\ME3493 MANUFACTURING TECHNOLOGY **L T P C**
3 0 0 3

OBJECTIVES:

1 To study the concepts and basic mechanics of metal cutting and the factors affecting machinability
2 To learn working of basic and advanced turning machines.
3 To teach the basics of machine tools with reciprocating and rotating motions and abrasive finishing processes.
4 To study the basic concepts of CNC of machine tools and constructional features of CNC.
5 To learn the basics of CNC programming concepts to develop the part programme for Machine centre and turning centre

UNIT - I THEORY OF METAL CUTTING 9

Mechanics of chip formation, forces in machining, Types of chip, cutting tools – single point cutting tool nomenclature, orthogonal and oblique metal cutting, thermal aspects, cutting tool materials, tool wear, tool life, surface finish, cutting fluids and Machinability.

UNIT - II TURNING MACHINES 9

Centre lathe, constructional features, specification, operations – taper turning methods, thread cutting methods, special attachments, surface roughness in turning, machining time and power estimation. Special lathes - Capstan and turret lathes- tool layout – automatic lathes: semi-automatic – single spindle: Swiss type, automatic screw type – multi spindle

UNIT - III RECIPROCATING MACHINE TOOLS 9

Reciprocating machine tools: shaper, planer, slotter: Types and operations- Hole making: Drilling, reaming, boring, tapping, type of milling operations-attachments- types of milling cutters– machining time calculation - Gear cutting, gear hobbing and gear shaping – gear finishing methods Abrasive processes: grinding wheel – specifications and selection, types of grinding process – cylindrical grinding, surface grinding, centerless grinding, internal grinding - micro finishing methods

UNIT - IV CNC MACHINES 9

Computer Numerical Control (CNC) machine tools, constructional details, special features – Drives, Recirculating ball screws, tool changers; CNC Control systems – Open/closed, point-to-point/continuous - Turning and machining centres – Work holding methods in Turning and machining centres, Coolant systems, Safety features.

UNIT - V PROGRAMMING OF CNC MACHINE TOOLS 9

Coordinates, axis and motion, Absolute vs Incremental, Interpolators, Polar coordinates, Program planning, G and M codes, Manual part programming for CNC machining centers and Turning centers – Fixed cycles, Loops and subroutines, Setting up a CNC machine for machining.

TOTAL : 45 PERIODS

OUTCOMES: At the end of the course the students would be able to

1. Apply the mechanism of metal removal process and to identify the factors involved in improving machinability.
2. Describe the constructional and operational features of centre lathe and other special purpose lathes.
3. Describe the constructional and operational features of reciprocating machine tools.
4. Apply the constructional features and working principles of CNC machine tools.
5. Demonstrate the Program CNC machine tools through planning, writing codes and setting up CNC machine tools to manufacture a given component.

TEXT BOOKS:

1. Kalpakjian. S, "Manufacturing Engineering and Technology", Pearson Education India,7th Edition, 2018.
2. Michael Fitzpatrick, Machining and CNC Technology, McGraw-Hill Education; 4th edition, 2018.

REFERENCES:

1. Roy. A. Lindberg, Processes and materials of manufacture, PHI / Pearson education, 2006.
2. Geofrey Boothroyd, "Fundamentals of Metal Machining and Machine Tools", McGraw Hill, 1984.
3. Rao. P.N "Manufacturing Technology," Metal Cutting and Machine Tools, Tata McGraw- Hill, New Delhi, 2009.
4. A. B. Chattopadhyay, Machining and Machine Tools, Wiley, 2nd edition, 2017.
5. Peter Smid, CNC Programming Handbook, Industrial Press Inc.,; Third edition, 2007

UNIT V

CNC MACHINES

5.1 Numerical Control (NC) Machine Tools

Numerical Control (NC) refers to the method of controlling the manufacturing operation by means of directly inserted coded numerical instructions into the machine tool. It is important to realize that NC is not a machining method, rather, it is a concept of machine control. Although the most popular applications of NC are in machining, NC can be applied to many other operations, including welding, sheet metalworking, riveting, etc.

The major advantages of NC over conventional methods of machine control are as follows:

Higher precision

Machining of complex three-dimensional shapes

Better quality

Higher productivity

Multi-operational machining

Low operator qualification

5.2 Types of NC systems

Machine controls are divided into three groups,

Traditional numerical control (NC);

Computer numerical control (CNC);

Distributed numerical control (DNC).

The original numerical control machines were referred to as NC machine tool. They have “hardwired” control, whereby control is accomplished through the use of punched paper (or plastic) tapes or cards. Tapes tend to wear, and become dirty, thus causing misreadings. Many other problems arise from the use of NC tapes, for example the need to manual reload the NC tapes for each new part and the lack of program editing abilities, which increases the lead time. The end of NC tapes was the result of two competing developments, CNC and DNC.

CNC refers to a system that has a local computer to store all required numerical data. While CNC was used to enhance tapes for a while, they eventually allowed the use of other storage media, magnetic tapes and hard disks. The advantages of CNC systems include but are not limited to the possibility to store and execute a number of large programs (especially if a three or more dimensional machining of complex shapes is considered), to allow editing of programs, to execute cycles of machining commands, etc.

The development of CNC over many years, along with the development of local area networking, has evolved in the modern concept of DNC. Distributed numerical control is similar to CNC, except a remote computer is used to control a number of machines. An off-site mainframe host computer holds programs for all parts to be produced in the DNC facility. Programs are downloaded from the mainframe computer, and then the local controller feeds instructions to the hardwired NC machine.

The recent developments use a central computer which communicates with local CNC computers (also called Direct Numerical Control)

Controlled axes

NC system can be classified on the number of directions of motion they are capable to control simultaneously on a machine tool. Each free body has six degree of freedom, three positive or negative translations along x, y, and z-axis, and three rotations clockwise or counter clockwise about these axes. Commercial NC systems are capable of controlling simultaneously two, two and half, three, four and five degrees of freedom, or axes. The NC systems which control three linear translations (3-axis systems), or three linear translations and one rotation of the worktable (4-axis systems) are the most common.

Although the directions of axes for a particular machine tool are generally agreed as shown in the figure, the coordinate system origin is individual for each part to be machined and has to be decided in the very beginning of the process of CNC part programming.

Point-to-point vs. continuous systems

The two major types of NC systems are (see the figure):

Point-to-point (PTP) system, and

Contouring system.

PTP is a NC system, which controls only the position of the components. In this system, the path of the component motion relative to the workpiece is not controlled. The travelling between different positions is performed at the traverse speed allowable for the machine tool and following the shortest way.

Contouring NC systems are capable of controlling not only the positions but also the component motion, i.e., the travelling velocity and the programmed path between the desired positions.

Computer numerical control (CNC)

Numerical control (NC) is the automation of machine tools that are operated by precisely programmed commands encoded on a storage medium, as opposed to controlled manually via hand wheels or levers, or mechanically automated via cams alone. Most NC today is computer numerical control (CNC), in which computers play an integral part of the control.

In modern CNC systems, end-to-end component design is highly automated using computer-aided design (CAD) and computer-aided manufacturing (CAM) programs. The programs produce a computer file that is interpreted to extract the commands needed to operate a particular machine via a post processor, and then loaded into the CNC machines for production. Since any particular component might require the use of a number of different tools – drills, saws, etc., modern machines often combine multiple tools into a single "cell". In other installations, a number of different machines are used with an external controller and human or robotic operators that move the component from machine to machine. In either case, the series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.

The first NC machines were built in the 1940s and 1950s, based on existing tools that were modified with motors that moved the controls to follow points fed into the system on punched tape. These early servomechanisms were rapidly augmented with analog and digital computers, creating the modern CNC machine tools that have revolutionized the machining processes.

Modern CNC mills differ little in concept from the original model built at MIT in 1952. Mills typically consist of a table that moves in the X and Y axes, and a tool spindle that moves in the Z (depth). The position of the tool is driven by motors through a series of step-down gears in order to provide highly accurate movements, or in modern designs, direct-drive stepper motor or servo motors. Open-loop control works as long as the forces are kept small enough and speeds are not too great. On commercial metalworking machines closed loop controls are standard and required in order to provide the accuracy, speed, and repeatability demanded.

As the controller hardware evolved, the mills themselves also evolved. One change has been to enclose the entire mechanism in a large box as a safety measure, often with additional safety interlocks to ensure the operator is far enough from the working piece for safe operation. Most new CNC systems built today are completely electronically controlled.

CNC-like systems are now used for any process that can be described as a series of movements and operations. These include laser cutting, welding, friction stir welding, ultrasonic welding, flame and plasma cutting, bending, spinning, hole-punching, pinning, gluing, fabric cutting, sewing, tape and fiber placement, routing, picking and placing (PnP), and sawing.

Mills

CNC mills use computer controls to cut different materials. They are able to translate programs consisting of specific number and letters to move the spindle to various locations and depths. Many use G-code, which is a standardized programming language that many CNC machines understand, while others use proprietary languages created by their manufacturers. These proprietary languages while often simpler than G-code are not transferable to other machines.

Lathes

Lathes are machines that cut spinning pieces of metal. CNC lathes are able to make fast, precision cuts using indexable tools and drills with complicated programs for parts that normally cannot be cut on manual lathes. These machines often include 12 tool holders and coolant pumps to cut down on tool wear. CNC lathes have similar control specifications to

CNC mills and can often read G-code as well as the manufacturer's proprietary programming language.

Plasma cutters

CNC plasma cutting

Plasma cutting involves cutting a material using a plasma torch. It is commonly used to cut steel and other metals, but can be used on a variety of materials. In this process, gas (such as compressed air) is blown at high speed out of a nozzle; at the same time an electrical arc is formed through that gas from the nozzle to the surface being cut, turning some of that gas toplasma. The plasma is sufficiently hot to melt the material being cut and moves sufficiently fast to blow molten metal away from the cut.

Electric discharge machining

Electric discharge machining (EDM), sometimes colloquially also referred to as spark machining, spark eroding, burning, die sinking, or wire erosion, is a manufacturing process in which a desired shape is obtained using electrical discharges (sparks). Material is removed from the workpiece by a series of rapidly recurring current discharges between two electrodes, separated by a dielectric fluid and subject to an electric voltage. One of the electrodes is called the tool-electrode, or simply the "tool" or "electrode," while the other is called the workpiece-electrode, or "workpiece."

When the distance between the two electrodes is reduced, the intensity of the electric field in the space between the electrodes becomes greater than the strength of the dielectric (at least in some point(s)), which breaks, allowing current to flow between the two electrodes. This phenomenon is the same as the breakdown of a capacitor. As a result, material is removed from both the electrodes. Once the current flow stops (or it is stopped – depending on the type of generator), new liquid dielectric is usually conveyed into the inter-electrode volume enabling the solid particles (debris) to be carried away and the insulating proprieties of the dielectric to be restored. Adding new liquid dielectric in the inter-electrode volume is commonly referred to as flushing. Also, after a current flow, a difference of potential between the two electrodes is restored to what it was before the breakdown, so that a new liquid dielectric breakdown can occur.

Wire EDM

Also known as wire cutting EDM, wire burning EDM, or traveling wire EDM, this process uses spark erosion to machine or remove material with a traveling wire electrode from any electrically conductive material. The wire electrode usually consists of brass or zinc-coated brass material.

Sinker EDM

Sinker EDM, also called cavity type EDM or volume EDM, consists of an electrode and workpiece submerged in an insulating liquid—often oil but sometimes other dielectric fluids. The electrode and workpiece are connected to a suitable power supply, which generates an electrical potential between the two parts. As the electrode approaches the workpiece, dielectric breakdown occurs in the fluid forming a plasma channel) and a small spark jumps.

Water jet cutters

A water jet cutter, also known as a waterjet, is a tool capable of slicing into metal or other materials (such as granite) by using a jet of water at high velocity and pressure, or a mixture of water and an abrasive substance, such as sand. It is often used during fabrication or manufacture of parts for machinery and other devices. Waterjet is the preferred method when the materials being cut are sensitive to the high temperatures generated by other methods. It has found applications in a diverse number of industries from mining to aerospace where it is used for operations such as cutting, shaping, carving, and reaming.

Other CNC tools: Many other tools have CNC variants, including:

- Drills
- EDMs
- Embroidery machines
- Lathes
- Milling machines
- Wood routers
- Sheet metal works (Turret punch)
- Wire bending machines
- Hot-wire foam cutters
- Plasma cutters
- Water jet cutters
- Laser cutting
- Oxy-fuel
- Surface grinders
- Cylindrical grinders
- 3D Printing
- Induction hardening machines
- submerged welding
- knife cutting
- glass cutting

5.3 Programming Fundamentals CNC

Fanuc G-Code List (Lathe)

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G09	Exact stop

G10	Programmable data input
G20	Input in inch
G21	Input in mm
G22	Stored stroke check function on
G23	Stored stroke check function off
G27	Reference position return check
G28	Return to reference position
G32	Thread cutting
G40	Tool nose radius compensation cancel
G41	Tool nose radius compensation left
G42	Tool nose radius compensation right
G70	Finish machining cycle
G71	Turning cycle
G72	Facing cycle
G73	Pattern repeating cycle
G74	Peck drilling cycle
G75	Grooving cycle
G76	Threading cycle
G92	Coordinate system setting or max. spindle speed setting
G94	Feed Per Minute
G95	Feed Per Revolution
G96	Constant surface speed control
G97	Constant surface speed control cancel

Fanuc G-Code List (Mill)

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G17	X Y plane selection
G18	Z X plane selection
G19	Y Z plane selection
G28	Return to reference position

G30	2nd, 3rd and 4th reference position return
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation + direction
G44	Tool length compensation – direction
G49	Tool length compensation cancel
G53	Machine coordinate system selection
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G68	Coordinate rotation
G69	Coordinate rotation cancel
G73	Peck drilling cycle
G74	Left-spiral cutting circle
G76	Fine boring cycle
G80	Canned cycle cancel
G81	Drilling cycle, spot boring cycle
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute command
G91	Increment command
G92	Setting for work coordinate system or clamp at maximum spindle speed
G98	Return to initial point in canned cycle
G99	Return to R point in canned cycle

Fanuc M-Code List (Lathe)

M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M08	Coolant on
M09	Coolant off
M29	Rigid tap mode
M30	End of program reset
M40	Spindle gear at middle
M41	Low Gear Select
M42	High Gear Select
M68	Hydraulic chuck close
M69	Hydraulic chuck open
M78	Tailstock advancing
M79	Tailstock reversing
M94	Mirrorimage cancel
M95	Mirrorimage of X axis
M98	Subprogram call
M99	End of subprogram

Fanuc M-Code List (Mill)

M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M06	Tool change
M07	Coolant ON – Mist coolant/Coolant thru spindle
M08	Coolant ON – Flood coolant

M09	Coolant OFF
M19	Spindle orientation
M28	Return to origin
M29	Rigid tap
M30	End of program (Reset)
M41	Low gear select
M42	High gear select
M94	Cancel mirrorimage
M95	Mirrorimage of X axis
M96	Mirrorimage of Y axis
M98	Subprogram call
M99	End of subprogram

5.4 Manual Part Programming

Lathe

G02 G03 G Code Circular Interpolation

G02 G Code Clock wise Circular Interpolation.

G03 G Code Counter Clock wise Circular Interpolation.

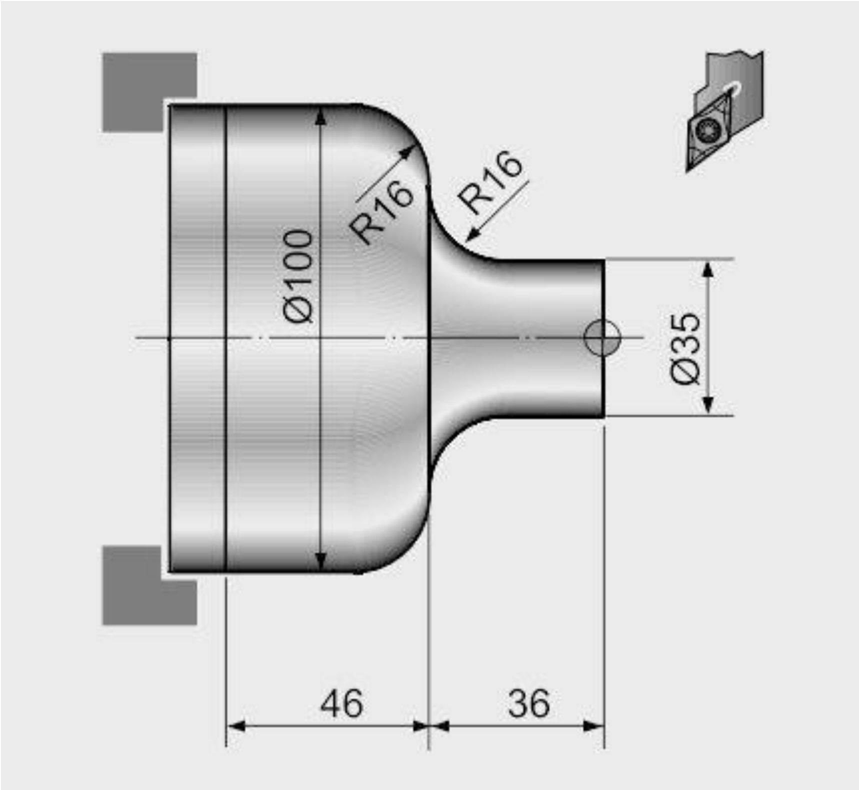
There are multiple articles/cnc program examples about G code circular interpolation, here is the list of few articles so that cnc machinists can easily navigate through different cnc programming articles.

G02 G03 G Code Example CNC Programs (G code Arc Examples)

- CNC Circular Interpolation Tutorial G02 G03
- Fanuc CNC Lathe Programming Example
- CNC Programming Example G Code G02 Circular Interpolation Clockwise
- Fanuc G20 Measuring in Inches with CNC Program Example
- CNC Arc Programming Exercise
- CNC Programming for Beginners a CNC Programming Example
- CNC Lathe Programming Example

Here is a new CNC programming examples which shows the use of G02 G03 G code circular interpolation.

G02 G03 G Code Example Program



G02 G03 G Code Circular Interpolation Example Program

```
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 R16.0
G01 X68.0 :
G03 X100.0 Z-52.0 R16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T0300
M30
```

G Code G02 G03 I & K Example Program

G02 G03 G Code Circular Interpolation can be programmed in two ways,

```
G02 X... Z... R...
G02 X... Z... I... K...
```

The below is the same cnc program but this version uses I & K with G02 G03 G code.

```
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 I16.0 K0
G01 X68.0 :
G03 X100.0 Z-52.0 I0 K-16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T0300
```

```
M30
```

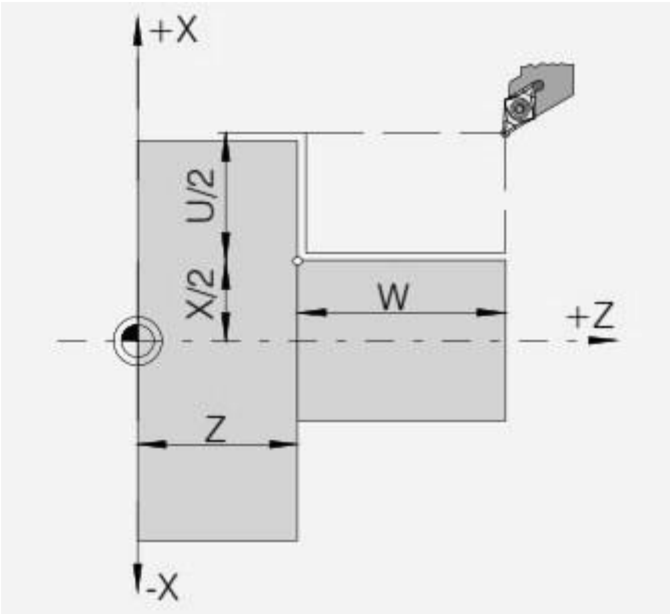
G20 Turning Cycle Format for Straight Turning

```
G20 X… Z… F…
```

or

```
G20 U… W… F…
```

X – Diameter to be cut (absolute).
Z – End point in z-axis (absolute).
F – Feed-rate.
U – Diameter to be cut (incremental).
W – End point in z-axis (incremental).



G20 Turning Cycle – CNC Lathe Fanuc 21 TB

G20 Turning Cycle Format for Taper Turning

```
G20 X… Z… R… F…
```

or

```
G20 U… W… R… F…
```

X – Diameter to be cut (absolute).
Z – End point in z-axis (absolute).
R – Incremental taper dimension in X with direction (+/-)
F – Feed-rate.
U – Diameter to be cut (incremental).
W – End point in z-axis (incremental).

As cnc machinists can use X or U value for the contour value, same way Z or W can be used or you can even mix both absolute (X, Z) and incremental (U, W) values.

G20 Turning Cycle Example CNC Program Code

```
G96 S200 M03
G00 X56.0 Z2.0
G20 X51.0 W-20.0 F0.25
X46.0
X41.0
X36.0
X31.0
X30.0
G00 X100 Z100
M30
```

CNC Program Code Explanation

As you can see in the above cnc program code,
Tool is at X56 Z2 point,
First cut is made at X51 and tool travels W-20 in Z-axis.
Second cut is made at X46
Third cut is made at X41
…
Last cut is made at X30

G20 Turning Cycle Function

As if you study the above cnc program code you will notice that,
1 – with G20 both absolute (X51.0) and incremental (W-20.0) values are used to make cuts.
2 – If above code also shows a very powerful functionality of G20 turning cycle which is that a cnc machinist can control depth-of-cut of every pass of G20 turning cycle which is impossible to achieve with other Turning Canned Cycle like G71 Rough Turning Cycle. So you will notice first five-cuts are of 5mm deep but the last one is just 1mm deep.

Cancellation of G20 Turning Cycle

G20 turning cycle is a modal G-code.
"Modal" G-code meaning that they stay in effect until they are cancelled or replaced by a contradictory G code.
It means G20 turning cycle remains active until another motion command is given like G00, G01 etc. As in above cnc program example G20 G code is cancelled with G00 G code.

Milling

Programming

```
G72.1 P... L... X... Y... R...
```

Parameters

Parameter	Description
P	Subprogram number
L	Number of times the operation is repeated
X	Center of rotation on the X axis

Y	Center of rotation on Y axis
R	Angular displacement (a positive value indicates a counter clockwise angular displacement. Specify an incremental value.)

G-Code Data

Modal/Non-Modal	G-Code Group
Non-Modal	00

Programming Notes

Notes

1. In the G72.1 block, addresses other than P, L, X, Y and R are ignored.
2. P, X, Y and R must always be specified.
3. If L is not specified, the figure is copied once.
4. The coordinate of the center of rotation is handled as an absolute value even if it is specified in the incremental mode.
5. Specify an increment in the angular displacement at address R. The angular displacement (degree) for the Nth figure is calculated as follows: Rx(N-1).

First block of the subprogram

Always specify a move command in the first block of a subprogram that performs a rotational copy. If the first block contains only the program number such as O00001234; and does not have a move command, movement may stop at the start point of the figure made by the n-th (n = 1,2, 3, …) copying.

Example of an incorrect program

```
O00001234 ;
G00 G90 X100.0 Y200.0 ;
;
;
M99 ;
```

Example of a correct program

```
O00001000 G00 G90 X100.0 Y200.0 ;
;
;
M99 ;
```

Limitation

Specifying two or more commands to copy a figure

G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, alarmPS0900 will occur).

In a subprogram that specifies rotational copy, however, linear copy (G72.2) can be specified. Similarly, in a subprogram that specifies linear copy, rotational copy can be specified.

Commands that must not be specified

Within a program that performs a rotational copy, the following must not be specified:

————————————Command for changing the selected plane (G17 to G19)

————————————Command for specifying polar coordinates (G16)

————————————Reference position return command(G28)

————————————Axis switching

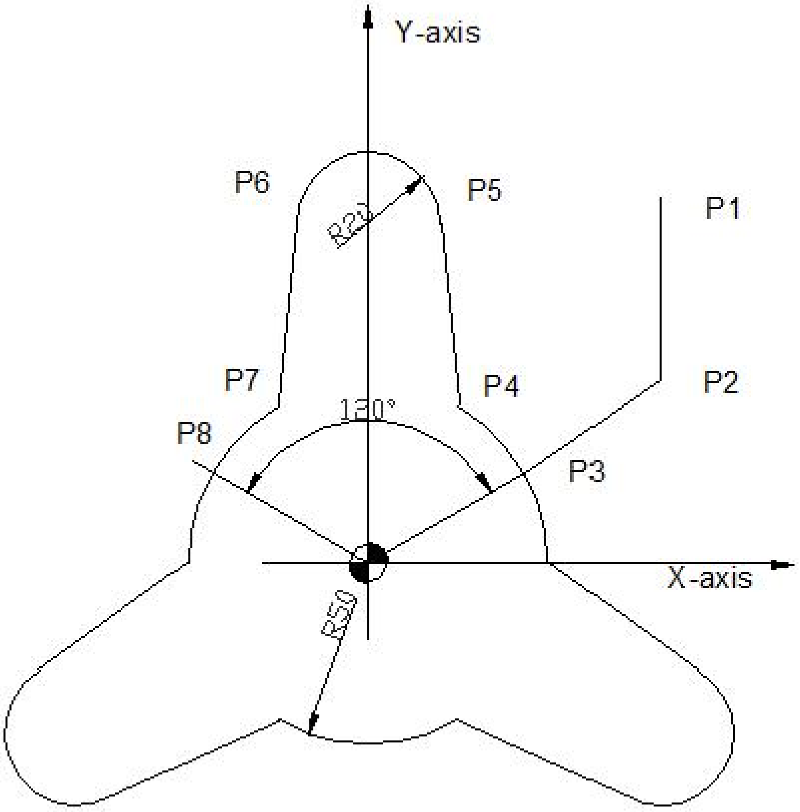
————————————————Coordinate system rotation (G68)
————————————————scaling (G51)
————————————————programmable mirror image (G51.1)

The command for rotational copying can be specified after a command for coordinate system rotation, scaling, or programm able mirror image is executed.

Single block

Single-block stops are not performed in a block with G721.1 or G72.2.

G72.1 Programming Example



Main program

```
O1000 ;
N10 G90 G00 X80. Y100. ;            (P1)
N20 Y50. ;                          (P2)
N30 G01 G17 G42 X43.301 Y25. D01 F100 ;(P3)
N40 G72.1 P1100 L3 X0 Y0 R120. ;
N50 G90 G40 G01 X80. Y50. ;         (P2)
N60 G00 X80. Y100. ;                (P1)
N70 M30 ;
```

Sub program

```
O1100 G91 G03 X-18.301 Y18.301 R50. ; (P4)
N100 G01 X-5. Y50. ;                 (P5)
N200 G03 X-40. I-20. ;               (P6)
N300 G01 X-5. Y-50. ;                (P7)
N400 G03 X-18.301 Y-18.301 R50. ;    (P8)
N500 M99 ;
```

5.5 Micromachining

Superfinishing, a metalworking process for producing very fine surface finishes

Various micro electro mechanical systems

Bulk micromachining

Surface micromachining

High-aspect-ratio microstructure technologies

Bulk micromachining is a process used to produce micro machinery or micro electro mechanical systems (MEMS).

Unlike surface micromachining, which uses a succession of thin film deposition and selective etching, bulk micromachining defines structures by selectively etching inside a substrate. Whereas surface micromachining creates structures *on top* of a substrate, bulk micromachining produces structures *inside* a substrate.

Usually, silicon wafers are used as substrates for bulk micromachining, as they can be anisotropically wet etched, forming highly regular structures. Wet etching typically uses alkaline liquid solvents, such as potassium hydroxide (KOH) or tetramethylammonium hydroxide (TMAH) to dissolve silicon which has been left exposed by the photolithography masking step. These alkali solvents dissolve the silicon in a highly anisotropic way, with some crystallographic orientations dissolving up to 1000 times faster than others. Such an approach is often used with very specific crystallographic orientations in the raw silicon to produce V-shaped grooves. The surface of these grooves can be atomically smooth if the etch is carried out correctly, and the dimensions and angles can be precisely defined.

Bulk micromachining starts with a silicon wafer or other substrates which is selectively etched, using photolithography to transfer a pattern from a mask to the surface. Like surface micromachining, bulk micromachining can be performed with wet or dry etches, although the most common etch in silicon is the anisotropic wet etch. This etch takes advantage of the fact that silicon has a crystal structure, which means its atoms are all arranged periodically in lines and planes. Certain planes have weaker bonds and are more susceptible to etching. The etch results in pits that have angled walls, with the angle being a function of the crystal orientation of the substrate. This type of etching is inexpensive and is generally used in early, low-budget research.

Unlike Bulk micromachining, where a silicon substrate (wafer) is selectively etched to produce structures, surface micromachining builds microstructures by deposition and etching of different structural layers on top of the substrate. Generally polysilicon is commonly used as one of the layers and silicon dioxide is used as a sacrificial layer which is removed or etched out to create the necessary void in the thickness direction. Added layers are generally very thin with their size varying from 2-5 Micro metres. The main advantage of this machining process is the possibility of realizing monolithic microsystems in which the electronic and the mechanical components(functions) are built in on the same substrate. The surface micromachined components are smaller compared to their counterparts, the bulk micromachined ones.

As the structures are built on top of the substrate and not inside it, the substrate's properties are not as important as in bulk micromachining, and the expensive silicon wafers can be replaced by cheaper substrates, such as glass or plastic. The size of the substrates can also be much larger than a silicon wafer, and surface micromachining is used to produce TFTs on large area glass substrates for flat panel displays. This technology can also be used for the manufacture of thin film solar cells, which can be deposited on glass, but also on PET substrates or other non-rigid materials.

HARMST is an acronym for **H**igh **A**spect **R**atio **M**icrostructure **T**echnology that describes fabrication technologies, used to create high-aspect-ratio microstructures with heights between tens of micrometers up to a centimeter and aspect ratios greater than 10:1. Examples include the LIGA fabrication process, advanced silicon etch, and deep reactive ion etching.

5.6 Water Machining

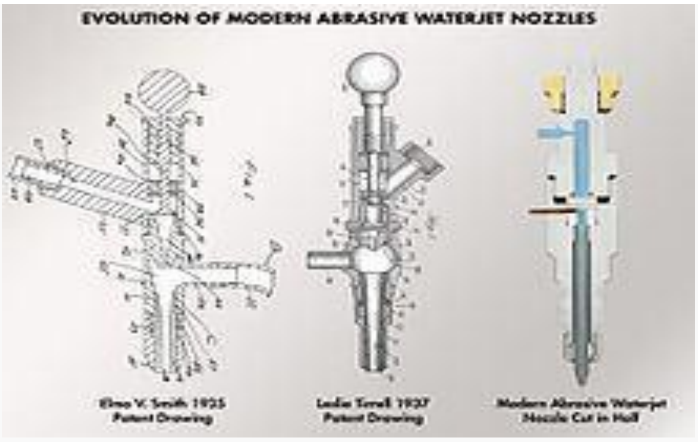
A water jet cutter, also known as a waterjet or waterjet, is an industrial tool capable of cutting a wide variety of materials using a very high-pressure jet of water, or a mixture of water and an abrasive substance. The term abrasive jet refers specifically to the use of a mixture of water and abrasive to cut hard materials such as metal or granite, while the terms pure waterjet and water-only cutting refer to waterjet cutting without the use of added abrasives, often used for softer materials such as wood or rubber. Waterjet cutting is often used during fabrication of machine parts. It is the preferred method when the materials being cut are sensitive to the high temperatures generated by other methods. Waterjet cutting is used in various industries, including mining andaerospace, for cutting, shaping, and reaming.

Water jet CNC cutting Machine

While using high-pressure water for erosion dates back as far as the mid-1800s with hydraulic mining, it was not until the 1930s that narrow jets of water started to appear as an industrial cutting device. In 1933, the Paper Patents Company in Wisconsin developed a paper metering, cutting, and reeling machine that used a diagonally moving waterjet nozzle to cut a horizontally moving sheet of continuous paper. These early applications were at a low pressure and restricted to soft materials like paper.

Waterjet technology evolved in the post-war era as researchers around the world searched for new methods of efficient cutting systems. In 1956, Carl Johnson of Durox International in Luxembourg developed a method for cutting plastic shapes using a thin stream high-pressure waterjet, but those materials, like paper, were soft materials.[3] In 1958, Billie Schwacha of North American Aviation developed a system using ultra-high-pressure liquid to cut hard materials.[4] This system used a 100,000 psi (690 MPa) pump to deliver ahypersonic liquid jet that could cut high strength alloys such as PH15-7-MO stainless steel. Used as a honeycomb laminate on the Mach 3 North American XB-70 Valkyrie, this cutting method resulted in delaminating at high speed, requiring changes to the manufacturing process. While not effective for the XB-70 project, the concept was valid and further research continued to evolve waterjet cutting. In 1962, Philip Rice of Union Carbideexplored using a pulsing waterjet at up to 50,000 psi (345 MPa) to cut metals, stone, and other materials. Research by S.J. Leach and G.L. Walker in the mid-1960s expanded on traditional coal waterjet cutting to determine ideal nozzle shape for high-pressure waterjet cutting of stone, and Norman Franz in the late 1960s focused on waterjet cutting of soft materials by dissolving long chain polymers in the water to improve the cohesiveness of the jet stream. In the early 1970s, the desire to improve the durability of the waterjet nozzle led Ray Chadwick, Michael Kurko, and Joseph Corriveau of the Bendix Corporation to come up with the idea of using corundum crystal to form a waterjet orifice, while Norman Franz expanded on this and created a waterjet nozzle with an orifice as small as 0.002 inches (0.05 mm) that operated at pressures up to 70,000 psi (483 MPa). John Olsen, along with George Hurlburt and Louis Kapcsandy at Flow Research (later Flow Industries), further improved the commercial potential of the waterjet by showing that treating the water beforehand could increase the operational life of the nozzle.

Abrasive waterjet



The Evolution of the Abrasive Waterjet Nozzle

While cutting with water is possible for soft materials, the addition of an abrasive turned the waterjet into a modern machining tool for all materials. This began in 1935 when the idea of adding an abrasive to the water stream was developed by Elmo Smith for the liquid abrasive blasting. Smith’s design was further refined by Leslie Tirrell of the Hydroblast Corporation in 1937, resulting in a nozzle design that created a mix of high-pressure water and abrasive for the purpose of wet blasting. Producing a commercially viable abrasive waterjet nozzle for precision cutting came next by Dr. Mohamed Hashish who invented and led an engineering research team at Flow Industries to develop the modern abrasive waterjet cutting technology. Dr. Hashish, who also coined the new term "Abrasive Waterjet" AWJ, and his team continued to develop and improve the AWJ technology and its hardware for many applications which is now in over 50 industries worldwide. A most critical development was creating a durable mixing tube that could withstand the power of the high-pressure AWJ, and it was Boride Products (now Kennametal) development of their ROCTEC line of ceramic tungsten carbide composite tubes that significantly increased the operational life of the AWJ nozzle. Current work on AWJ nozzles is on micro abrasive waterjet so cutting with jets smaller than 0.015 inch in diameter can be commercialized.

Applications

Because the nature of the cutting stream can be easily modified the water jet can be used in nearly every industry; there are many different materials that the water jet can cut. Some of them have unique characteristics that require special attention when cutting.

Materials commonly cut with a water jet include rubber, foam, plastics, leather, composites, stone, tile, metals, food, paper and much more. Materials that cannot be cut with a water jet are tempered glass, diamonds and certain ceramics. Water is capable of cutting materials over eighteen inches (45 cm) thick.

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**

- 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.

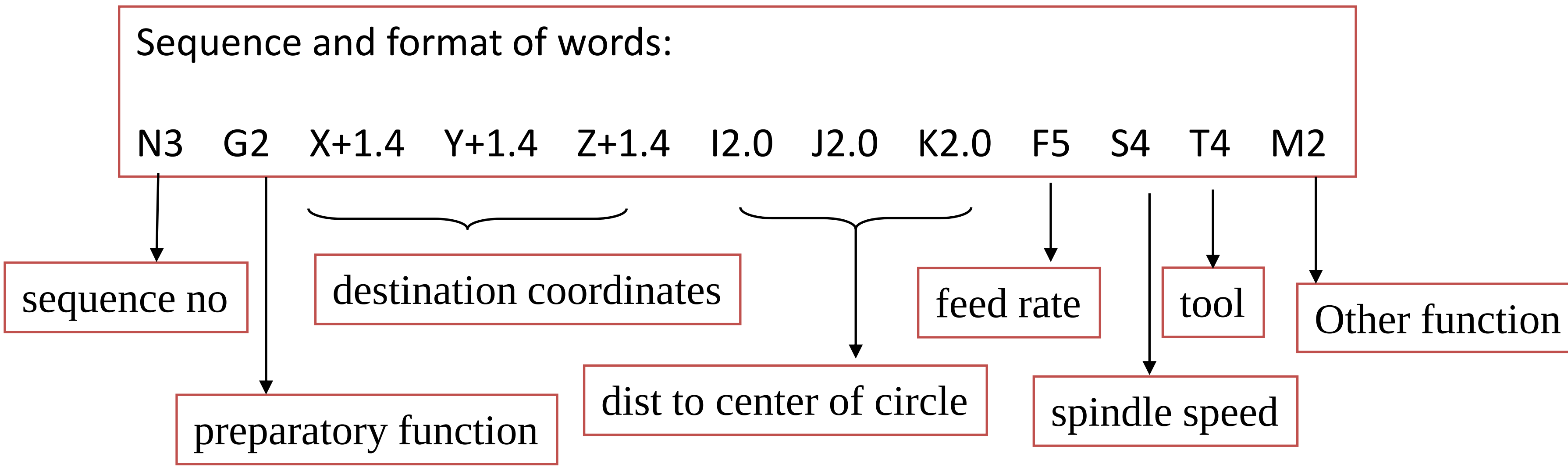
- F-CODE: Feed rate. The F code specifies the feed in the machining operation.

- 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.

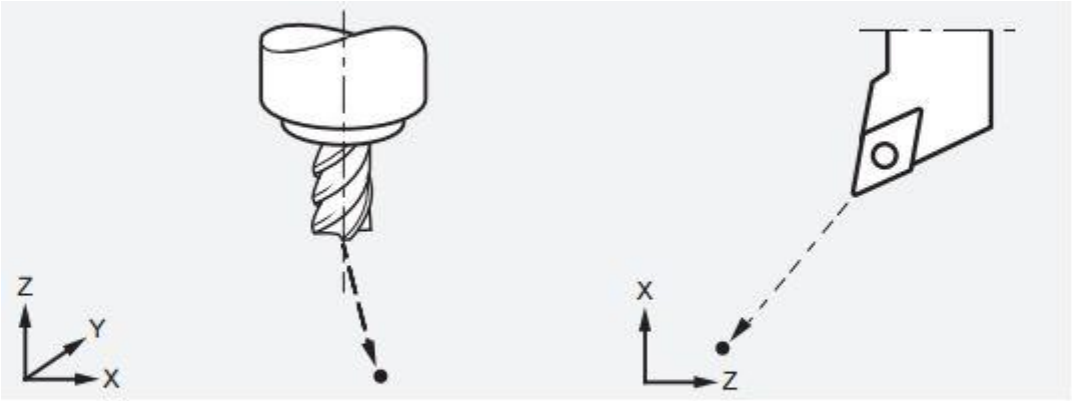
M-codes	Function
M00	Optional program stop automatic
M01	Optional program stop request
M02	Program end
M03	Spindle ON clock wise (CW)
M04	Spindle ON counter clock wise (CCW)
M05	Spindle stop
M06	Tool change
M07	Mist coolant ON (coolant 1 ON)
M08	Flood coolant ON (coolant 2 ON)
M09	Coolant OFF
M30	End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

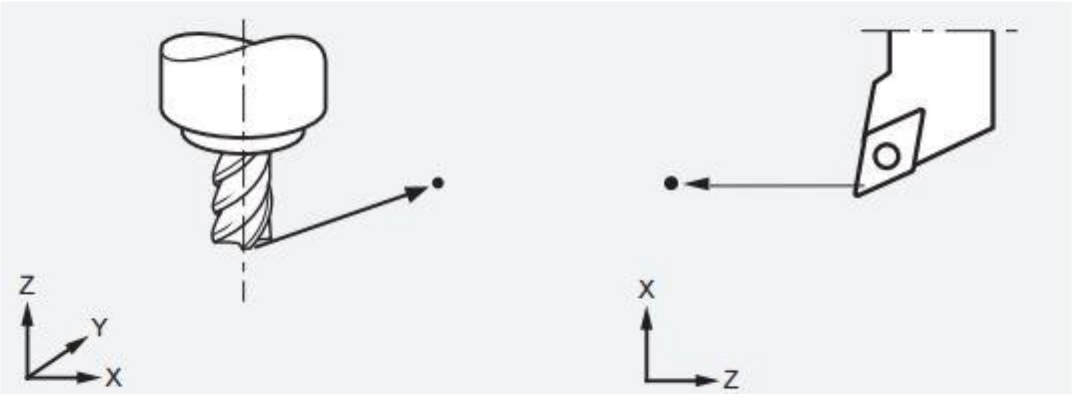
Syntax: N010 [G90/G91] G00 X10 Y10 Z5



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

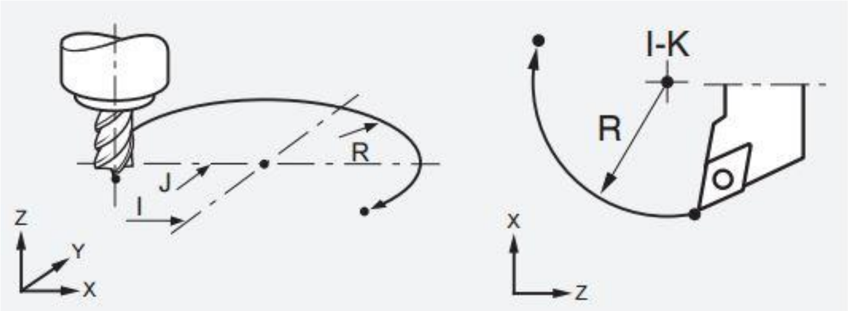
N__ G02/03 X__ Y__Z__ I__ J__K__ F__ using the arc center

or

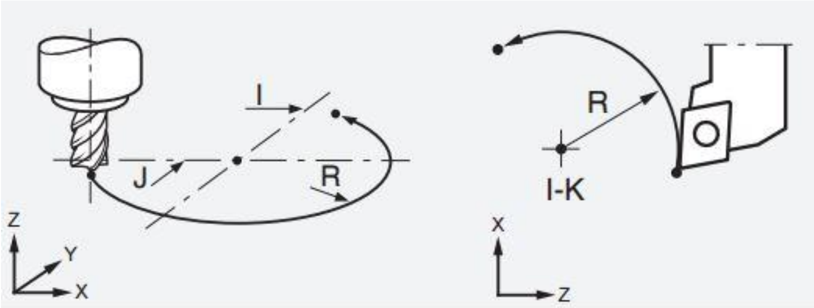
N__ G02/03 X__ Y__Z__ R__ F__ using the arc radius

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



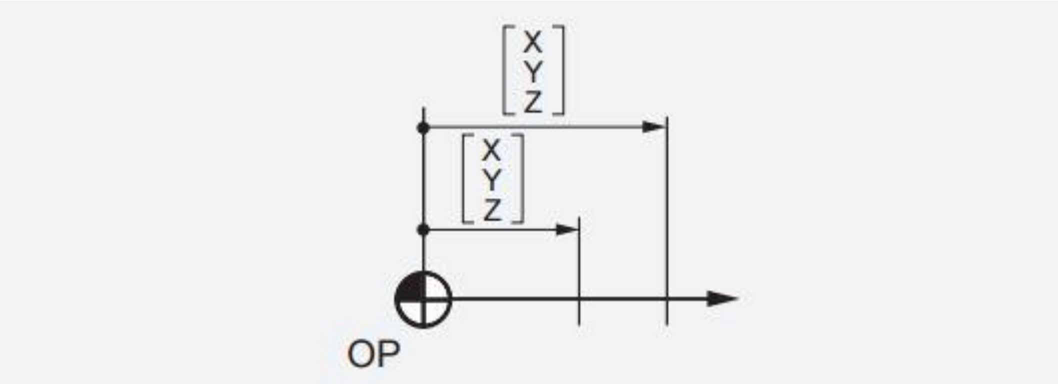
G03 moves along a CCW arc

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.

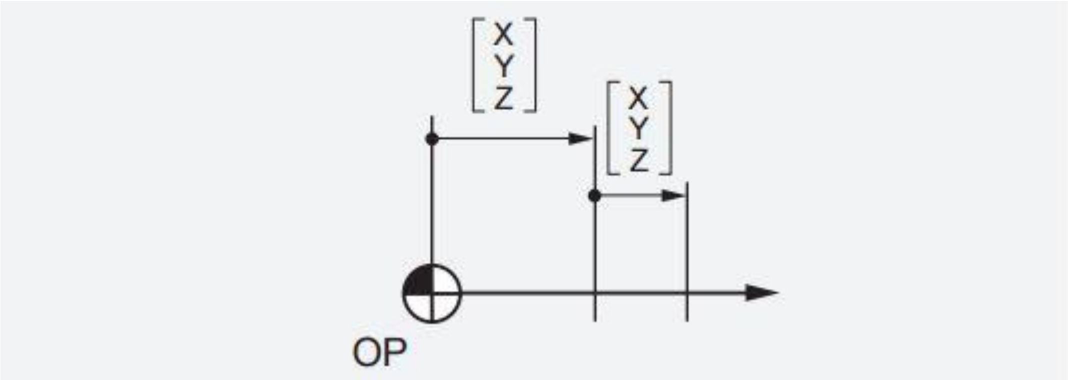


Syntax: N.. G90 X.. Y.. Z.. A.. B.. C..

G91 INCREMENTAL POSITION COMMAND

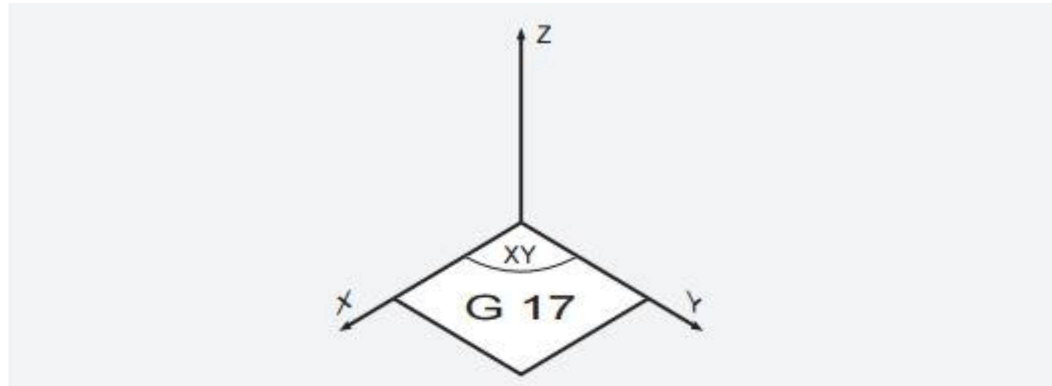
- This code is modal and changes the way axis motion commands are interpreted. G91 makes all subsequent commands incremental. Zero point shifts with the new position.

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

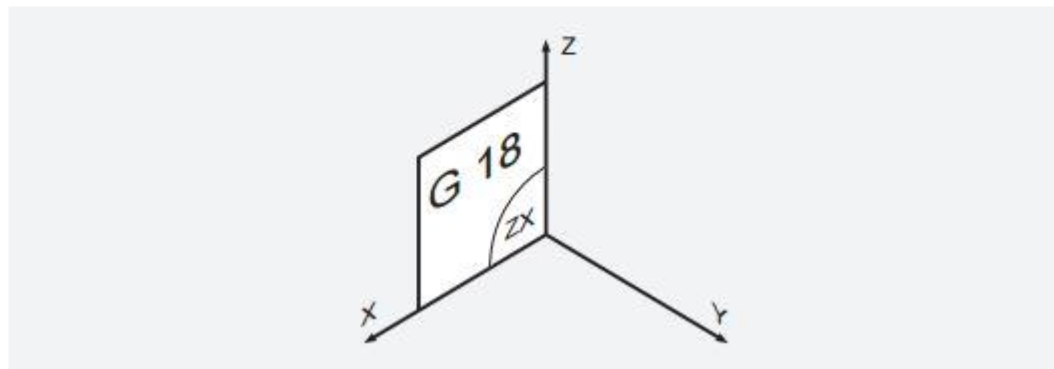


G 17 G18 G19 : PLANE SELECTION

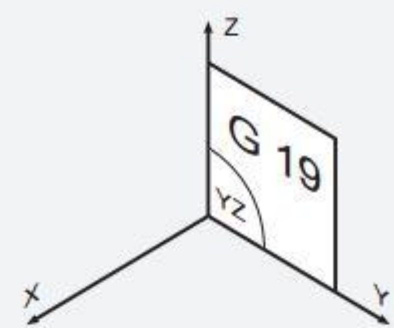
G 17 : XY plane selection
Syntax: N.. G17



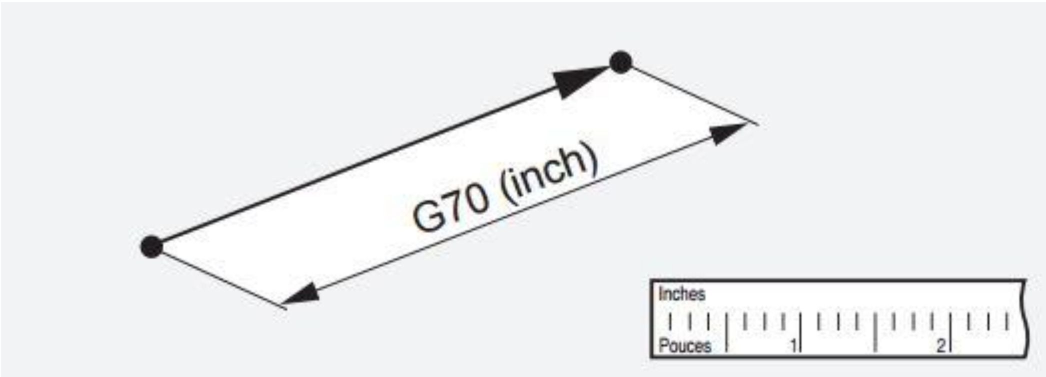
G 18 : ZX plane selection
Syntax: N.. G18



G 19 : ZY plane selection
Syntax: N.. G19

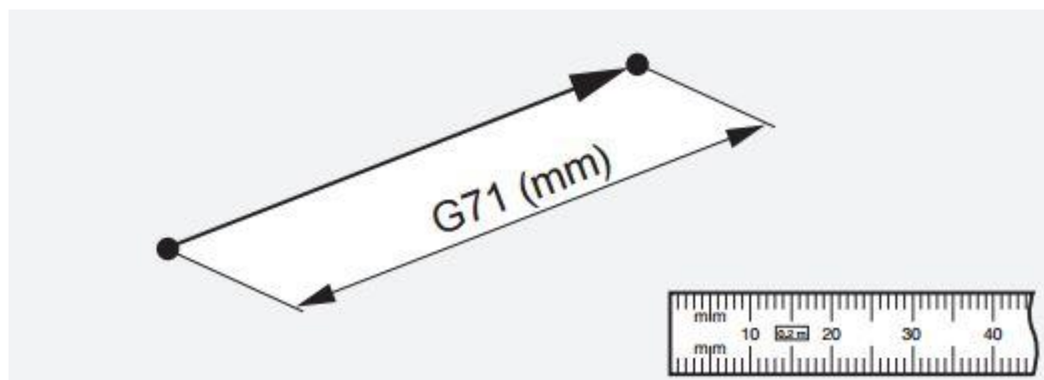


Syntax: N020 G17 G75 F6.0 S300 T1001 M08



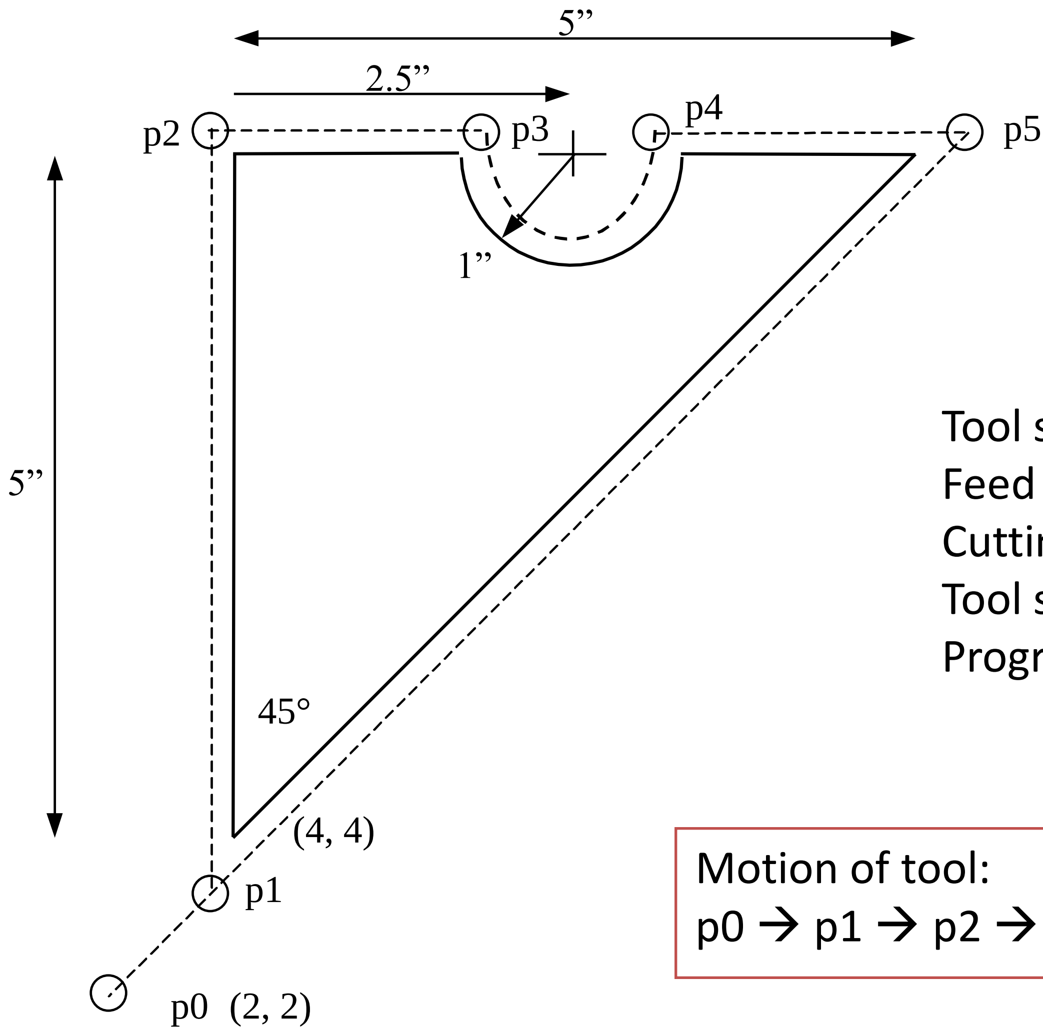
G 70 Inch data input

G 71 Metric data input



Syntax : N010 G70 G90 G94 G97 M04

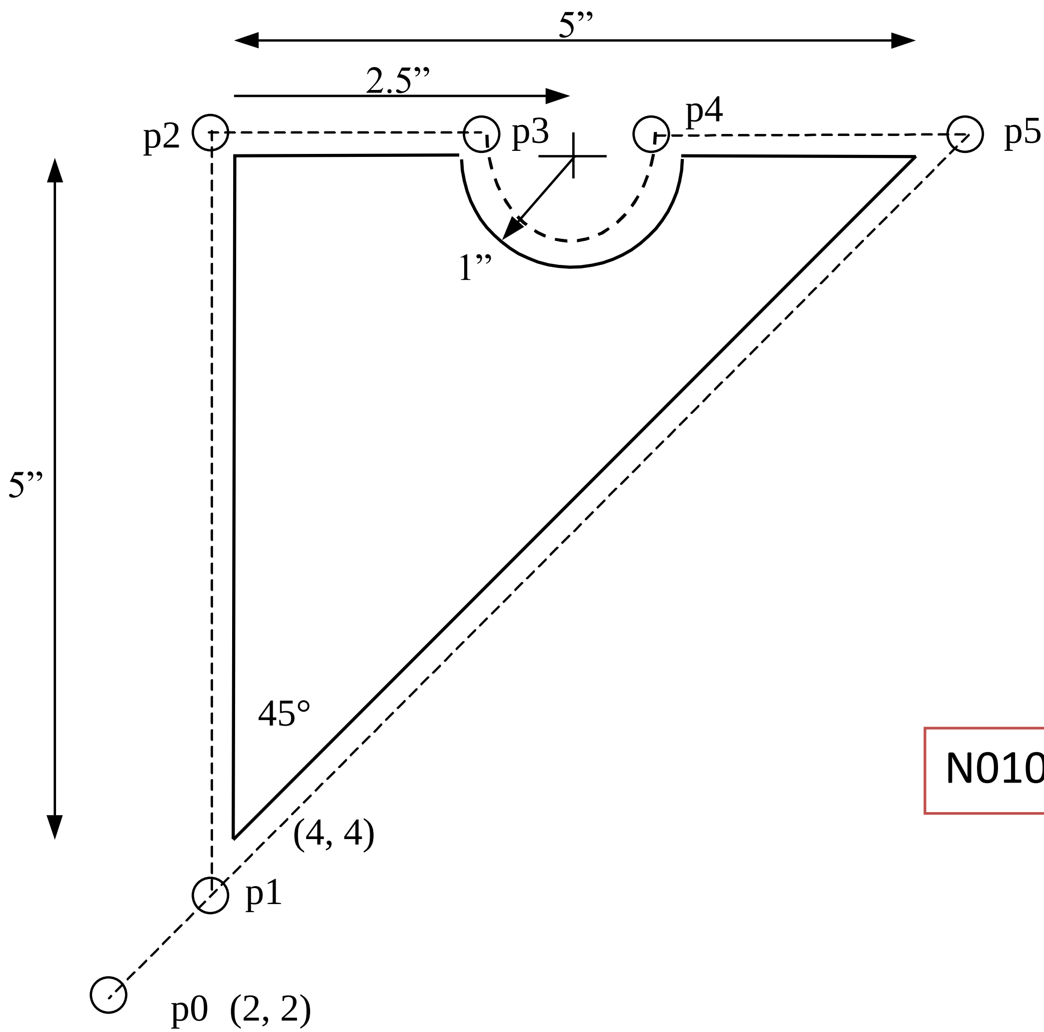
Manual Part Programming Example



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

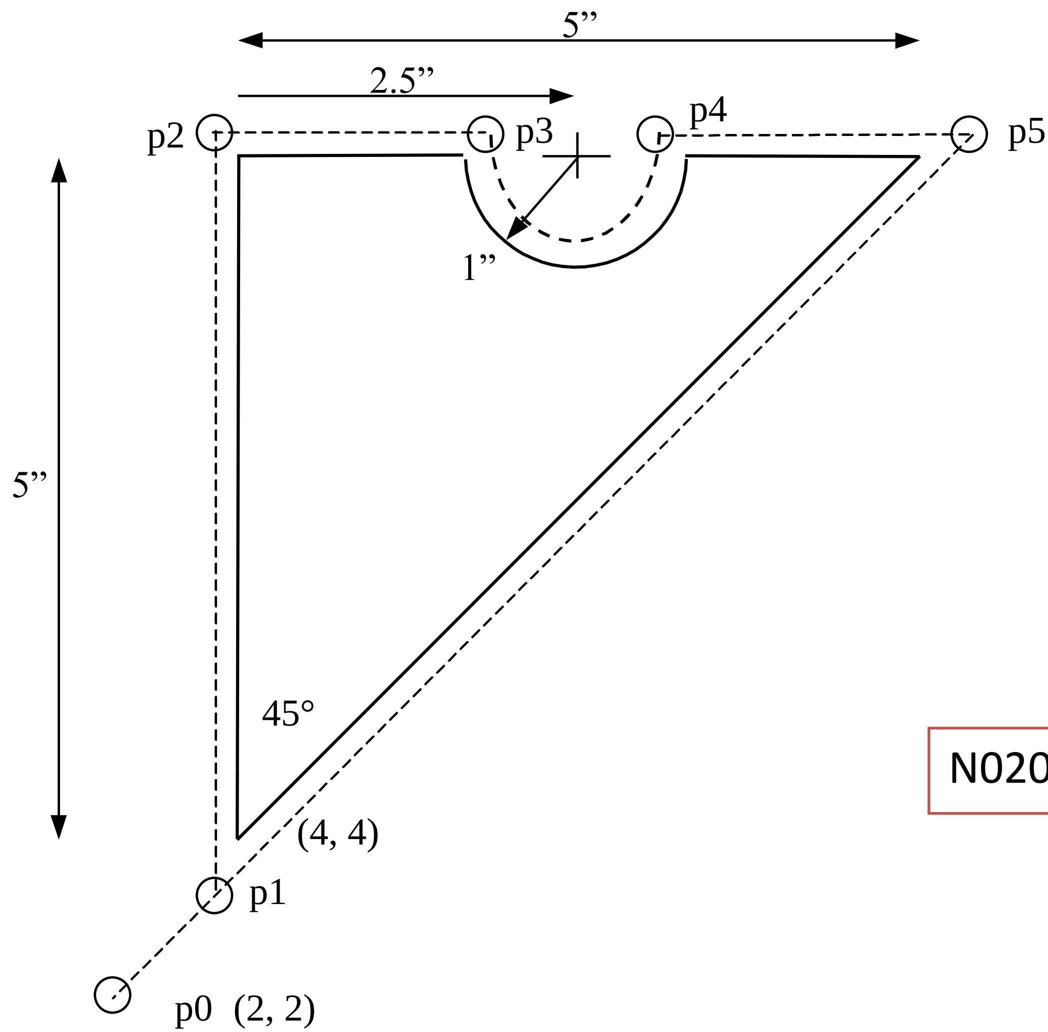
1. Set up the programming parameters

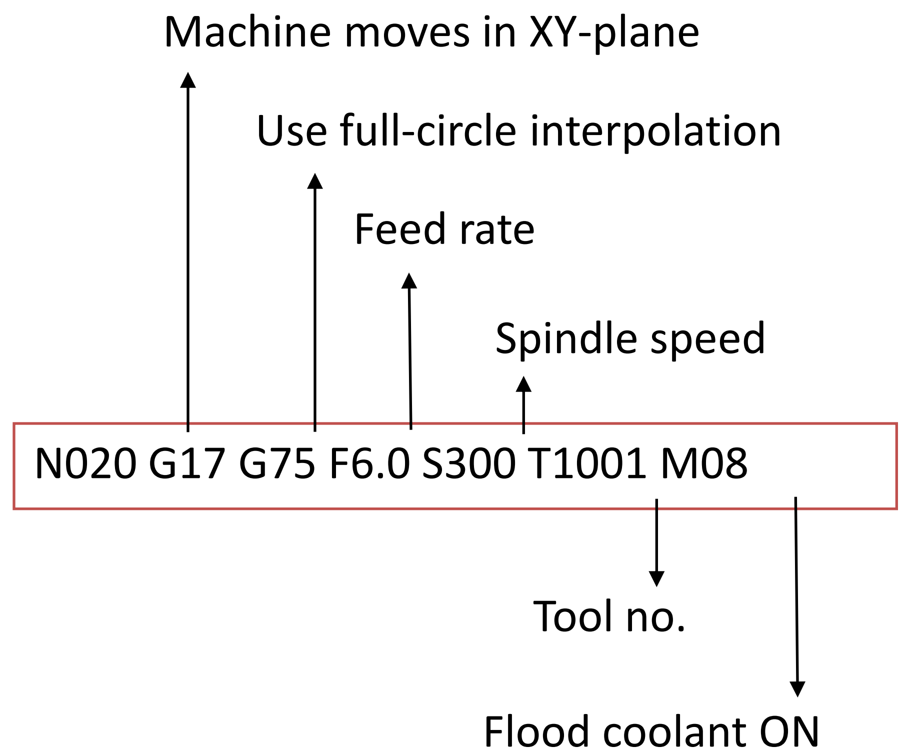


N010 G70 G90 G94 G97 M04

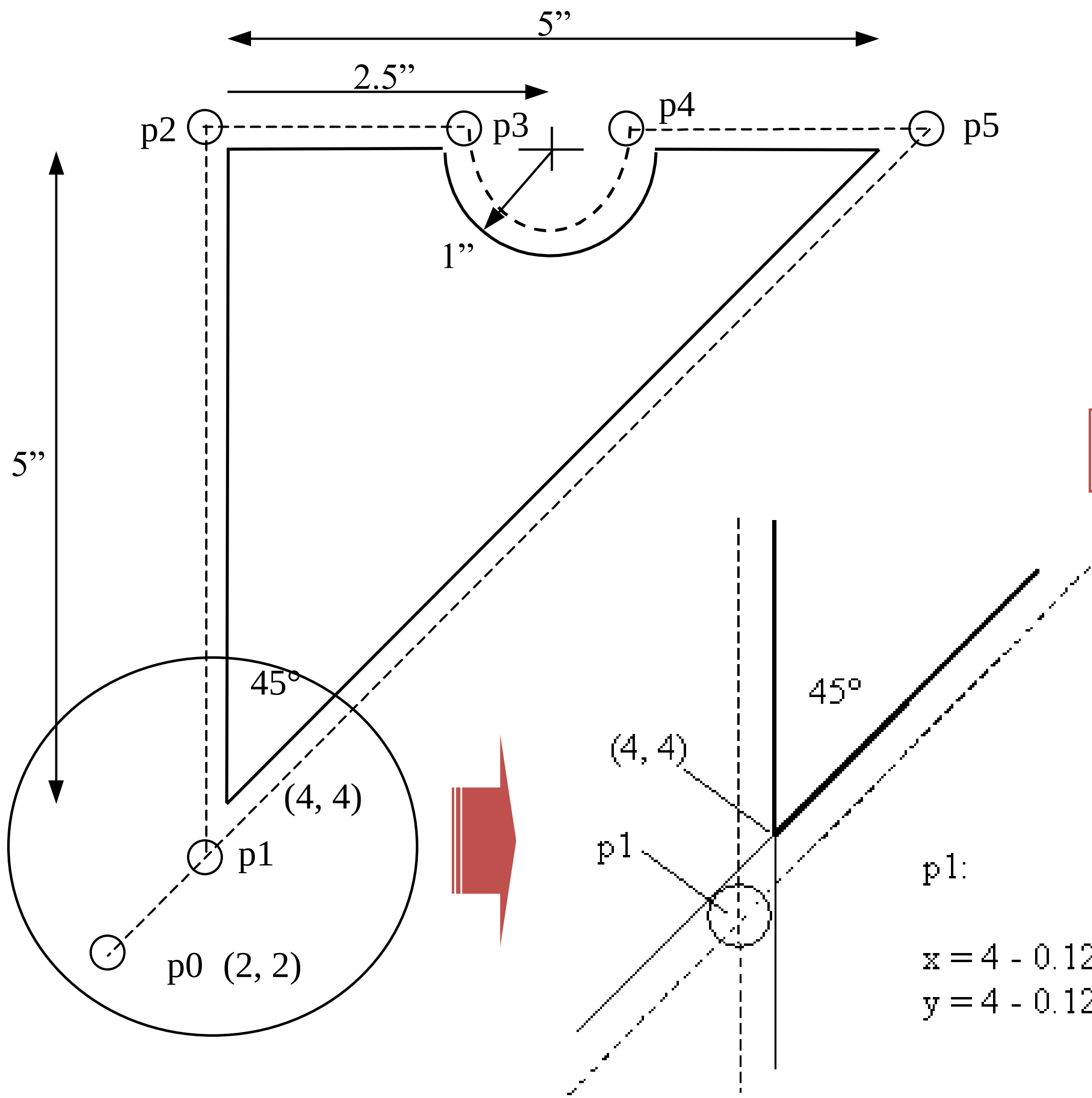
- G70: Programming in inches
- G90: Use absolute coordinates
- G94: Feed in ipm
- G97: Spindle speed in rpm
- M04: Spindle CCW

2. Set up the machining conditions





3. Move tool from p0 to p1 in straight line



Linear interpolation

target coordinates

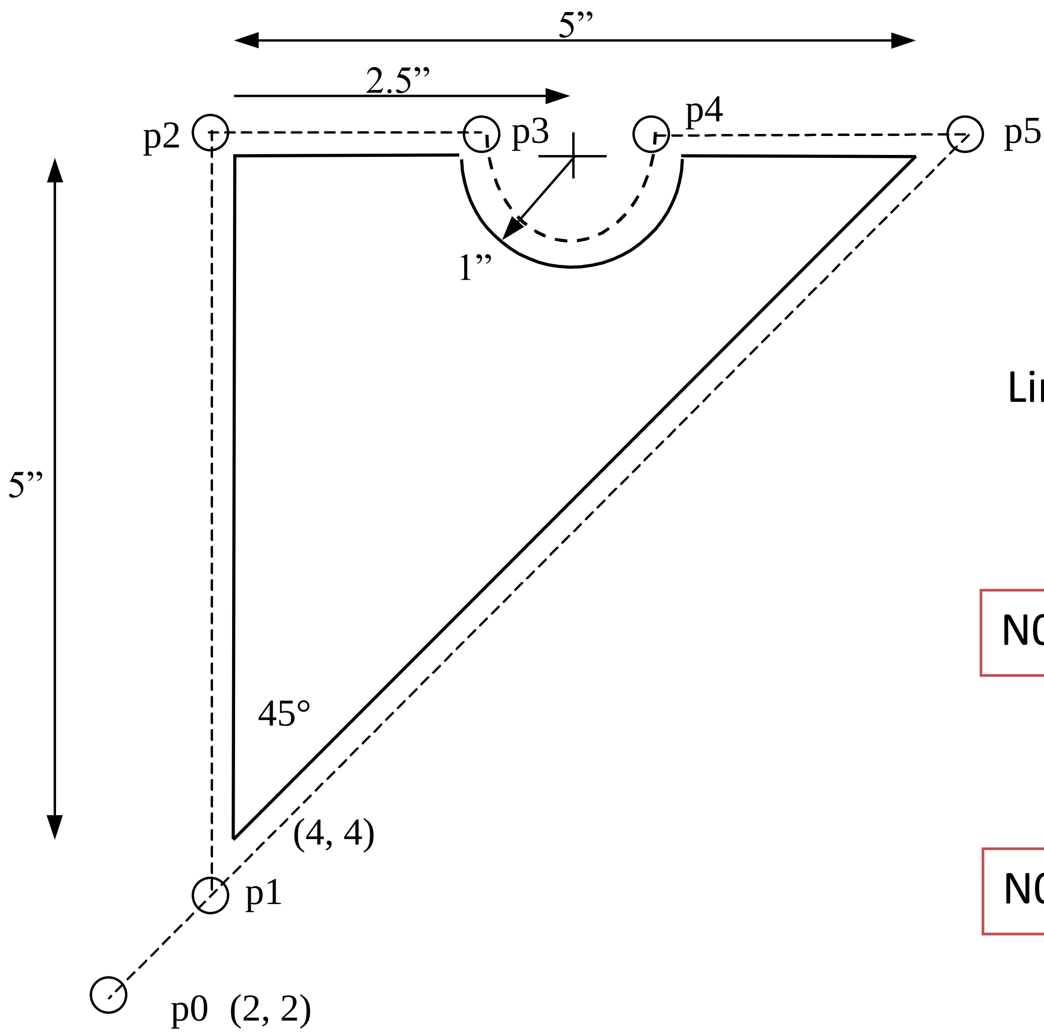
N030 G01 X3.875 Y3.698

p1:

$x = 4 - 0.125 = 3.875$

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

4. Cut profile from p1 to p2



Linear interpolation

target coordinates

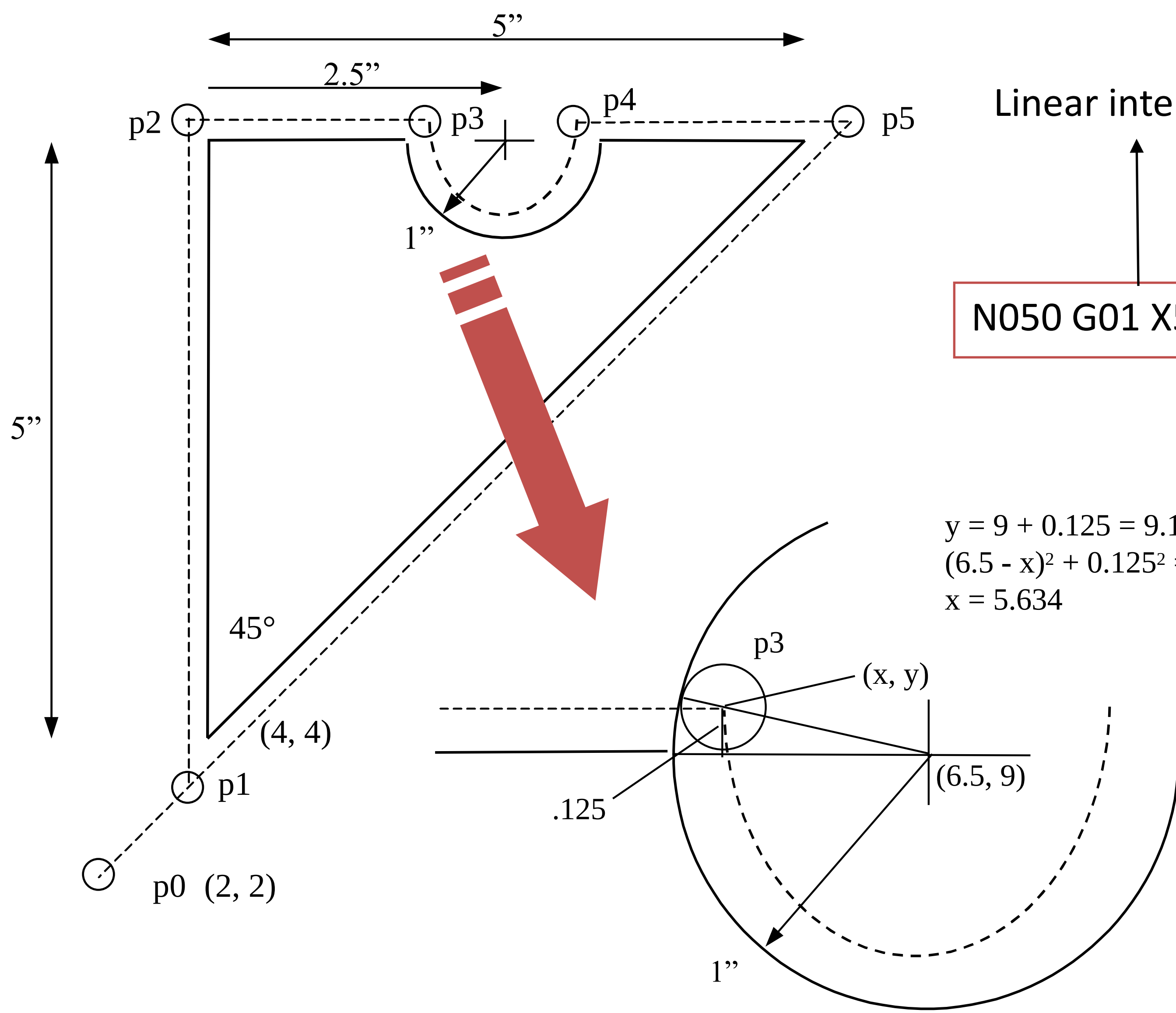
N040 G01 X3.875 Y9.125

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

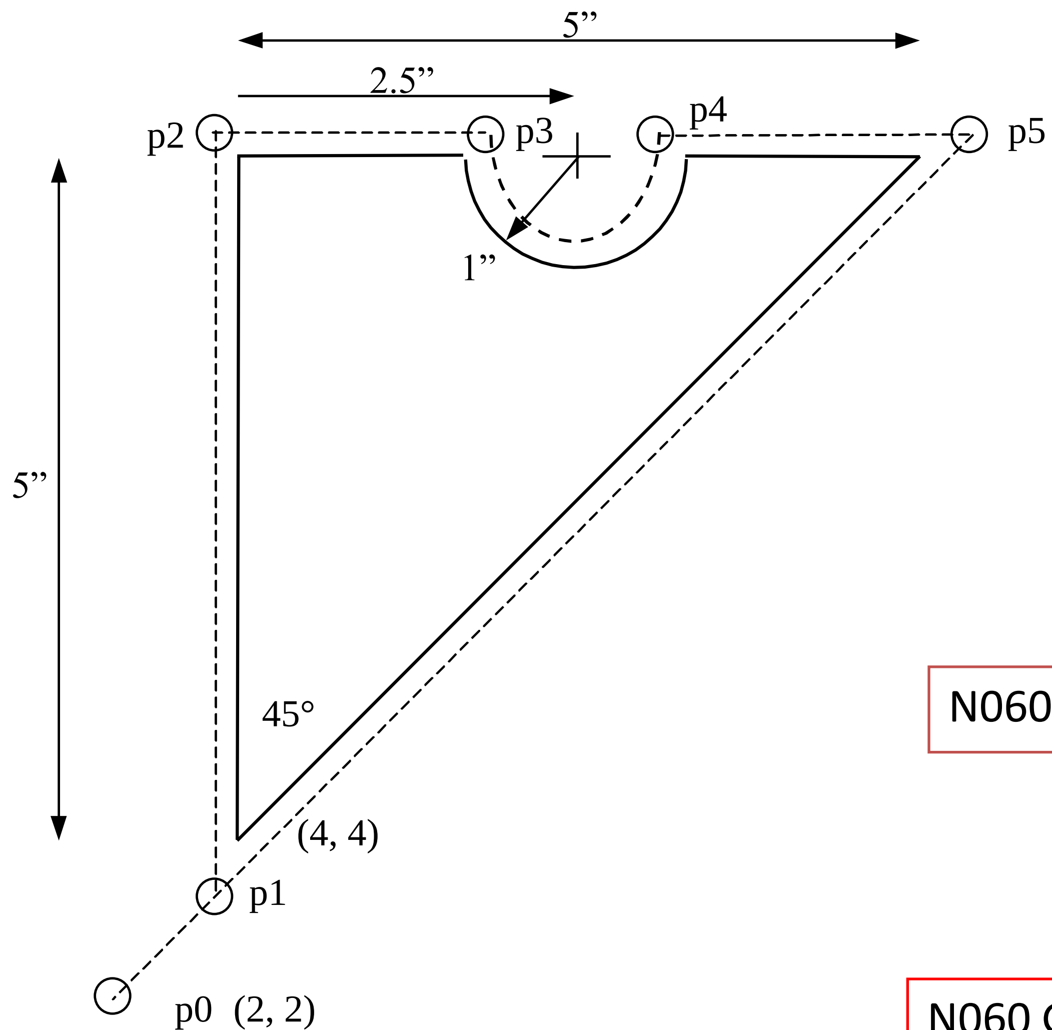
```
N050 G01 X5.634 Y9.125
```

$y = 9 + 0.125 = 9.125$

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

$x = 5.634$

6. Cut along circle from p3 to p4



circular interpolation, CCW motion

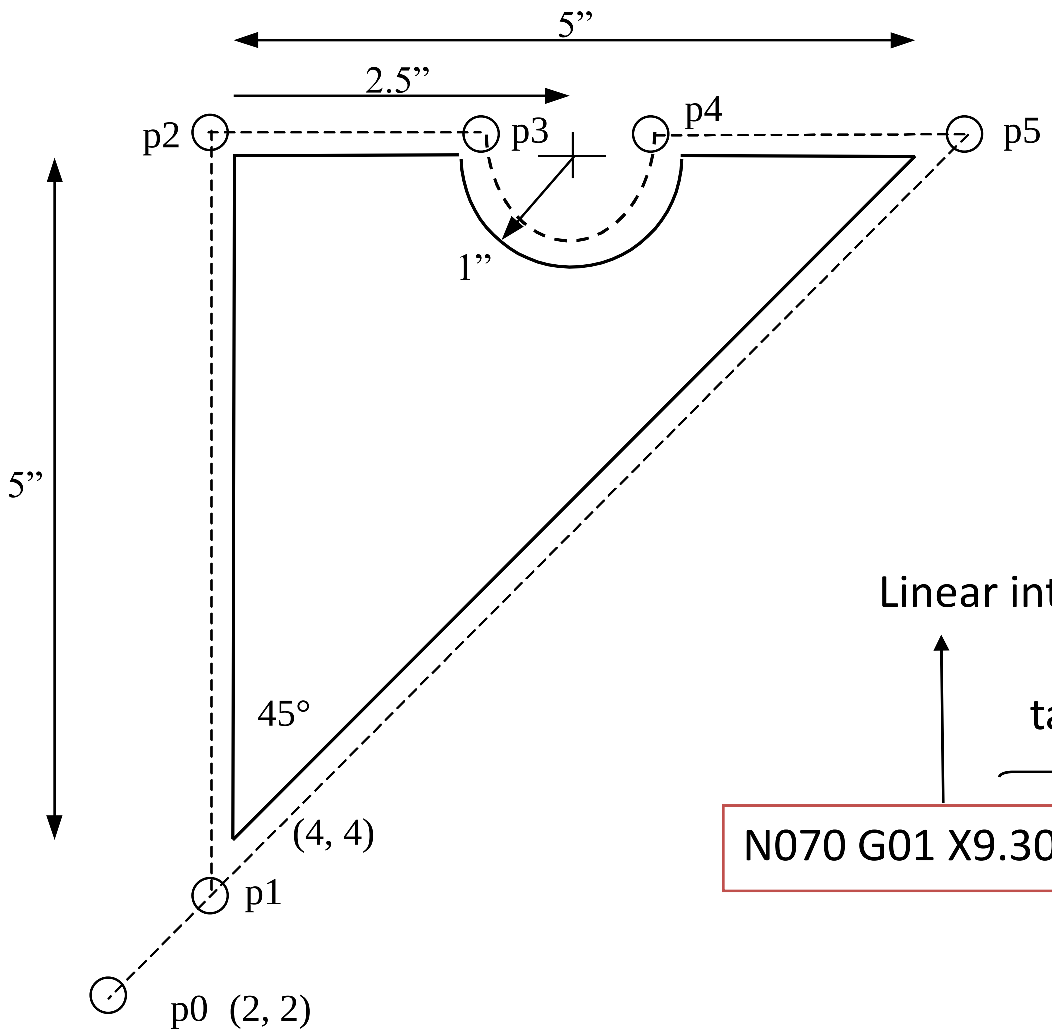
target coordinates

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

coordinates of center of circle

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

7. Cut from p4 to p5

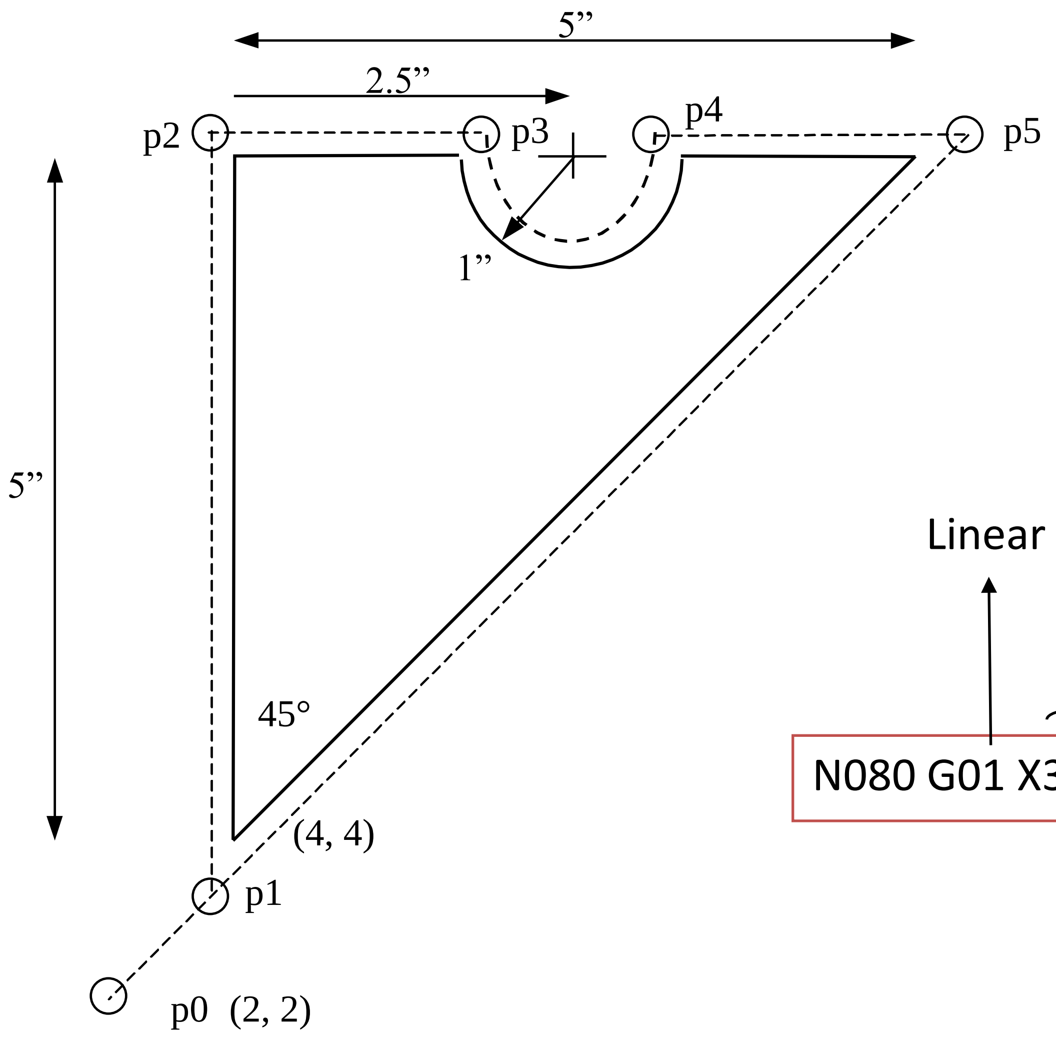


Linear interpolation

target coordinates (Y is unchanged)

N070 G01 X9.302

8. Cut from p5 to p1

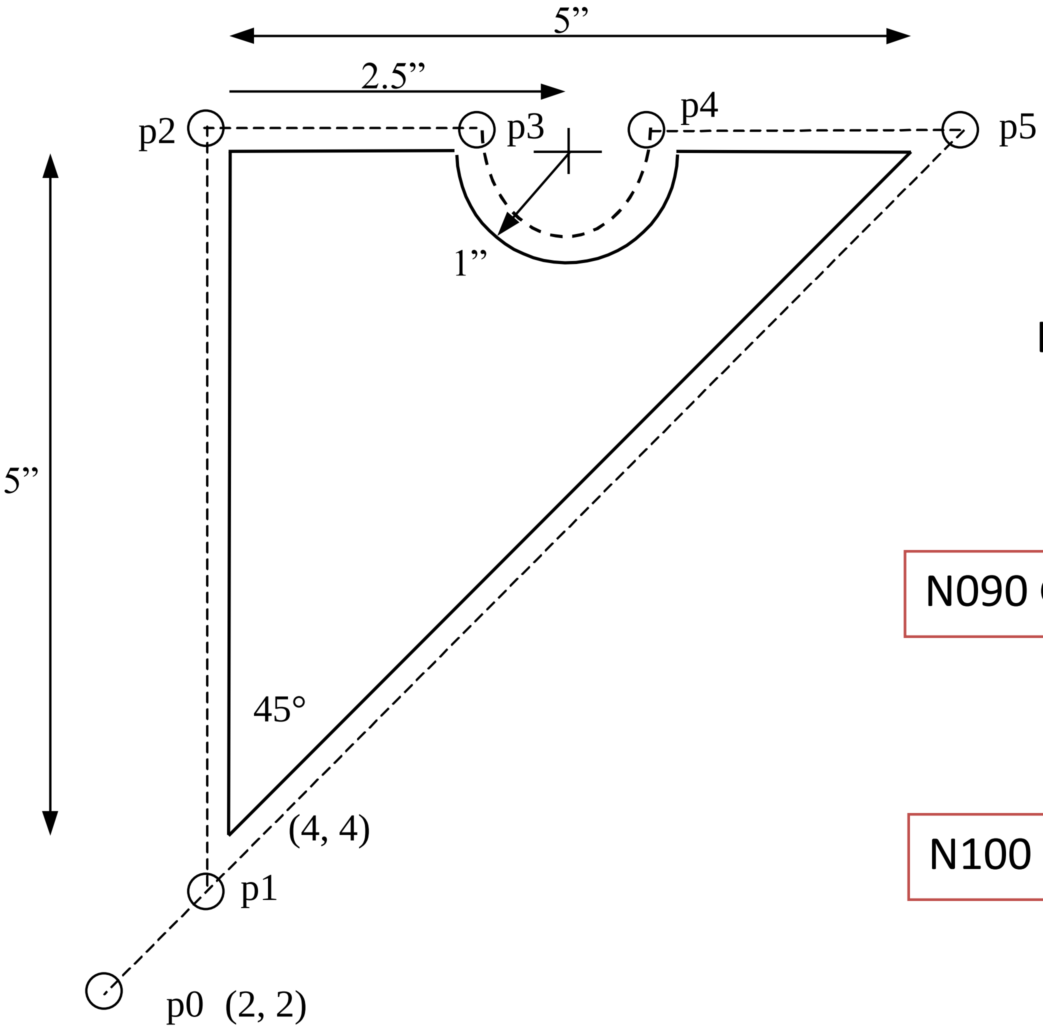


Linear interpolation

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

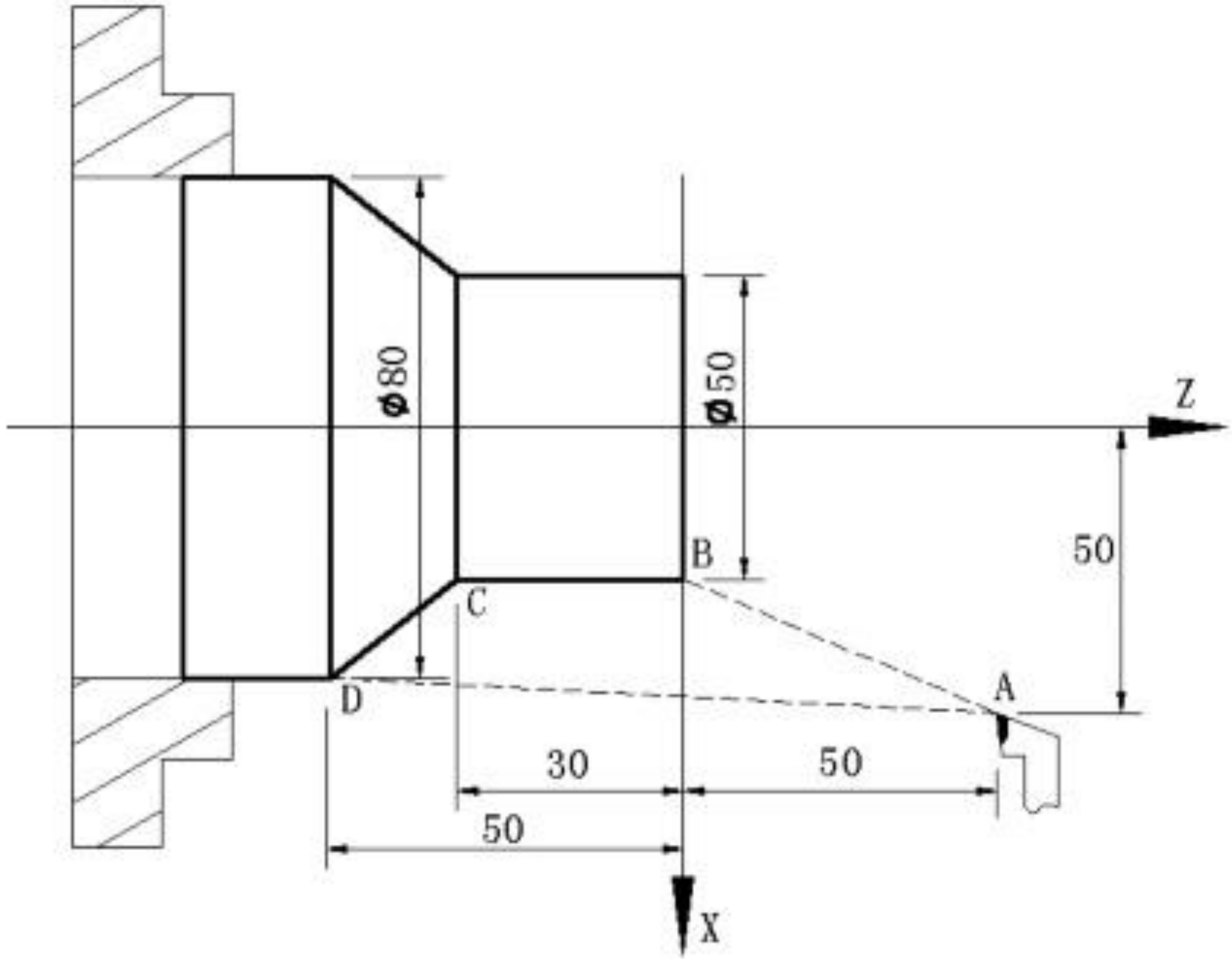
```
N100 M00
```

program stop

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

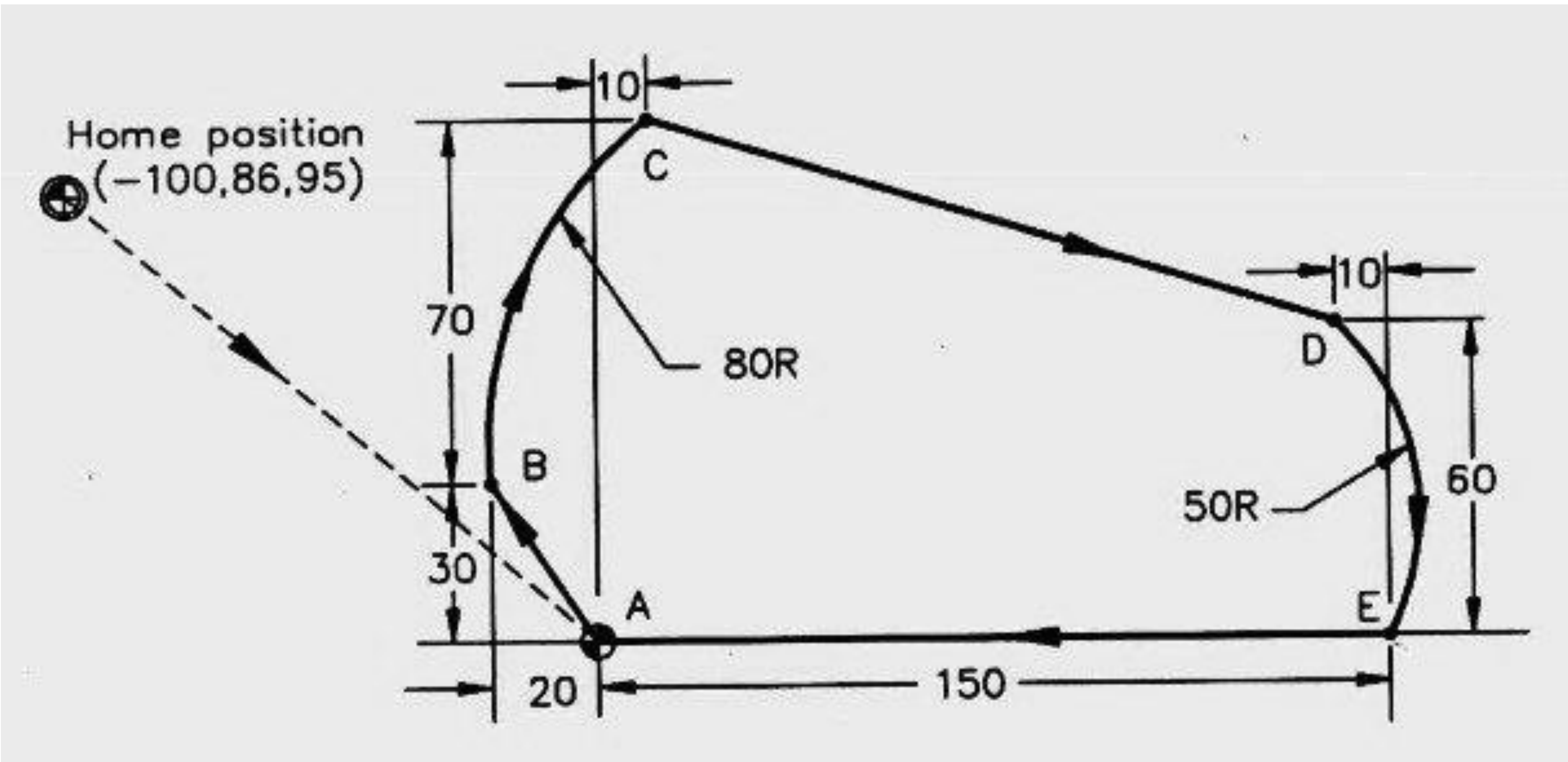
N80 Stopping the spindle

N90 Cooling OFF

N100 Releasing workpiece

N110 End of program, spindle stopping and Cooling OFF

CNC MILLING EXAMPLE

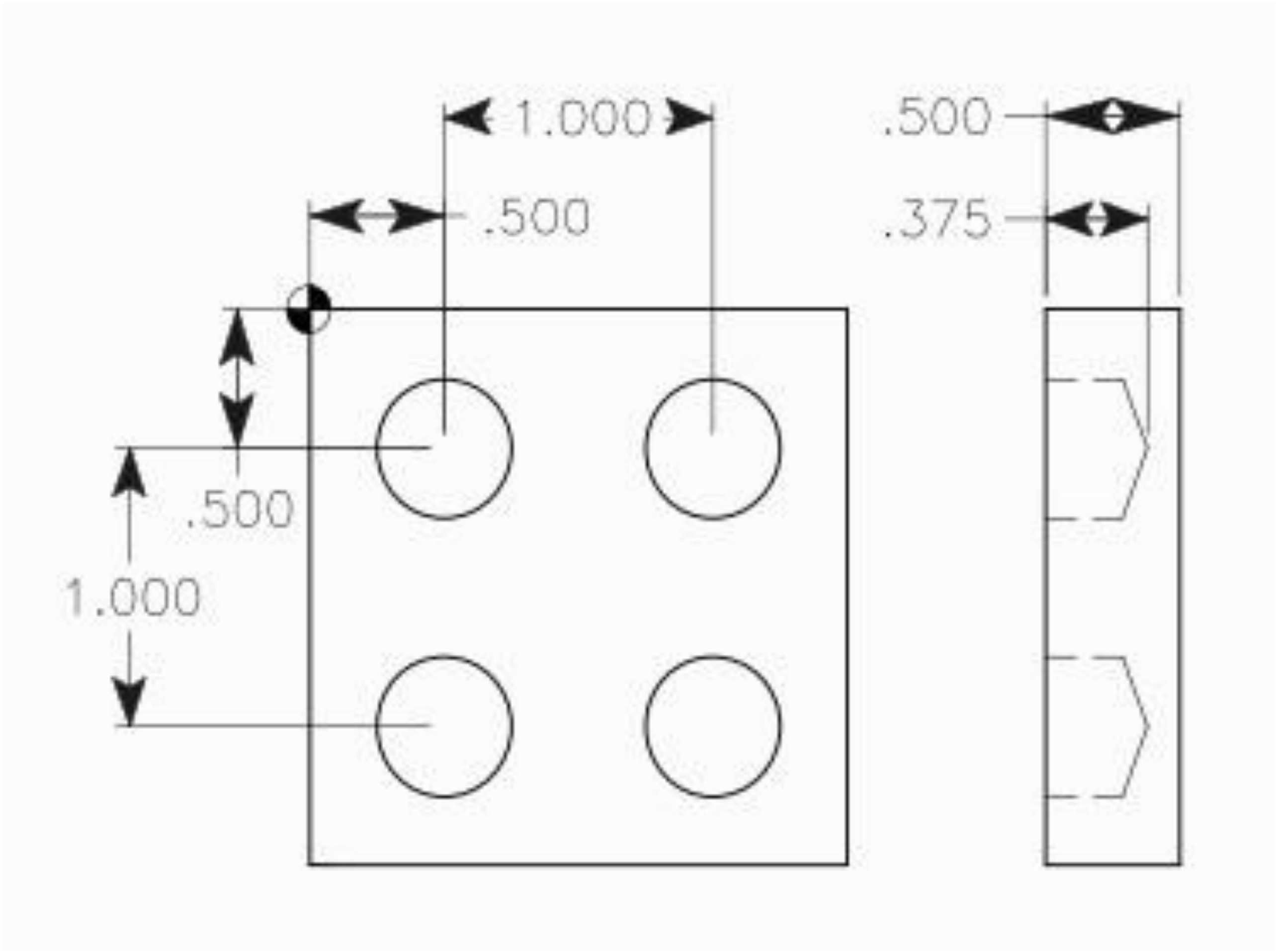


```
N5 G90 G71
N10 T1 M6
N15 G92 X-100 Y86 Z95
N20 G0 X0 Y0 S2500 M3
N25 Z12.5
N30 G1 Z-12.5 F150
N35 X-20 Y30
N40 G2 X10 Y100 R80
N45 G1 X140 Y60
N50 G2 X150 Y0 R50
N55 G1 X0 Y0
N60 G0 Z12.5
N65 G91 G28 Z0 M5
N70 G91 G28 X0 Y0
N75 M30
```

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.

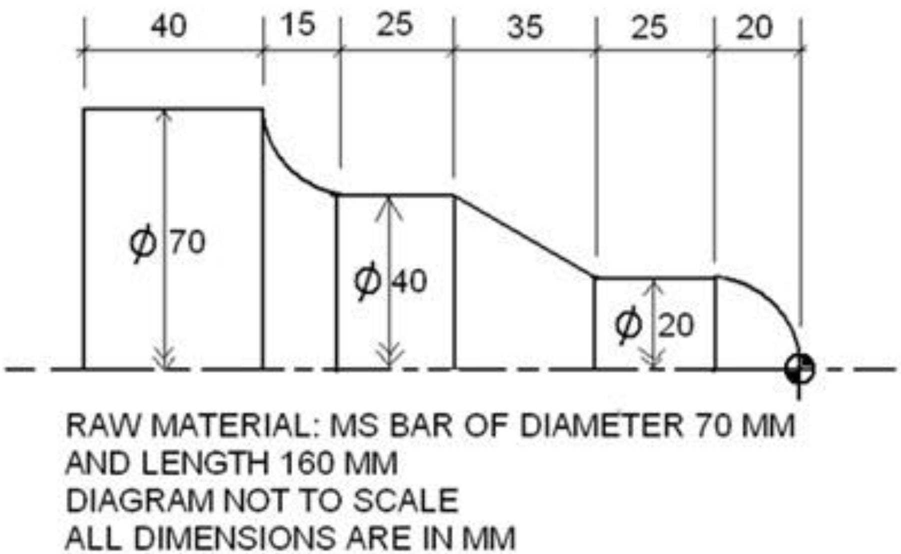


Figure A component to be turned.

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		%