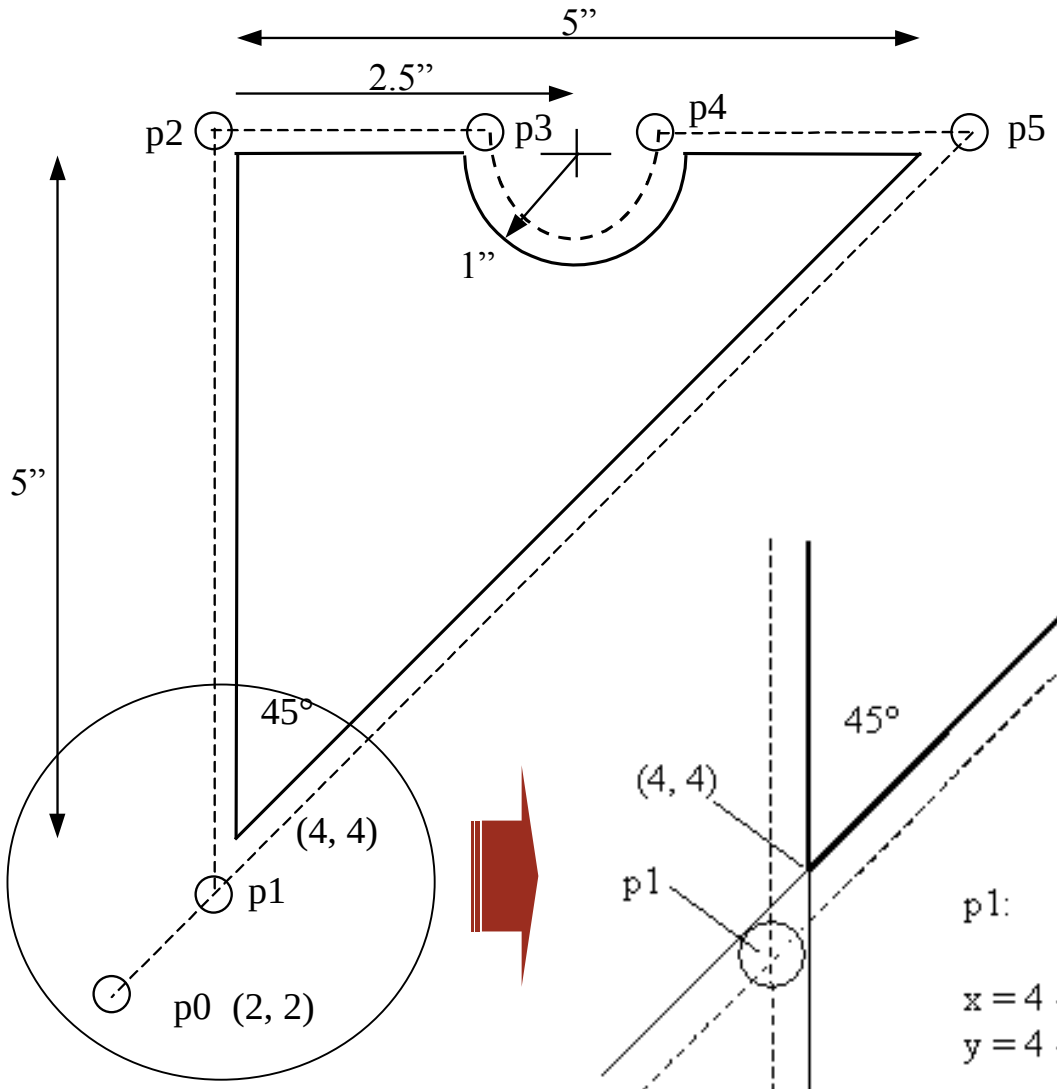
Spindle speed

N020 G17 G75 F6.0 S300 T1001 M08

Tool no.

Flood coolant ON

3. Move tool from p0 to p1 in straight line

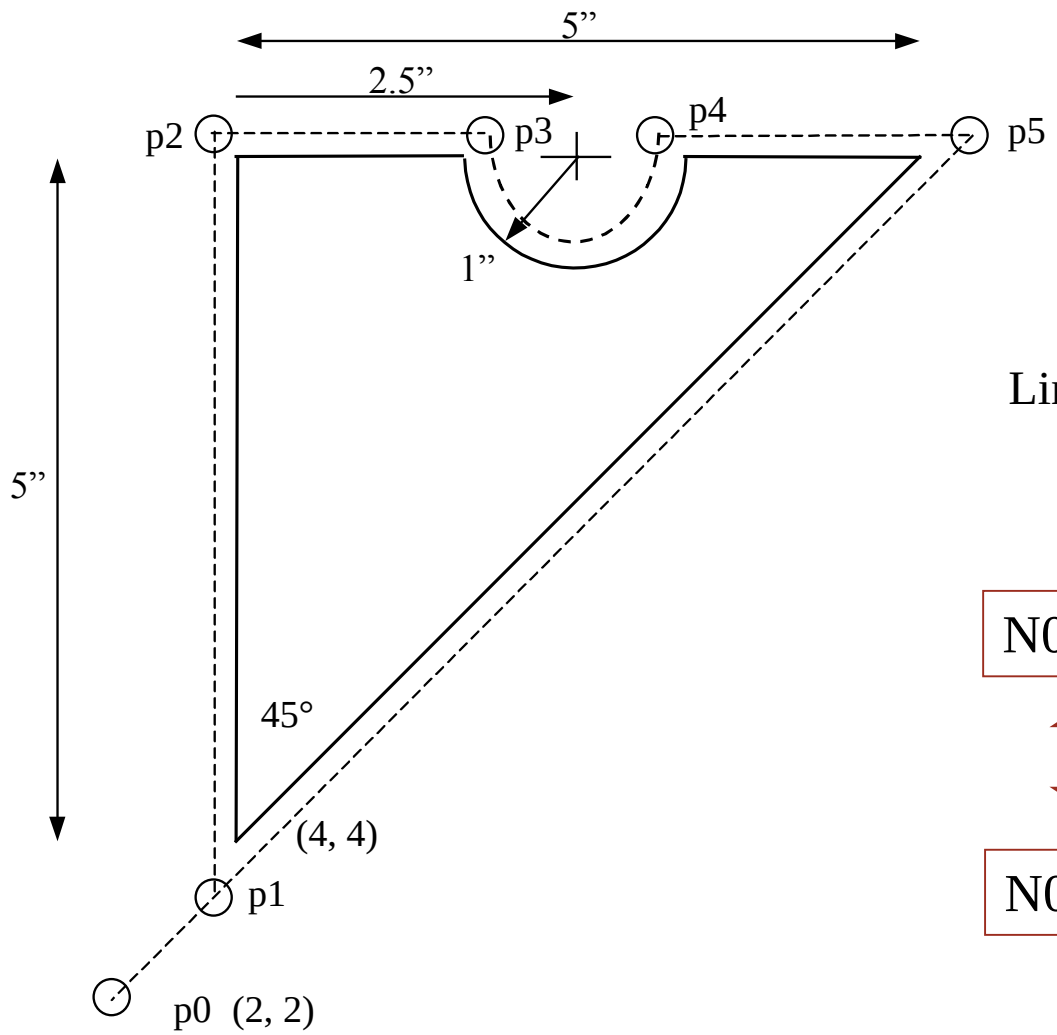


Linear interpolation

target coordinates

N030 G01 X3.875 Y3.698

4. Cut profile from p1 to p2



Linear interpolation

target coordinates

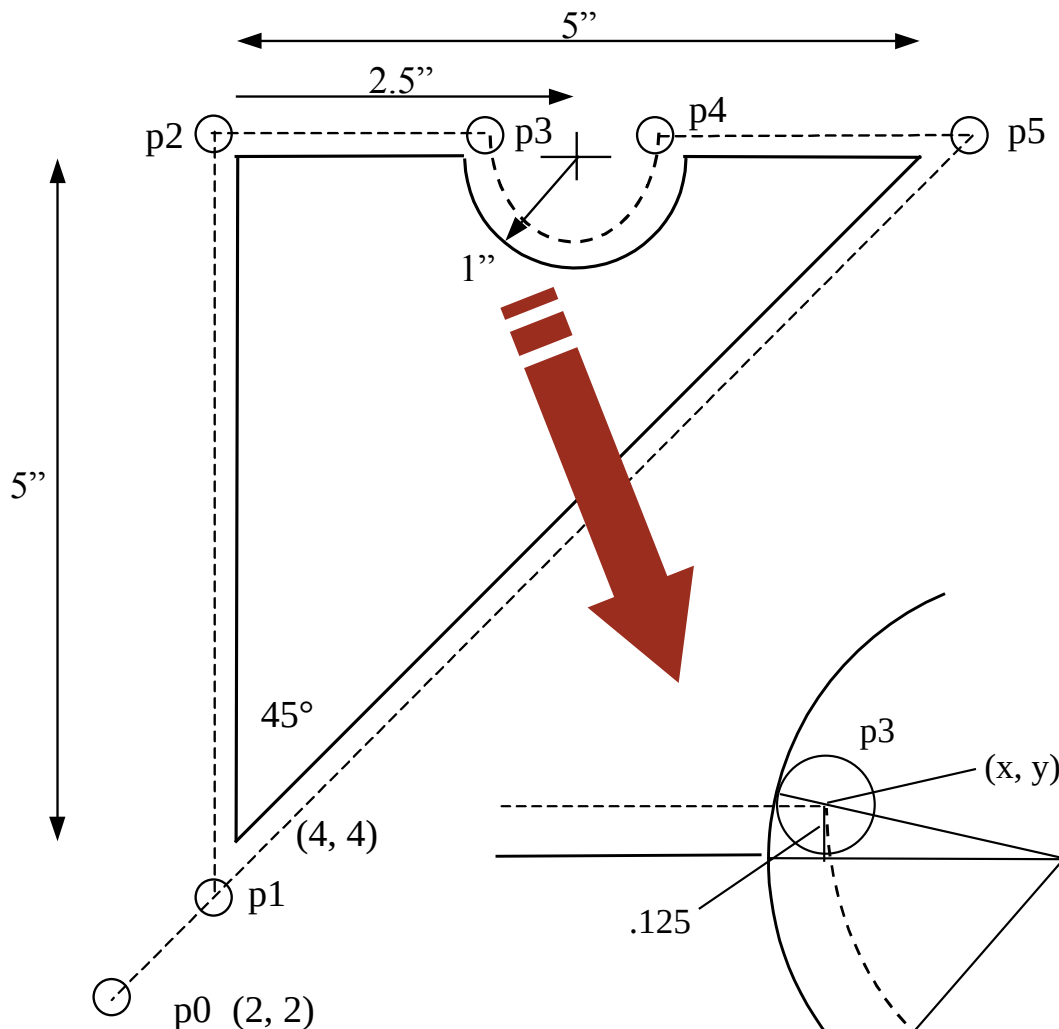
N040 G01 X3.875 Y9.125

or

N040 G01 Y9.125

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

5. Cut profile from p2 to p3



Linear interpolation

target coordinates

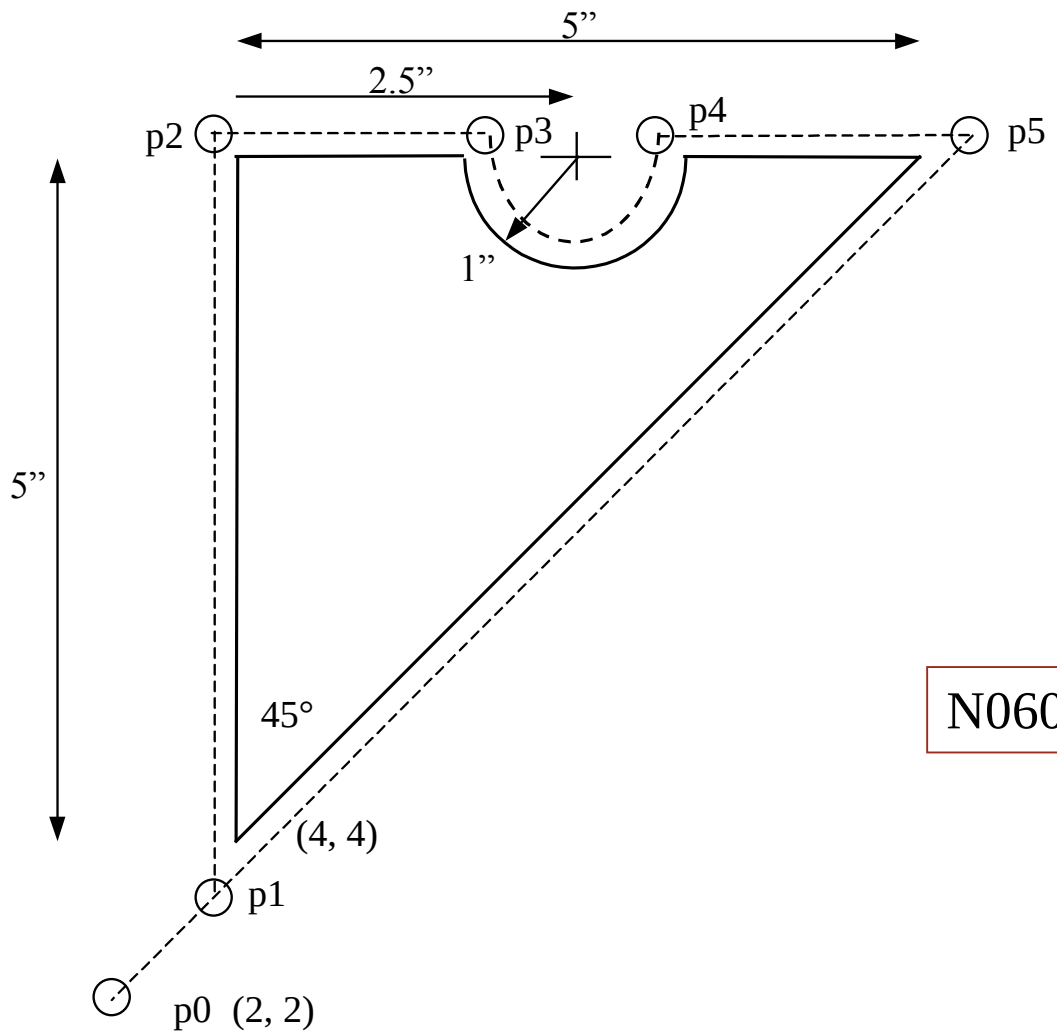
N050 G01 X5.634 Y9.125

$y = 9 + 0.125 = 9.125$

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

$x = 5.634$

6. Cut along circle from p3 to p4



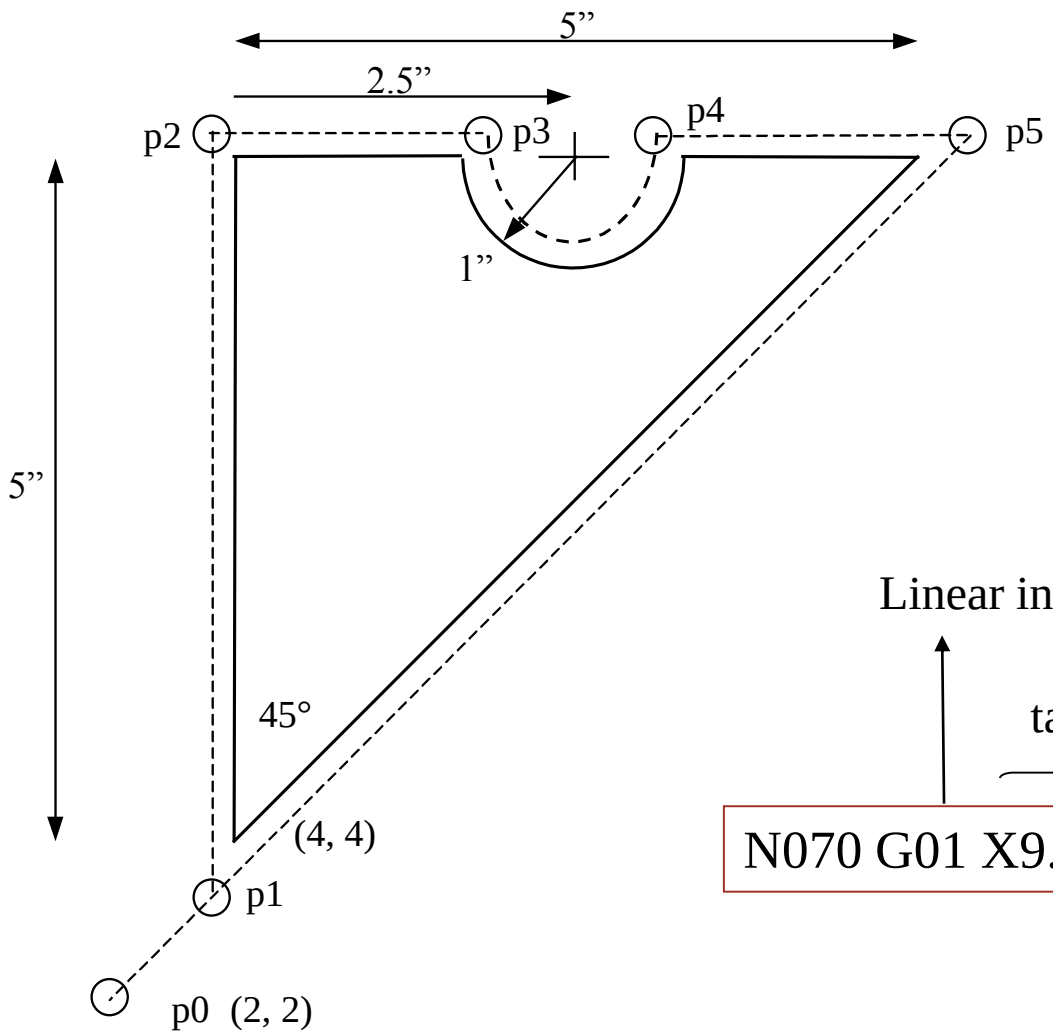
circular interpolation, CCW motion

target coordinates

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

coordinates of center of circle

7. Cut from p4 to p5

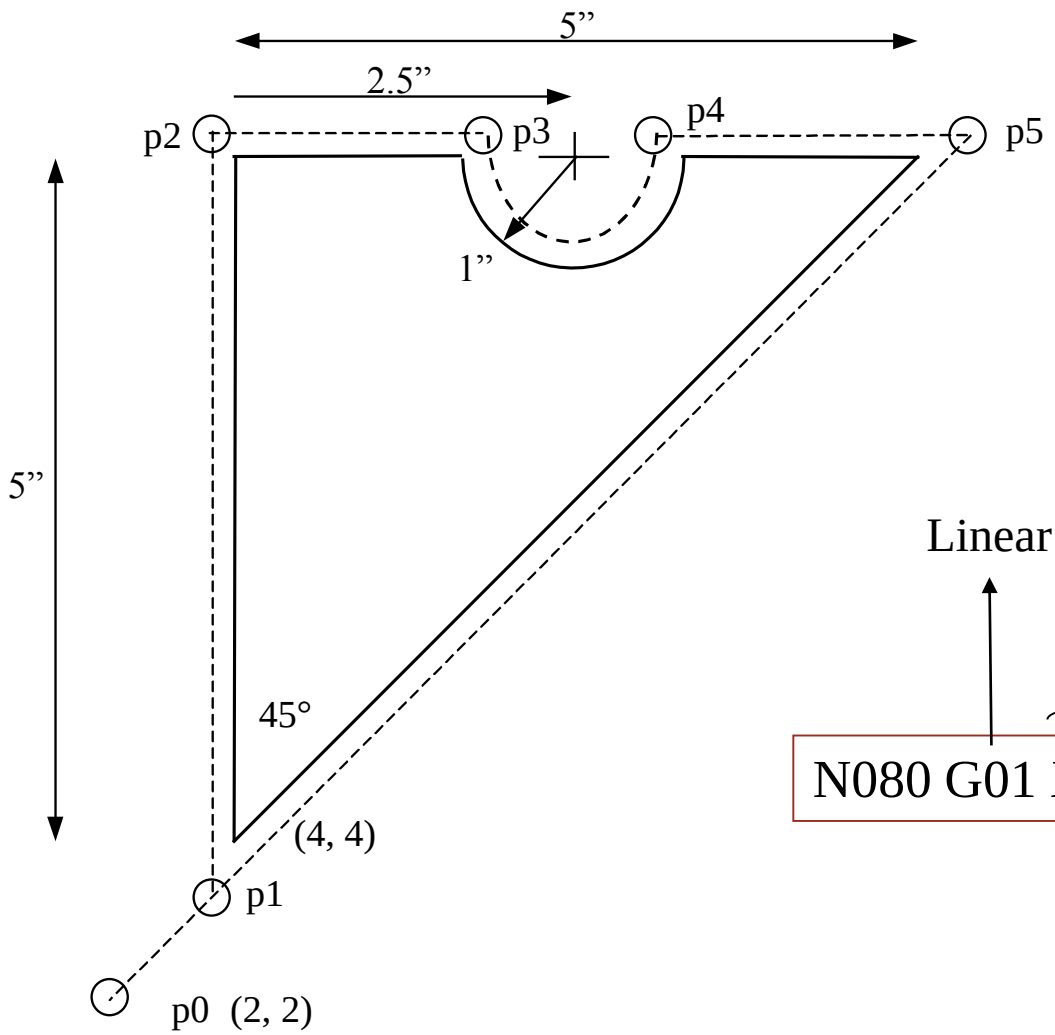


Linear interpolation

target coordinates (Y is unchanged)

N070 G01 X9.302

8. Cut from p5 to p1

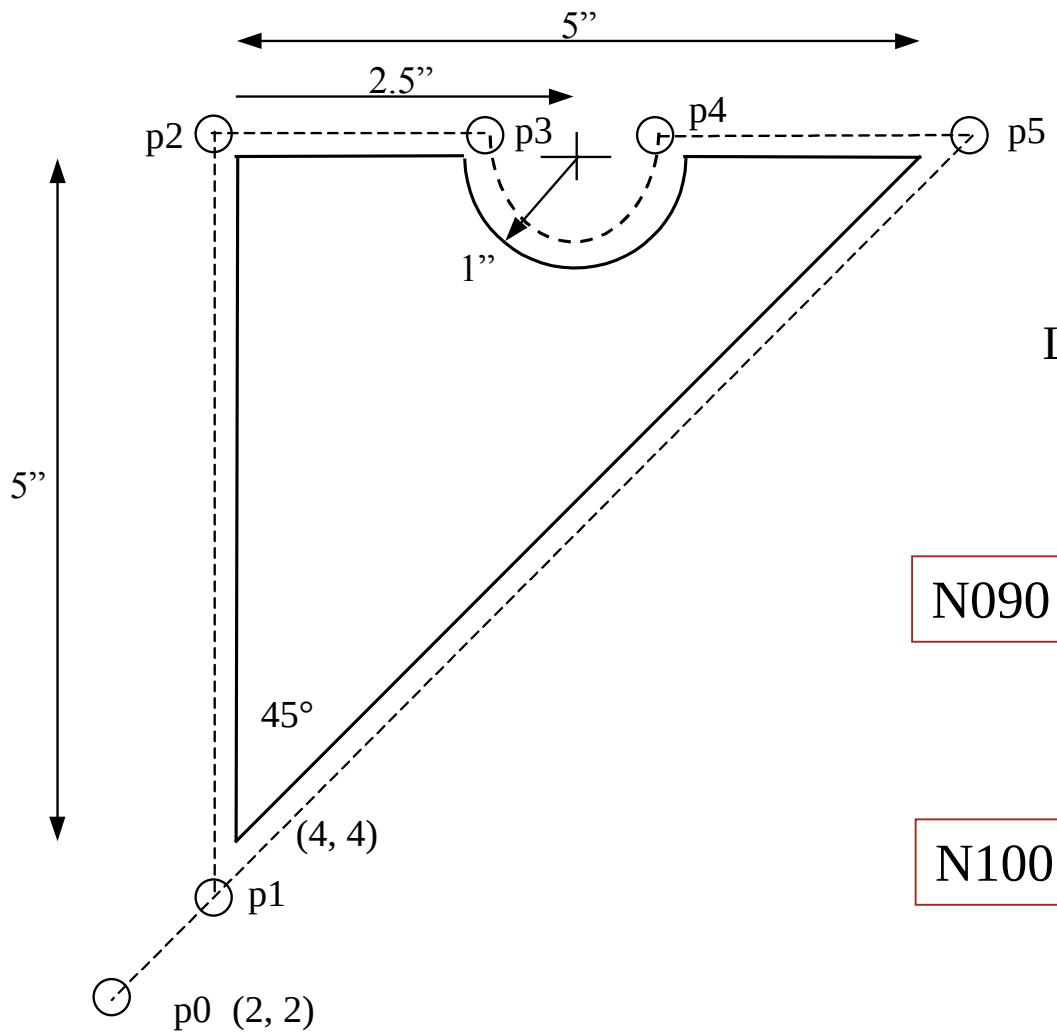


Linear interpolation

target coordinates (see step 3)

N080 G01 X3.875 Y3.698

9. Return to home position, stop program



Linear interpolation

target coordinates (see step 3)

N090 G01 X2.0 Y2.0 M30

end of data

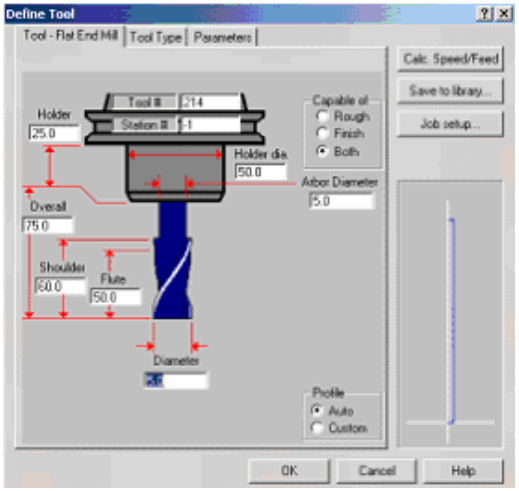
N100 M00

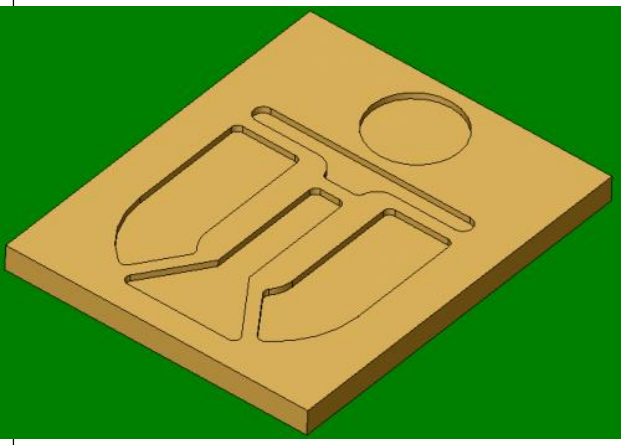
program stop

Automatic Part Programming

Software programs can automatic generation of CNC data

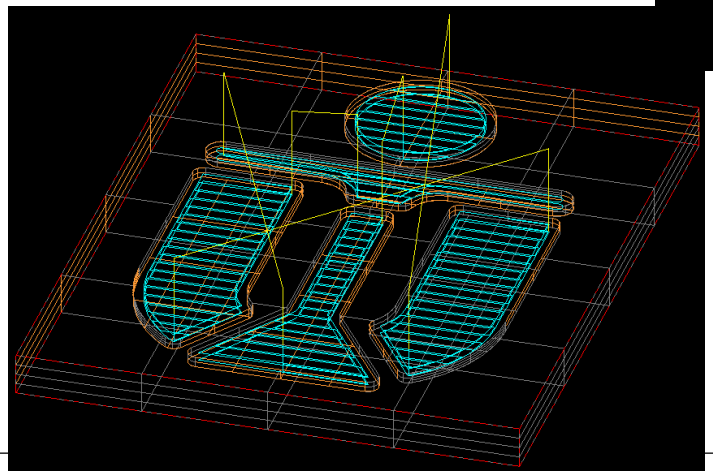
Define Tool

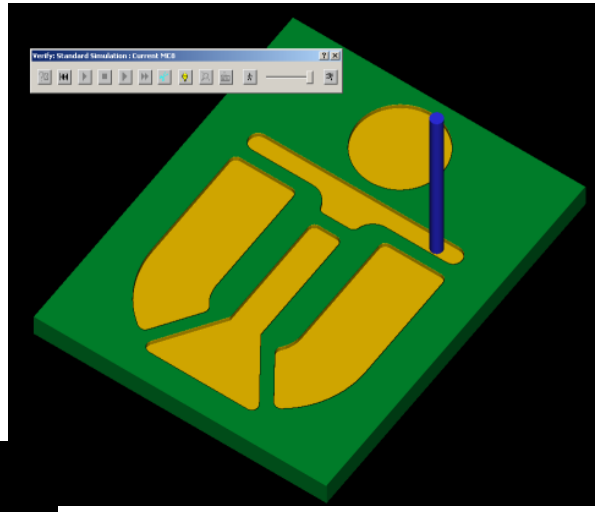




Make 3D model

CNC data





Simulate cutting

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

PART PROGRAM FOR LATHE OPERATION

The CNC lathe operation such as simple facing, turning, taper turning, thread, boring, parting off etc. The *X*-axis and *Z*-axis are taken as the direction of transverse motion of the tool post and the axis of the spindle respectively. To prepare part programs using G-codes and M-codes. The following examples illustrated the part program for different components.

Example

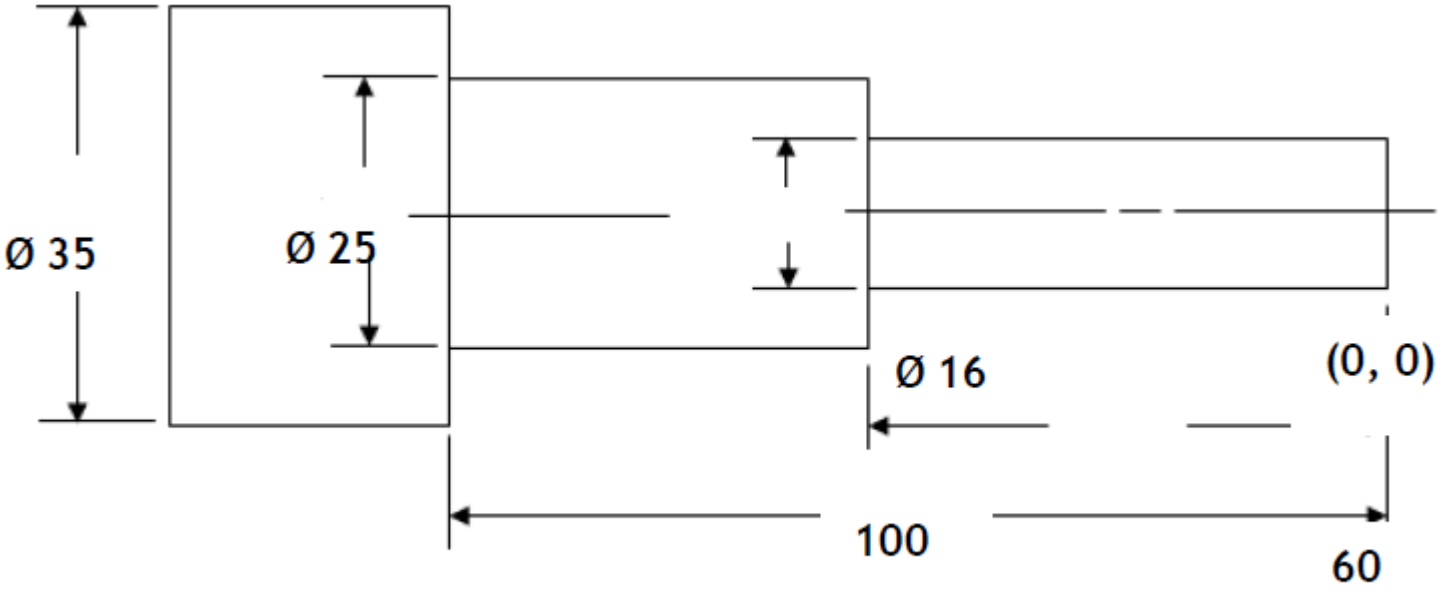


Figure 4.11 : Turning Operation

```
% 1000; (Main programme)
N01 G54 G90 G71 G94 M03 S800; (Parameters Setting)
N05 G01 X-12.5 Z0 F2; (Facing the job)
N10 G00 Z1; (Retrieval of tool)
N15 G00 X00; (Tool clearance)
N20 G01 Z-100; (Starting cut)
N25 G00 X1 Z1; (Clearance position)
N30 G00 X-2; (Position of cut)
N35 G01 Z-60; (Cutting length)
N40 G00 X-1 Z1; (Retrieval of tool)
N45 G00 X-3; (Position of cut)
N50 G01 Z-60; (Cutting length)
N55 G00 X-2 Z1; (Retrieval of tool)
N60 G00 X-4; (Position of cut)
N65 G01 Z-60; (Cutting length)
```

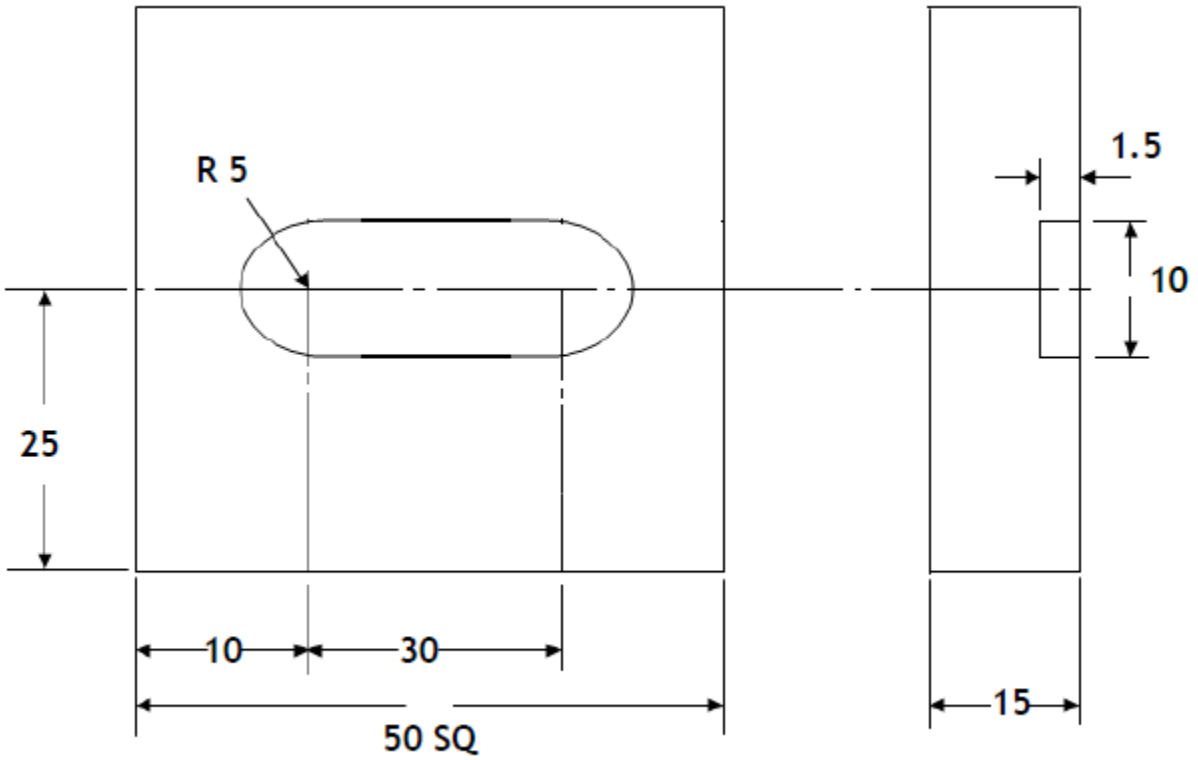
N70 G00 X-3 Z1; (Retrieval of tool)
N75 G00 X-4.5; (Position of cut)
N80 G01 Z-60; (Cutting length)
N85 G00 X5 Z5; (Final position of tool)
N90 M02; (End of programme)

PART PROGRAM FOR MACHINING CENTRES (MILLING)

The CNC milling machine, the motion is possible in three axes, *X*-axis, *Y*-axis and *Z*-axis. The movement of *Z*-axis is taken as positive when tool moves away from the job or vice versa.

Example

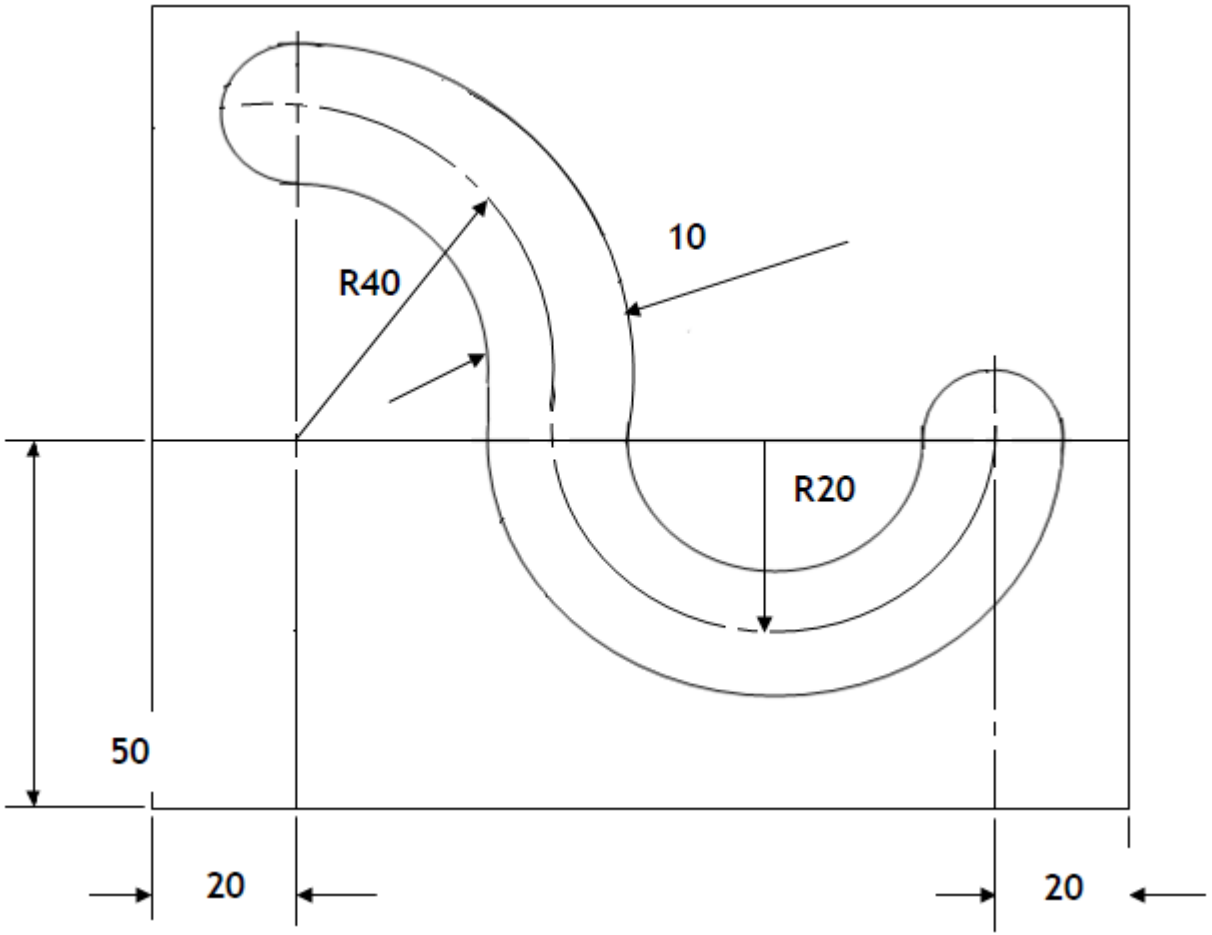
01 (All dimensions are in mm).



% 100; (Main programme)
N5 G17 G71 G90 G94 G54; (Parameters Setting)
N10 T2 L90; (Home position)
N15 G00 D2 Z50 M3 S700 X10 Y-25; (Position of tool)
N20 G01 Z-1.5; (Position of cut)
N25 G01 X4 F100 M8; (Cutting slat)
N30 G00 Z100 M9; (Final position of tool)
N35 M30; (Main programme end)

Example

02 (All dimensions are in mm).



%101; (Main programme)
N2 G17 G71 G90 G94 G54; (Parameters Setting)
N4 T1 L90; (Home position)
N6 G00 Z5 D5 M3 S500 X20 Y90; (Position of tool)
N8 G01 Z-2 F50; (Position of cut)
N10 G02 X60 Y50 I0 J-40; (Circular interpolation clockwise-CW)
N12 G03 X80 Y50 I20 J0; (Circular interpolation clockwise-CCW)
N14 G00 Z100; (Final position of tool)
N16 M02; (End of programme)