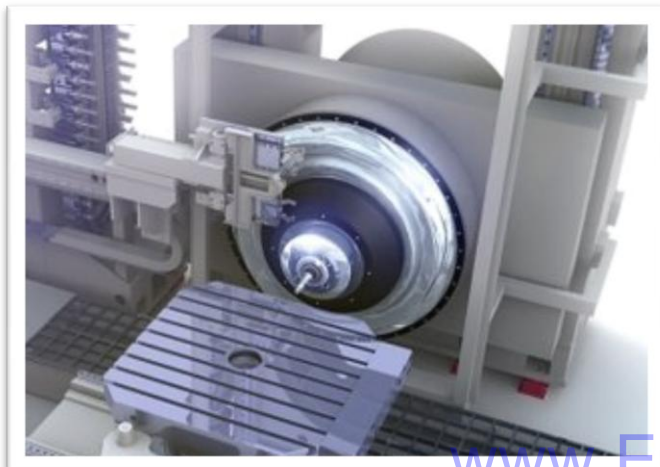


CNC Programming

CAD/CAM course | MSc. Production Engineering | By: Dr. Laith Abdullah Mohammed
Department of Production Engineering & Metallurgy, University of Technology



- Manual [Write code directly]
- Computer-assisted [Draw cutter path]
- CAD/CAM [Draw the part, Cutter path is generated]

- Offline programming linked to CAD programs.
- Conversational programming by the operator.
- MDI ~ Manual Data Input.
- Word-Address Coding using standard G-codes and M-codes.

Basics of NC Part Programming:

During secondary motion either the tool motion, moves relative to the workpiece or the workpiece moves relative to the tool.

In NC programming, it is always assumed that the tool moves relative to the workpiece no matter what the real situation is.

The position of the tool is described by using a **Cartesian coordinate system**. If (0,0,0) position can be described by the operator, then it is called *floating zero*.

Structure of an NC Part Program:

Commands are input into the controller in units called *blocks or statements*.

Block Format:

1. Fixed sequential format
2. Tab sequential format
3. **Word address format**

Example: Assume that a drilling operation is to be programmed as:

1. The tool is positioned at (25.4,12.5,0) by a rapid movement.
2. The tool is then advanced -10 mm in the z direction at a feed rate of 500 mm/min., with the flood coolant on
3. The tool is then retracted back 10 mm at the rapid feed rate, and the coolant is turned off.

Word address format

```
N50 G00 X25400 Y125 Z0 F0  
N60 G01 Z-10000 F500 M08  
N70 G00 Z0 M09
```

INFORMATION NEEDED by a CNC

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

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

BLOCK FORMAT

Sample Block

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

Restrictions on CNC blocks

- Each may contain only one tool move
- Each may contain any number of non-tool move Gcodes
- Each may contain only one feedrate
- Each may contain only one specified tool or spindle speed
- The block numbers should be sequential
- Both the program start flag and the program number must be independent of all other commands (on separate lines)
- The data within a block should follow the sequence shown in the above sample block

Example CNC Program

```
N5 G90 G20
```

```
N10 M06 T3
```

```
N15 M03 S1250
```

```
N20 G00 X1 Y1
```

```
N25 Z0.1
```

```
N30 G01 Z-0.125 F5
```

```
N35 X3 Y2 F10
```

```
N40 G00 Z1
```

```
N45 X0 Y0
```

```
N50 M05
```

```
N55 M30
```

Each instruction to the machine consists of a letter followed by a number.

Each letter is associated with a specific type of action or piece of information needed by the machine.

Letters used in Codes

N,G,X,Y,Z,I,J,K,F,S,T,M

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
G91 Incremental 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

N Codes

Gives an identifying number for each block of information.

It is generally good practice to increment each block number by 5 or 10 to allow additional blocks to be inserted if future changes are required.

X,Y, and Z Codes

X, Y, and Z codes are used to specify the coordinate axis.

Number following the code defines the coordinate at the end of the move relative to an incremental or absolute reference point.

I,J, and K Codes

I, J, and K codes are used to specify the coordinate axis when defining the center of a circle.

Number following the code defines the respective coordinate for the center of the circle.

F-code: used to specify the feed rate

S-code: used to specify the spindle speed

T-code: used to specify the tool identification number associated with the tool to be used in subsequent operations.

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)

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

preparatory function

destination coordinates

dist to center of circle

feed rate

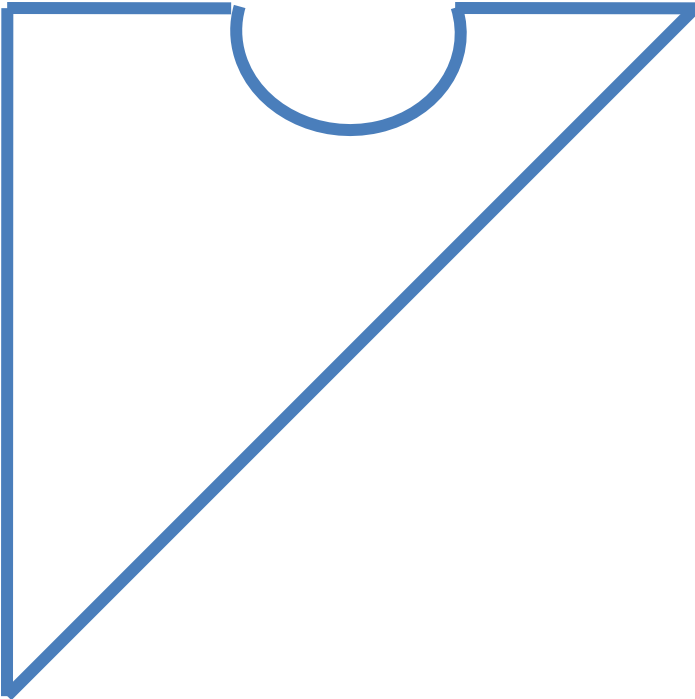
spindle speed

tool

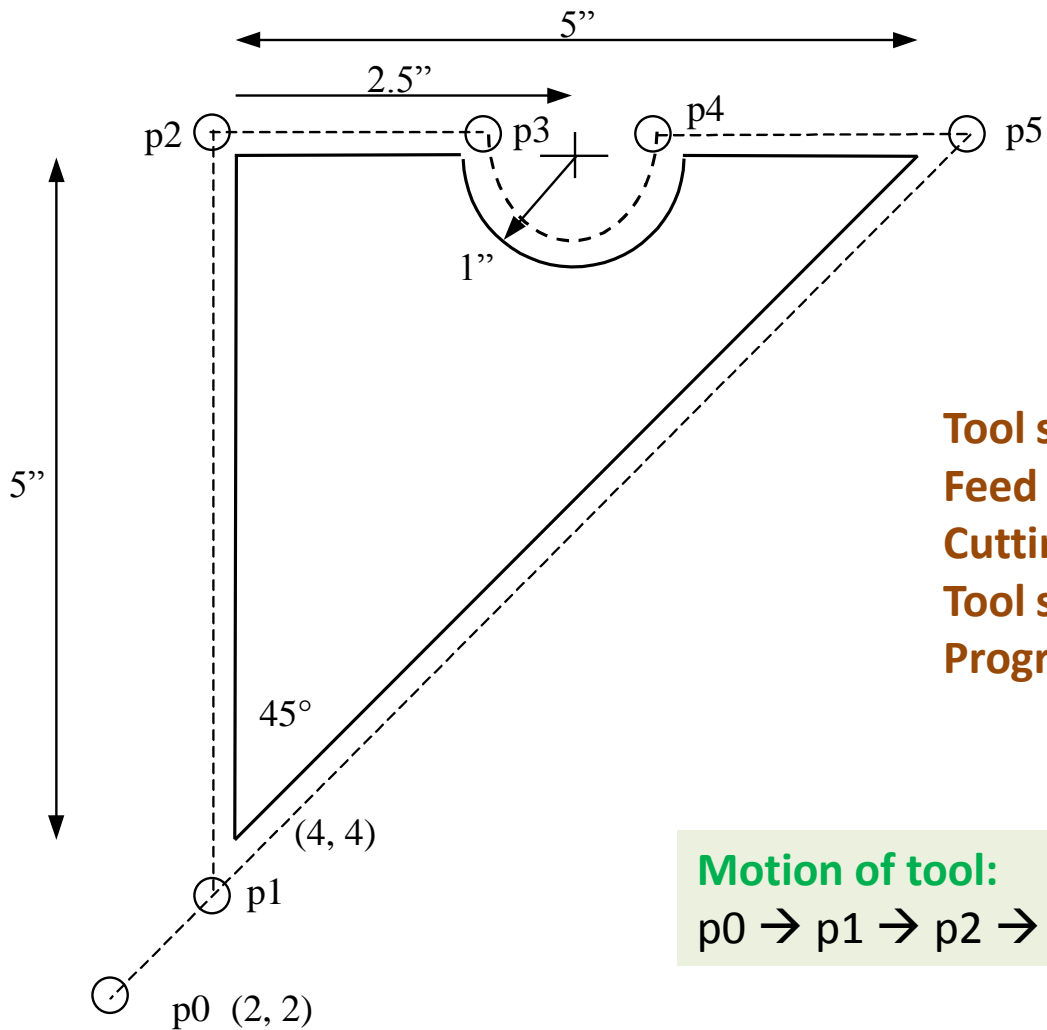
miscellaneous function

Manual Part Programming Example

Write a G-code program for the part shown below



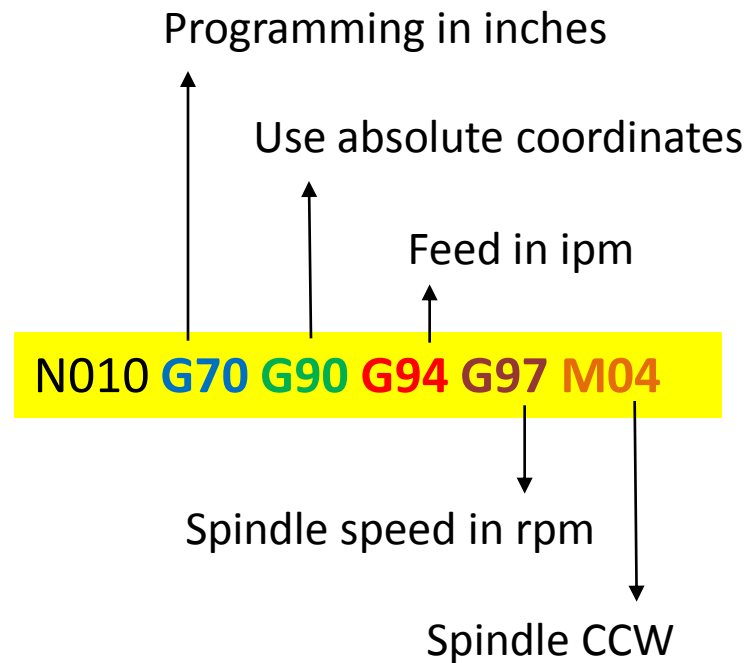
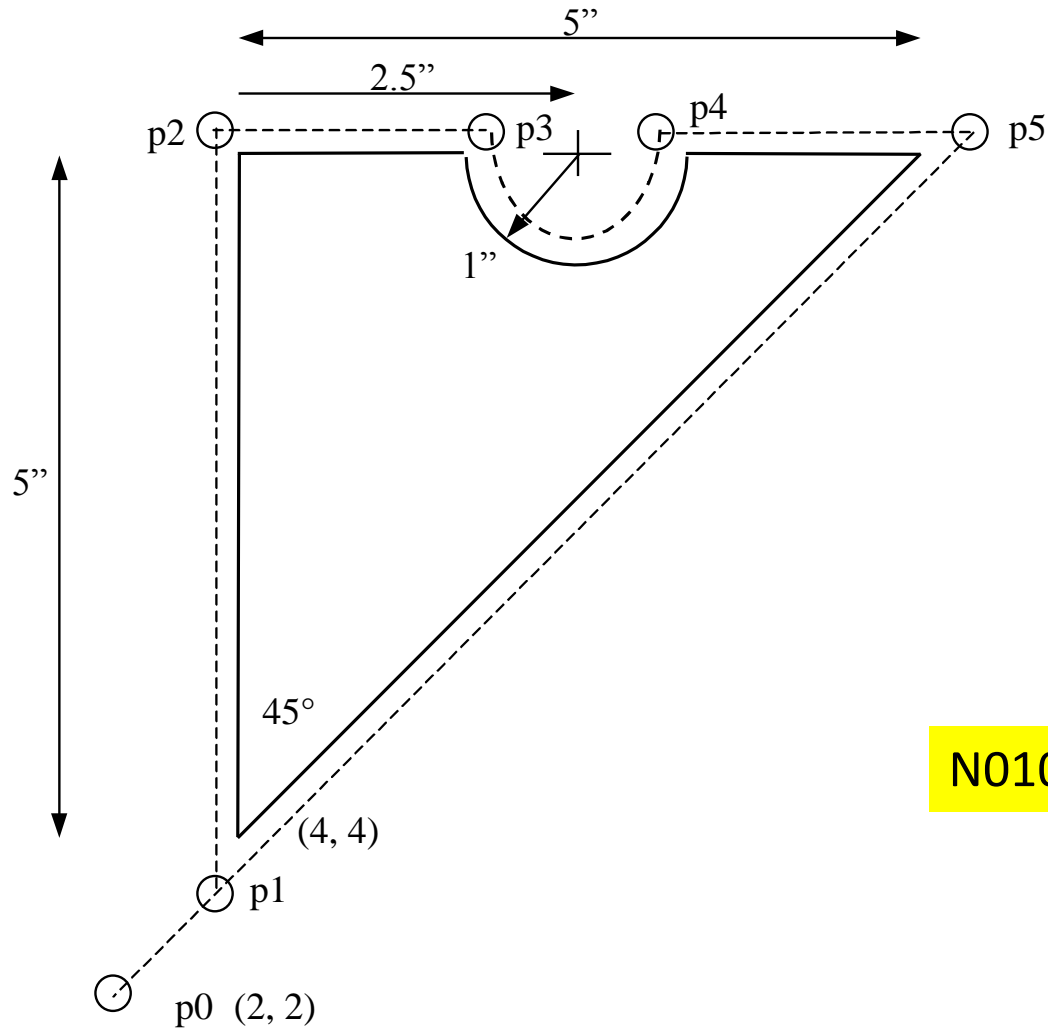
```
N010 G70 G90 G94 G97 M04  
N020 G17 G75 F6.0 S300 T1001 M08  
N030 G01 X3.875 Y3.698  
N040 G01 X3.875 Y9.125  
N050 G01 X5.634 Y9.125  
N060 G03 X7.366 Y9.125 I6.5 J9.0  
N070 G01 X9.302  
N080 G01 X3.875 Y3.698  
N090 G01 X2.0 Y2.0 M30  
N100 M00
```



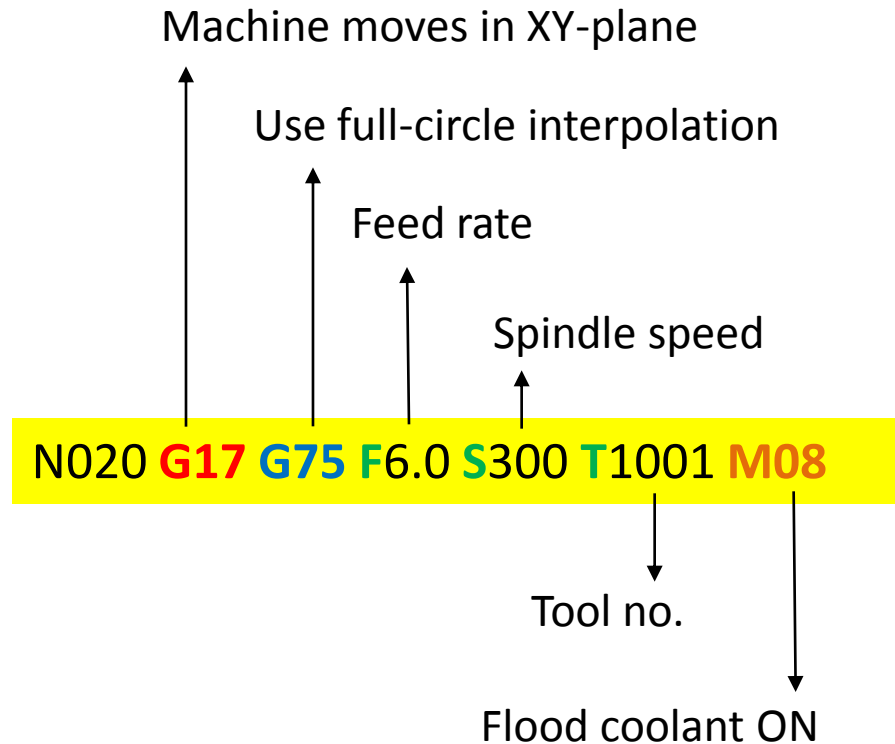
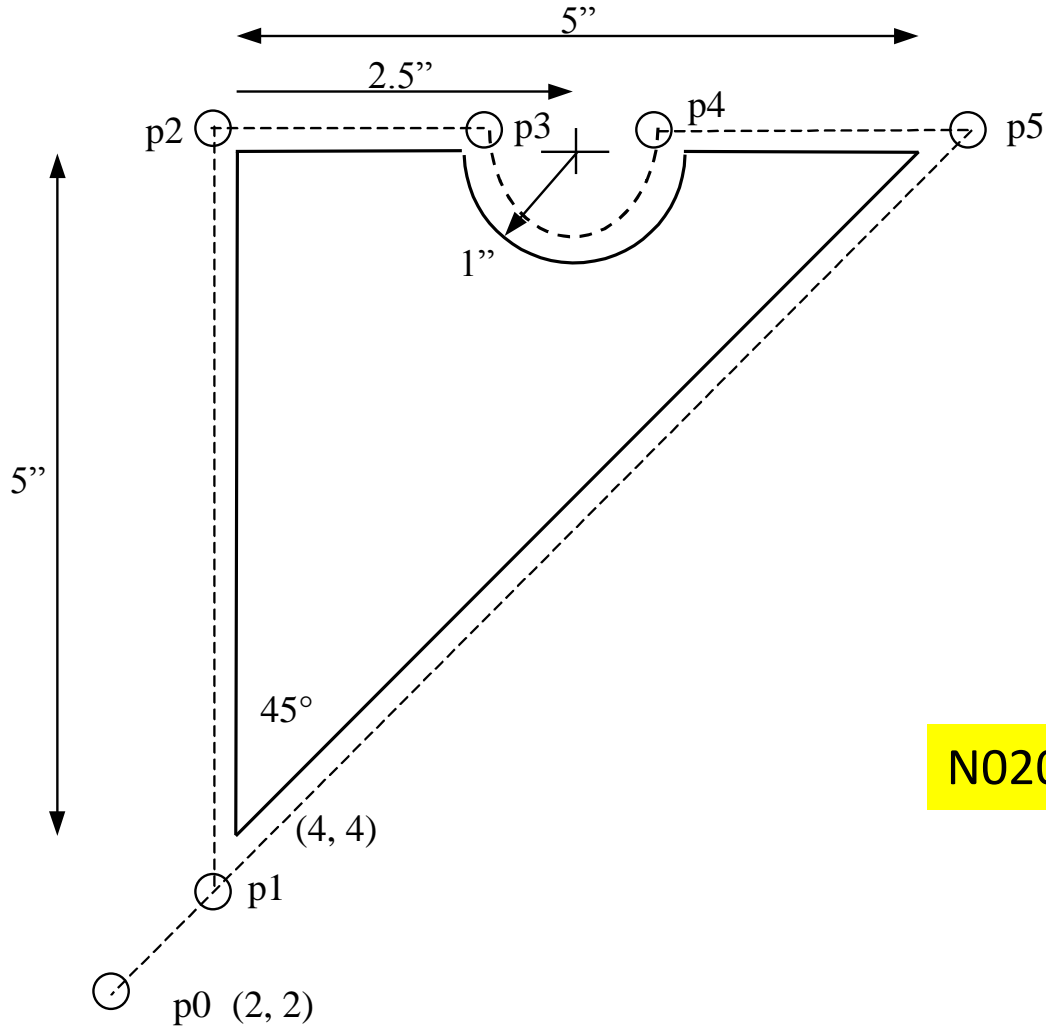
Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

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

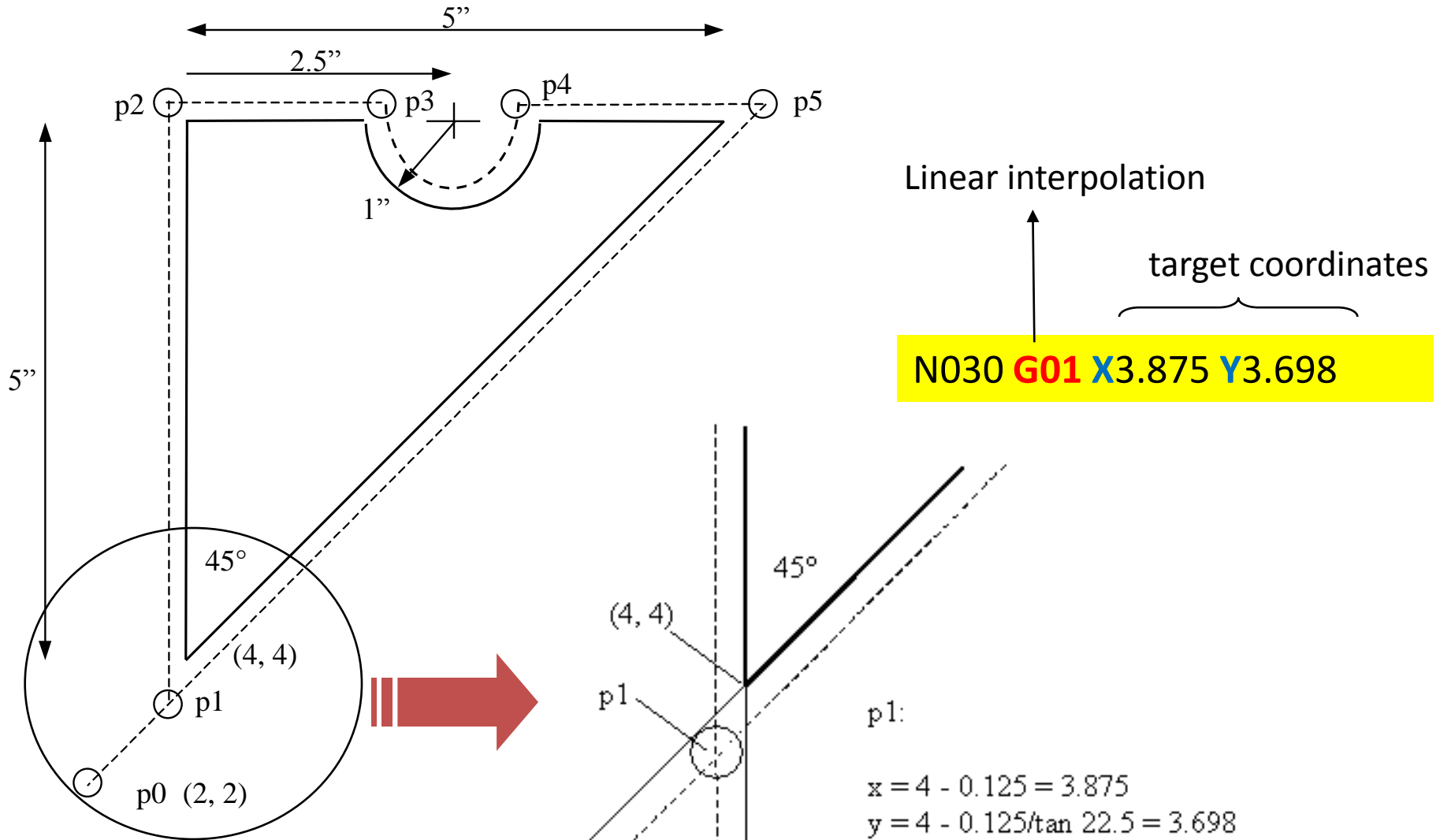
1. Set up the programming parameters



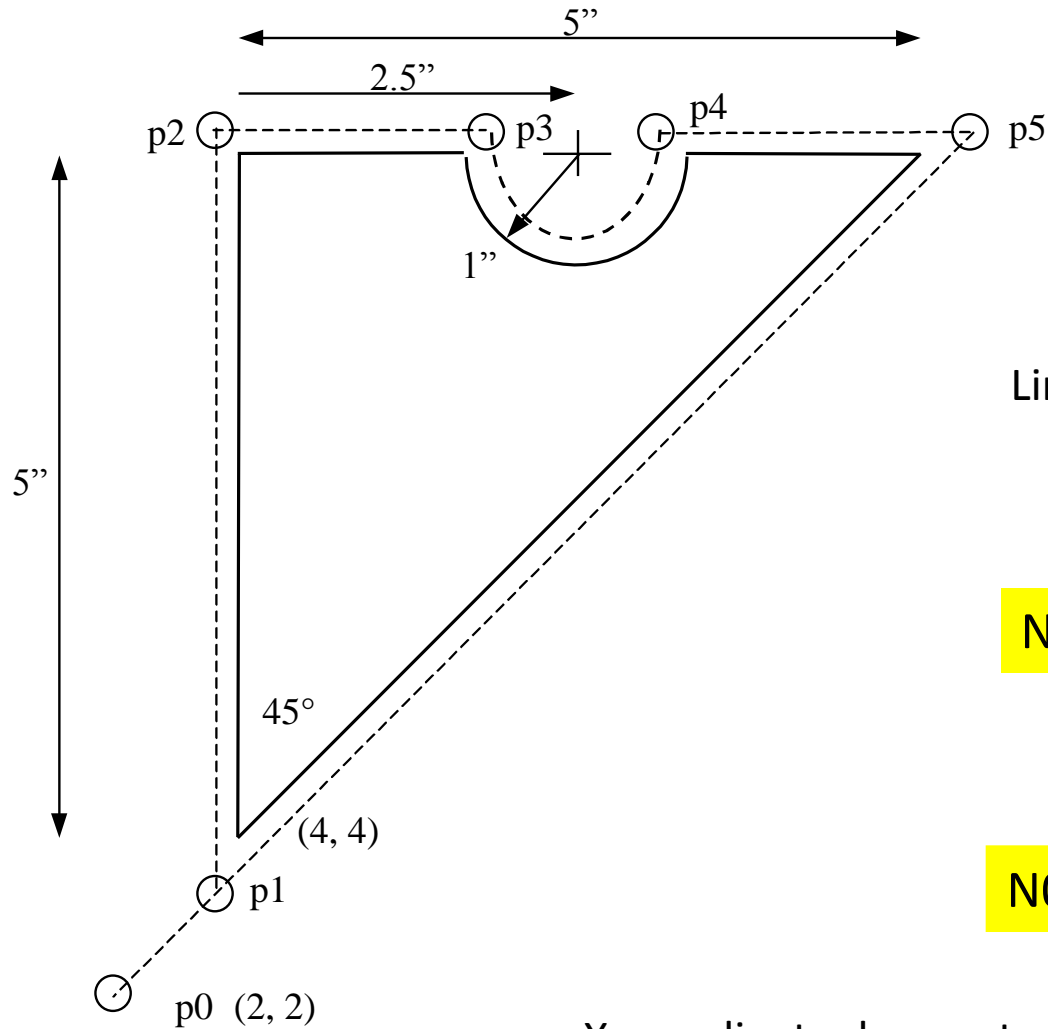
2. Set up the machining conditions



3. Move tool from p0 to p1 in straight line



4. Cut profile from p1 to p2



Linear interpolation

target coordinates

N040 G01 X3.875 Y9.125

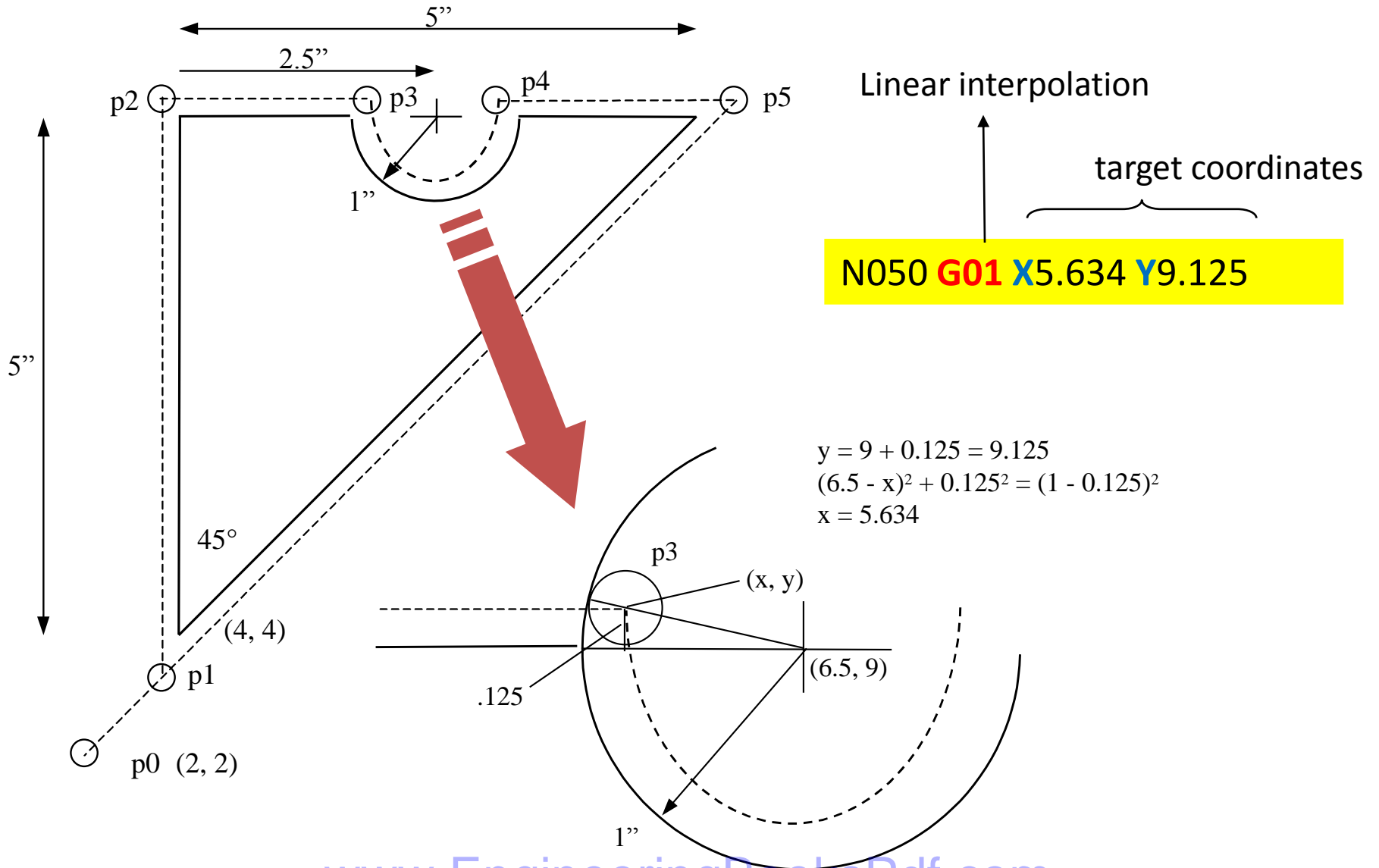


or

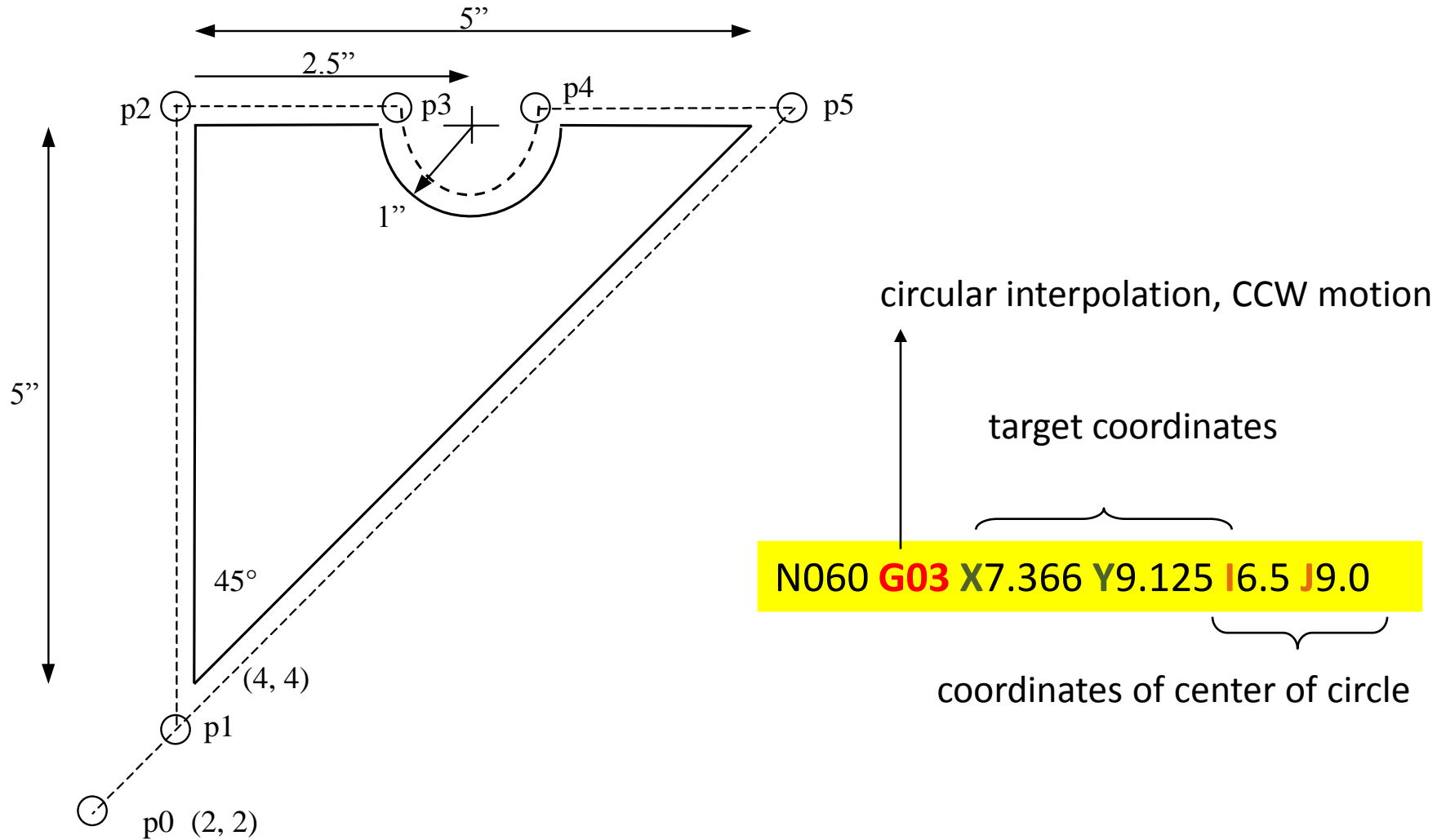
N040 G01 Y9.125

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

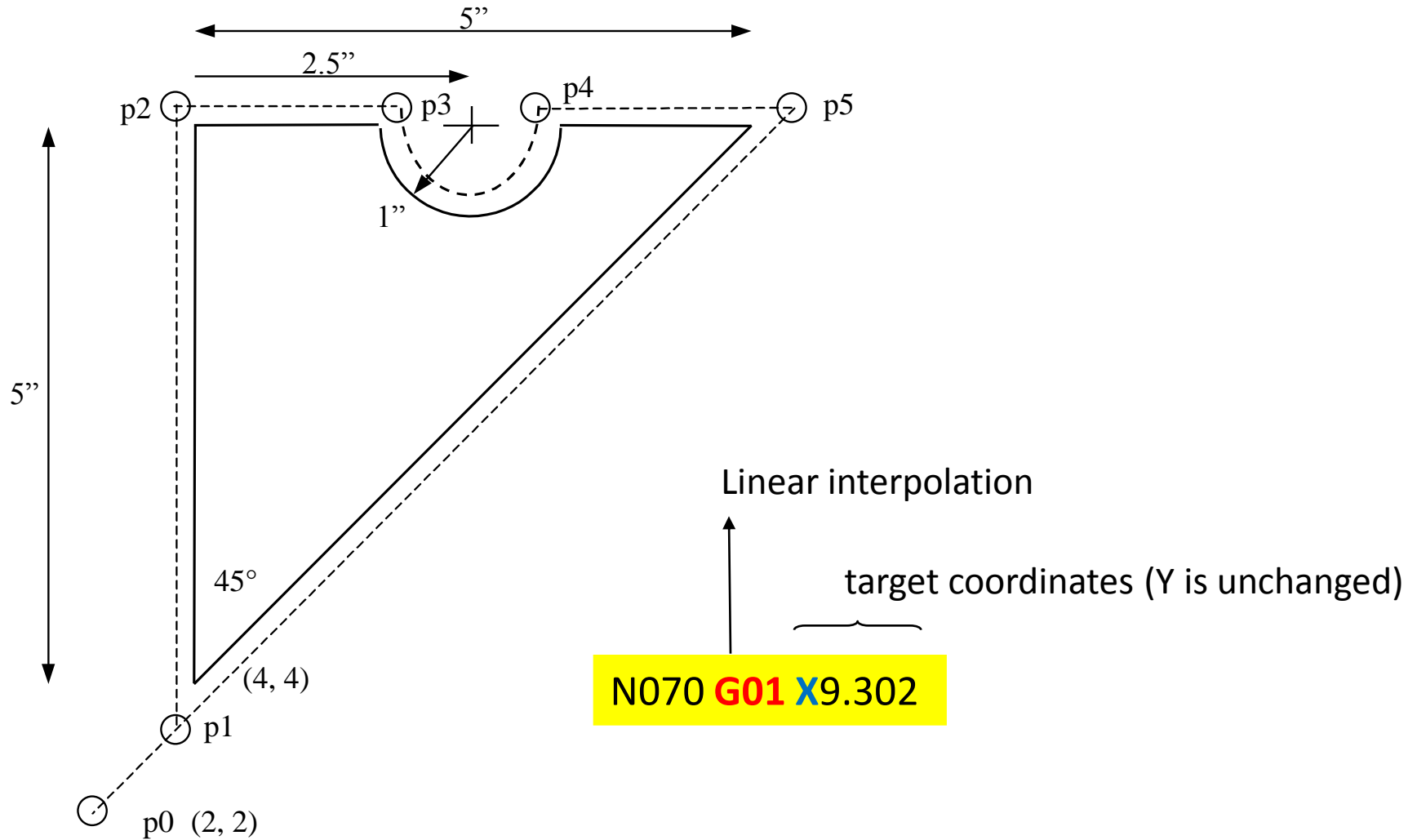
5. Cut profile from p2 to p3



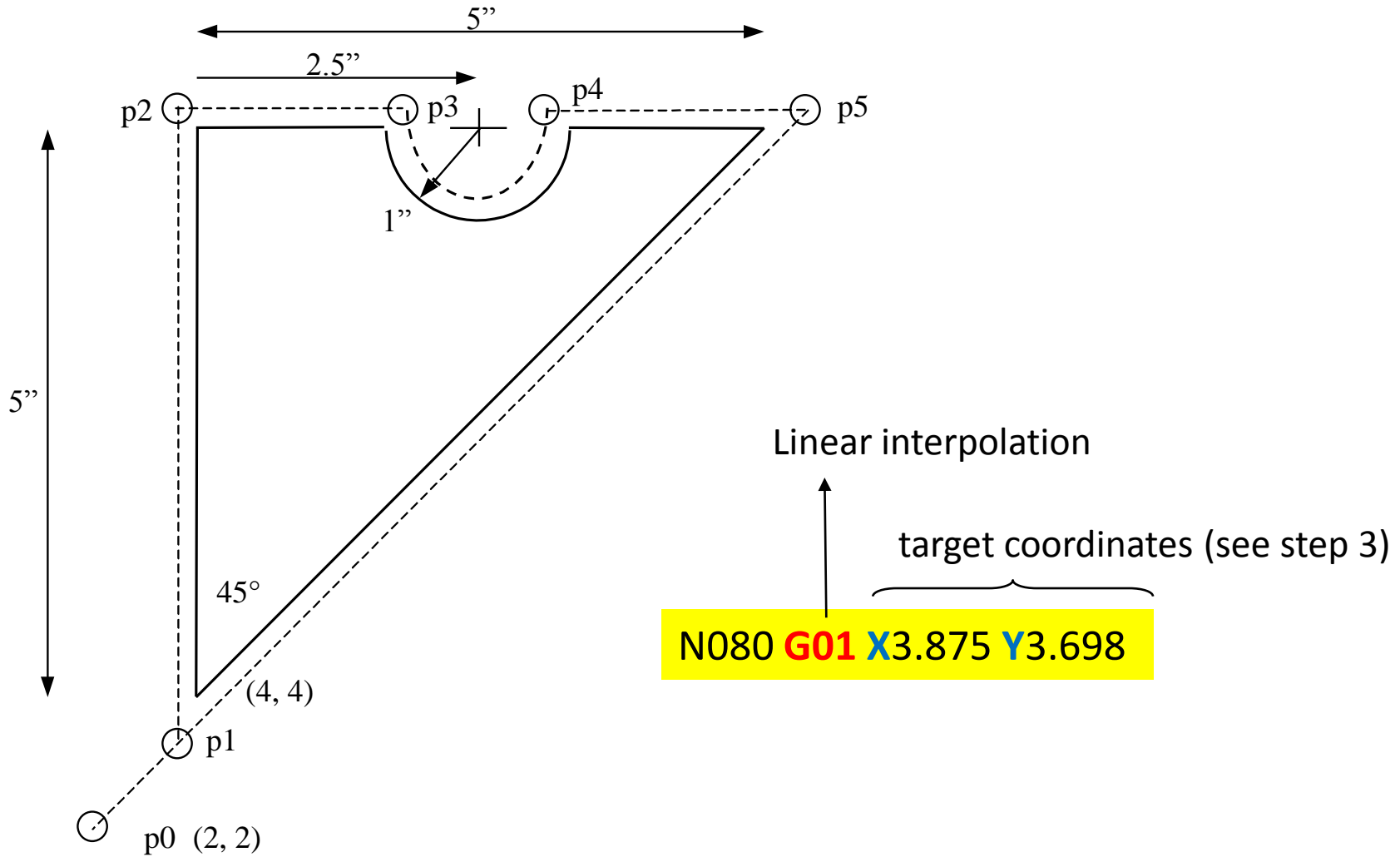
6. Cut along circle from p3 to p4



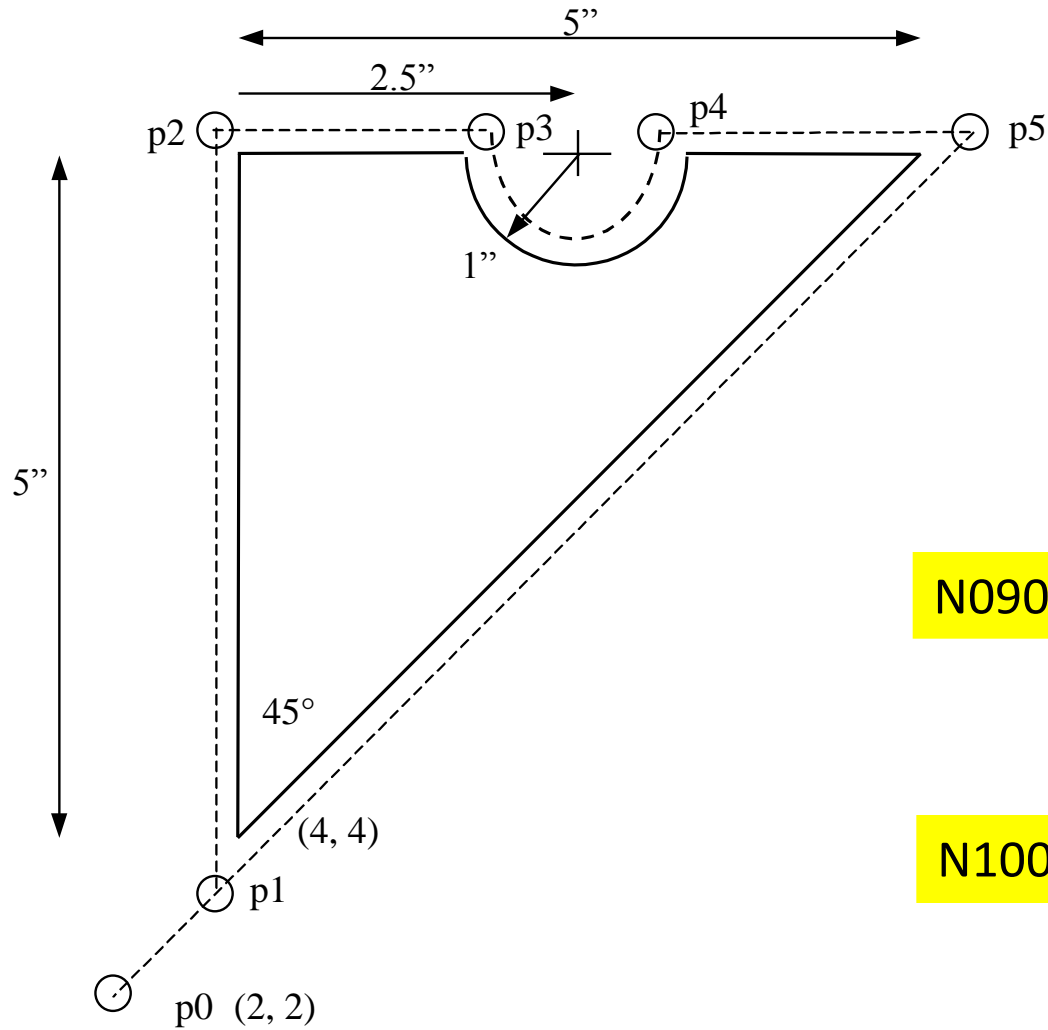
7. Cut from p4 to p5



8. Cut from p5 to p1



9. Return to home position, stop program



Linear interpolation

target coordinates (see step 3)

```
N090 G01 X2.0 Y2.0 M30
```

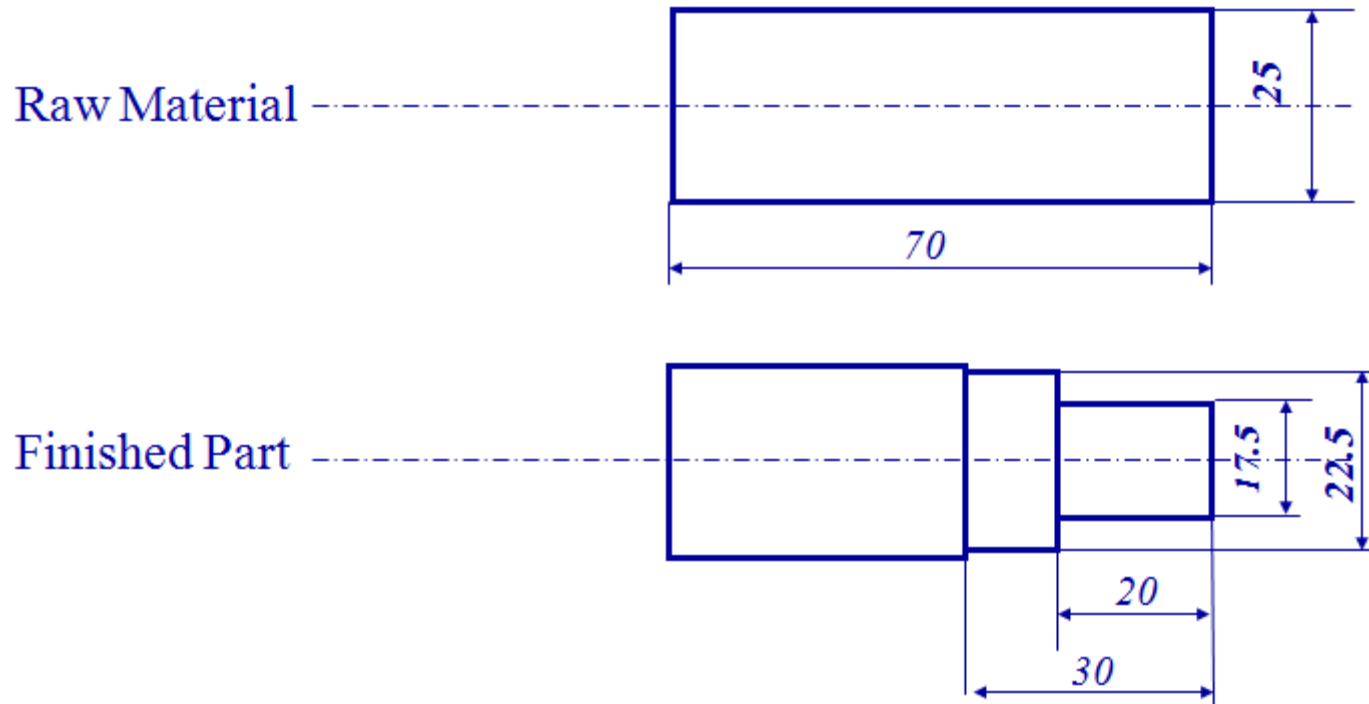
end of data

```
N100 M00
```

program stop

CNC Programming Example [2]

Cylindrical Part

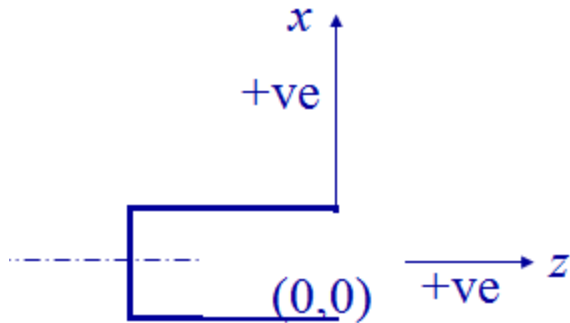


N0005 G53
N0010 T0404

To cancel any previous working zero point
N0010 Sequence number
T0404 Select tool number 404

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 from zero)
Z0.0 Z location
S500 Spindle speed is 500 rpm
M04 Rotate spindle counterclockwise



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.



N0040 G00 Z2.0

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

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.

N0060 T0404

T0404 Select tool number 404

N0070 G57 G00 X22.50 Z2.0 S500

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

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.



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.

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)



N0110 G00 Z2.0

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

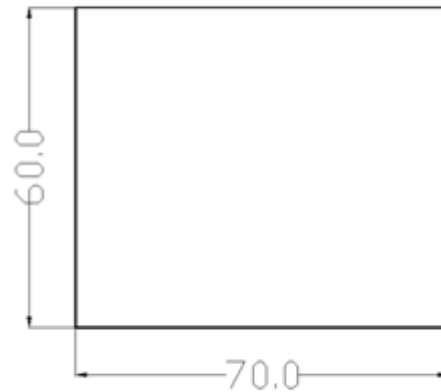
N0120 X50.0 Z50.0

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

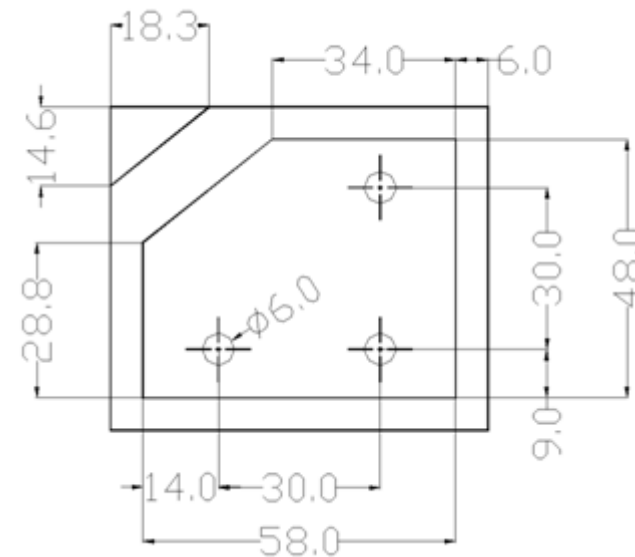
N0130 M30

M30 Program End

CNC Programming Example [3]



Raw Material



Finished Part

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

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

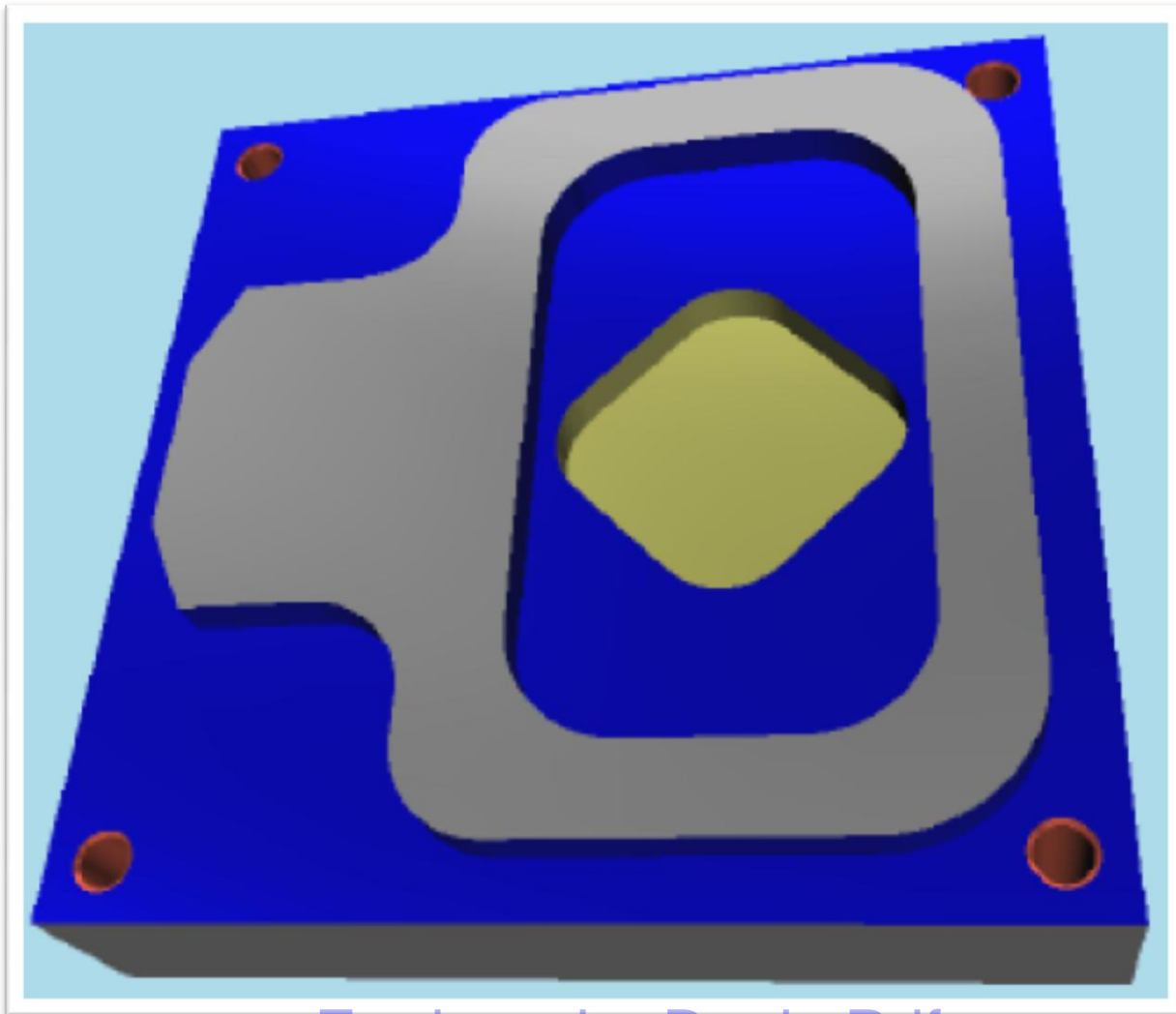
Flow of Computer-Aided CNC Processing

- Develop or obtain the 3D geometric model of the part, using CAD.
- Decide which machining operations and cutter-path directions are required (computer assisted).
- Choose the tooling required (computer assisted).
- Run CAM software to generate the CNC part program.
- Verify and edit program.
- Download the part program to the appropriate machine.
- Verify the program on the actual machine and edit if necessary.
- Run the program and produce the part.



Exercise CNC Milling Programming: Contact plate

The contact plate on the drawing is to be produced on a CNC vertical milling machine from a blank of AlMg1 dimensioned 100 x 100 x 25 mm. Prepare, test and correct the manufacturing process with the MasterCAM CNC Milling Simulator. Define the workpart zero, work out the process layout, set-up form and NC program.



Exercise CNC Turning Programming: Drill sleeve

The drill sleeve is to be produced on a CNC lathe as to the drawing from a blank made of AlMg1 dimensioned $\varnothing 90 \times 128$ mm. The manufacturing process is to be prepared with the MasterCAM CNC Simulator including all planning documentation. Use the compound fixed cycle G71. Test, correct and print the NC program.

