PART PROGRAMMING

INTRODUCTION

A group of commands given to the CNC for operating the machine is called the program.

It consists of:

- Informa on about part geometry
- Mo on statements to move the cu ng tool
- Cu ng speed
- Feed
- Auxiliary func ons such as coolant on and o , spindle direc on

❑ CNC program structure

There are four basic terms used in CNC programming **Character -> Word -> Block -> Program**

- Character is the smallest unit of CNC program. It can have Digit / Letter / Symbol.
- Word is a combination of alpha-numerical characters. This creates a single instruction to the CNC machine. Each word begins with a capital letter, followed by a numeral. These are used to represent axes positions, federate, speed, preparatory commands, and miscellaneous functions.
- A program block may contain multiple words, sequenced in a logical order of processing.
- The program comprises of multiple lines of instructions, ‘blocks’ which will be executed by the machine control unit (MCU).

FIXED ZERO v/s FLOATING ZERO

Fixed zero:

- Origin is always located at some position on M/C table (usually at south west corner/Lower left-hand) of the tables & all tool location are defined W.R.T. this zero

Floating Zero:

- Very common with CNC M/C used now a days.
- Operator sets zero point at any convenient position on M/C table.
- The Coordinate system is knows as work coordinate system (WCS)

Modal and Non modal commands

- Commands issued in the NC program may stay in effect indefinitely (until they explicitly cancelled or changed by some other command), or they may be effective for only the one time that they are issued.
- The former are referred as Modal commands. Examples include feed rate selection and coolant selection.
- Commands that are effective only when issued and whose effects are lost for subsequent commands are referred to as non-modal commands.
- A dwell command, which instructs the tool to remain in a given configuration for a given amount of time, is an example of a non-modal command.

Structure of an NC part program

- An NC part program is made up of a series of commands that are input into the MCU in a serial manner.
- The MCU interprets these commands and generates the necessary signals to each of the drive units of the machine to accomplish the required action.
- The NC program is required to have a particular structure that the controller can understand and it must follow a specific syntax.
- Commands are inputs into the controller in units called blocks or statements.
- Each block is made up of one or more machine commands.

- In general, several commands are grouped together to accomplish a specific machining operation, hence the use of a block of information for each operation.

- Each command gives a specific element of control data, such as dimension or a feed rate. Each command within a block is also called a word.

- The way in which words are arranged within the block is called block format.

- Three different block formats are commonly used, (Fixed sequential format, Tab sequential format and Word address format)

Word Sequential Format : Used on virtually all modern controllers.

N50 G00 X50 Y25 Z0 F0
N60 G01 Z-1 F50 M08
N70 Z0 M09

- With this type of format, each type of word is assigned as address that is identified by a letter code within the part program.
- Thus the letter code specifies the type of word that follows and then its associated numeric data is given.
- For example, the code T represents a tool number. Thus a word of the form T01 would represent tool number 1.
- Theoretically, with this approach, the words in a given block can be entered in any sequence and the controller should be able to interpret them correctly.

- With the word address format only the needed words for a given operation have to be included within the block.
- The command to which the particular numeric data applies is identified by the preceding address code.
- Word format has the advantage of having more than one particular command in one block something that would be impossible in the other two formats.

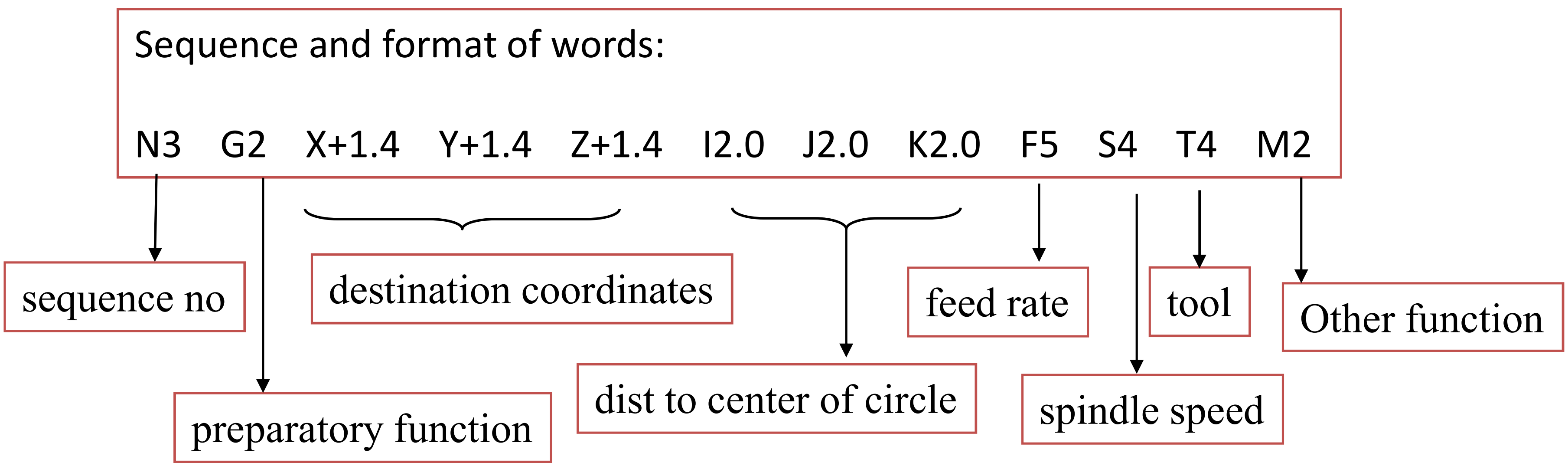
COMMONLY USED WORD ADDRESSES

- N-CODE: Sequence number, used to identify each block with in an NC program and provides a means by which NC commands may be rapidly located. It is program line number. It is a good practice to increment each block number by 5 to 10 to allow additional blocks to be inserted if future changes are required.

- G-CODE: Preparatory Word, used as a communication device to prepare the MCU. The G-code indicates that a given control function such as G01, linear interpolation, is to be requested.

- X, Y & Z-CODES: Coordinates. These give the coordinate positions of the tool.

- F-CODE: Feed rate. The F code specifies the feed in the machining operation.
- S-CODE: Spindle speed. The S code specifies the cutting speed of the machining process.
- T-CODE: Tool selection. The T code specifies which tool is to be used in a specific operation.
- M-CODE: Miscellaneous function. The M code is used to designate a particular mode of operation for an NC machine tool.
- I, J & K-CODES: They specify the centre of arc coordinates from starting.


Sequence and format of words:
N3 G2 X+1.4 Y+1.4 Z+1.4 I2.0 J2.0 K2.0 F5 S4 T4 M2
sequence no
preparatory function
destination coordinates
dist to center of circle
feed rate
spindle speed
tool
Other function

G00	Rapid Linear Positioning	G55	Work Coordinate System 2 Selection
G01	Linear Feed Interpolation	G56	Work Coordinate System 3 Selection
G02	CW Circular Interpolation	G57	Work Coordinate System 4 Selection
G03	CCW Circular Interpolation	G58	Work Coordinate System 5 Selection
G04	Dwell	G59	Work Coordinate System 6 Selection
G07	Imaginary Axis Designation	G60	Single Direction Positioning
G09	Exact Stop	G61	Exact Stop Mode
G10	Offset Value Setting	G64	Cutting Mode
G17	XY Plane Selection	G65	Custom Macro Simple Call
G18	ZX Plane Selection	G66	Custom Macro Modal Call
G19	YZ plane Selection	G67	Custom Macro Modal Call Cancel
G20	Input In Inches	G68	Coordinate System Rotation On
G21	Input In Millimeters	G69	Coordinate System Rotation Off
G22	Stored Stroke Limit On	G73	Peck Drilling Cycle
G23	Stored Stroke Limit Off	G74	Counter Tapping Cycle
G27	Reference Point Return Check	G76	Fine Boring
G28	Return To Reference Point	G80	Canned Cycle Cancel
G29	Return From Reference Point	G81	Drilling Cycle, Spot Boring
G30	Return To 2nd, 3rd and 4th Ref. Point	G82	Drilling Cycle, Counter Boring
G31	Skip Cutting	G83	Peck Drilling Cycle
G33	Thread Cutting	G84	Tapping Cycle
G40	Cutter Compensation Cancel	G85	Boring Cycle
G41	Cutter Compensation Left	G86	Boring Cycle
G42	Cutter Compensation Right	G87	Back Boring Cycle
G43	Tool Length Compensation + Direction	G88	Boring Cycle
G44	Tool Length Compensation - Direction	G89	Boring Cycle
G45	Tool Offset Increase	G90	Absolute Programming
G46	Tool Offset Double	G91	Incremental Programming
G47	Tool Offset Double Increase	G92	Programming Of Absolute Zero
G48	Tool Offset Double Decrease	G94	Feed Per Minute
G49	Tool Length Compensation Cancel	G95	Feed Per Revolution
G50	Scaling Off	G96	Constant Surface Speed Control
G51	Scaling On	G97	Constant Surface Speed Control Cancel
G52	Local Coordinate System Setting	G98	Return To Initial Point In Canned Cycles
G54	Work Coordinate System 1 Selection	G99	Return To R Point In Canned Cycles

List of M codes

M codes vary from machine to machine depending on the functions available on it. They are decided by the manufacturer of the machine. The M codes listed below are the common ones.

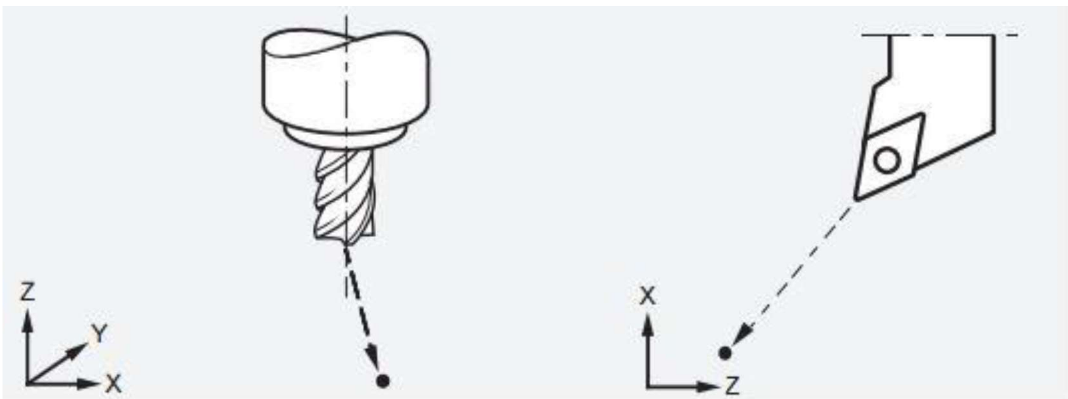
M-codes	Function
M00	Optional program stop automatic
M01	Optional program stop request
M02	Program end
M03	Spindle ON clock wise (CW)
M04	Spindle ON counter clock wise (CCW)
M05	Spindle stop
M06	Tool change
M07	Mist coolant ON (coolant 1 ON)
M08	Flood coolant ON (coolant 2 ON)
M09	Coolant OFF
M30	End of program, Reset to start
M98	Sub program call
M99	Sub program end

G00 Rapid traverse

When the tool being positioned at a point preparatory to a cutting motion, to save time it is moved along a straight line at Rapid traverse, at a fixed traverse rate which is pre-programmed into the machine's control system.

Typical rapid traverse rates are 10 to 25 m /min., but can be as high as 80 m/min.

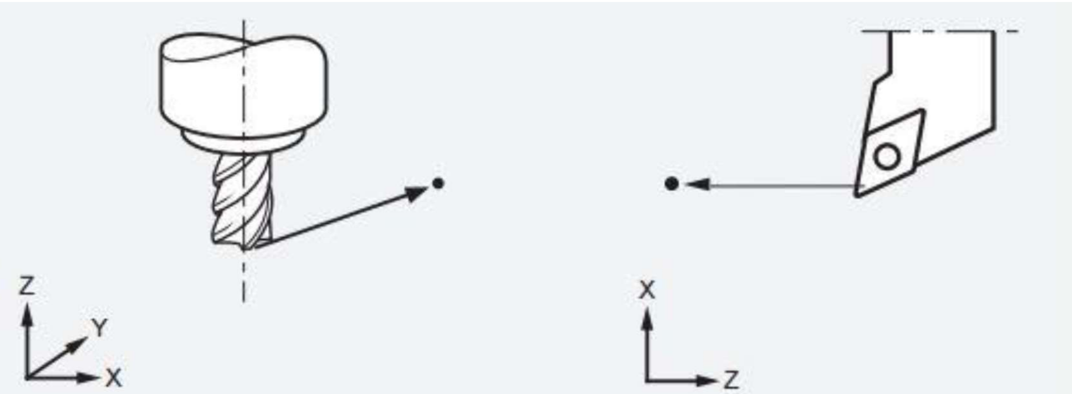
Syntax: N010 [G90/G91] G00 X10 Y10 Z5



G01 Linear interpolation (feed traverse)

The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the feed rate.

Syntax: N010[G90/G91] G01 X10 Y10 Z5 F25



G02/G03 Circular interpolation

Format

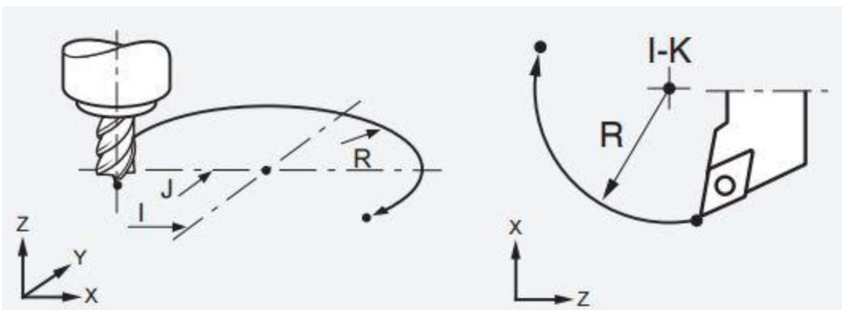
N__ G02/03 X__ Y__Z__ I__ J__K__ F__ using the arc center

or

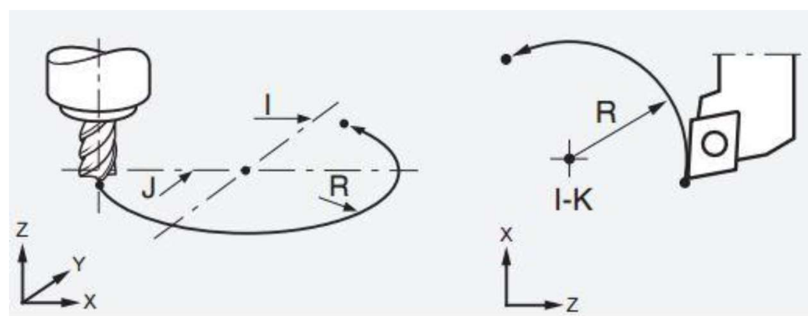
N__ G02/03 X__ Y__Z__ R__ F__ using the arc radius

Arc center

The arc center is specified by addresses I, J and K. I, J and K are the X, Y and Z co-ordinates of the arc center with reference to the arc start point.



G02 moves along a CW arc



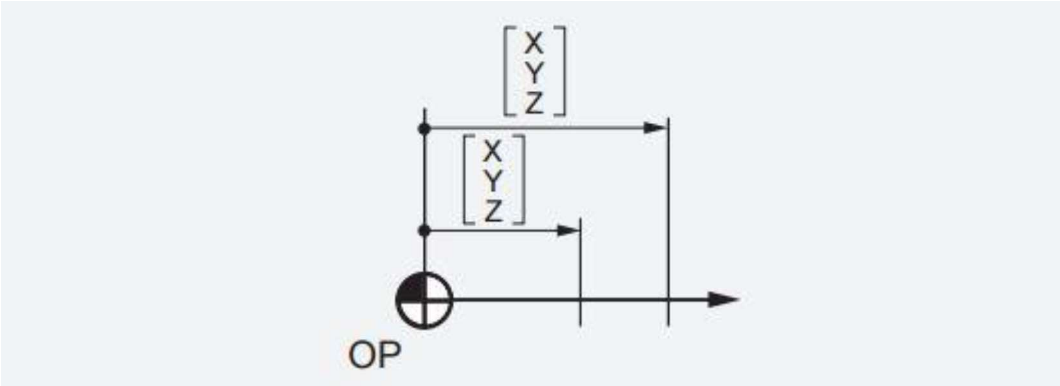
G03 moves along a CCW arc

G90 ABSOLUTE POSITION COMMAND

- When using a G90 absolute position command, each dimension or move is referenced from a fixed point, known as ABSOLUTE ZERO (part zero).

- Absolute zero is usually set at the corner edge of a part, or at the center of a square or round part, or an existing bore. ABSOLUTE ZERO is where the dimensions of a part program are defined from.

- Absolute dimensions are referenced from a known point on the part, and can be any point the operator chooses, such as the upper-left corner, center of a round part, or an existing bore.

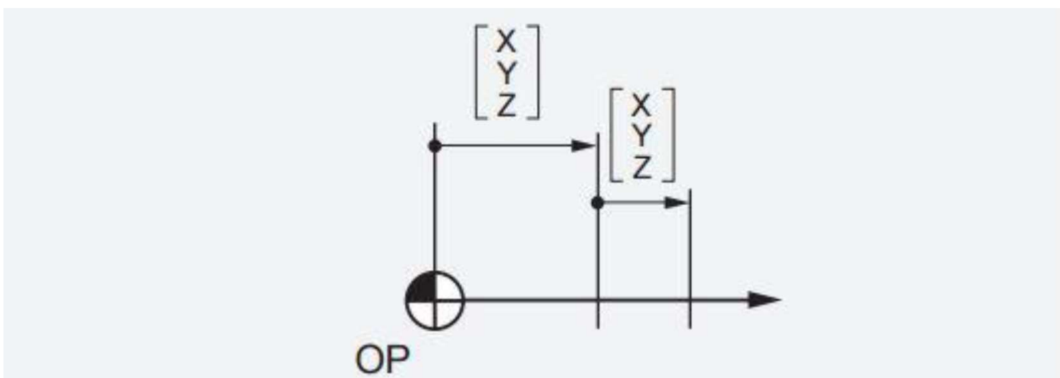


Syntax: N.. G90 X.. Y.. Z.. A.. B.. C..

G91 INCREMENTAL POSITION COMMAND

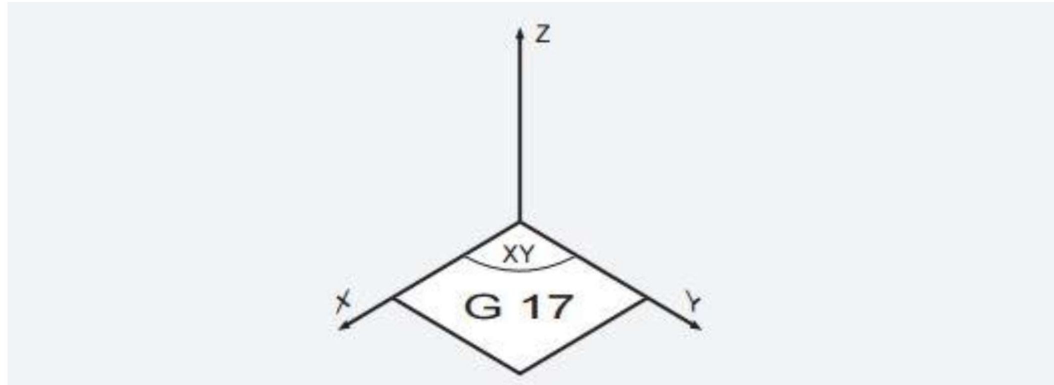
- This code is modal and changes the way axis mo on commands are interpreted. G91 makes all subsequent commands incremental. Zero point shi s with the new posi on.

Syntax: N.. G91 X.. Y.. Z.. A.. B.. C..

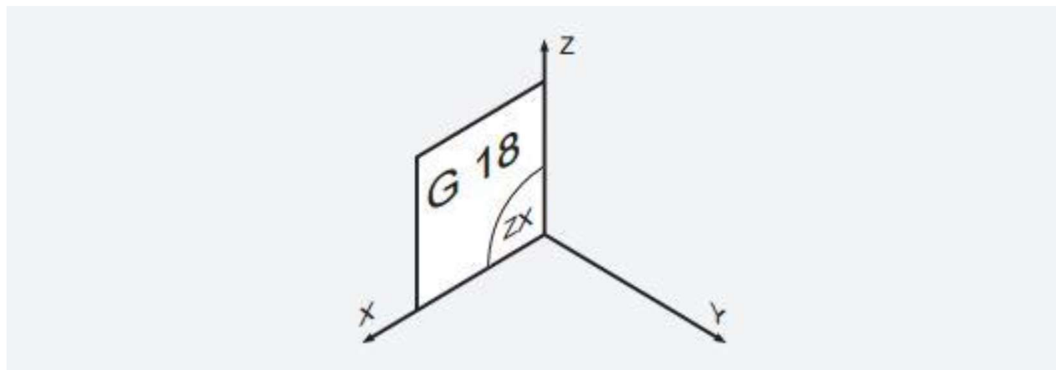


G 17 G18 G19 : PLANE SELECTION

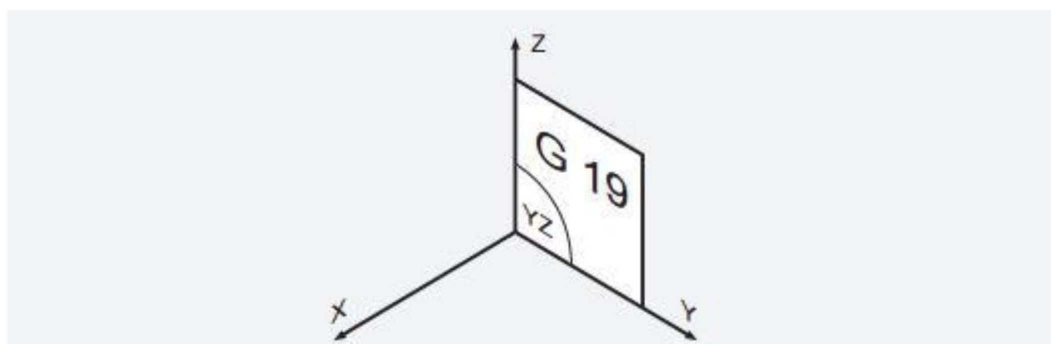
G 17 : XY plane selec on
Syntax: N.. G17



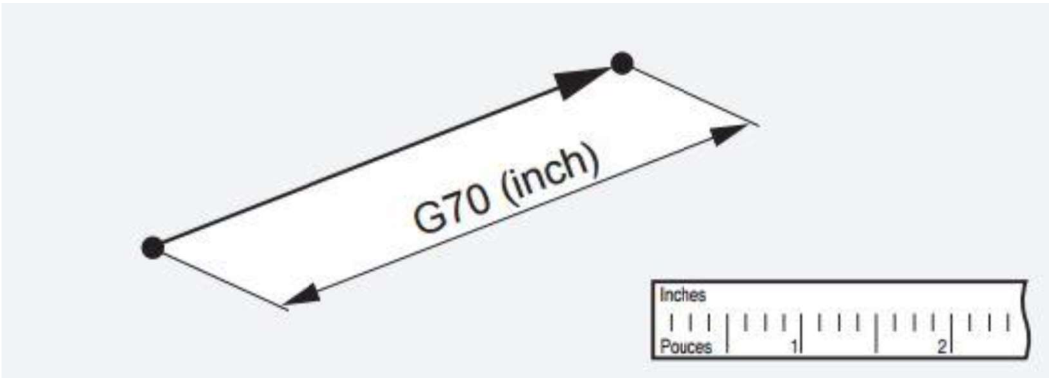
G 18 : ZX plane selec on
Syntax: N.. G18



G 19 : ZY plane selec on
Syntax: N.. G19

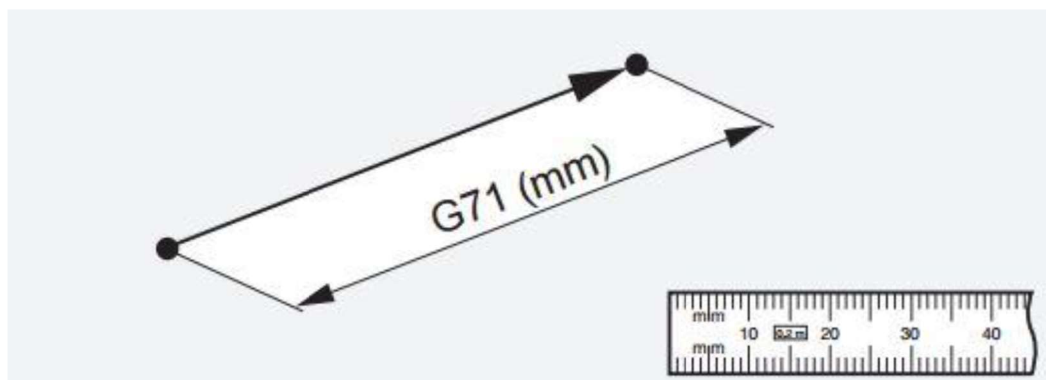


Syntax: N020 G17 G75 F6.0 S300 T1001 M08



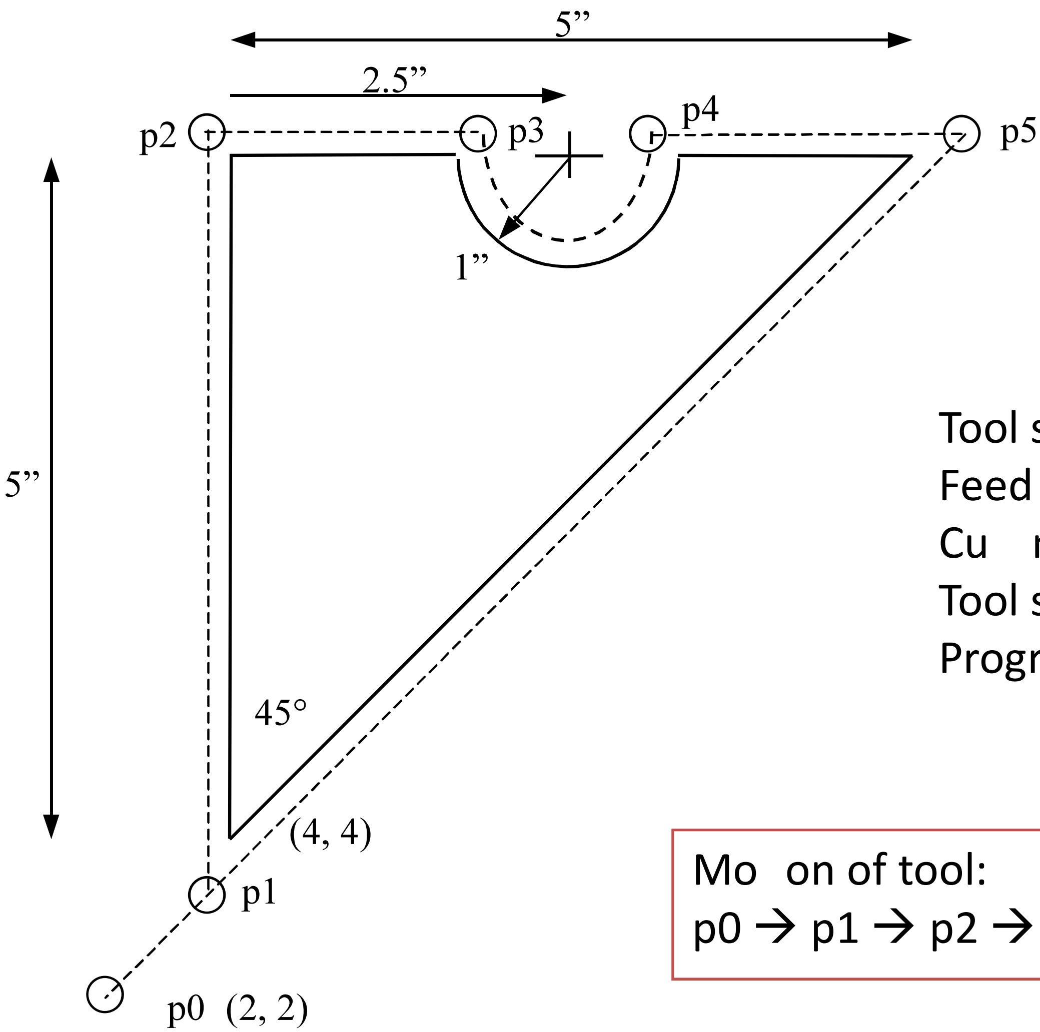
G 70 Inch data input

G 71 Metric data input



Syntax : N010 G70 G90 G94 G97 M04

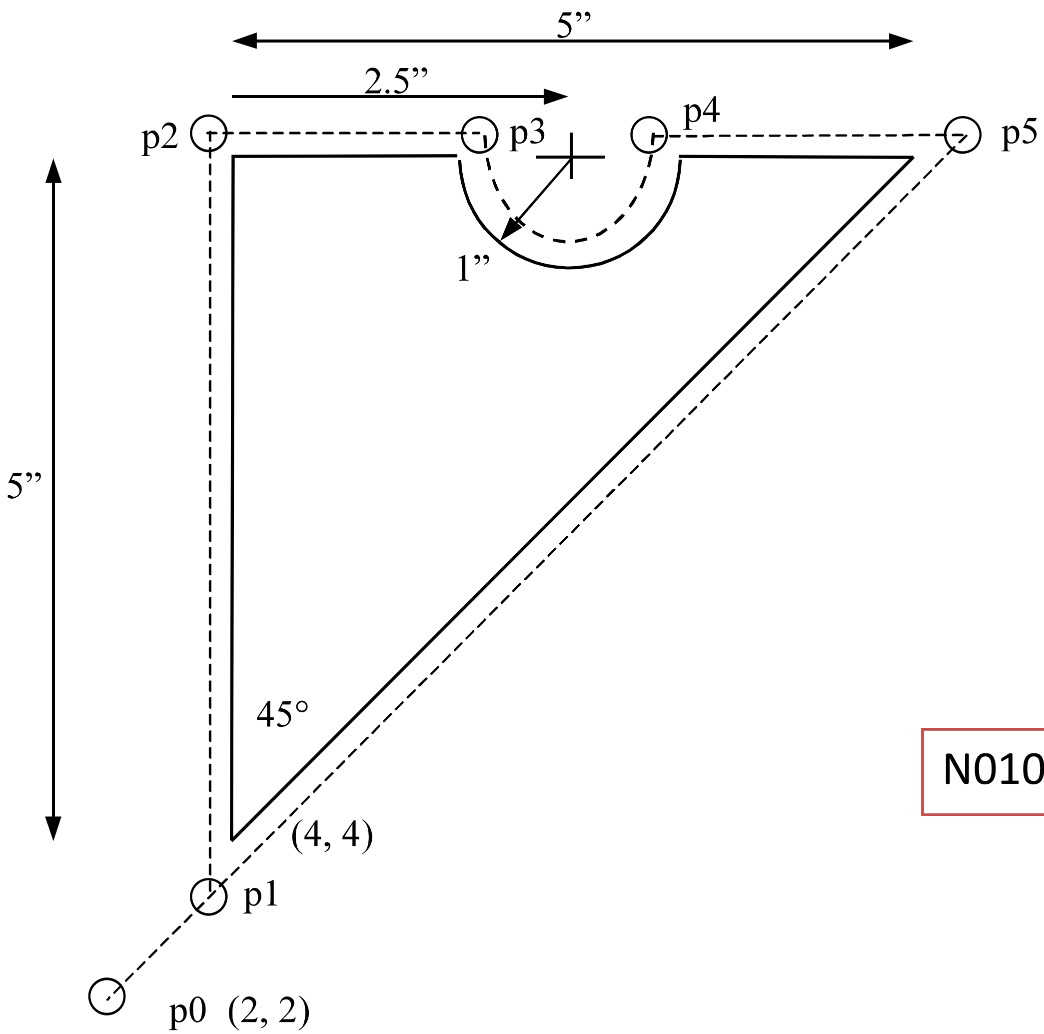
Manual Part Programming Example



Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cu ng speed = 300 rpm,
Tool start posi on: 2.0, 2.0
Programming in inches

Mo on of tool:
p0 → p1 → p2 → p3 → p4 → p5 → p1 → p0

1. Set up the programming parameters



Programming in inches

Use absolute coordinates

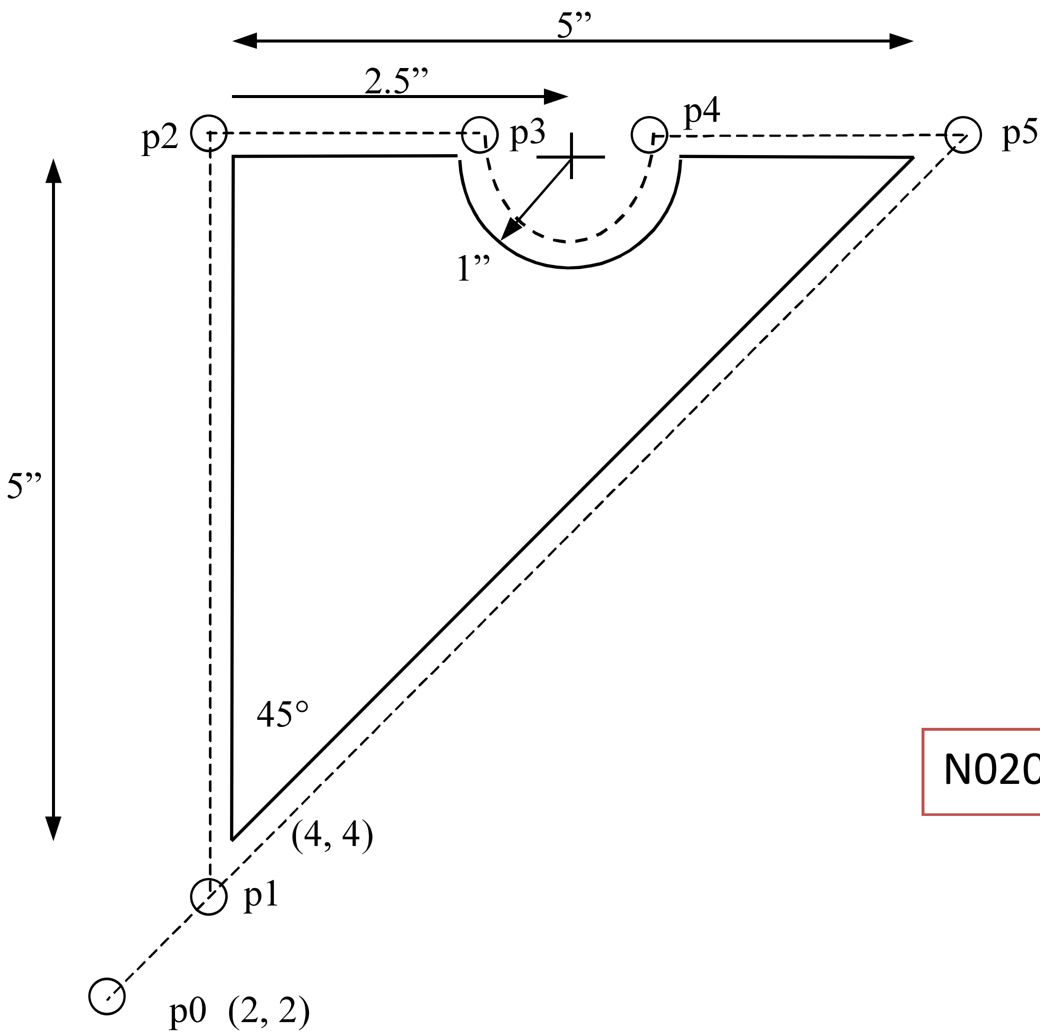
Feed in ipm

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 interpola on

Feed rate

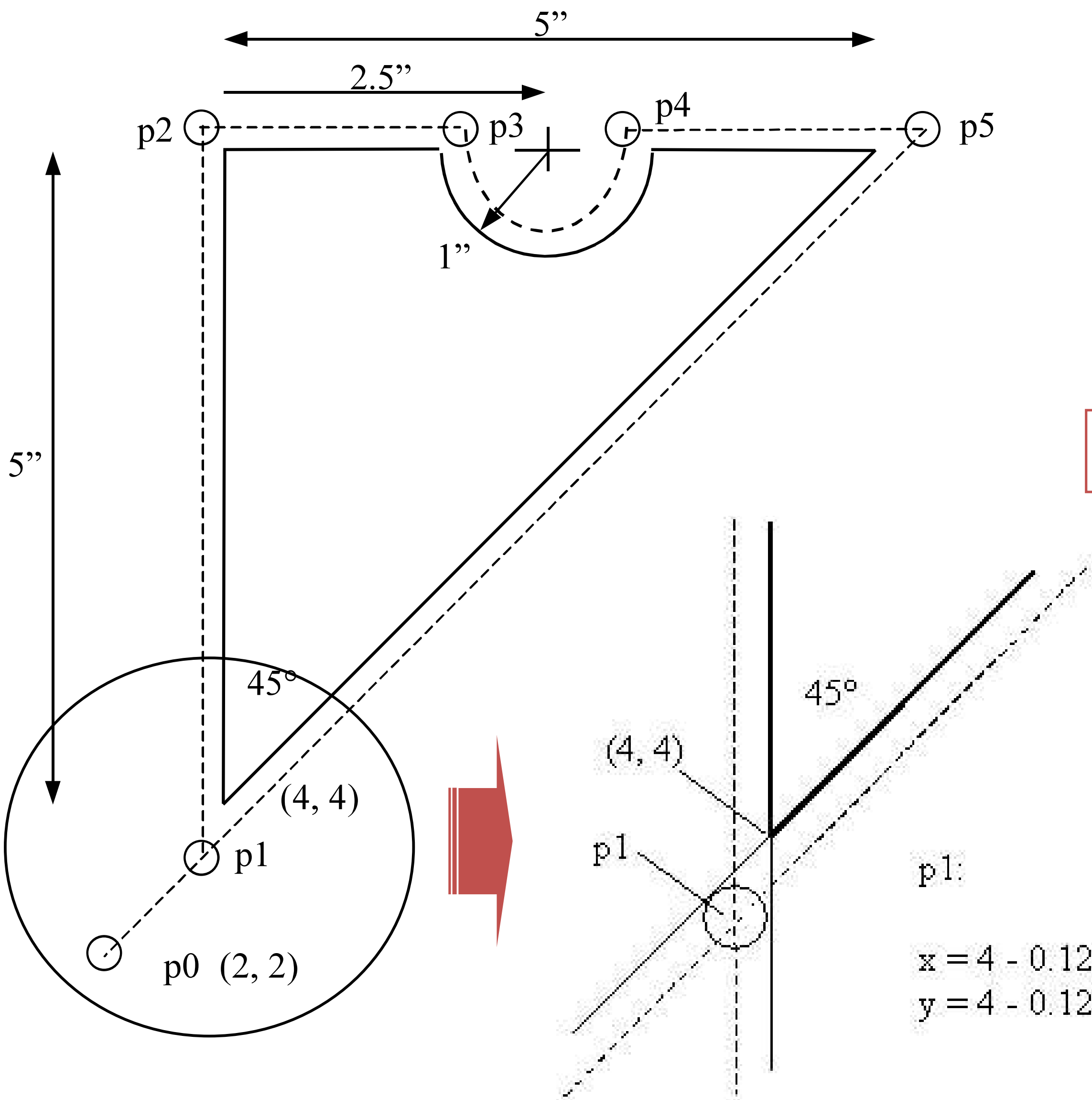
Spindle speed

`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

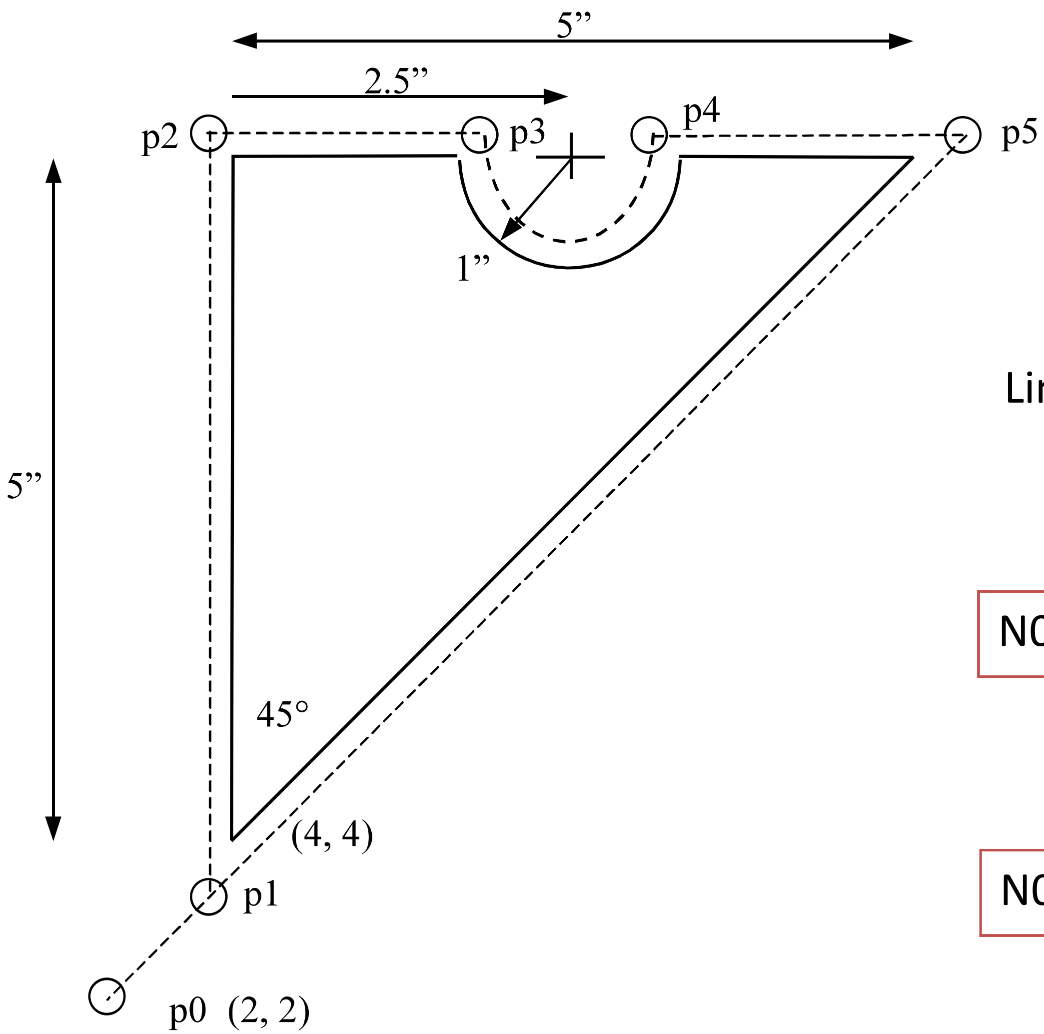
N030 G01 X3.875 Y3.698

p1:

$x = 4 - 0.125 = 3.875$

$y = 4 - 0.125/\tan 22.5 = 3.698$

4. Cut profile from p1 to p2



Linear interpola on

target coordinates

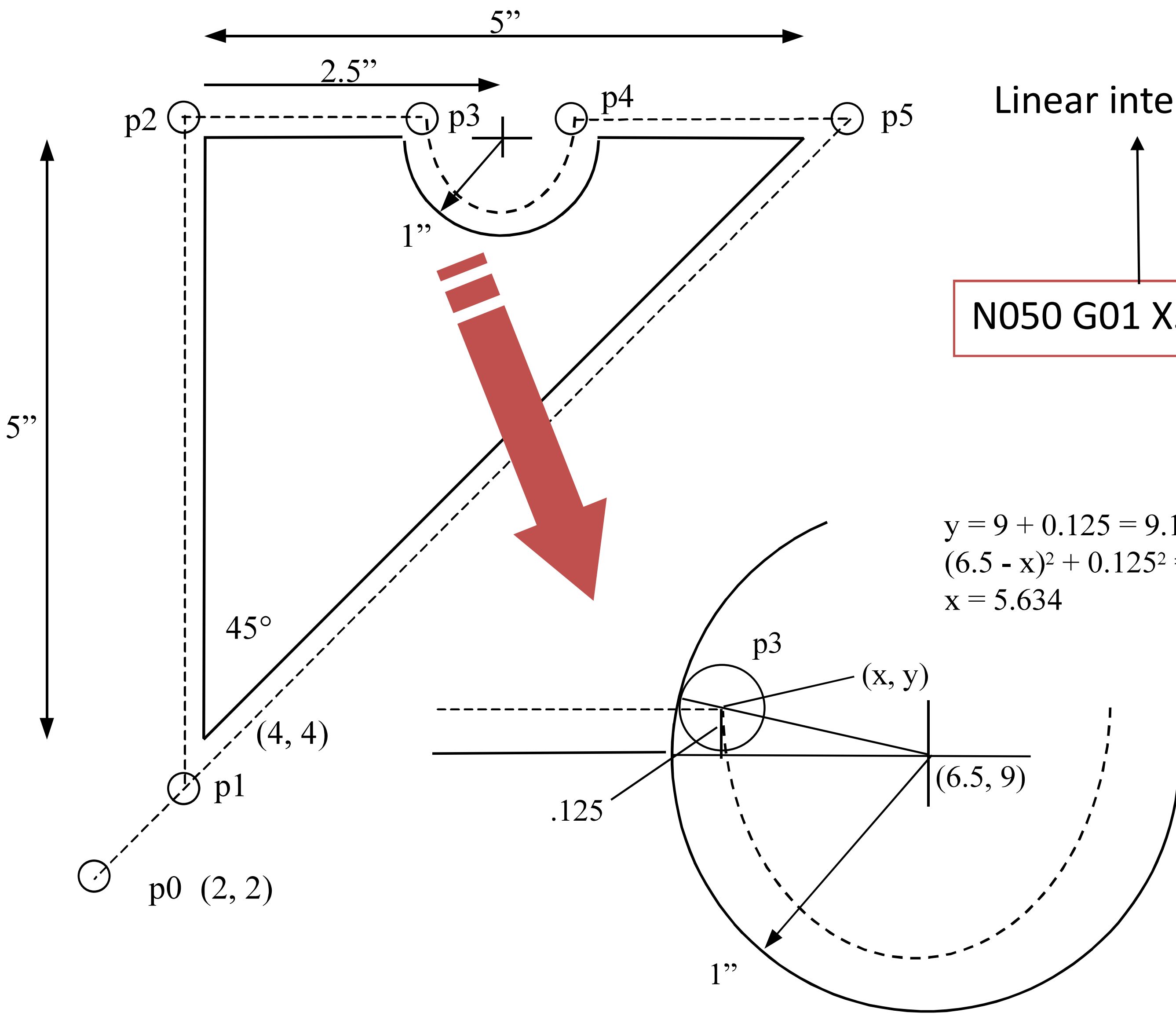
N040 G01 X3.875 Y9.125

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

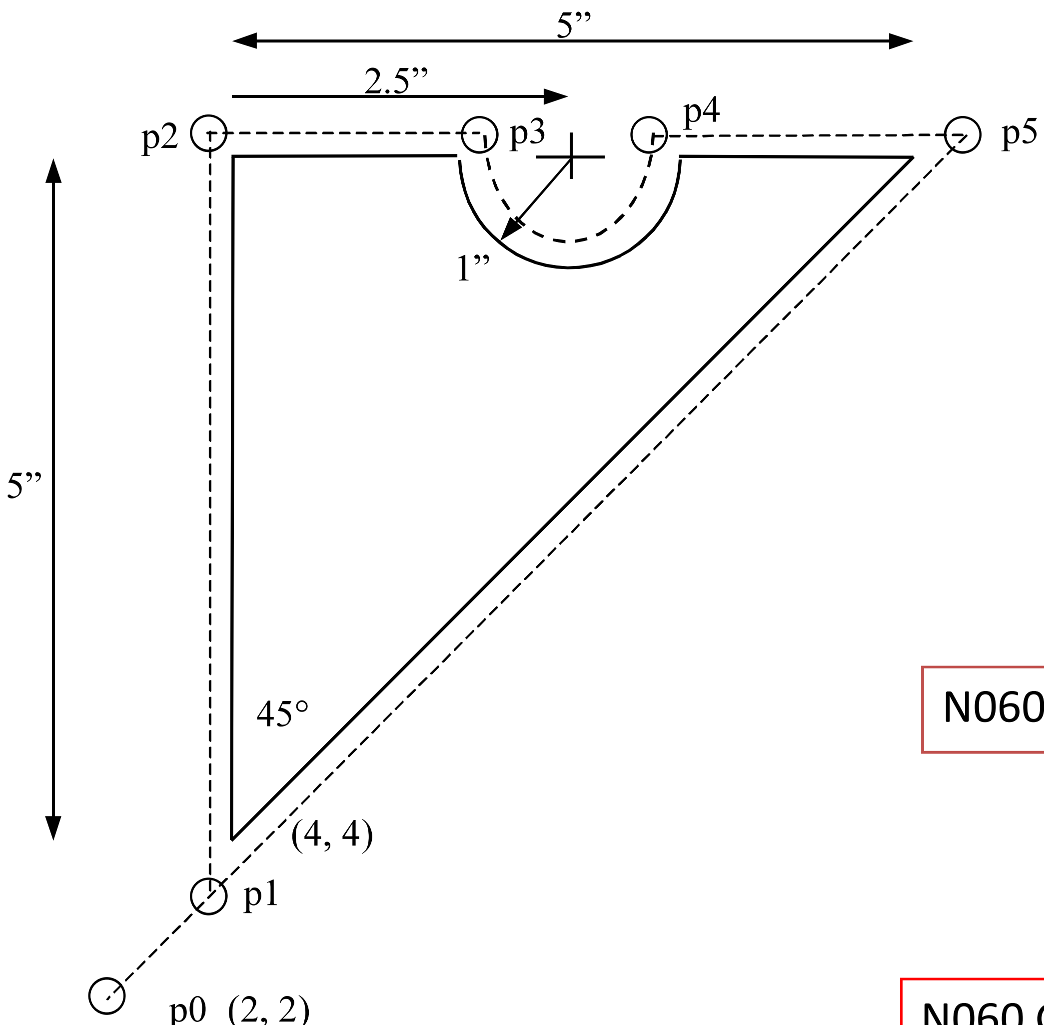
```
N050 G01 X5.634 Y9.125
```

$y = 9 + 0.125 = 9.125$

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

$x = 5.634$

6. Cut along circle from p3 to p4



circular interpola on, CCW mo on

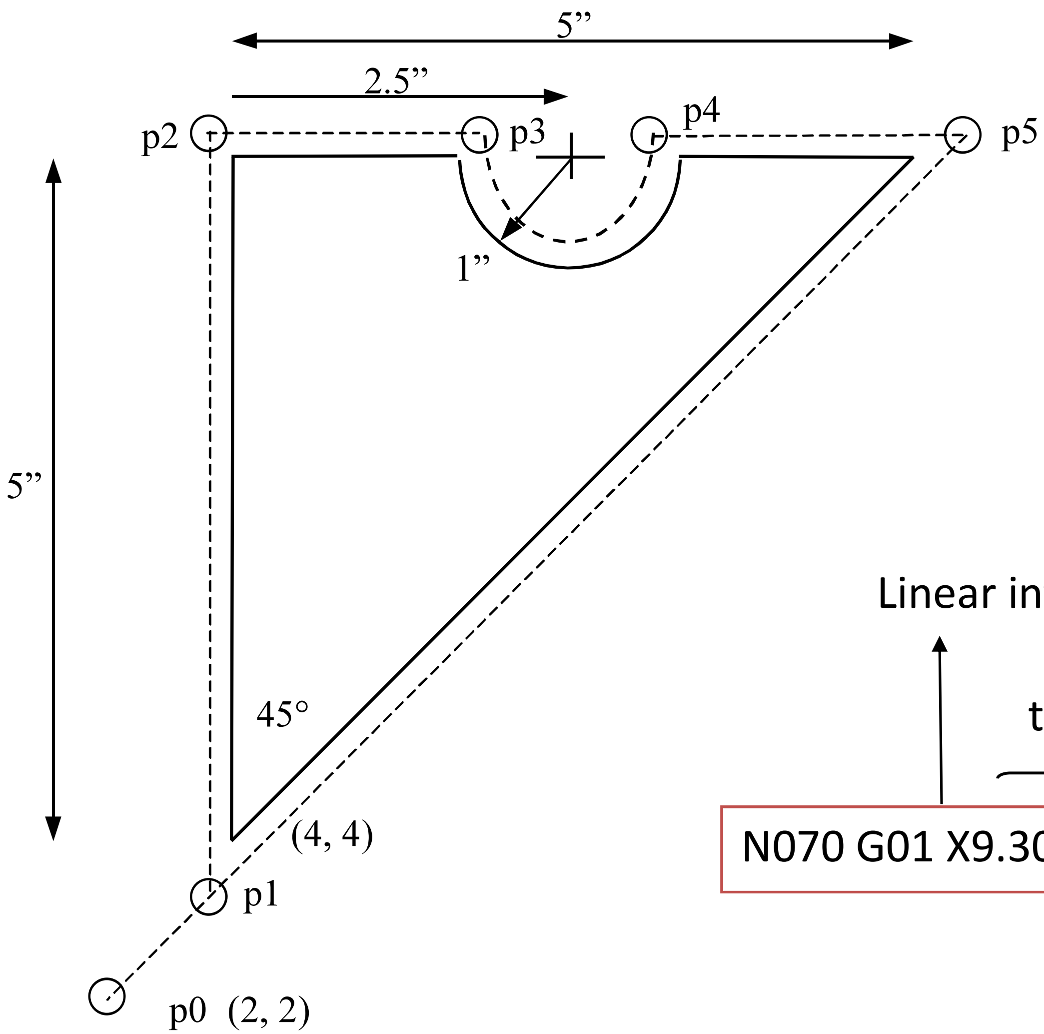
target coordinates

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

coordinates of center of circle

N060 G03 X7.366 Y9.125 I0.866 J-0.125

7. Cut from p4 to p5

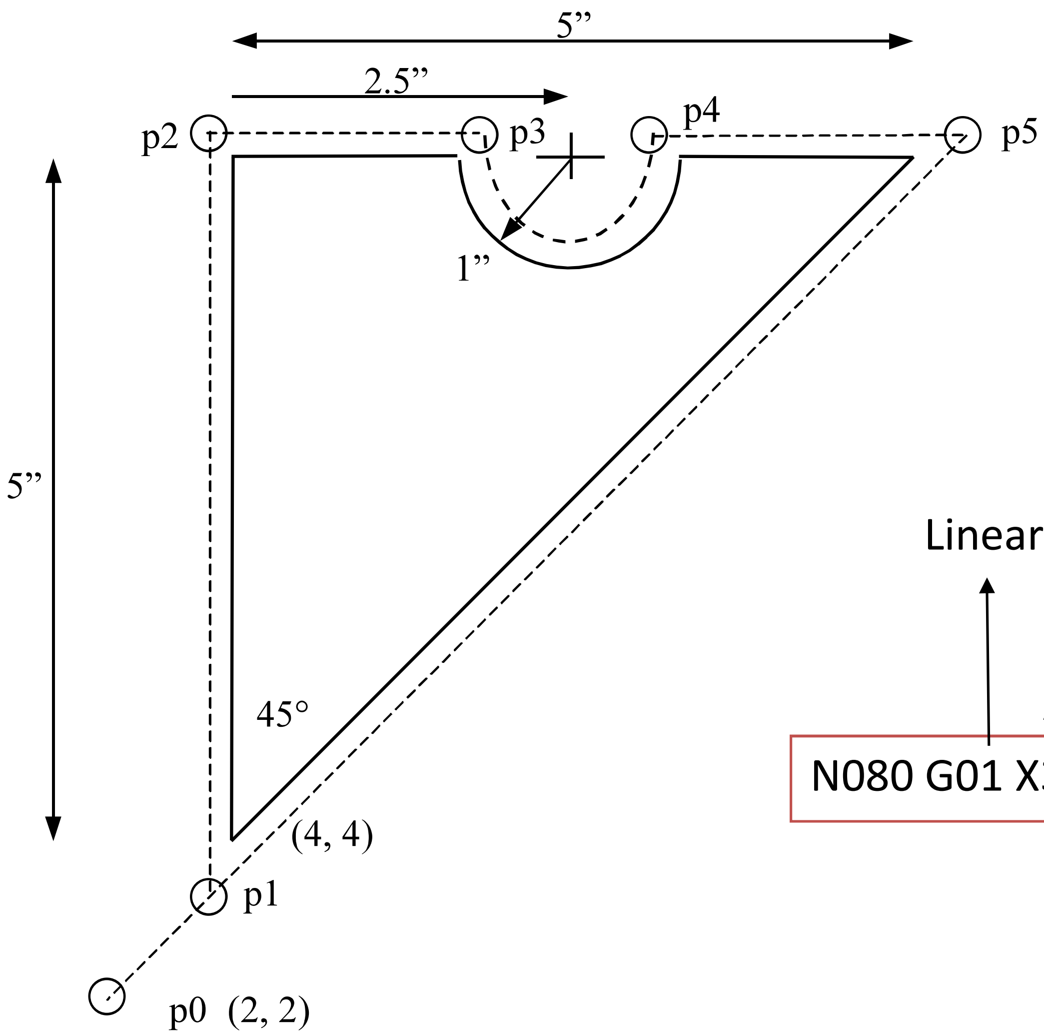


Linear interpola on

target coordinates (Y is unchanged)

N070 G01 X9.302

8. Cut from p5 to p1

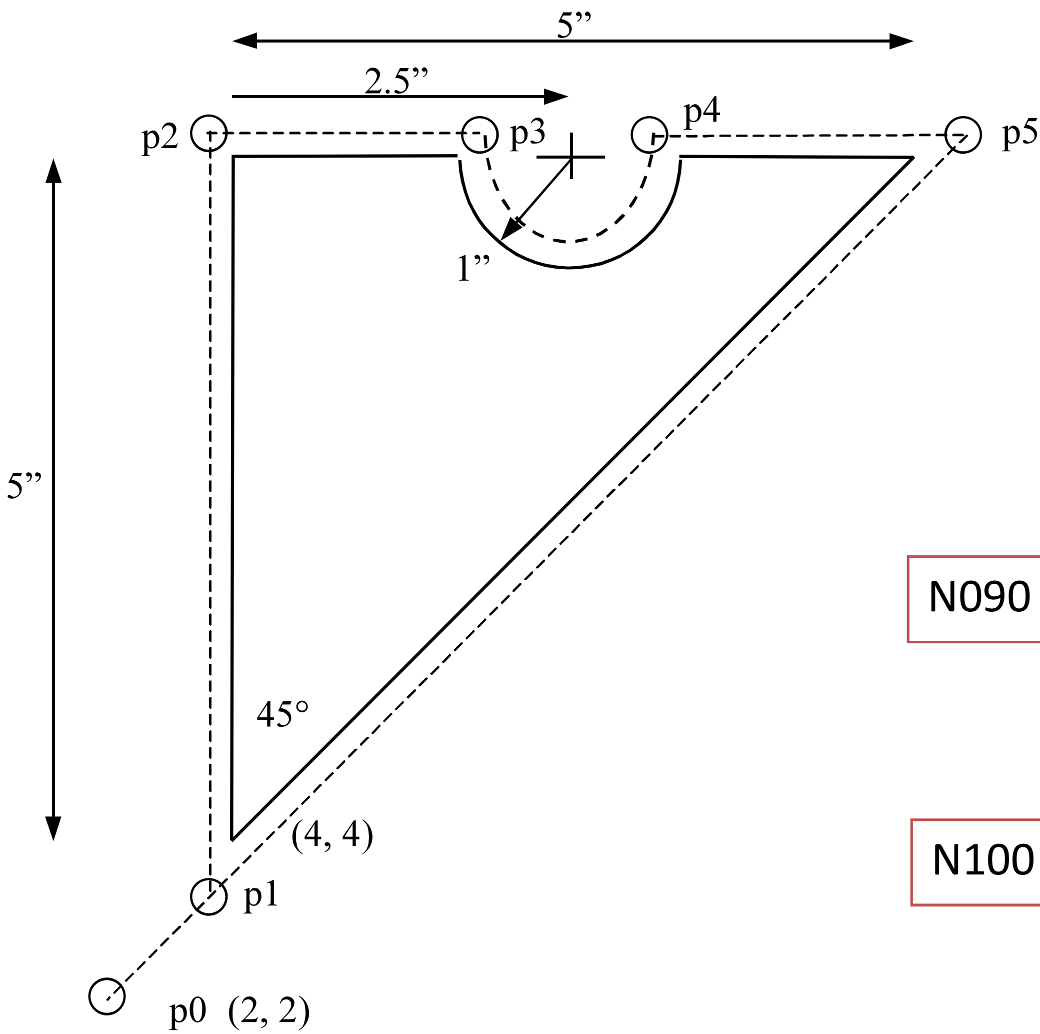


Linear interpola on

target coordinates (see step 3)

N080 G01 X3.875 Y3.698

9. Return to home position, stop program



Linear interpola on

target coordinates (see step 3)

N090 G01 X2.0 Y2.0 M30

end of data

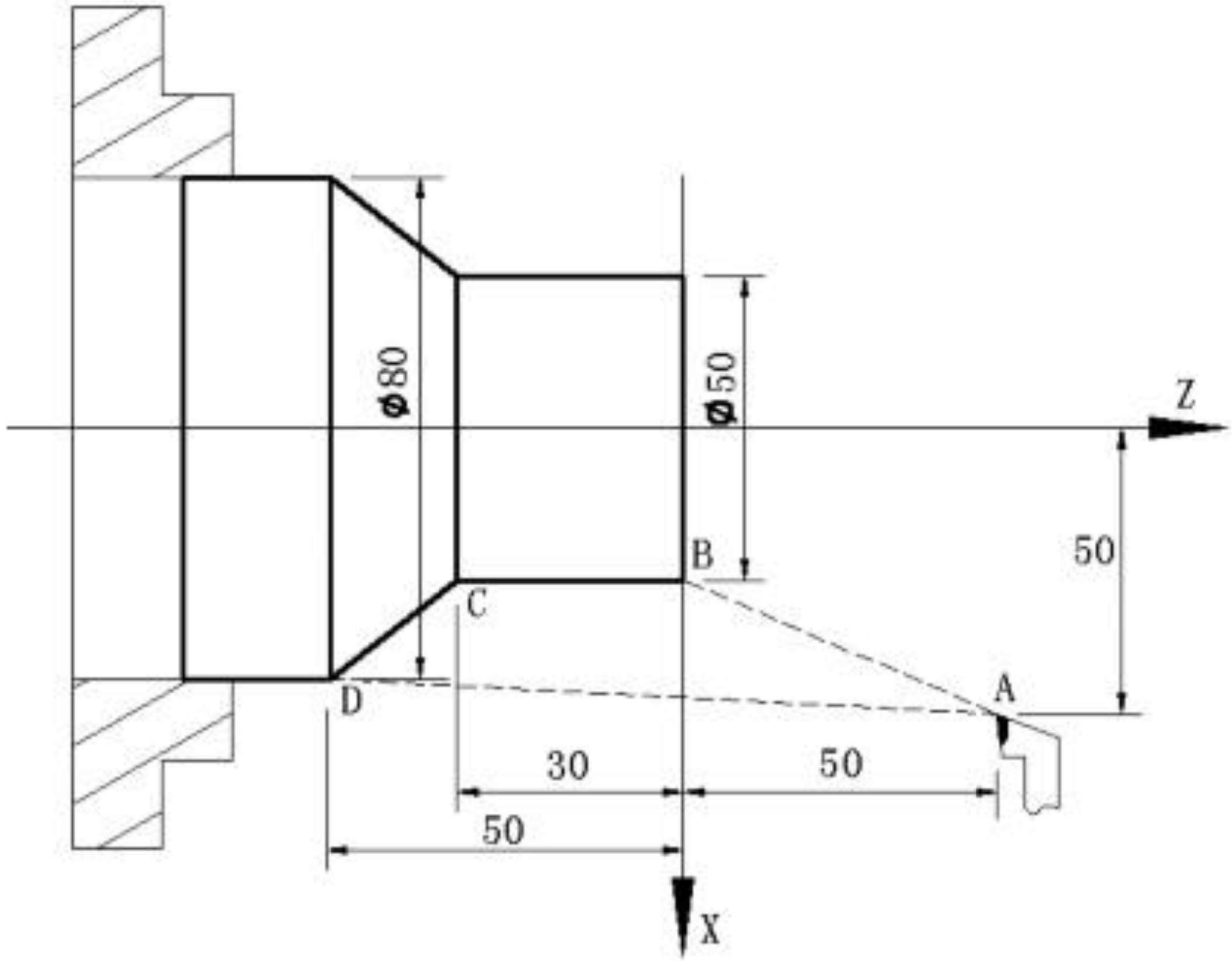
N100 M00

program stop

10. Complete RS-274 program

```
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 I0.866 J-0.125
N070 G01 X9.302
N080 G01 X3.875 Y3.698
N090 G01 X2.0 Y2.0 M30
```

Simple G Code Example CNC Lathe



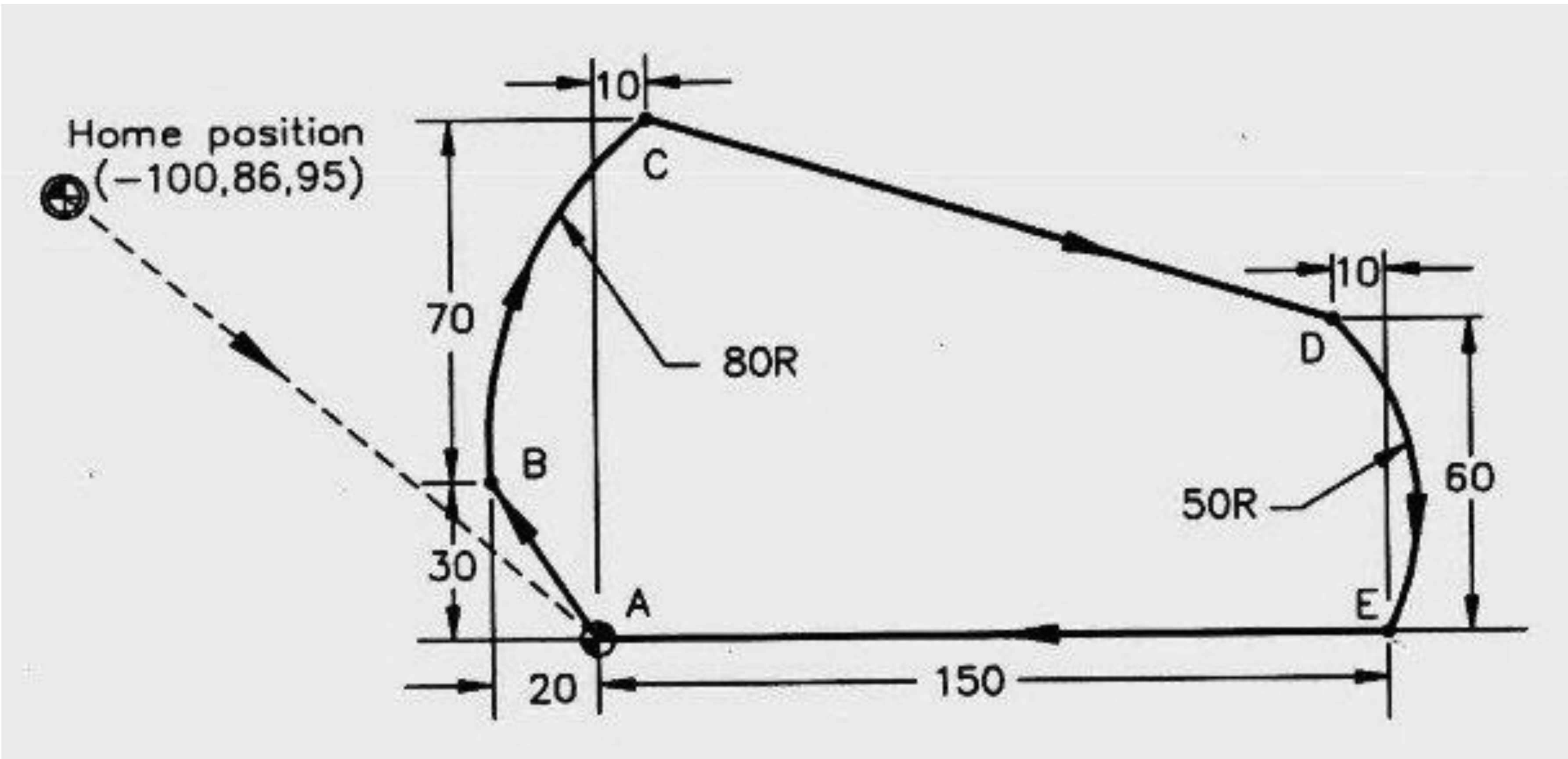
PART PROGRAM

```
N5 M12
N10 T0101
N15 G0 X100 Z50
N20 M3 S600
N25 M8
N30 G1 X50 Z0 F600
N40 Y-30 F200
N50 X80 Y-20 F150
N60 G0 X100 Z50
N70 T0100
N80 M5
N90 M9
N100 M13
N110 M30
```

Code Explanation

N5 Clamping workpiece
N10 Changing No.1 tool and executing its offset
N15 Rapidly positioning to A point
N20 Starting the spindle with 600 r/min
N25 Cooling ON
N30 Approaching B point with 600mm/min
N40 Cutting from B point to C point
N50 Cutting from C point to D point
N60 Rapidly retracting to A point
N70 Cancelling the tool offset
N80 Stopping the spindle
N90 Cooling OFF
N100 Releasing workpiece
N110 End of program, spindle stopping and Cooling OFF

CNC MILLING EXAMPLE

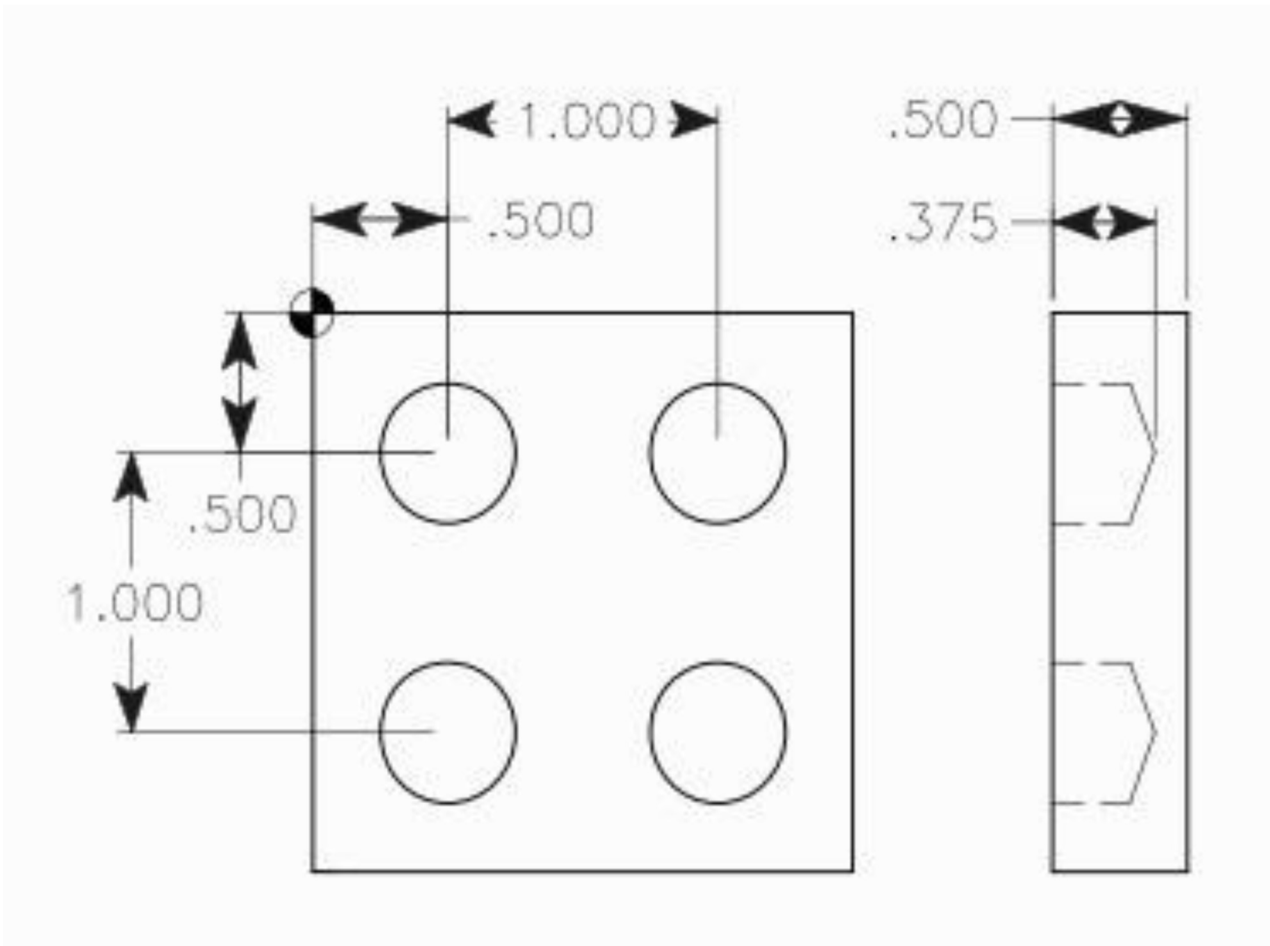


```
N5 G90 G71
N10 T1 M6
N15 G92 X-100 Y86 Z95
N20 G0 X0 Y0 S2500 M3
N25 Z12.5
N30 G1 Z-12.5 F150
N35 X-20 Y30
N40 G2 X10 Y100 R80
N45 G1 X140 Y60
N50 G2 X150 Y0 R50
N55 G1 X0 Y0
N60 G0 Z12.5
N65 G91 G28 Z0 M5
N70 G91 G28 X0 Y0
N75 M30
```

CODE EXPLANATION

N5 absolute posi oning, metric unit
N10 tool change to T1
N15 de ne work zero point at A
N20 rapid traverse to A, spindle on (2500 RPM, CW)
N25 rapid plunge to 12.5 mm above Z0
N30 feed to Z-12.5, feed rate 150 MMPM
N35 cut line AB to B
N40 cut arc BC to C
N45 cut line CD to D
N50 cut arc DE to E
N55 cut line EA to A
N60 rapid retract to Z12.5
N65 reference point return in Z direc on, spindle o
N70 reference point return in X and Y direc ons
N75 end of program

SAMPLE PROGRAM ON DRILLING



```
N1 T16 M06
N2 G90 G54 G00 X0.5 Y-0.5
N3 S1450 M03
N4 G43 H16 Z1. M08
N5 G81 G99 Z-0.375 R0.1 F9.
N6 X1.5
N7 Y-1.5
N8 X0.5
N9 G80 G00 Z1. M09
N10 G53 G49 Z0. M05
N11 M30
```

CODE EXPLANATION

N1- Tool change (M06) to tool no.16

N2- Tool rapidly moves (G00) to rst drilling posi on X0.5 Y-0.5 while taking into account Zero-o set-no. 1 (G54)

N3- Drill starts rota ng clockwise (M03) with 1450 rpm (S1450).

N4- Drill takes depth Z1. taking into account tool length compensa on (G43 H16), coolant is turned on (M08).

N5- Drilling cycle (G81) parameters, drill depth (Z) and cu ng feed (F) are given, with this command rst drill is made at current posi on (X0.5 Y-0.5).

N6- As drilling cycle con nues it’s work with every axis movement so next drill is done at X1.5

N7- Third drilling hole at Y-1.5

N8- Fourth drill at X0.5

N9- Drilling cycle is cancelled (G80), Coolant is turned o   (M09).

N10- Taking Machine-coordinate-system (G53) into account the drill is taken to Z0 posi on. Tool length compensa on is cancelled (G49), cu   er rota on is stopped (M05).

N11- CNC part-program is ended.

Typical PROGRAMMING - TURNING OPERATIONS

Write a part program for turning operations being carried out on a CNC turning center. Let us take an exercise:

Figure shows the final profile to be generated on a bar stock by using a CNC turning center. After studying the required part geometry and features, the main program can be written as follows.

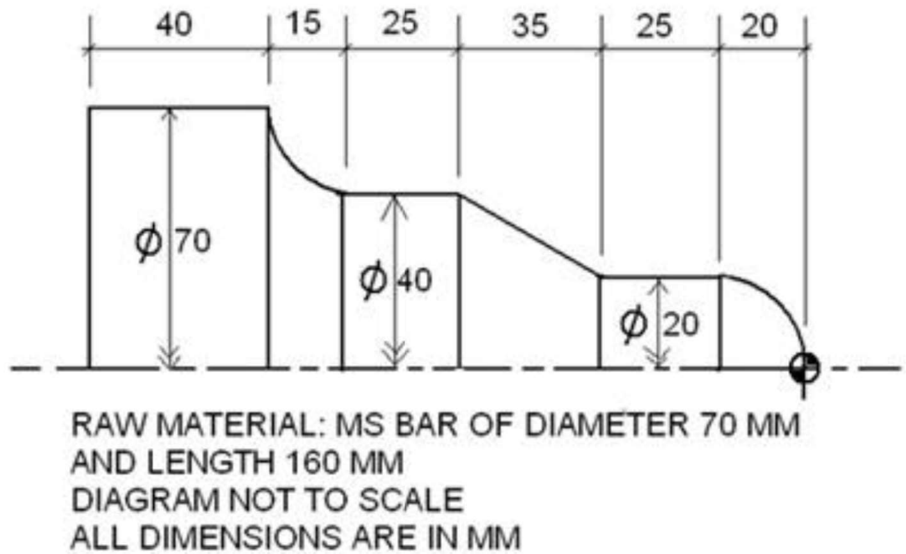


Figure A component to be turned.

Block 1		%
2		O0004
3	N10	G21
4	N20	G40 G90
5	N30	G54 X… Z…
6	N40	T0100 M42
7	N50	G96 S450 M03
8	N60	G00 G41 X72 Z0 T0101 M08
9	N70	G01 X0
10	N80	G00 Z5
11	N90	G42 X72
12	N100	G71 U1 R3
13	N110	G71 P120 Q190 U1 W1 F0.05
14	N120	G00 X0
15	N130	G01 Z0
16	N140	G03 X20 Z-20
17	N150	G01 Z-45
18	N160	X40 Z-80
19	N170	Z-105
20	N180	G02 X70 Z-120
21	N190	G01 X75
22	N200	G00 X100 Z20
23	N210	G70 P120 Q190 F0.03
24	N220	G00 G40 X100 Z20 T0100
25	N230	M09
26	N240	M30
27		%