

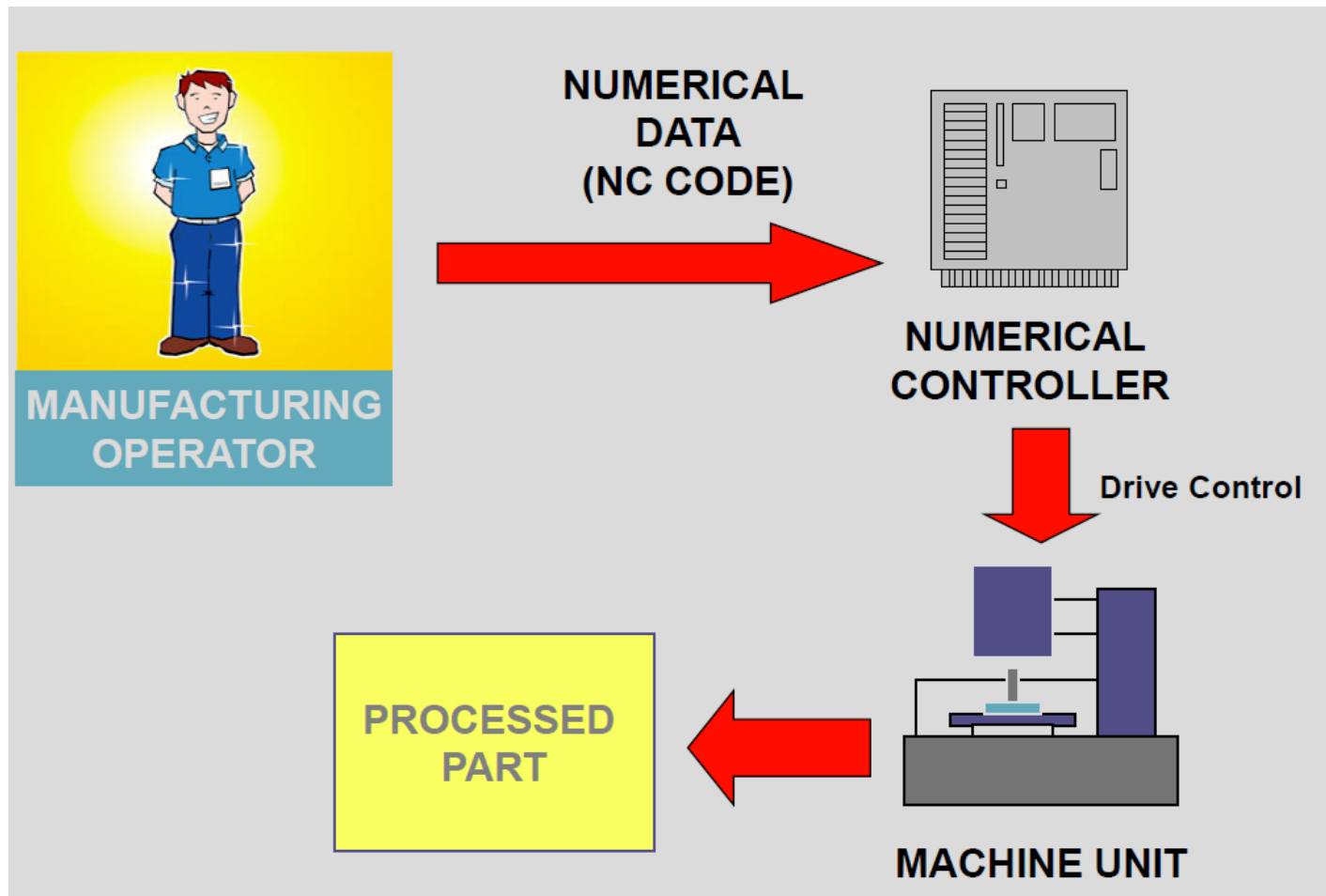
Computer Numerical Control (CNC)



Prof. Sounak K. Choudhury,
Mechanical Engineering Department, IIT Kanpur



Computer Numerical Control



History

- 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)
- 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



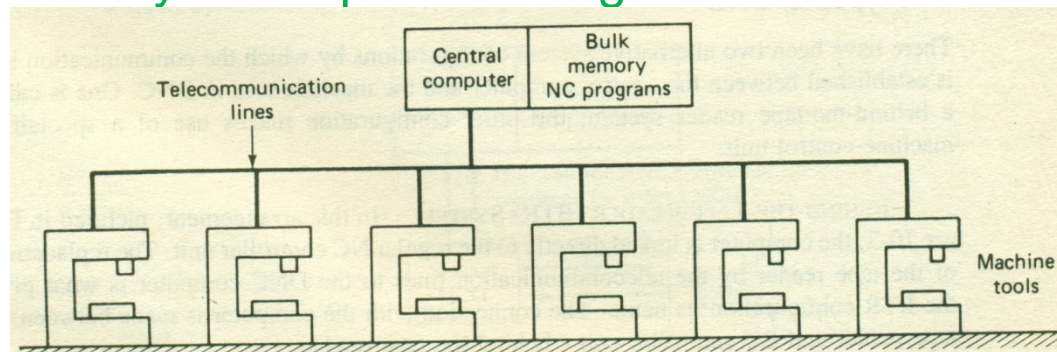
Computer Numerical Control (CNC)

➤ Numerical Control (NC) is a form of Programmable Automation in which the processing equipment is controlled by means of numbers, letters and other symbols which are coded in an appropriate format to define a program of instructions for a particular workpart or job.

Types of Numerical Control

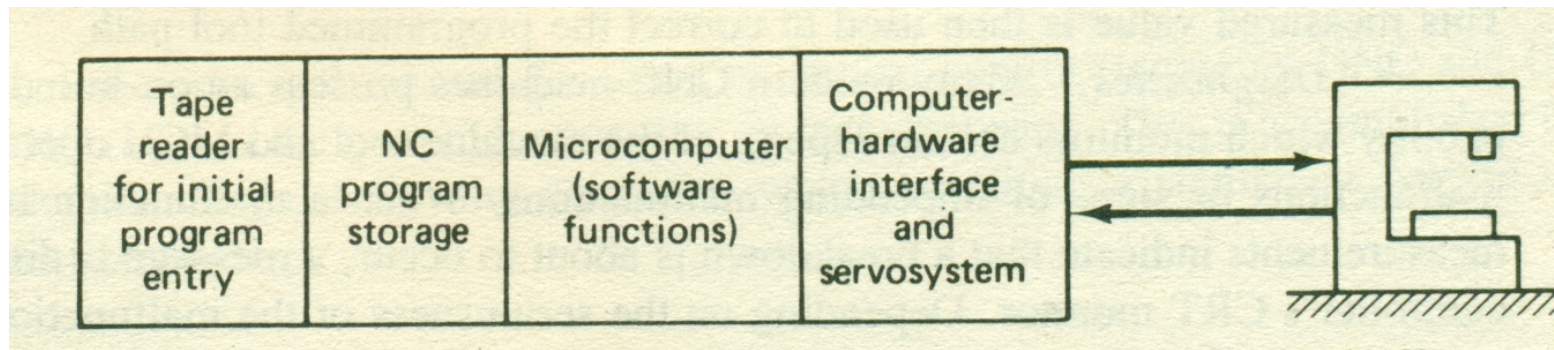
- ❑ **Conventional Numerical Control (NC)**
- ❑ **Direct Numerical Control (DNC)**
- ❑ **Computer Numerical Control (CNC)**

➤ Direct Numerical Control (DNC) is a Manufacturing System in which a number of machines are controlled by a computer through direct connection and in real time.



Computer Numerical Control (CNC)

Computer Numerical Control (CNC) is an NC system using a dedicated microcomputer as the Machine Control Unit. The conventional NC controller unit is hard-wired (control features can not be easily altered to incorporate improvement into the unit).

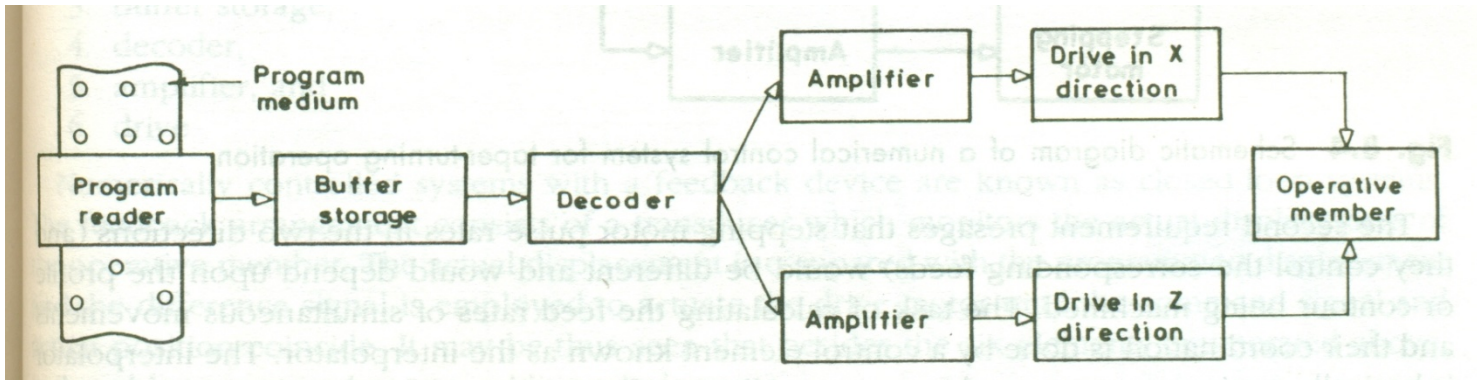
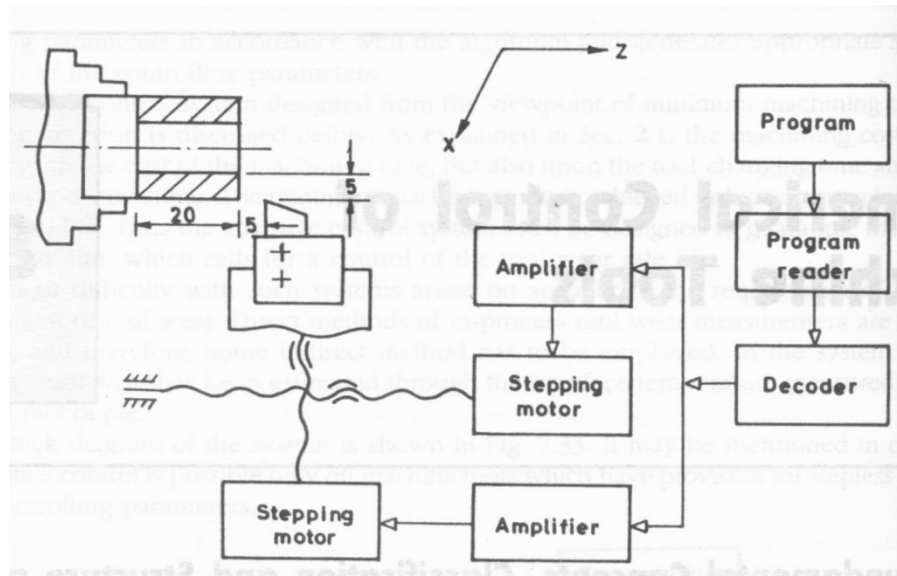


Advantages of CNC

- Reduces time for delivery of part
- Reduces scrap rate of material
- Reduces tooling costs
- Reduces layout time
- Increases machine and tool life
- Reduces storage problems
- Less setup time
- Reduces actual machining time Allows rapid design changes in part Less jigs and fixtures are needed



Principle of NC Machines



MAJOR COMPONENTS RELATED TO CNC MACHINE TOOLS

Any CNC machine tool essentially consists of the following parts:

❑ Part program:

- 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.



□ Program input device

- The program input device is the means for part program to be entered into the CNC control.
- Three commonly used program input devices are punch tape reader, magnetic tape reader, and computer via RS-232-C communication.



□ Machine Control Unit

The machine control unit (MCU) is the heart of a CNC system. It is used to perform the following functions:

- To read the coded instructions.
- To decode the coded instructions.
- To implement interpolations (linear, circular, and helical) to generate axis motion commands.
- To feed the axis motion commands to the amplifier circuits for driving the axis mechanisms.
- To receive the feedback signals of position and speed for each drive axis.
- To implement auxiliary control functions such as coolant or spindle on/off and tool change.



□ Machine Tool

- CNC controls are used to control various types of machine tools.
- Regardless of which type of machine tool is controlled, it always has a slide table and a spindle to control position and speed.
- The machine table is controlled in the X and Y axes, while the spindle runs along the Z axis.



❑ Feed Back System

- The feedback system is also referred to as the measuring system.
- It uses position and speed transducers to continuously monitor the position at which the cutting tool is located at any particular instant.
- The MCU uses the difference between reference signals and feedback signals to generate the control signals for correcting position and speed errors.



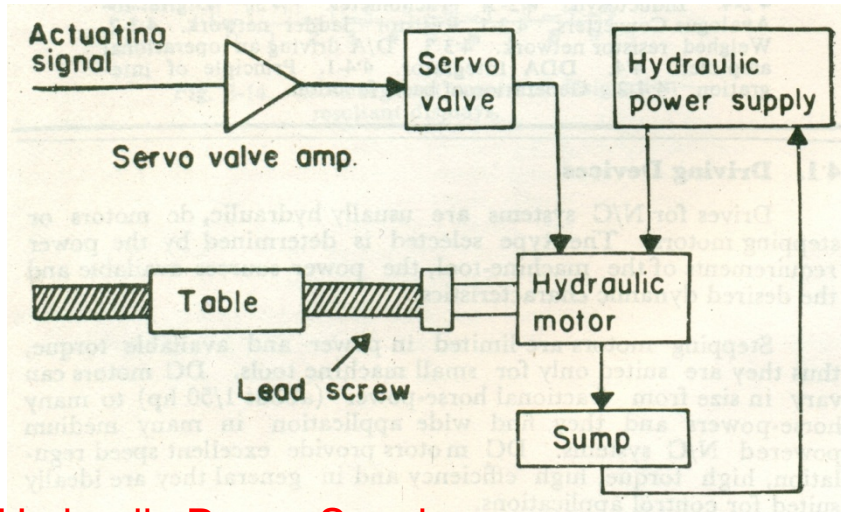
□ Drive System

- Drives are used to provide controlled motion to CNC elements
- A drive system consists of amplifier circuits, drive motors, and ball lead-screws.
- The MCU feeds the control signals (position and speed) of each axis to the amplifier circuits.
- The control signals are augmented to actuate drive motors which in turn rotate the ball lead-screws to position the machine table.



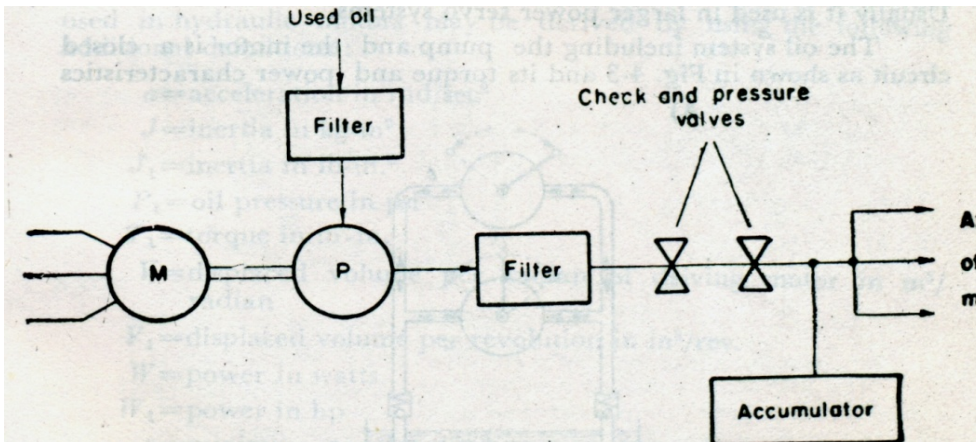
Drives of NC Machines

Hydraulic Drives: Used in relatively higher-powered NC systems.



- Can deliver large power being relatively small in size.
- Leakage of oil is a disadvantage
- Oil must be kept clean

Hydraulic Power Supply:

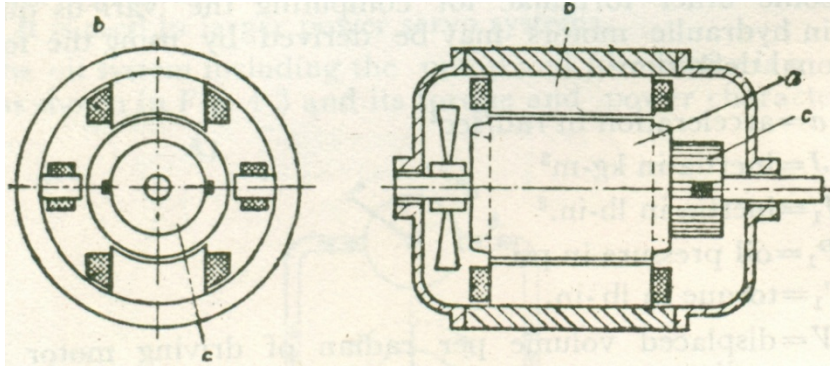


- Check valve for restricting flow from the accumulator back into the pump
- Pressure valve for controlling the supply pressure to servo system
- Accumulator stores hydraulic energy and for smoothing the pulsating flow.



Drives of NC Machines

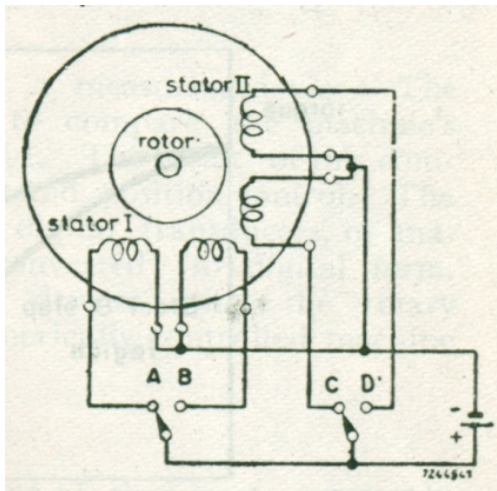
DC Motors:



a – Rotor; b – Stator Winding; c - Commutator

- Mostly used drive for NC machines since its voltage and speed can be varied smoothly and easily regulated.
- Principle of operation is based on a rotation of an armature winding within a magnetic field.
- Can be used as a generator or as a motor.

Stepper Motors:



- An incremental digital control device.
- Translates an input pulse sequence into a proportional angular movement, rotating one angular increment (step) for each input pulse.
- The shaft position is determined by the number of pulses, and its velocity is determined by the pulse frequency.
- The shaft speed in steps/sec is equal to the incoming frequency in pulse/sec.
- An electronic switch is used as a driving unit.



Types of CNC

Based on Motion Type:

- Point-to-Point, Straight Cut or Continuous path

Based on Control Loops:

- Open loop or Closed loop

Based on Power Supply:

- Electric or Hydraulic or Pneumatic

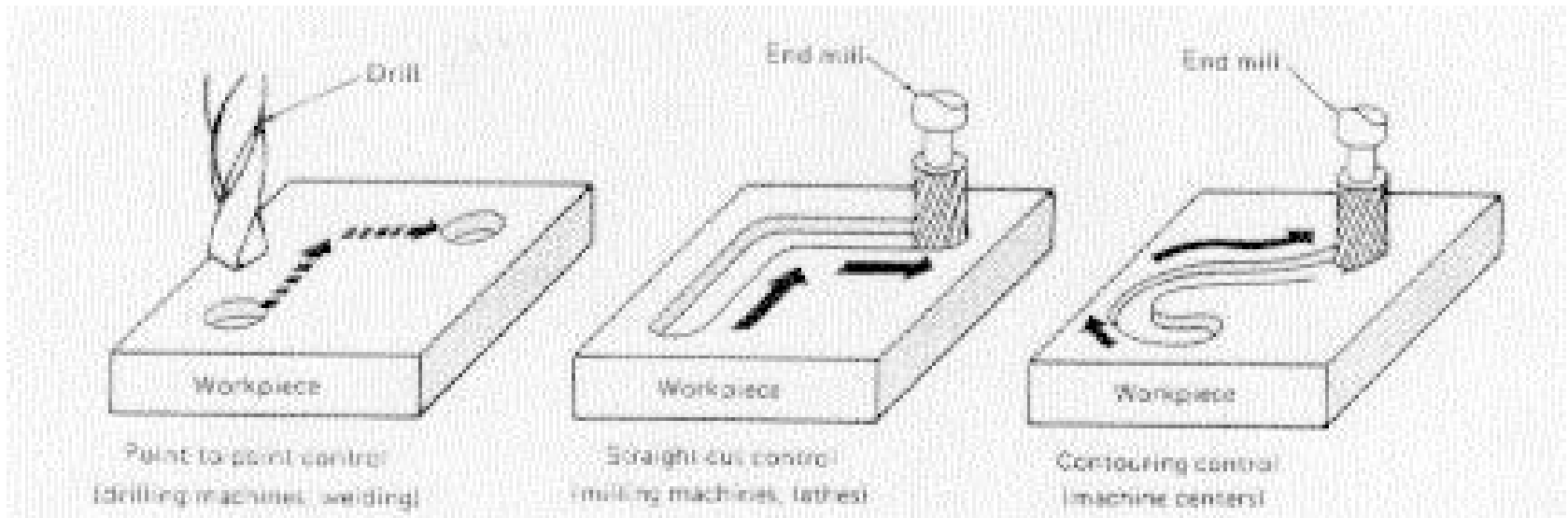
Based on Positioning System

- Incremental or Absolute

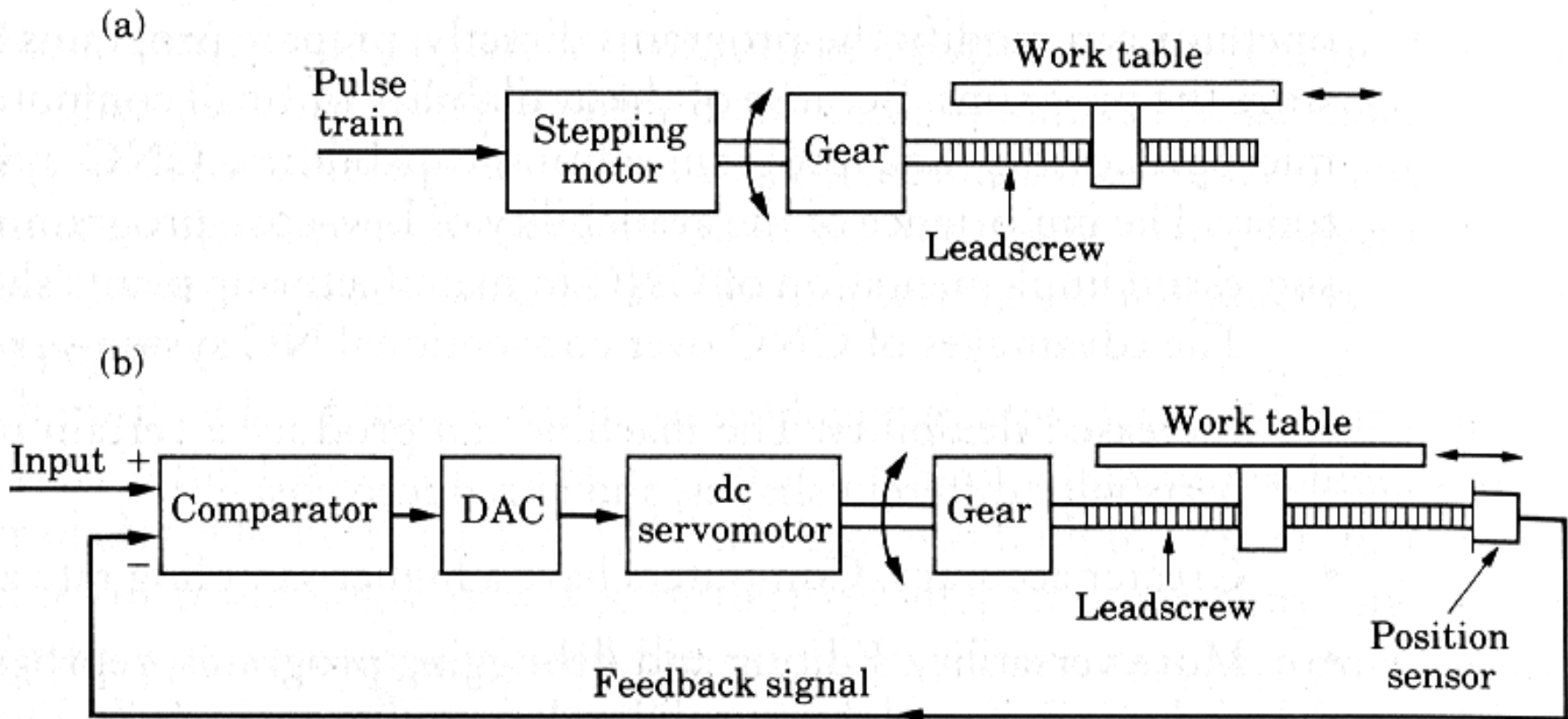


Type of CNC based on Motion Type

- **Point to Point** - No contouring capability
- **Straight cut control**- one axis motion at a time is controlled for machining
- **Contouring**- multiple axis's controlled simultaneously



Type of CNC based on Control Loop



(a) Open loop system (b) Closed loop system



Open Loop vs. Closed Loop System

Open-Loop Limitations

- Control unit “assumes” desired position is achieved
- No positioning compensation
- Typically, a lower torque motor

Open-Loop Advantages

- Less complex, Less costly, and lower maintenance costs



Open Loop vs. Closed Loop System

Closed-Loop Advantages

- DC motors have the ability to reverse instantly to adjust for position error
- Error compensation allows for greater positional accuracy (0.0001")
- DC motors have higher torque ranges against stepper motors

Closed-loop limitations

- High Cost



PART PROGRAMMING



INTRODUCTION

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

It consists of:

- Information about part geometry
- Motion statements to move the cutting tool
- Cutting speed
- Feed
- Auxiliary functions such as coolant on and off, spindle direction



❑ CNC program structure

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



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 known 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

destination coordinates

preparatory function

dist to center of circle

feed rate

spindle speed

tool

Other function



Various Codes in CNC programming

TABLE 5.A1 Address Words Used In NC Programming

Address	Meaning
A	Rotation about the x-axis
B	Rotation about the y-axis
C	Rotation about the z-axis
F	Feed rate command
G	Preparatory function
I	Circular interpolation: x-axis offset
J	Circular interpolation: y-axis offset
K	Circular interpolation: z-axis offset
M	Miscellaneous commands
N	Sequence number
R	Radius of arc or circle
S	Spindle speed
T	Tool number
U	Supplemental coordinate parallel to x-axis
V	Supplemental coordinate parallel to y-axis
W	Supplemental coordinate parallel to z-axis
X	x-axis data
Y	y-axis data
Z	z-axis data

Note: Additional nonstandard address words may be used by an individual manufacturer to meet specific needs.

TABLE 6.A3 Miscellaneous Functions

Code	Usage
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle on—clockwise rotation
M04	Spindle on—counterclockwise rotation
M05	Spindle off
M06	Tool change
M07	Mist coolant on
M08	Flood coolant on
M09	Coolant off
M10	Clamp on
M11	Clamp off
M13	Spindle on CW* and coolant on
M14	Spindle on CCW* and coolant on
M15	Rapid slide motion in positive direction
M16	Rapid slide motion in negative direction
M19	Stop spindle in oriented position
M30	End of tape—rewind
M31	Interlock bypass
M32–M35	Constant cutting speed codes
M36	Feed range 1
M37	Feed range 2
M38	Spindle speed range 1
M39	Spindle speed range 2
M50	Coolant no. 3 on
M51	Coolant no. 4 on
M55	Linear tool shift position 1
M56	Linear tool shift position 2
M60	Change workpiece
M61	Linear workpiece shift position 1
M62	Linear workpiece shift position 2
M68	Clamp workpiece
M69	Unclamp workpiece
M71	Angular workpiece shift position 1
M72	Angular workpiece shift position 2
M78	Clamp slide
M79	Unclamp slide

Note: Omitted sequences represent nonstandard codes that may be used by individual manufacturers to implement other functions deemed important.

* CW = clockwise; CCW = counterclockwise.



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



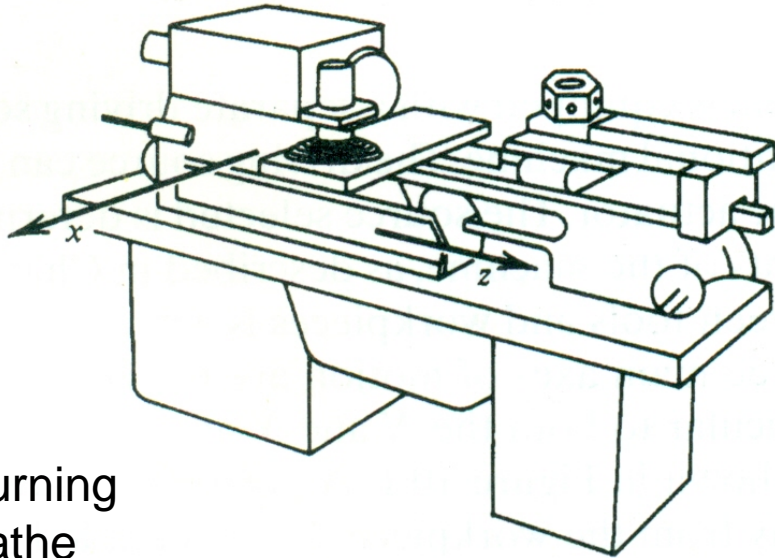
List of M codes

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.

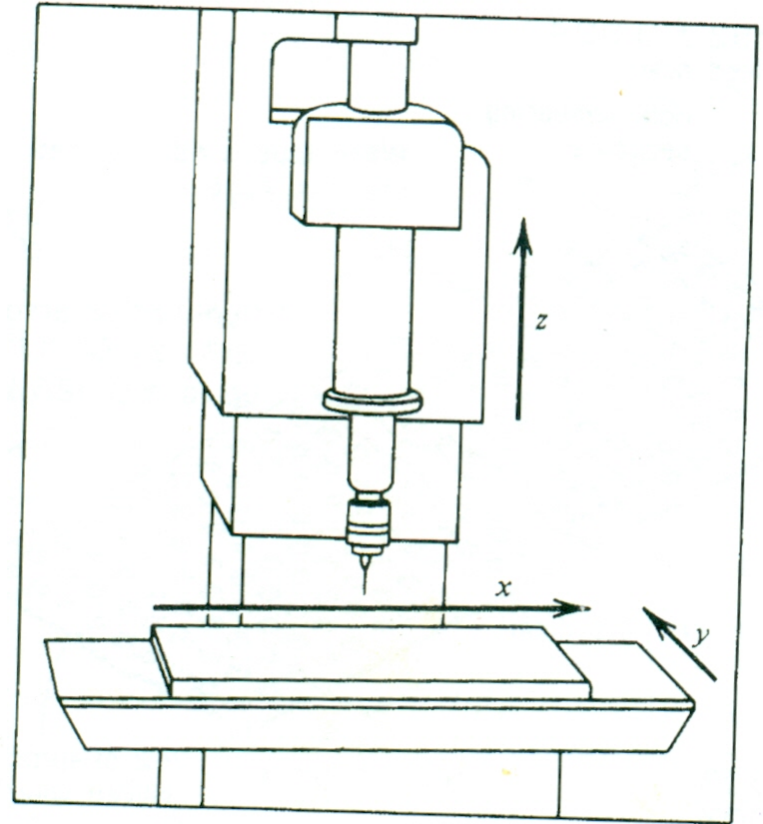
M-codes	Function
M00	Optional program stop automatic
M01	Optional program stop request
M02	Program end
M03	Spindle ON clock wise (CW)
M04	Spindle ON counter clock wise (CCW)
M05	Spindle stop
M06	Tool change
M07	Mist coolant ON (coolant 1 ON)
M08	Flood coolant ON (coolant 2 ON)
M09	Coolant OFF
M30	End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



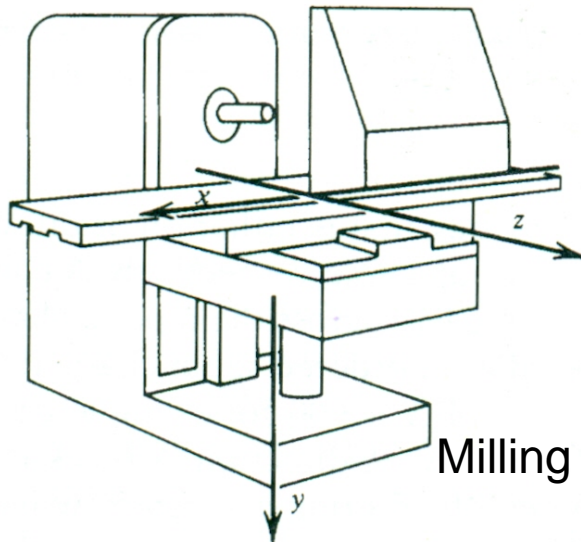
CNC Coordinates



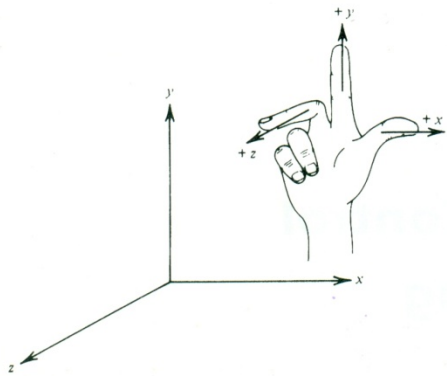
Turning
Lathe



Drilling



Milling

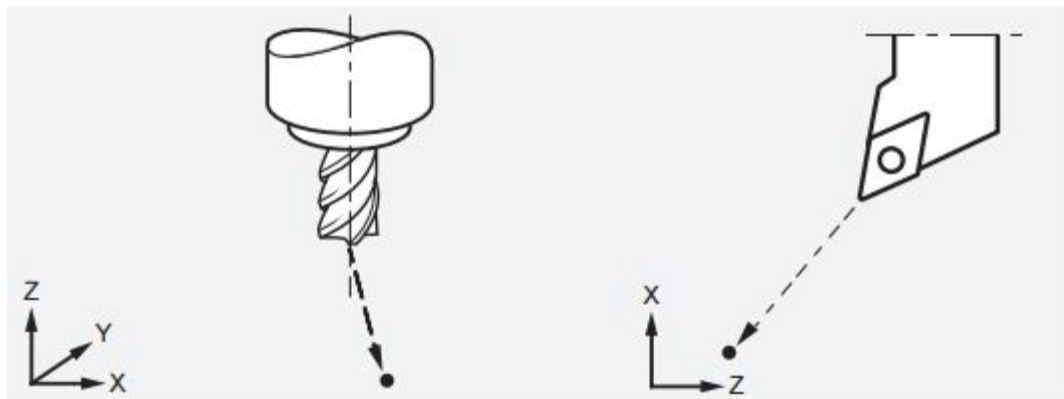


G00 Rapid traverse

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.

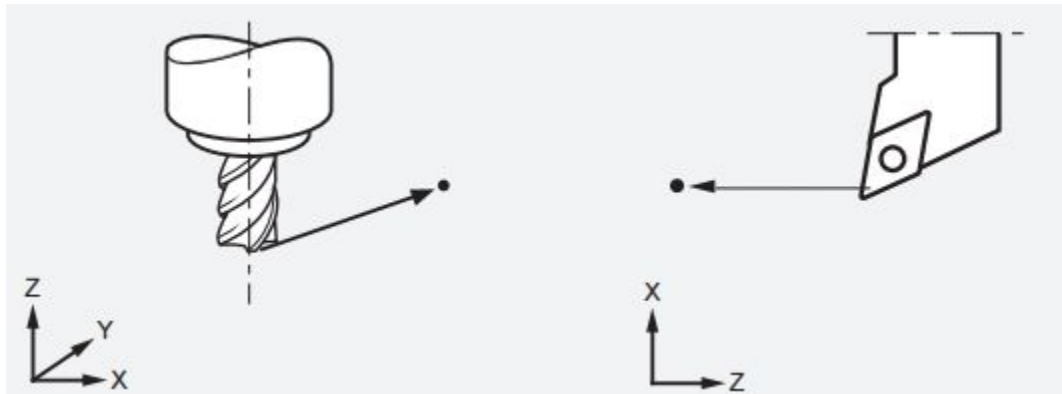
Syntax: N010 [G90/G91] G00 X10 Y10 Z5



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

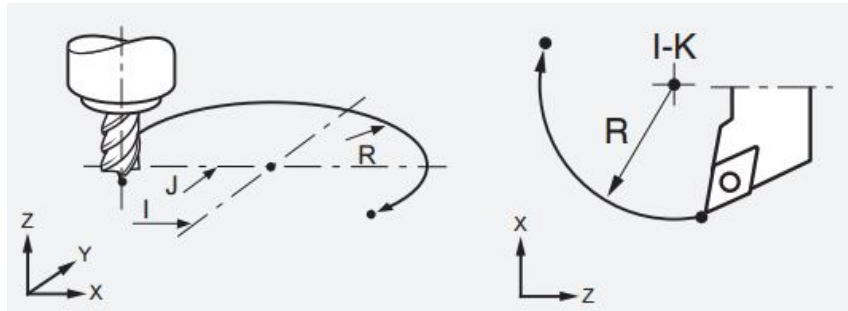
N__ G02/03 X__ Y__ Z__ I__ J__ K__ F__ using the arc center

or

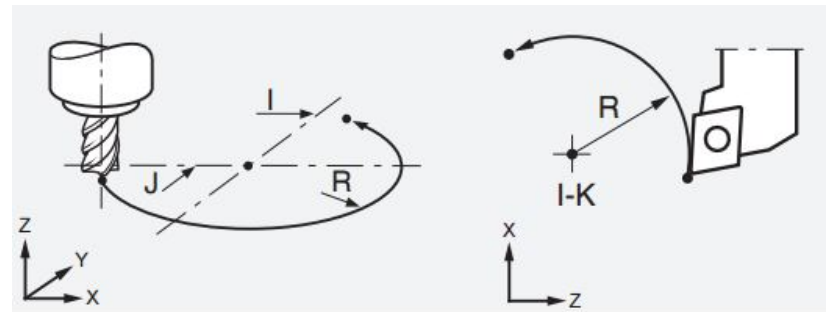
N__ G02/03 X__ Y__ Z__ R__ F__ using the arc radius

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

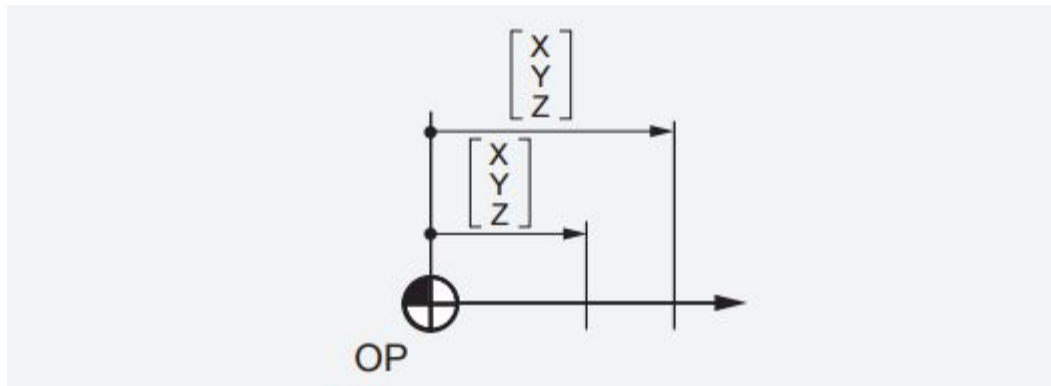


G03 moves along a CCW arc



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.



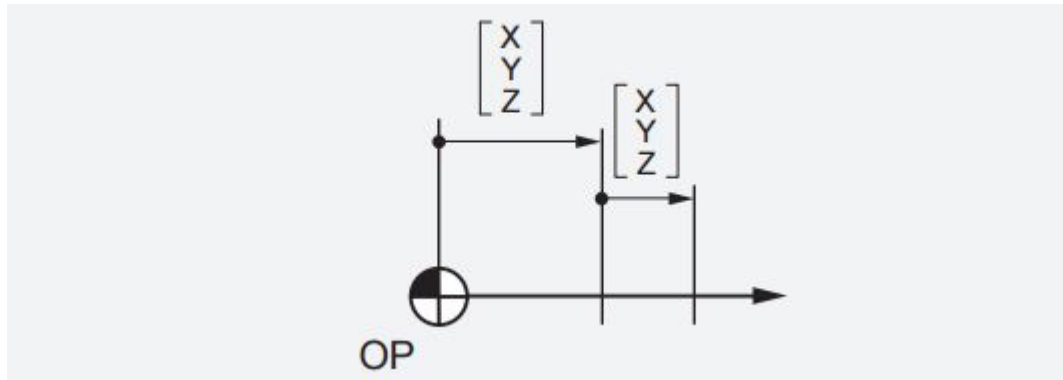
Syntax: N.. G90 X.. Y.. Z.. A.. B.. C..



G91 INCREMENTAL POSITION COMMAND

- This code is modal and changes the way axis motion commands are interpreted. G91 makes all subsequent commands incremental. Zero point shifts with the new position.

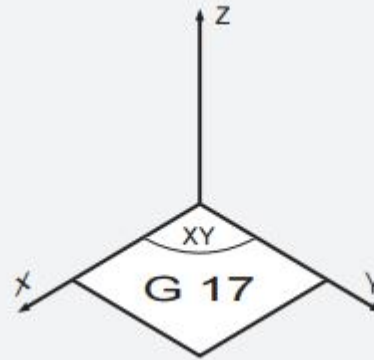
Syntax: N.. G91 X.. Y.. Z.. A.. B.. C..



G 17 G18 G19 : PLANE SELECTION

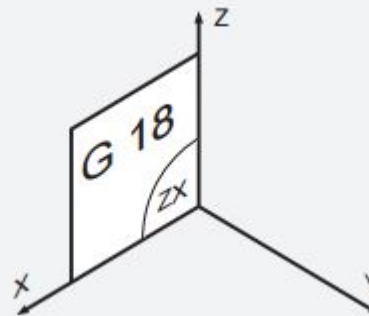
G 17 : XY plane selection

Syntax: N.. G17



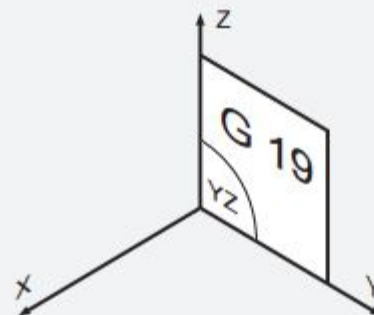
G 18 : ZX plane selection

Syntax: N.. G18



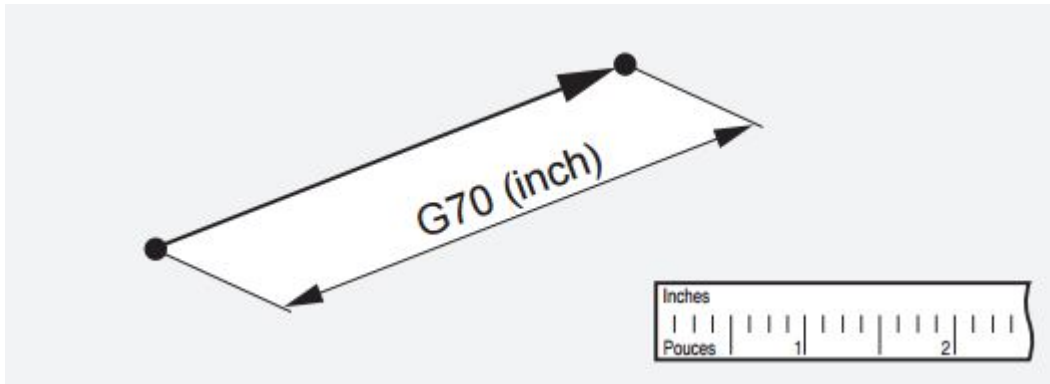
G 19 : ZY plane selection

Syntax: N.. G19



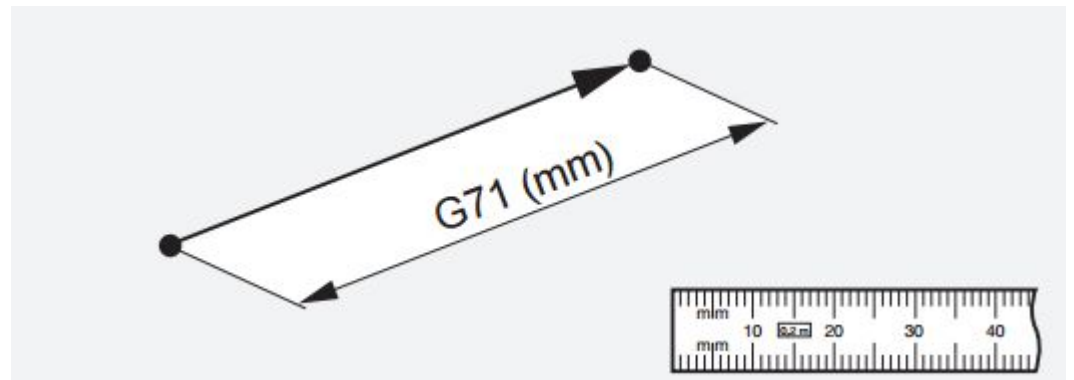
Syntax: N020 G17 G75 F6.0 S300 T1001 M08





G 70 Inch data input

G 71 Metric data input



Syntax : N010 G70 G90 G94 G97 M04



CNC Programming Example

N010 G90 G71
 N020 G97 G95 T01
 N030 G00 X51 Z 66
 N040 M03 S1000 M08
 N050 G01 X46 F1
 N060 Z 23
 N070 X 46.5 Z 23.5
 N080 G00 Z66
 N090 G01 X42 F1
 N100 Z 25
 N110 X 42.5 Z25.5

Absolute, Metric
 Speed Rev/min, Feed mm/rev
 Rapid Move to tool start position
 Turn on spindle and coolant
 Position tool for first cut
 First rough cut
 disengage the tool from workpiece
 Rapid move to position to tool for the nest cut
 Position the tool for next cut
 Second rough cut
 disengage the tool from workpiece

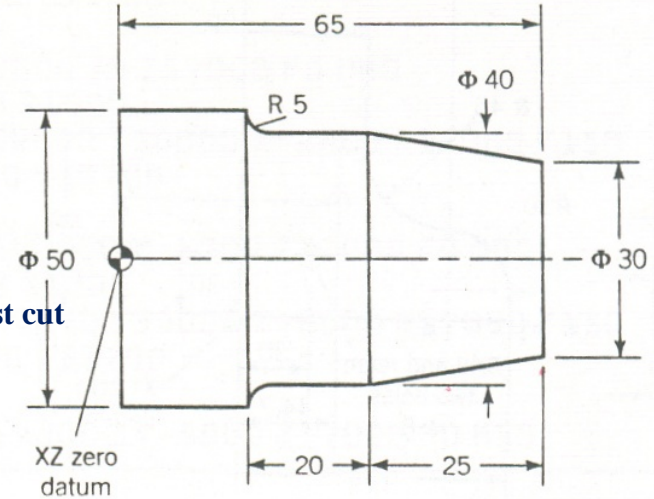
N120 G00 Z66
 N130 G01 X 36 F1
 N140 X42 Z50
 N150 X42.5 Z50.5
 N160 G00 Z66
 N170 G01 X32 F1
 N180 X42 Z40
 N190 X42.5 Z40.5

Position the tool for start of rough taper cut
 First rough taper cut
 disengage the tool from workpiece
 Position the tool for second rough taper cut
 Second rough taper cut
 disengage the tool from workpiece

N200 G00 Z66
 N205 M06 T05
 N207 S2000 M03 F0.5
 N210 G01 X30 F1
 N220 X40 Z40
 N230 Z25
 N240 G03 X50 Z20 R5
 N250 X50.5 Z20.5 M09

Change Tool
 For finish pass
 Position the tool for finish cut
 Finish taper
 Finish cut (40 mm dia)
 Finish 5mm radius
 disengage the tool from workpiece
 and Turn off coolant

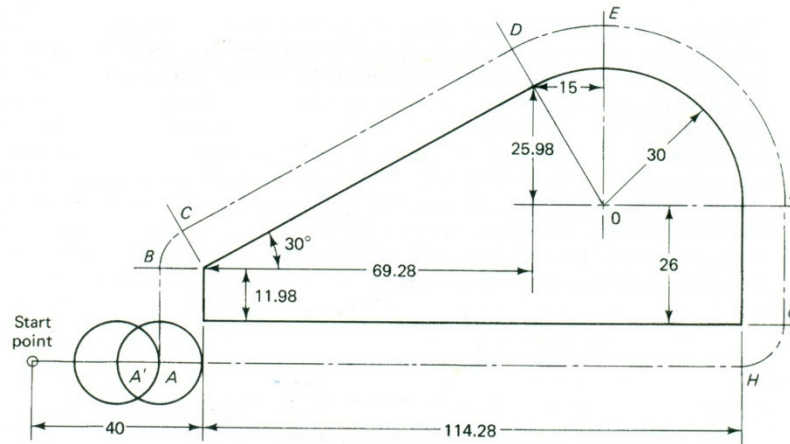
N260 G00 Z66
 N270 M30



A 50 mm diameter, 65 mm long rod is chosen as the workpiece material. Process sequence to be used: Rough cut of 40 mm dia in two passes, rough turn taper in two passes, and finish machine to final dimension. Use absolute programming. Spindle speed has to be specified in rev/mm, X values are programmed as diameter (default in all the NC controllers)



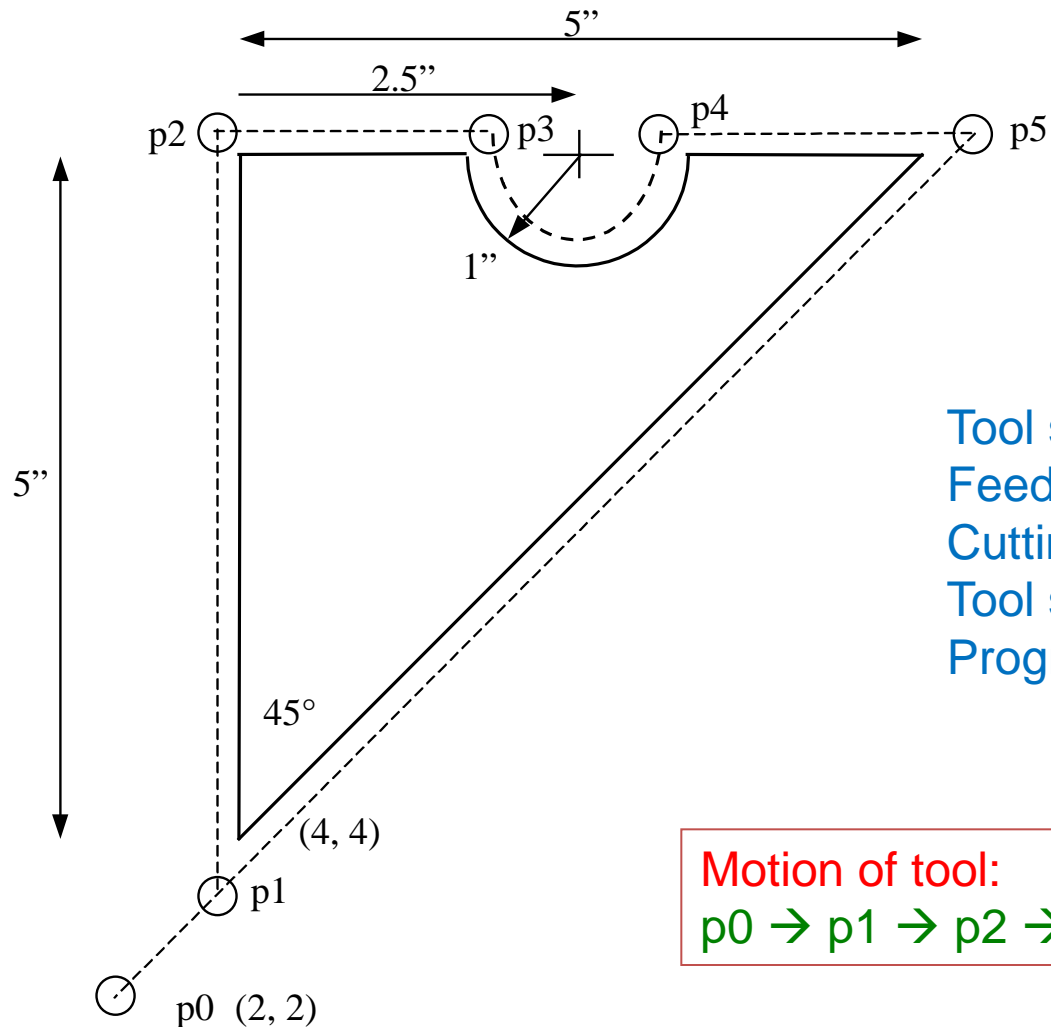
CNC Programming Example



N005	G71 G90 G97 G94 T01	Global Parameters, Metric, Absolute, V[rev/min], f[mm/min]
N010	G00 X -10 Y -10	Movement of Tool from 0 to A
N015	G01 Z -5 F 2	Movement of tool from above the part to A
N020	M03 S2000	Spindle ON, CW, Speed 2000 rpm
N025	X 114.28 (G01, Y -10, Z -5 may not be repeated)	F 0.3 From A to H
N030	G03 X (114.28+10) Y0 R10	From H to G, circular interpolation, CCW, Y value changes from -10 to 0
N035	G01 Y26	From G to F
N040	G03 X(69.28-10Sin30°) Y(26+25.98+10Cos30°) R40	From F to D
N045	G01 X(0-10Sin30°) Y(11.98+10Cos30°)	From D to C
N050	G03 X -10 Y 11.98 R 10	From C to B
N055	G01 X -10 Y -10	From B to A
N060	Z 5	
N065	M02	End of Program



Manual Part Programming Example



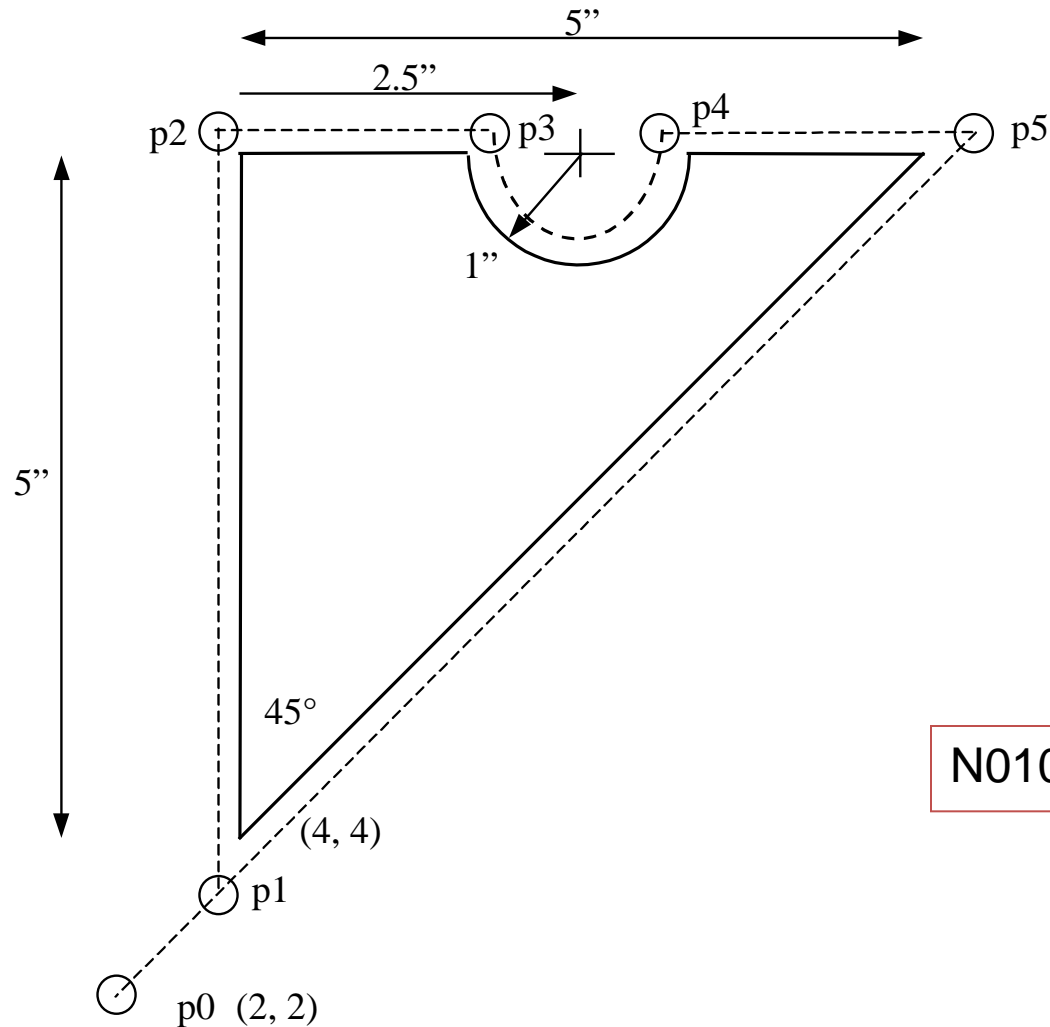
Tool size = 0.25 inch,
Feed rate = 6 inch per minute,
Cutting speed = 300 rpm,
Tool start position: 2.0, 2.0
Programming in inches

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

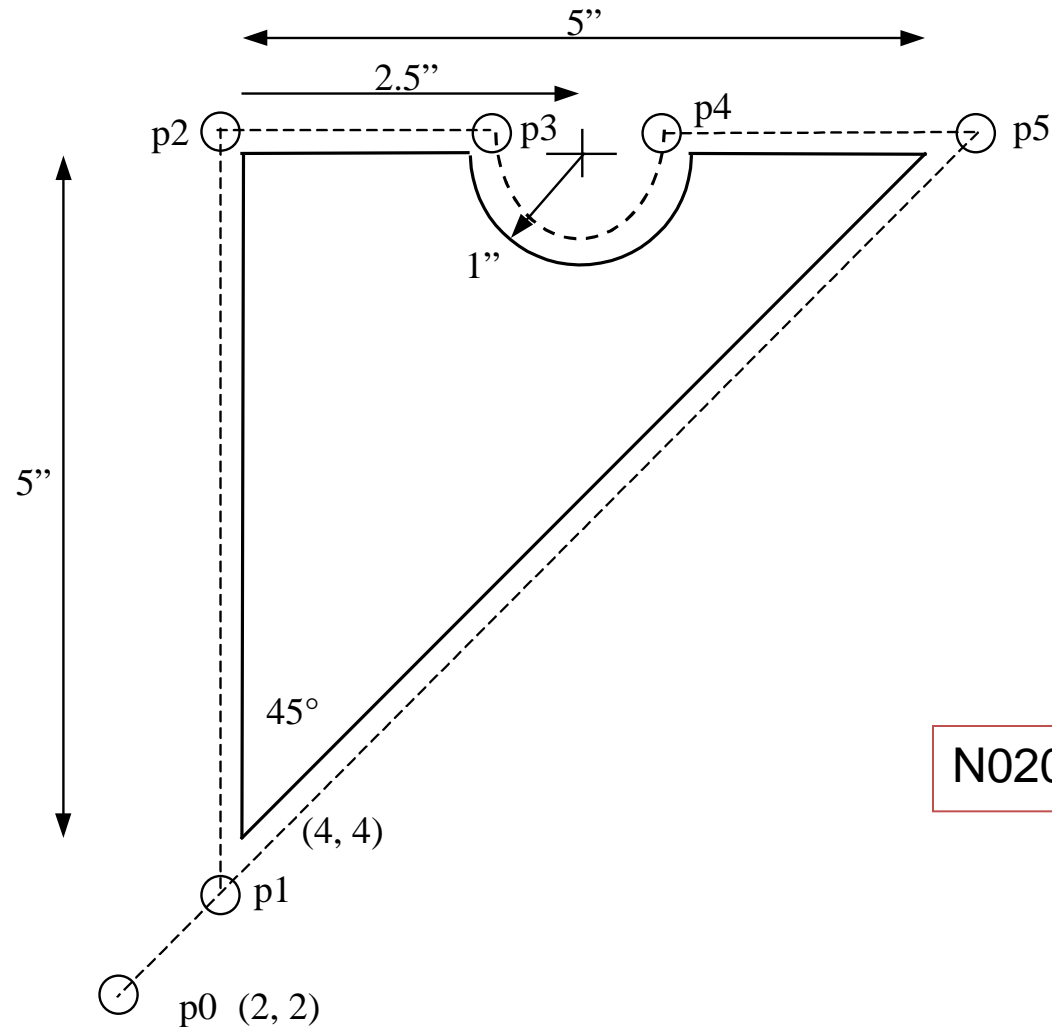
N010 G70 G90 G94 G97 M04

Spindle speed in rpm

Spindle CCW



2. Set up the machining conditions



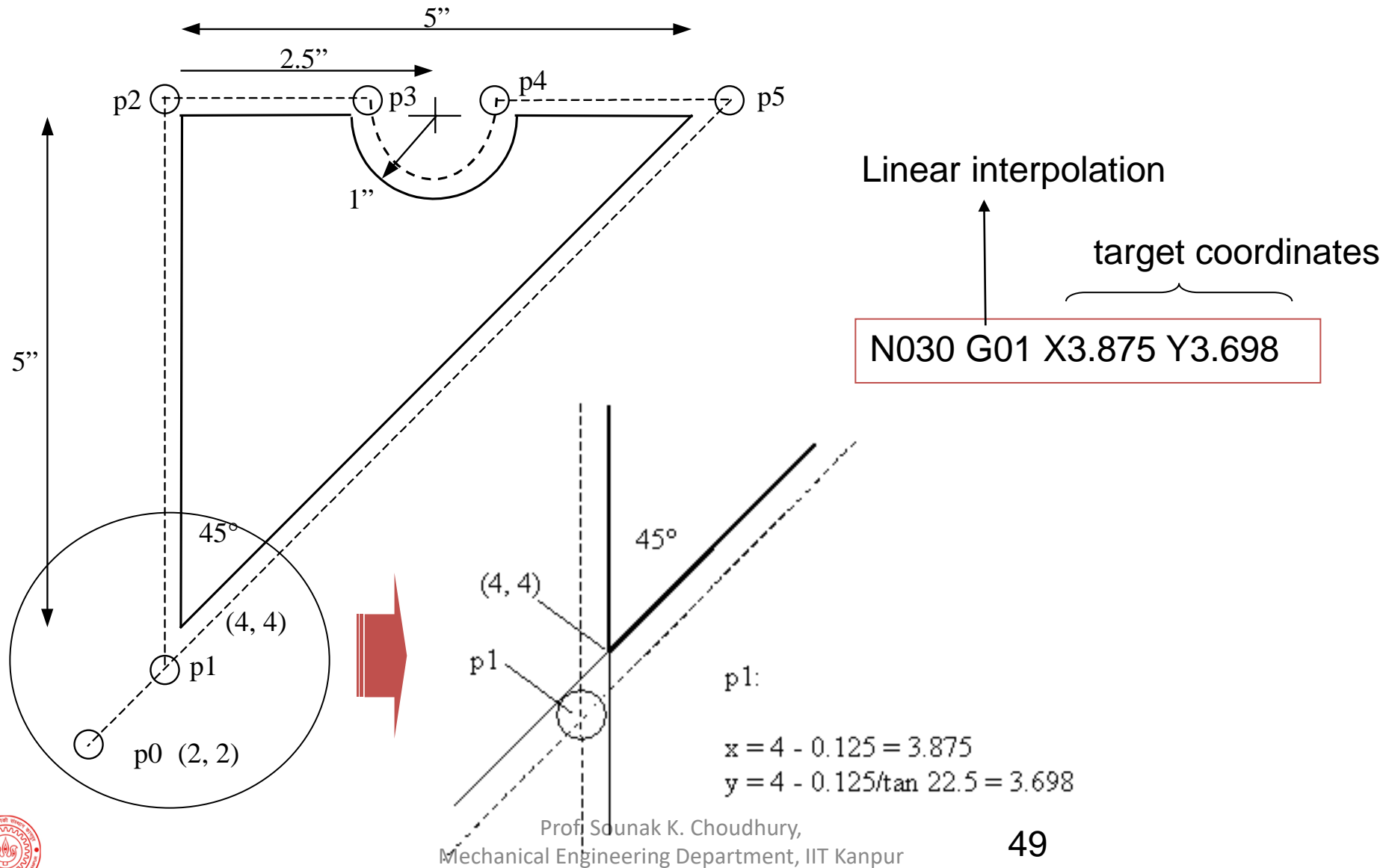
Machine moves in XY-plane
Use full-circle interpolation
Feed rate
Spindle speed

N020 G17 G75 F6.0 S300 T1001 M08

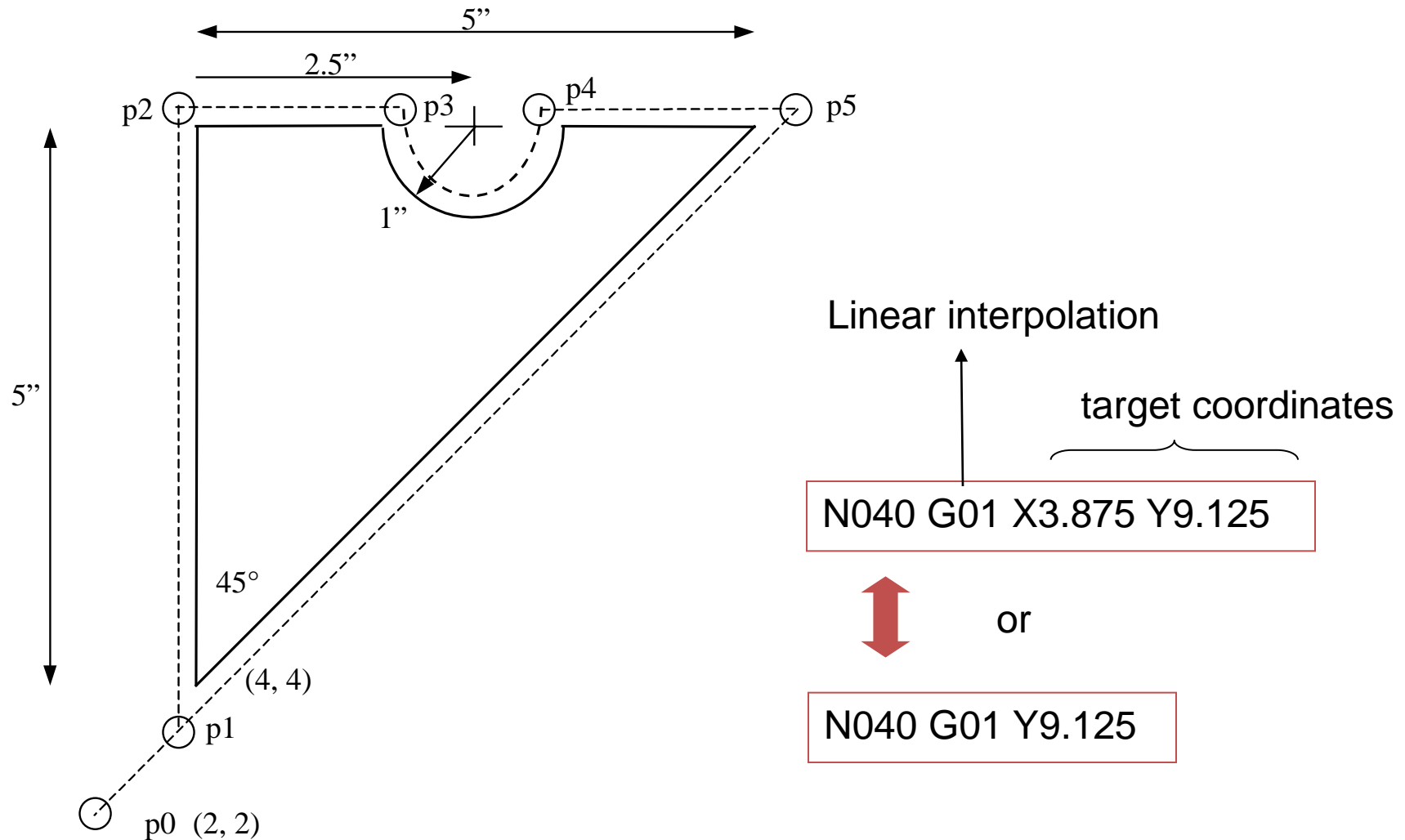
Tool no.
Flood coolant ON



3. Move tool from p0 to p1 in straight line



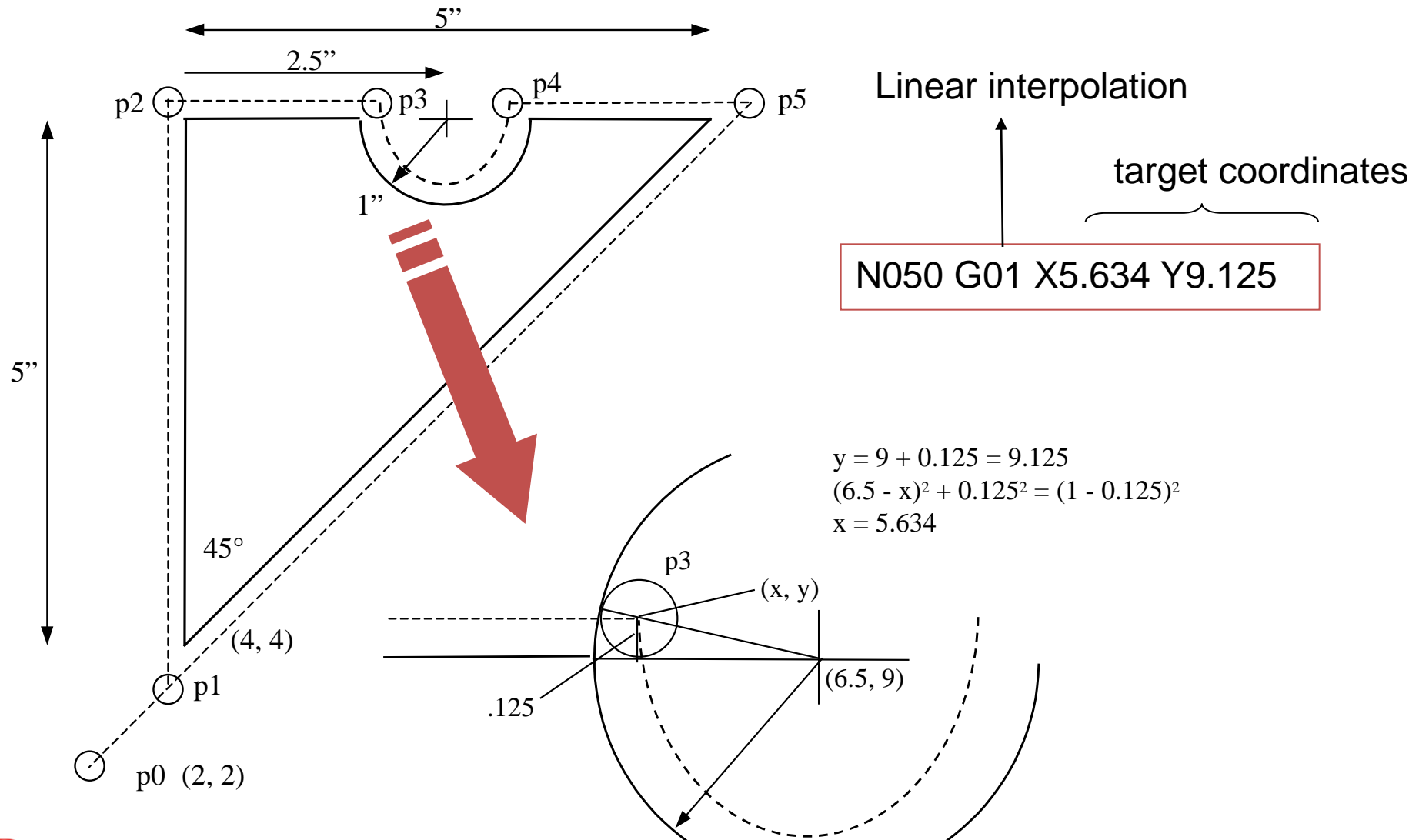
4. Cut profile from p1 to p2



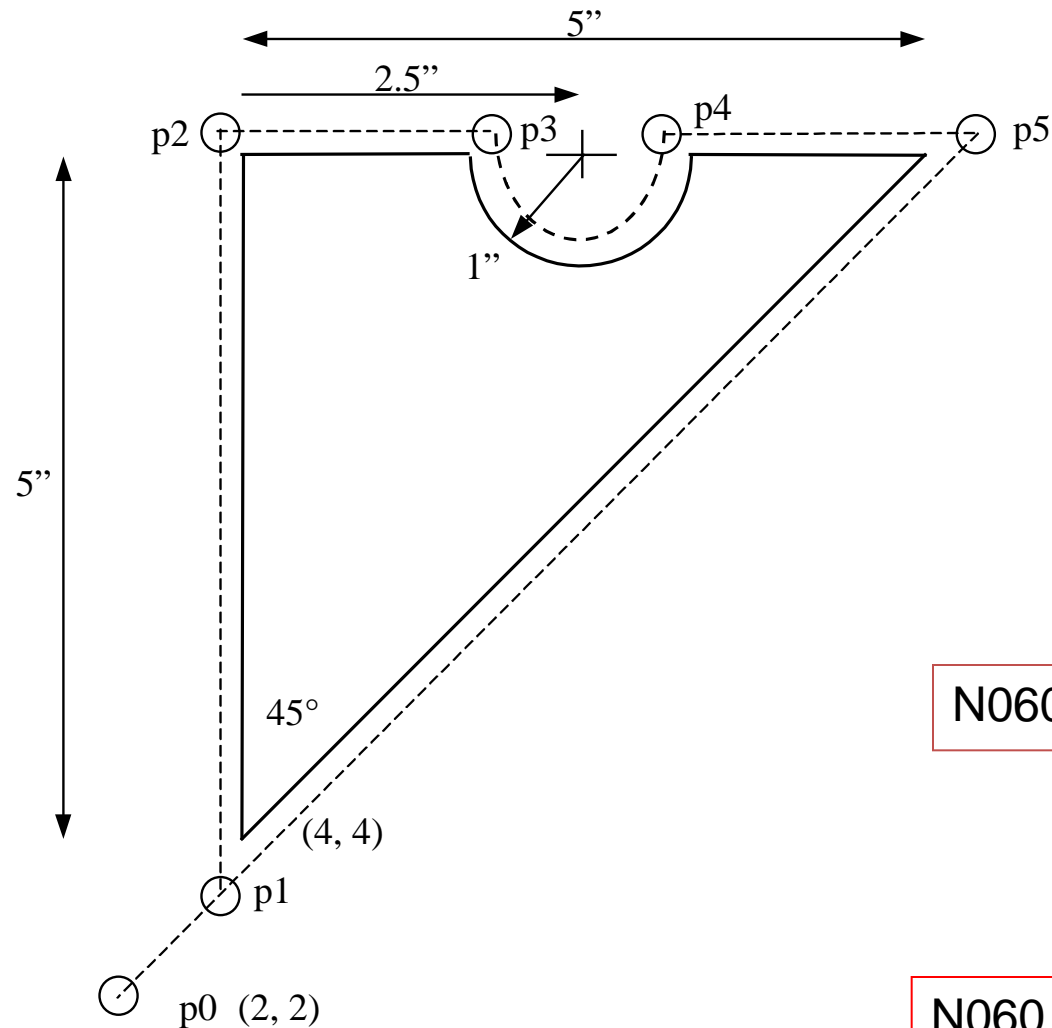
X-coordinate does not change → no need to program it



5. Cut profile from p2 to p3



6. Cut along circle from p3 to p4



circular interpolation, CCW motion

target coordinates

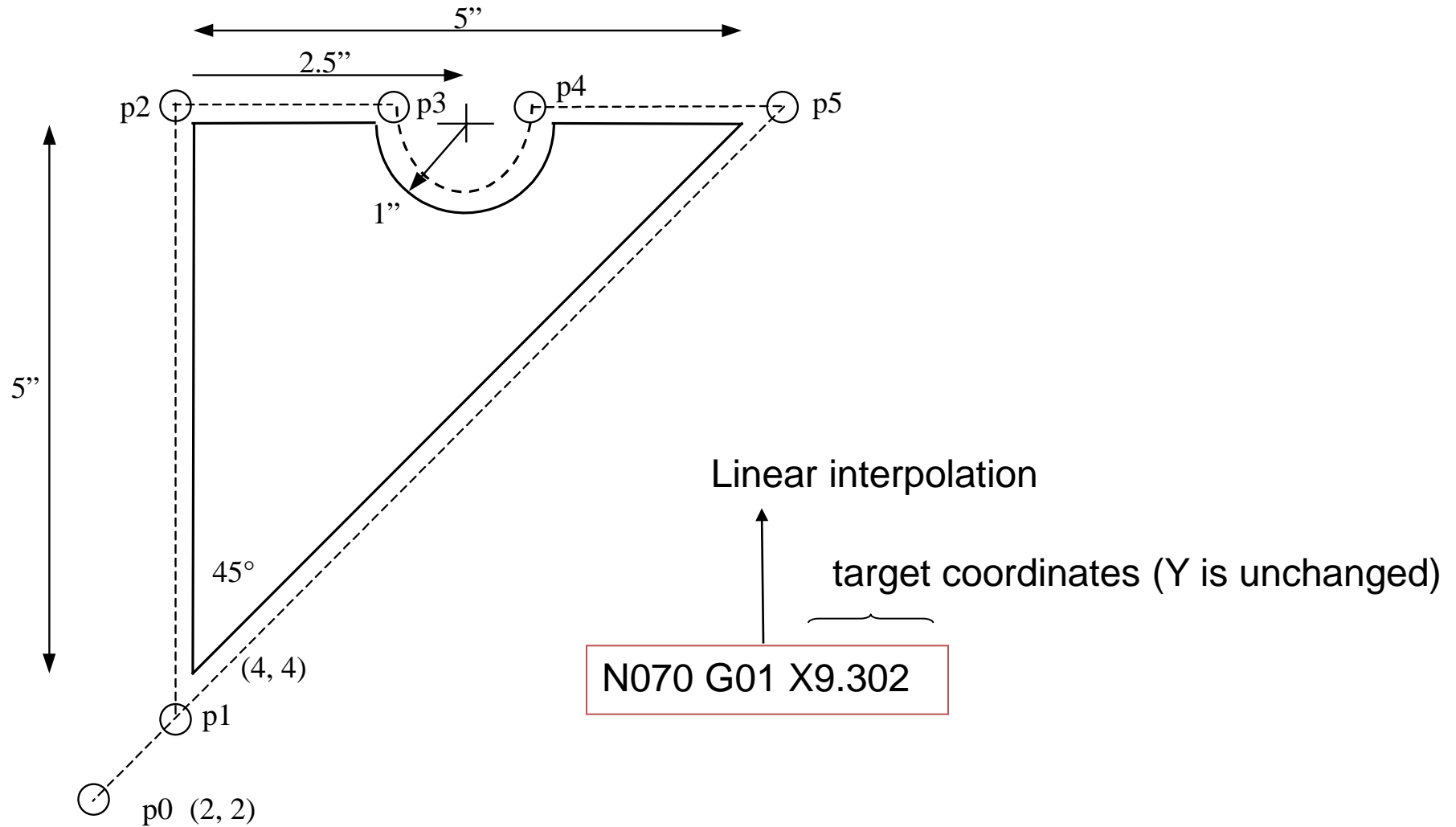
N060 G03 X7.366 Y9.125 I6.5 J9.0

coordinates of center of circle

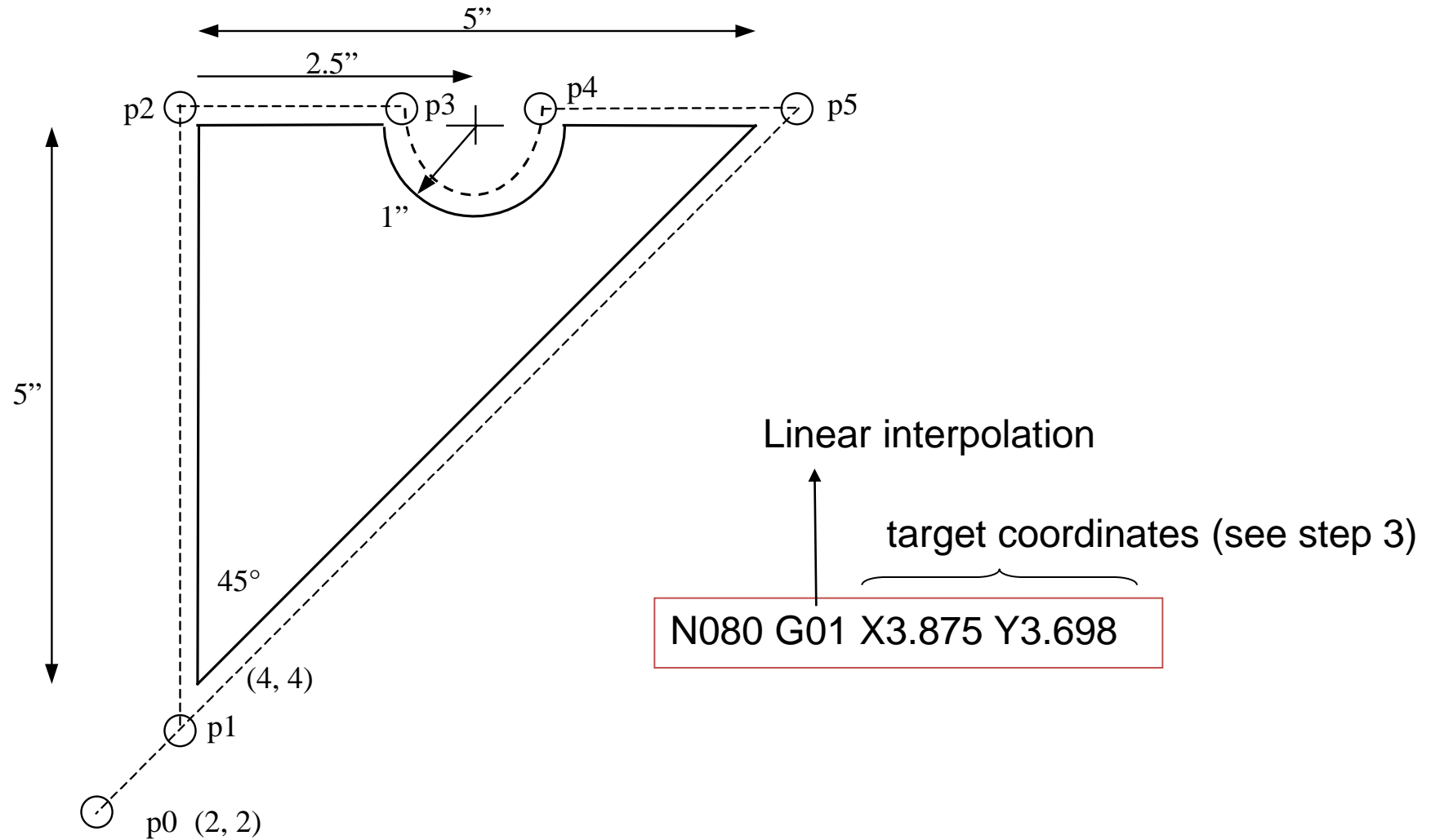
N060 G03 X7.366 Y9.125 I0.866 J-0.125



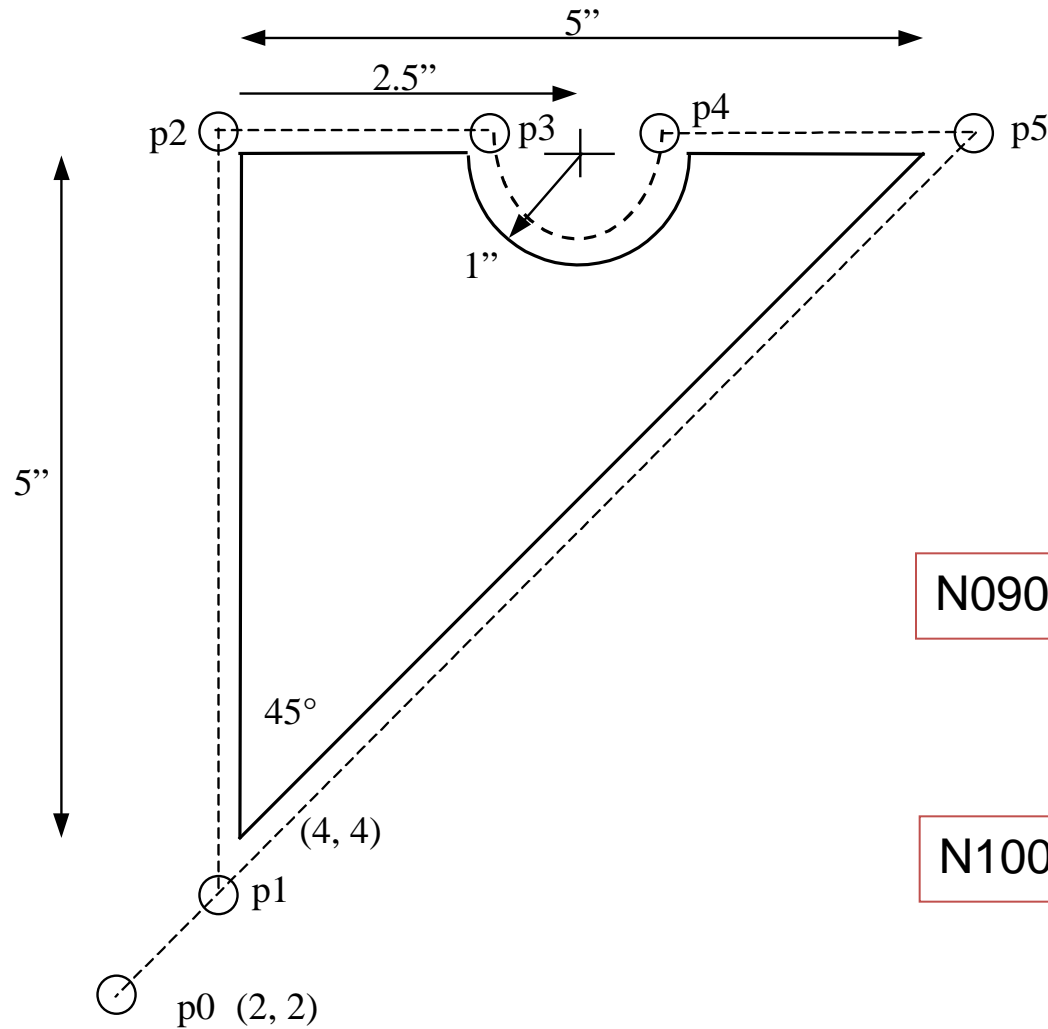
7. Cut from p4 to p5



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

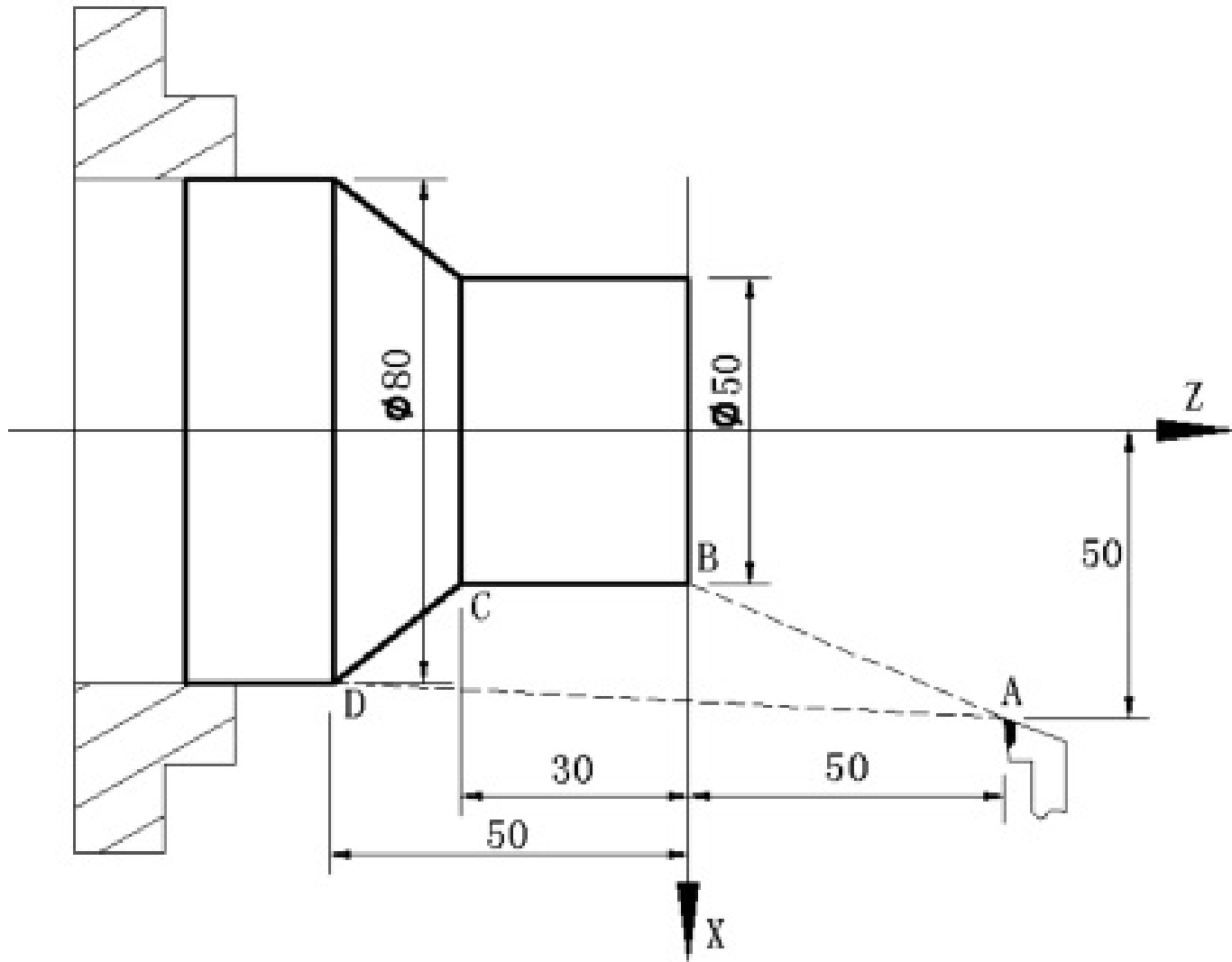


10. Complete RS-274 program

```
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

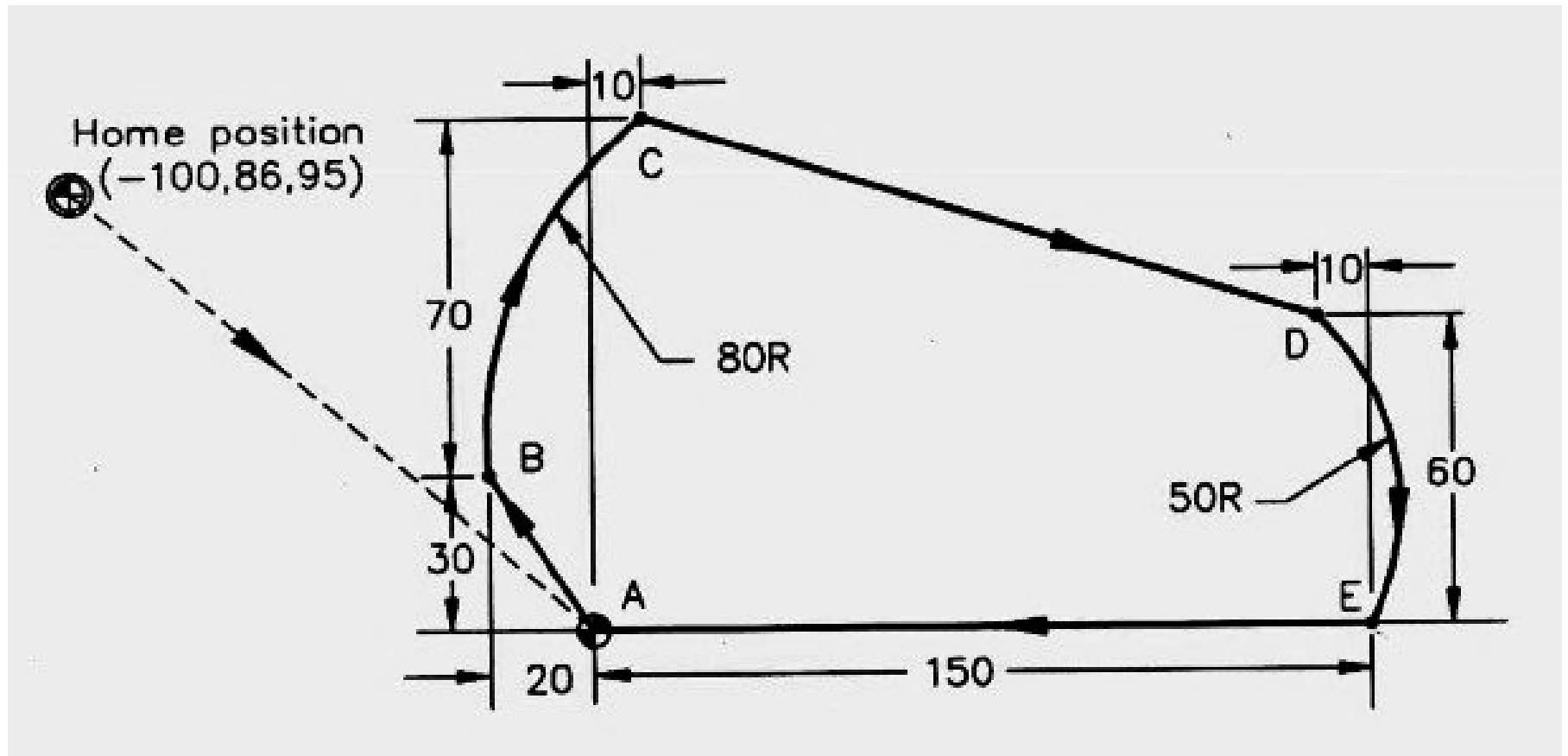
N90 Cooling OFF

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 positioning, metric unit

N10 tool change to T1

N15 define 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

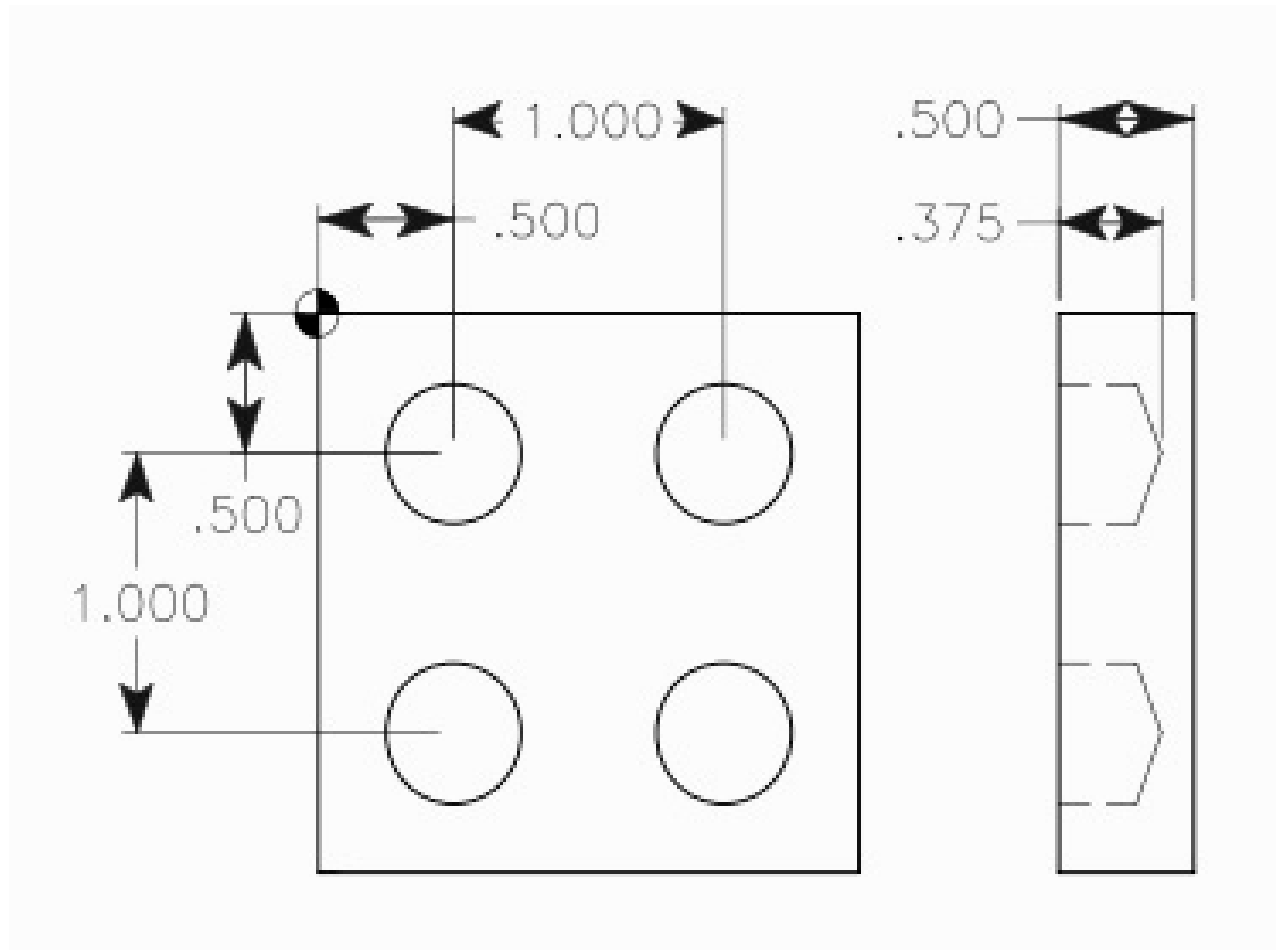
N65 reference point return in Z direction, spindle off

N70 reference point return in X and Y directions

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 first drilling position X0.5 Y-0.5 while taking into account Zero-offset-no. 1 (G54)

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

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

N5- Drilling cycle (G81) parameters, drill depth (Z) and cutting feed (F) are given, with this command first drill is made at current position (X0.5 Y-0.5).

N6- As drilling cycle continues 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 off (M09).

N10- Taking Machine-coordinate-system (G53) into account the drill is taken to Z0 position. Tool length compensation is cancelled (G49), cutter rotation 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.

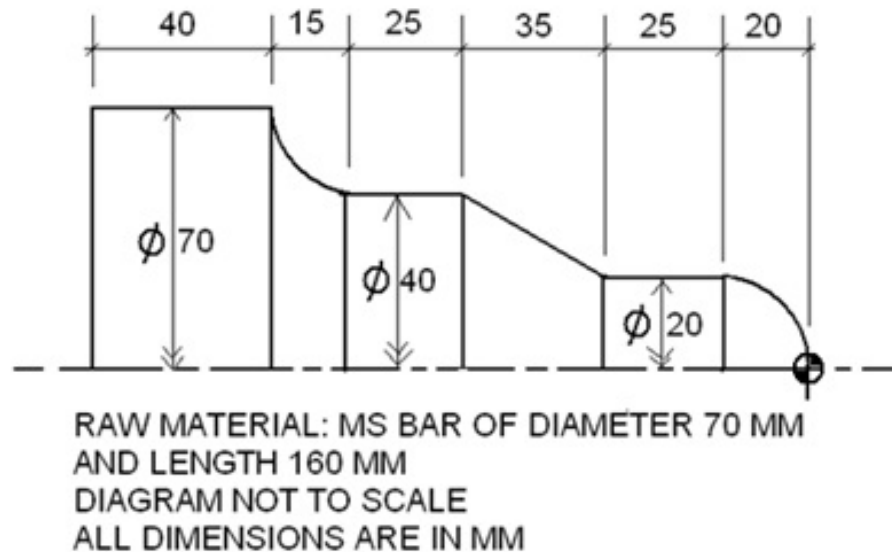


Figure A component to be turned.



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		%

