



Lab Sheet for CNC Laboratory

Department of Production Engineering and Metallurgy

Prepared by: *Dr. Laith Abdullah Mohammed*

Numerical Control (NC) Fundamentals

■ What is Numerical Control (NC)?

Form of programmable automation in which the processing equipment (e.g., machine tool) is controlled by coded instructions using numbers, letter and symbols

- Numbers form a set of instructions (or NC program) designed for a particular part.
- Allows new programs on same machined for different parts.
- Most important function of an NC system is positioning (tool and/or work piece).

■ When is it appropriate to use NC?

1. Parts from similar raw material, in variety of sizes, and/or complex geometries.
2. Low-to-medium part quantity production.
3. Similar processing operations & sequences among work pieces.
4. Frequent changeover of machine for different part numbers.
5. Meet tight tolerance requirements (compared to similar conventional machine tools).

■ Advantages of NC over conventional systems:

Flexibility with accuracy, repeatability, reduced scrap, high production rates, good quality. Reduced tooling costs. Easy machine adjustments. More operations per setup, less lead time, accommodate design change, reduced inventory. Rapid programming and program recall, less paperwork. Faster prototype production. Less-skilled operator, multi-work possible.

■ Limitations of NC:

- Relatively high initial cost of equipment.
- Need for part programming.
- Special maintenance requirements.
- More costly breakdowns.

■ Advantages of CNC over conventional NC:

- Control using software (executive) rather than hard-wired
- Increased flexibility (variety of mixed operations & functions)
- Elimination of tape reader (or tape read only once per program)
- Part program storage (computer memory (multiple programs) & storage media)
- Display shows instructions being executed & other operational data
- Greater accuracy (faster control solutions)
- More versatility (e.g., program editing (at the machine), reprogramming, tool path plotting, metric conversion, cutter dimension compensation)
- Fixed (subroutine) cycles (e.g., pocket milling, pecking)
- Manual data input (MDI) (even while another program is running) and remote data transfer
- System integration capability (connect to robots & other computer- or microprocessor-based equipment, create cells)
- Machine diagnostics (gives error message or identifies problem)

■ Distributed (Direct) Numerical Control (DNC):

- (Direct) Central computer stores programs & directs NC operations; NC machines dependent on central computer.
- (Distributed) Central computer stores programs and transfers programs to CNC machines.
- Central computer provides management functions (e.g., part of MIS) Programs stored as cutter location (CL) files and post-processed for the machine assigned the job.
- Major components: central computer; bulk memory; telecommunication (EDI); CNC machine tools.

■ Three basic components of an NC system:

1. Input medium:

- Part program or instructions needed to drive the machine tool components
- Instructions are prepared manually or by use of computer
- Instructions include machining parameters (feed rate, cutting speed); sequence of actions (e.g., positioning & machine functions)
- Instructions are stored in the form of tape (paper, magnetic); floppy diskettes; DNC download to CNC RAM

2. Machine control unit (MCU):

- Electronics & control hardware
- Interpret instruction set
- Execute instructions
- Monitor results & correct where appropriate

3. Machine tool:

- Mechanical structure that performs the machining, including the components that drive each axis of motion (e.g., AC or DC motor; hydraulic actuator; stepper motor — choice affects speed of response, accuracy and power capacity).

■ What is meant by axis of motion?

Axis of motion describes the relative motion that occurs between the cutting tool and the workpiece. Three main axes of motion for machine tools are referred to as the x, y, and z axes that form a right hand coordinate system.

■ Open loop vs. closed loop control:

- Open loop — control signals are given to actuators by the MCU, but the movements and final destinations of the positioning system are not checked for accuracy.
- Closed loop — equipped with transducers and sensors to measure positions, compare with control signals, and correct positions as necessary.

■ Types of control systems:

- Point-to-point (positioning) system: tool or workpiece is driven to predefined location without specifying the speed or path of motion; no tool-workpiece contact during positioning; simple, inexpensive control system; used mainly in drilling, punching, straight milling
- Contouring (continuous path) system: positioning (path) and speed are simultaneously controlled at specified values; contour path based on interpolation between points; complex, flexible, expensive control system; capacity for simultaneous control of more

than one axis of motion; used on lathes, milling machines, grinders, welding machinery, machining centers.

■ Types of interpolation:

- Linear: tool or work piece moves in a straight line from specified start point to endpoint in two or three axes
- Circular: path of motion is derived from specified coordinates of end points, coordinates of arc center, radius of arc and direction of motion; tool path approximated by series of straight line segments.

■ Accuracy in NC:

- Accuracy: measure of control system's capacity to position the tool or workpiece to a specified set of coordinates.
- Repeatability: measure of agreement among repeated positioning movements under the same operating conditions.
- Resolution: smallest increment of positioning that is possible

■ Numerical control programming:

Steps in NC/CNC programming procedure:

1. Interpret part drawing
 - Define zero point
 - define x-, y-, z-axes
2. Determine machining requirements
 - Determine required operations & sequence
 - Determine tooling requirements
 - Determine feeds, speeds, depth(s) of cut
3. Complete part program
4. Complete post processor
5. Store part program (e.g., punch tape, disk file)
6. Verify completed program: "check it out;" is it accurate?

■ Post-processors and beyond:

- Post-processor: software used to translate NC program into code that is specific to the machine that will run the program.
- EIA Standard 494 (32-Bit Binary CL Exchange Input Format for Numerical Controlled Machines (BCL for short)): specifies data structure for CL data (integer code) to be downloaded to a BCL equipped MCU (before post-processing; converted to machine-specific code).

■ Cartesian coordinate system:

Almost everything that can be produced on a conventional machine tool can be produced on a CNC machine tool, with its many advantages. The machine tool movements used in producing a product are of two basic types: point -to-point and continuous path (contouring movements). The Cartesian, or rectangular, coordinate system was devised by the French mathematician and philosopher Rene' Descartes. With this system, any specific point can be described in mathematical terms from any other point along three perpendicular axes. This concept fits machine tools perfectly since their construction is generally based on three axes of motion (X,Y,Z) plus an axis of rotation. On a plain vertical milling machine, the X axis is the horizontal

movement (right or left) of the table, the Y axis is the table cross movement (toward or away from the column), and the Z axis is the vertical movement of the knee or the spindle. CNC systems rely heavily on the use of rectangular coordinates because the programmer can locate every point on a job precisely. When points are located on a work piece, two straight intersecting lines, one vertical and one horizontal, are used. These lines must be at right angles to each other, and the point where they cross is called the origin, or zero point (Fig. 1)

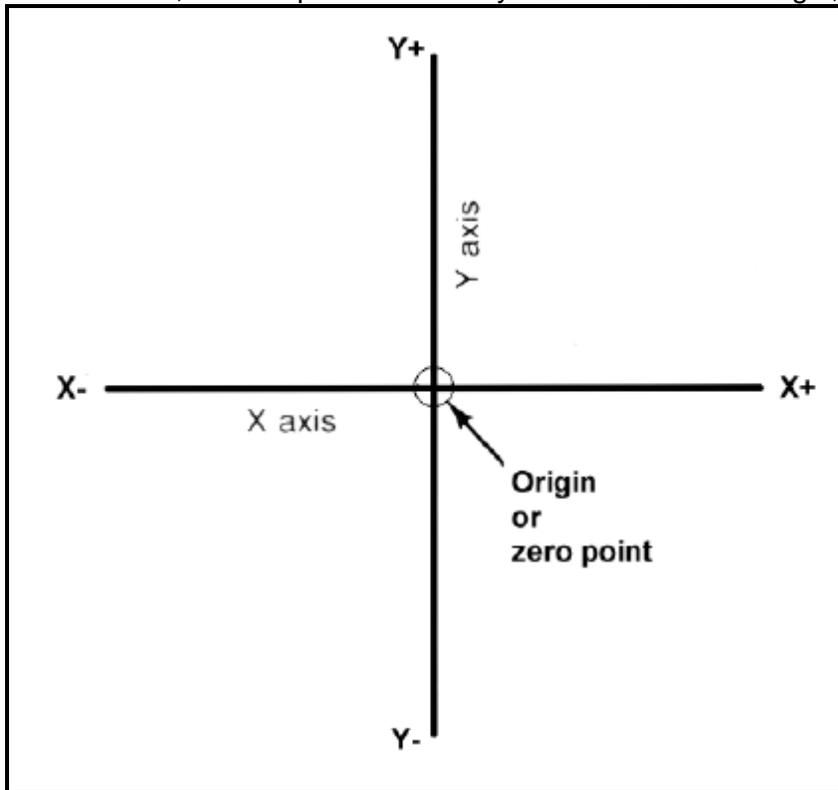


Fig.(1)

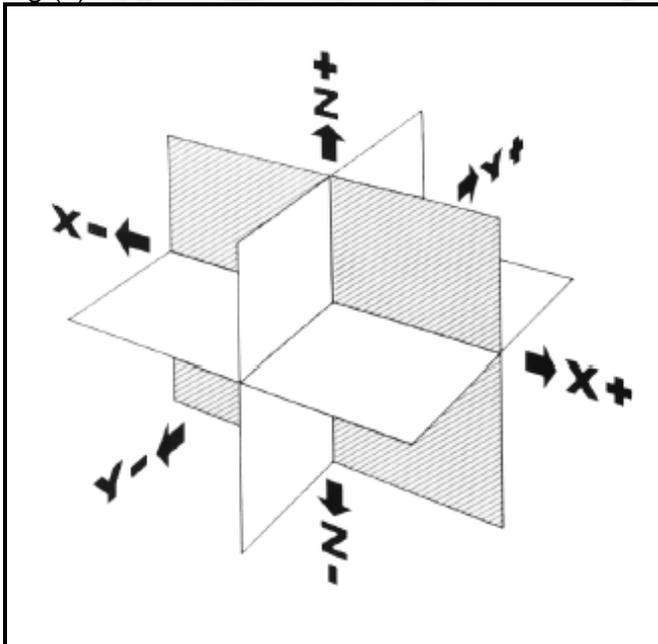


Fig.(2)

The three-dimensional coordinate planes are shown in Fig. 2. The X and Y planes (axes) are horizontal and represent horizontal machine table motions. The Z plane or axis represents the vertical tool motion. The plus (+) and minus (-) signs indicate the direction from the zero point

(origin) along the axis of movement. The four quadrants formed when the XY axes cross are numbered in a counterclockwise direction (Fig. 3). All positions located in quadrant 1 would be positive X (X+) and positive Y (Y+). In the second quadrant, all positions would be negative X (X-) and positive Y (Y+). In the third quadrant, all locations would be negative X (X-) and negative Y (Y-). In the fourth quadrant, all locations would be positive X (X+) and negative Y (Y-).

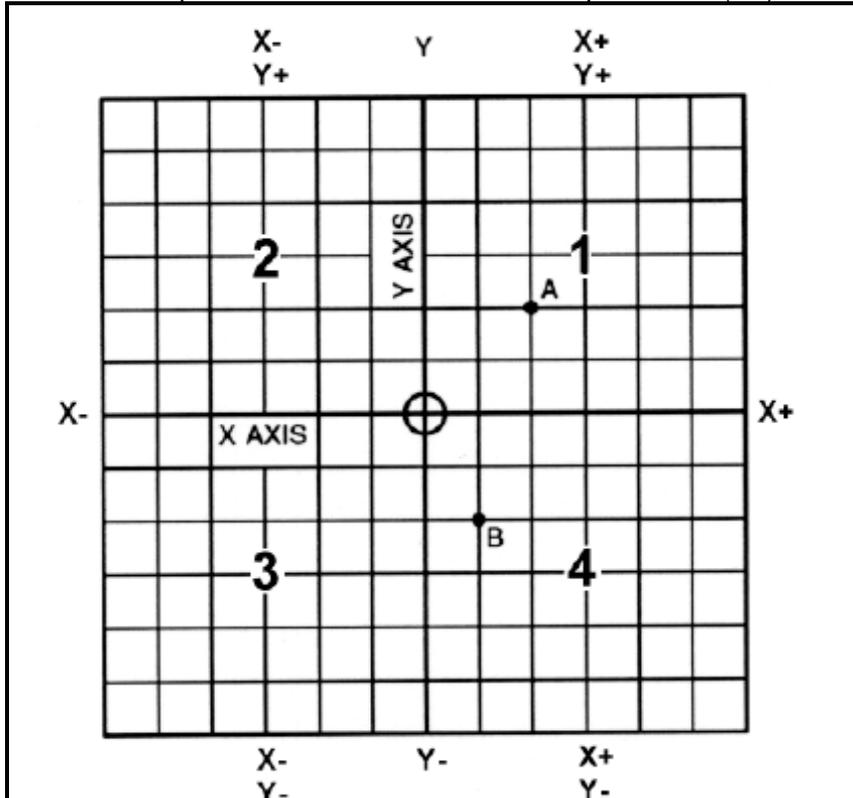


Fig.(3)

In Fig. 3, point A would be 2 units to the right of the Y axis and 2 units above the X axis. Assume that each unit equals 1.000. The location of point A would be X + 2.000 and Y + 2.000. For point B, the location would be X + 1.000 and Y - 2.000. In CNC programming it is not necessary to indicate plus (+) values since these are assumed. However, the minus (-) values must be indicated. For example, the locations of both A and B would be indicated as follows:

A X2.000 Y2.000

B X1.000 Y-2.000

■ Machines Using CNC:

Machine Types:

Lathe

The engine lathe, one of the most productive machine tools, has always been an efficient means of producing round parts (Fig.4). Most lathes are programmed on two axes.

- The X axis controls the cross motion of the cutting tool. Negative X (X-) moves the tool towards the spindle centerline; positive X moves the tool away from the spindle centerline.
- The Z axis controls the carriage travel toward or away from the headstock.

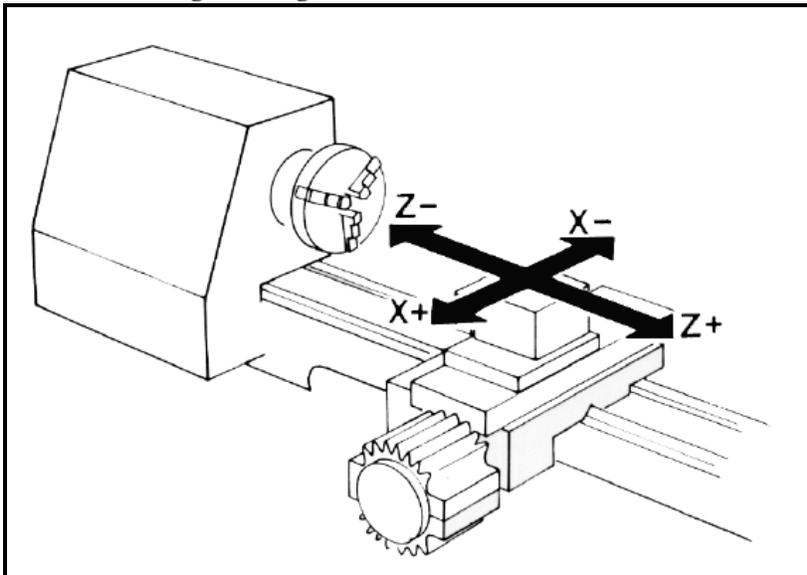


Fig. 4: The main axes of a lathe or turning center.

Milling Machine

The milling machine has always been one of the most versatile machine tools used in industry (Fig. 5). Operations such as milling, contouring, gear cutting, drilling, boring, and reaming are only a few of the many operations which can be performed on a milling machine. The milling machine can be programmed on three axes:

- The X axis controls the table movement left or right.
- The Y axis controls the table movement toward or away from the column.
- The Z axis controls the vertical (up or down) movement of the knee or spindle.

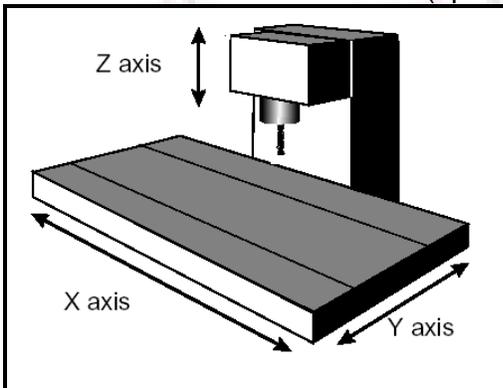
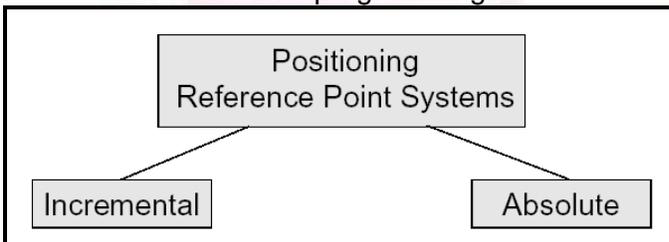


Fig. 5: The main axes of a vertical machining center.

■ Programming Systems:

Two types of programming modes, the incremental system and the absolute system, are used for CNC. Both systems have applications in CNC programming, and no system is either right or wrong all the time. Most controls on machine tools today are capable of handling either incremental or absolute programming.



Incremental program locations are always given as the distance and direction from the immediately preceding point (Fig. 6). Command codes which tell the machine to move the table, spindle, and knee are explained here using a vertical milling machine as an example:

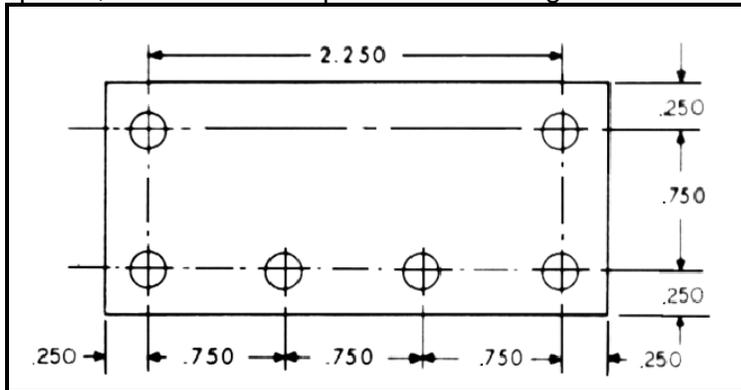


Fig. 6: A workpiece dimensioned in the incremental system mode.

- A “X plus” (X+) command will cause the cutting tool to be located to the right of the last point.
- A “X minus” (X-) command will cause the cutting tool to be located to the left of the last point.
- A “Y plus” (Y+) command will cause the cutting tool to be located toward the column.
- A “Y minus” (Y-) will cause the cutting tool to be located away from the column.
- A “Z plus” (Z+) command will cause the cutting tool or spindle to move up or away from the workpiece.
- A “Z minus” (Z-) moves the cutting tool down or into the workpiece.

In incremental programming, the G91 command indicates to the computer and MCU (Machine Control Unit) that programming is in the incremental mode.

Absolute program locations are always given from a single fixed zero or origin point (Fig. 7). The zero or origin point may be a position on the machine table, such as the corner of the worktable or at any specific point on the workpiece. In absolute dimensioning and programming, each point or location on the workpiece is given as a certain distance from the zero or reference point.

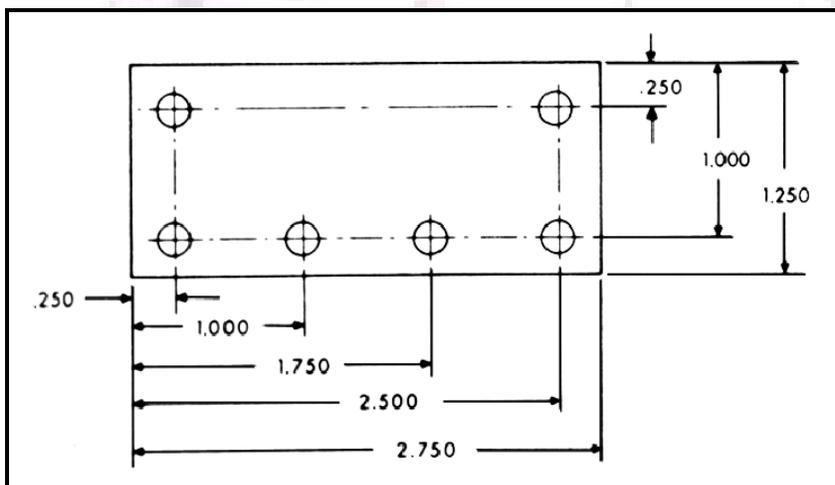


Fig. 7: A workpiece dimensioned in the absolute system mode. Note: All dimensions are given from a known point of reference.

- A “X plus” (X+) command will cause the cutting tool to be located to the right of the zero or origin point.
- A “X minus” (X-) command will cause the cutting tool to be located to the left of the zero or origin point.
- A “Y plus” (Y+) command will cause the cutting tool to be located toward the column.

- A “Y minus” (Y-) command will cause the cutting tool to be located away from the column.

In absolute programming, the G90 command indicates to the computer and MCU that the programming is in the absolute mode.

■ Point-to-Point or Continuous Path

CNC programming falls into two distinct categories (Fig. 8). The difference between the two categories was once very distinct. Now, however, most control units are able to handle both point-to-point and continuous path machining. A knowledge of both programming methods is necessary to understand what applications each has in CNC.

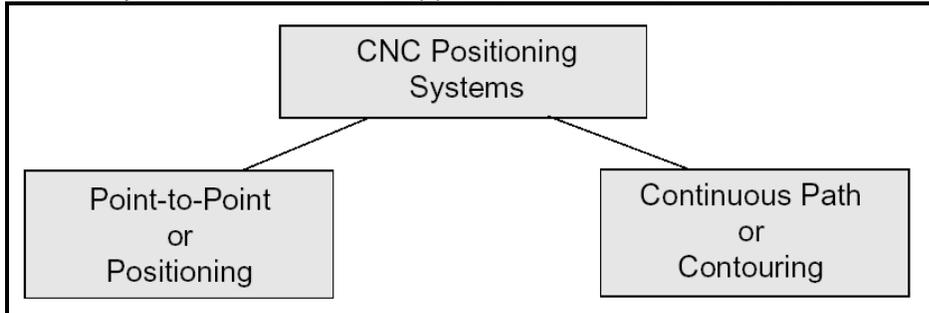


Fig. 8: Types of CNC positioning systems.

■ Point-to-Point Positioning

Point-to-point positioning is used when it is necessary to accurately locate the spindle, or the workpiece mounted on the machine table, at one or more specific locations to perform such operations as drilling, reaming, boring, tapping, and punching (Fig.9). Point-to-point positioning is the process of positioning from one coordinate (XY) position or location to another, performing the machining operation, and continuing this pattern until all the operations have been completed at all programmed locations.

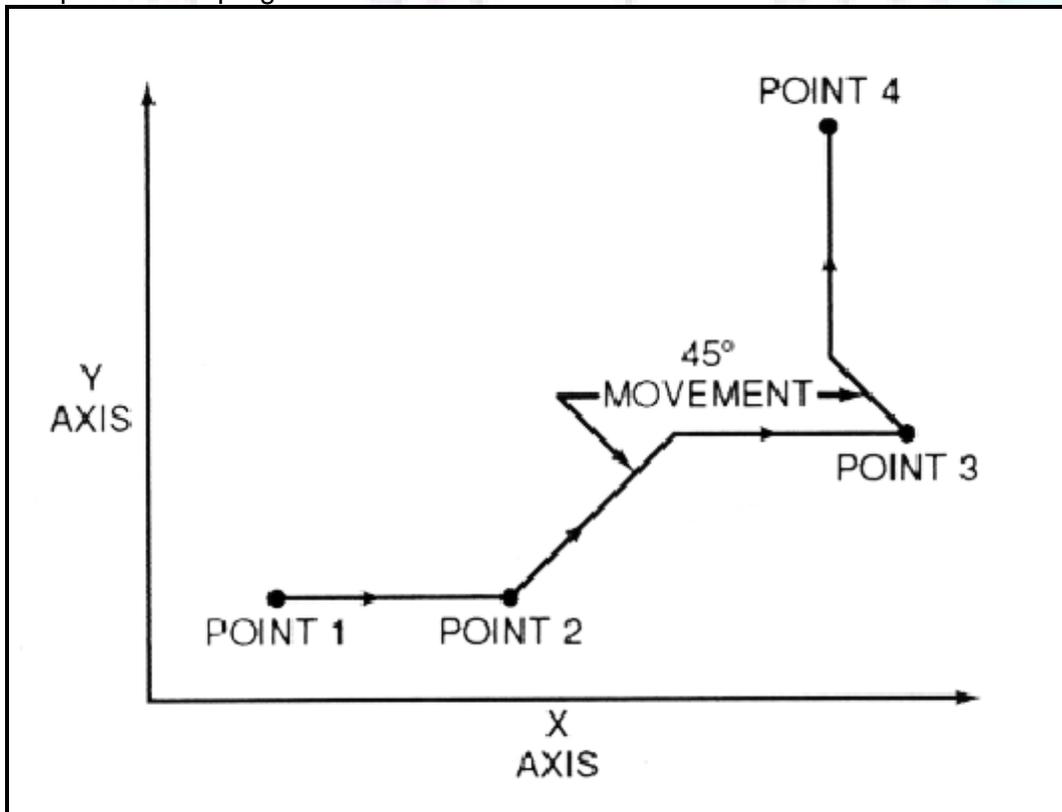


Fig. 9: The path followed by point-to-point positioning to reach various programmed points (machining locations) on the XY axis.

In Fig. 9 point 1 to point 2 is a straight line, and the machine moves only along the X axis; but points 2 and 3 require that motion along both the X and Y axes takes place. As the distance in the X direction is greater than in the Y direction, Y will reach its position first, leaving X to travel in a straight line for the remaining distance. A similar motion takes place between points 3 and 4.

■ Continuous Path (Contouring)

Contouring, or continuous path machining, involves work such as that produced on a lathe or milling machine, where the cutting tool is in contact with the workpiece as it travels from one programmed point to the next. Continuous path positioning is the ability to control motions on two or more machine axes simultaneously to keep a constant cutter-workpiece relationship. The programmed information in the CNC program must accurately position the cutting tool from one point to the next and follow a predefined accurate path at a programmed feed rate in order to produce the form or contour required (Fig. 10)

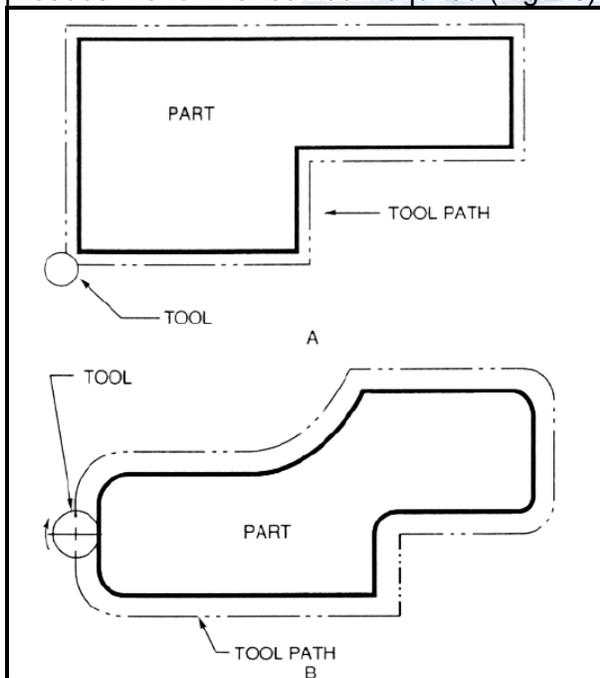


Fig. 10 Types of contour machining (A) Simple contour; (B) complex contour

■ Interpolation:

The method by which contouring machine tools move from one programmed point to the next is called interpolation. This ability to merge individual axis points into a predefined tool path is built into most of today's MCUs. There are five methods of interpolation:

linear, circular, helical, parabolic, and cubic. All contouring controls provide linear interpolation, and most controls are capable of both linear and circular interpolation. Helical, parabolic, and cubic interpolation are used by industries that manufacture parts which have complex shapes, such as aerospace parts and dies for car bodies.

Linear Interpolation:

Linear Interpolation consists of any programmed points linked together by straight lines, whether the points are close together or far apart (Fig. 11). Curves can be produced with linear interpolation by breaking them into short, straight-line segments. This method has limitations, because a very large number of points would have to be programmed to describe the curve in order to produce a contour shape. A contour programmed in linear interpolation requires the coordinate positions (XY positions in two-axis work) for the start and finish of each line segment. Therefore, the end point of one line or segment becomes the start point for the next segment, and so on, throughout the entire program.

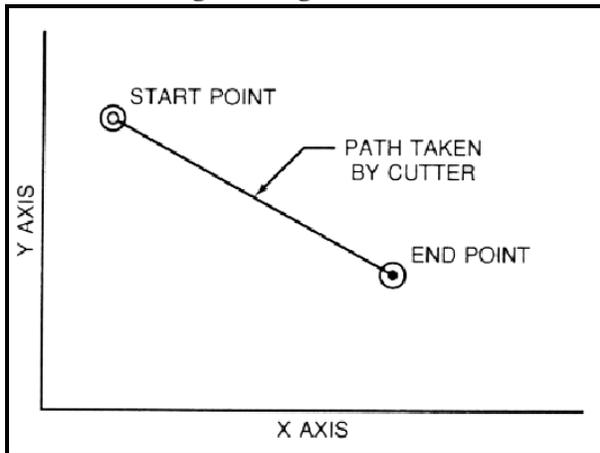


Fig. 11: An example of two-axis linear interpolation.

Circular Interpolation:

The development of MCUs capable of circular interpolation has greatly simplified the process of programming arcs and circles. To program an arc (Fig. 12), the MCU requires only the coordinate positions (the XY axes) of the circle center, the radius of the circle, the start point and end point of the arc being cut, and the direction in which the arc is to be cut (clockwise or counterclockwise) See Fig. 12.

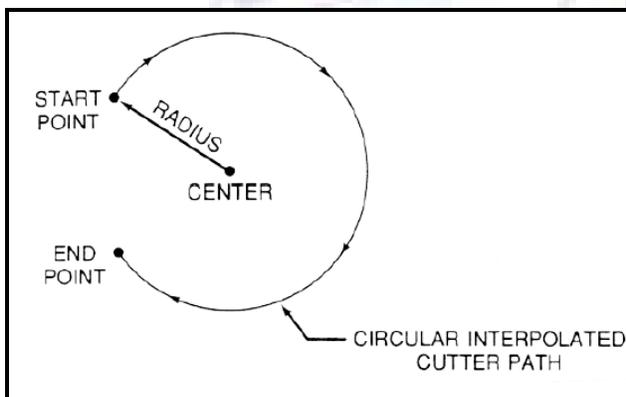


Fig. 12: For two-dimensional circular interpolation the MCU must be supplied with the XY axis, radius, start point, end point, and direction of cut.

■ Programming Format

Word address is the most common programming format used for CNC programming systems. This format contains a large number of different codes that transfers program information from the part print to machine servos, relays, micro-switches, etc., to manufacture a part. These codes, which conform to EIA (Electronic Industries Association) standards, are in a logical sequence called a block of information. Each block should contain enough information to perform one machining operation.

Word Address Format

Every program for any part to be machined must be put in a format that the machine control unit can understand. The format used on any CNC machine is built in by the machine tool builder and is based on the type of control unit on the machine. A variable-block format which uses words (letters) is most commonly used. Each instruction word consists of an address character, such as X, Y, Z, G, M, or S. Numerical data follows this address character to identify a specific function such as the distance, feed rate, or speed value. The address code G90 in a program,

tells the control that all measurements are in the absolute mode. The code G91, tells the control that measurements are in the incremental mode.

Codes

The most common codes used when programming CNC machines tools are G-codes (preparatory functions), and M codes (miscellaneous functions). Other codes such as F, S, D, and T are used for machine functions such as feed, speed, cutter diameter offset, tool number, etc. G-codes are sometimes called cycle codes because they refer to some action occurring on the X, Y, and/or Z axis of a machine tool, The G-codes are grouped into categories such as Group 01, containing codes G00, G01, G02, G03. which cause some movement of the machine table or head. Group 03 includes either absolute or incremental programming, while Group 09 deals with canned cycles. A G00 code rapidly positions the cutting tool while it is above the workpiece from one point to another point on a job. During the rapid traverse movement, either the X or Y axis can be moved individually or both axes can be moved at the same time. Although the rate of rapid travel varies from machine to machine, it ranges between (5 and 20 m/min).

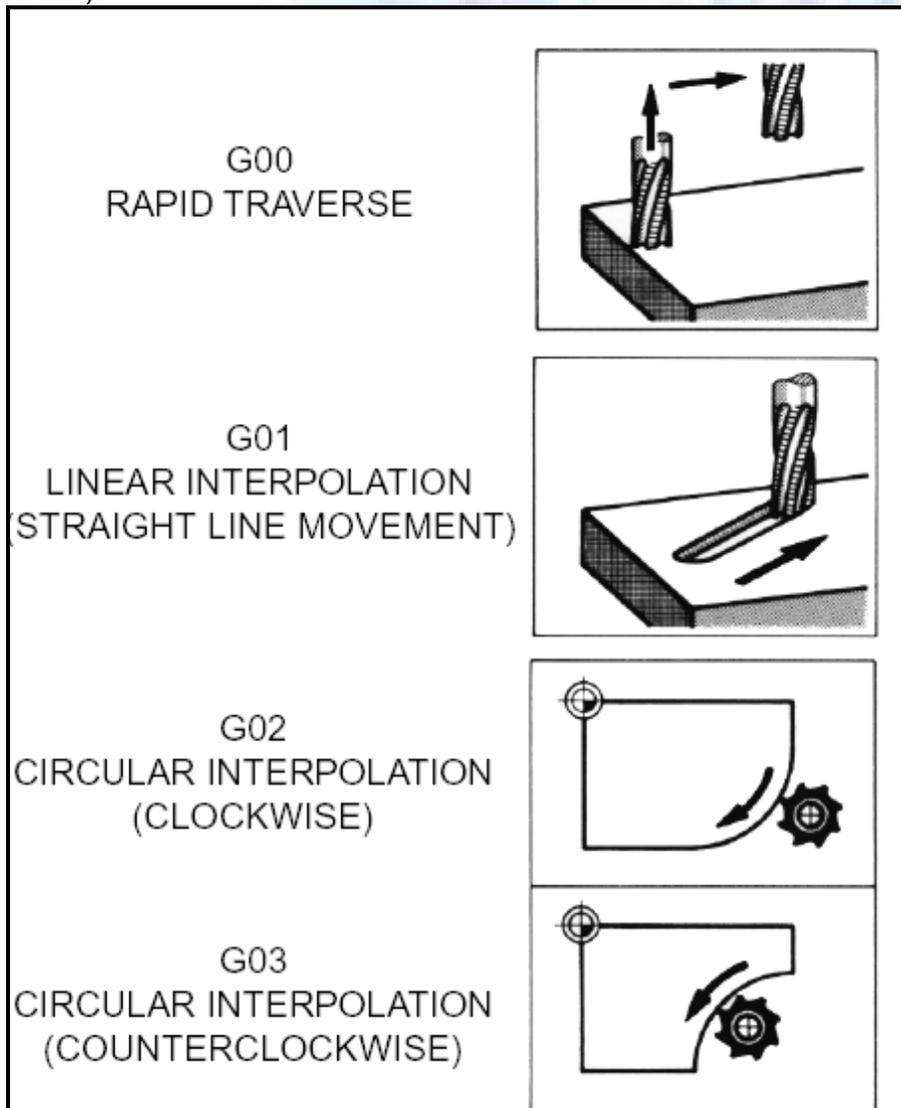


Fig. 13: The functions of a few common G-codes.

The G01, G02, and G03 codes move the axes at a controlled feedrate.

- G01 is used for straight-line movement (linear interpolation).
- G02 (clockwise) and G03 (counterclockwise) are used for arcs and circles (circular interpolation).

<u>Group</u>	<u>Code</u>	<u>Function</u>
01	G00	Rapid positioning
01	G01	Linear interpolation
01	G02	Circular interpolation clockwise (CW)
01	G03	Circular interpolation counterclockwise (CCW)
06	G20*	Inch input (in.)
06	G21*	Metric input (mm)
	G24	Radius programming (**)
00	G28	Return to reference point
00	G29	Return from reference point
	G32	Thread cutting (**)
07	G40	Cutter compensation cancel
07	G41	Cutter compensation left
07	G42	Cutter compensation right
08	G43	Tool length compensation positive (+) direction
08	G44	Tool length compensation minus (-) direction
08	G49	Tool length compensation cancel
	G84	Canned turning cycle (**)
03	G90	Absolute programming
03	G91	Incremental programming

(*) - on some machines and controls, these may be G70 (inch) and G71 (metric)

(**) - refers only to CNC lathes and turning centers.

Fig. 14: Some of the most common G-codes used in CNC programming.

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

- M03 turns the spindle on clockwise
- M04 turns the spindle on counterclockwise
- M05 turns the spindle off

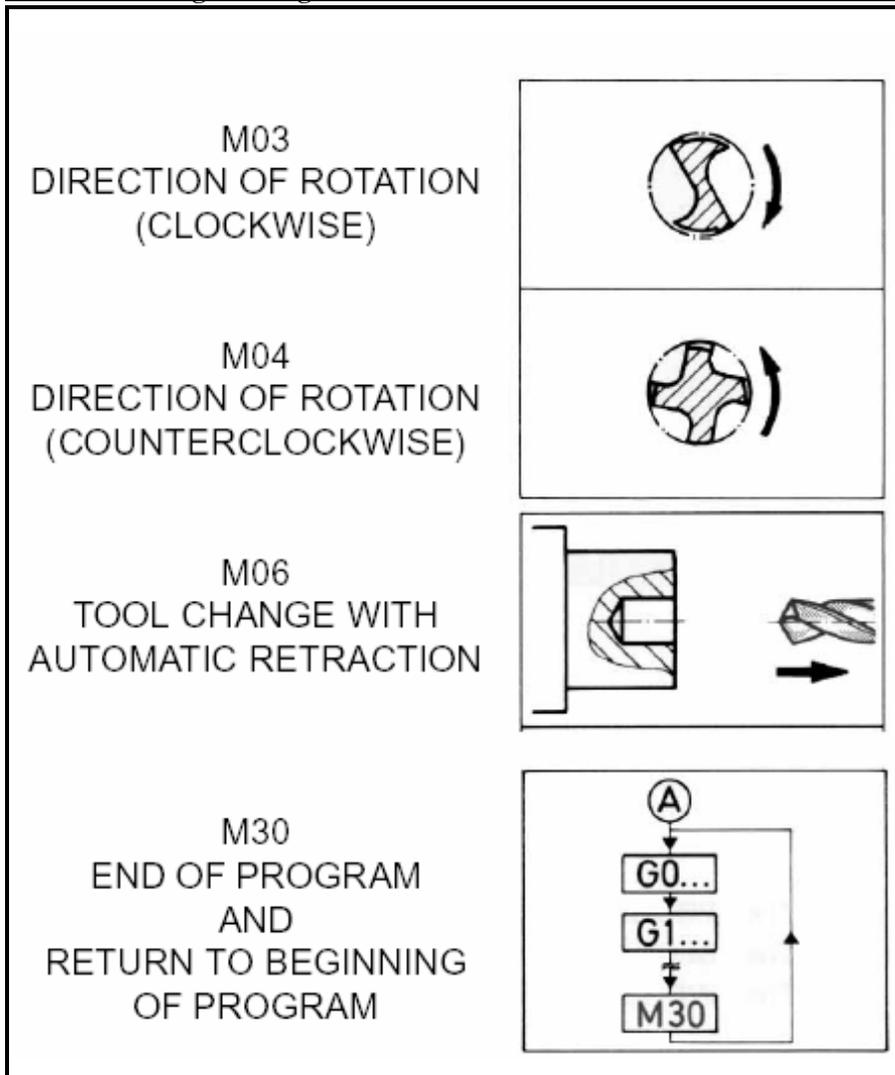


Fig. 15: The functions of a few common M-codes.

<u>Code</u>	<u>Function</u>
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck - clamping (**)
M11	Chuck - unclamping (**)
M12	Tailstock spindle out (**)
M13	Tailstock spindle in (**)
M17	Toolpost rotation normal (**)
M18	Toolpost rotation reverse (**)
M30	End of tape and rewind
M98	Transfer to subprogram
M99	End of subprogram

(**) - refers only to CNC lathes and turning centers.

Fig. 16: Some of the most common M-codes used in CNC programming.

Block of Information

CNC information is generally programmed in blocks of five words. Each word conforms to the EIA standards and they are written on a horizontal line. If five complete words are not included in each block, the machine control unit (MCU) will not recognize the information, therefore the control unit will not be activated. Using the example shown in Fig. 17, the five words are as follows:

- N001** represents the sequence number of the operation.
- G01** represents linear interpolation
- X12345** will move the table 1.2345 in. in a positive direction along the X axis.
- Y06789** will move the table 0.6789 in. along the Y axis.
- M03** Spindle on CW.

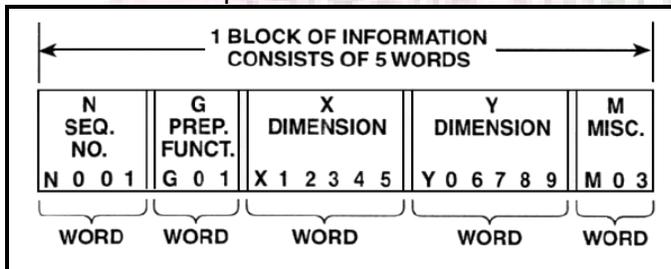


Fig. 17: A complete block of information consists of five words.

■ Programming for Positioning:

Before starting to program a job, it is important to become familiar with the part to be produced. From the engineering drawings, the programmer should be capable of planning the machining sequences required to produce the part. Visual concepts must be put into a written manuscript as the first step in developing a part program, Fig. 18. It is the part program that will be sent to the machine control unit by the computer, tape, diskette, or other input media. The programmer must first establish a reference point for aligning the workpiece and the machine tool for programming purposes. The manuscript must include this along with the types of cutting tools and work-holding devices required, and where they are to be located.

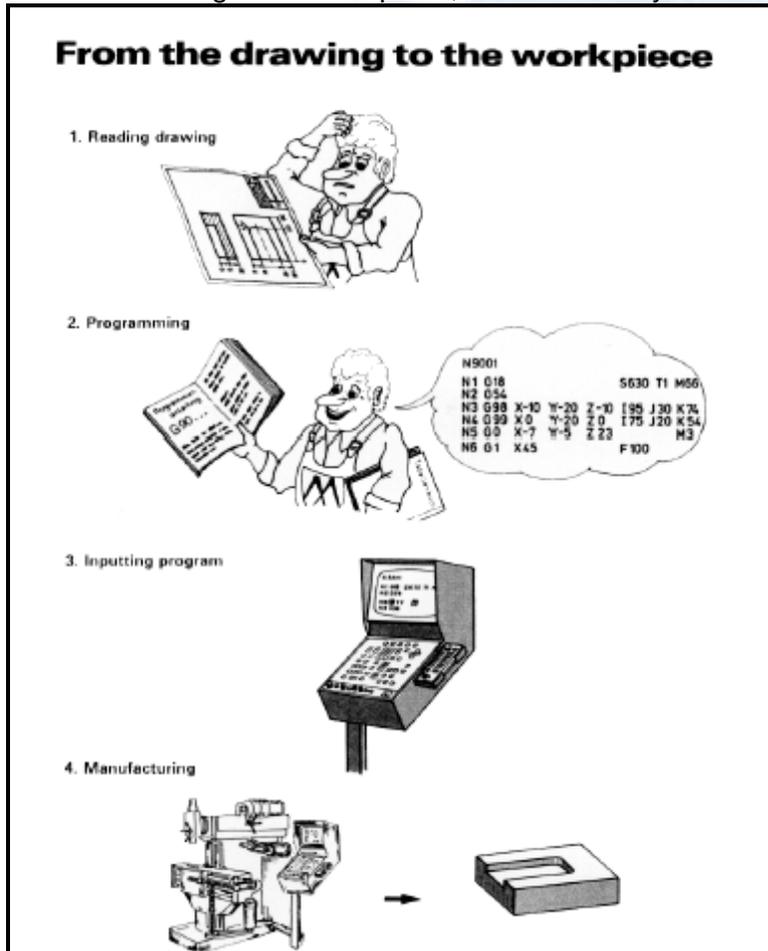


Fig. 18 The first step in producing a CNC program is to take the information from the print and produce a program manuscript.

Dimensioning Guidelines

The system of rectangular coordinates is very important to the successful operation of CNC machines. Certain guidelines should be observed when dimensioning parts for CNC machining. The following guidelines will insure that the dimensioning language means exactly the same thing to the design engineer, the technician, the programmer, and the machine operator.

1. Define part surfaces from three perpendicular reference planes.
2. Establish reference planes along part surfaces which are parallel to the machine axes.
3. Dimension from a specific point on the part surface.
4. Dimension the part clearly so that its shape can be understood without making mathematical calculations or guesses.
5. Define the part so that a computer numerical control cutter path can be easily programmed.

Machine Zero Point

The machine zero point can be set by three methods—by the operator, manually by a programmed absolute zero shift, or by work coordinates, to suit the holding fixture or the part to be machined.

MANUAL SETTING - The operator can use the MCU controls to locate the spindle over the desired part zero and then set the X and Y coordinate registers on the console to zero.

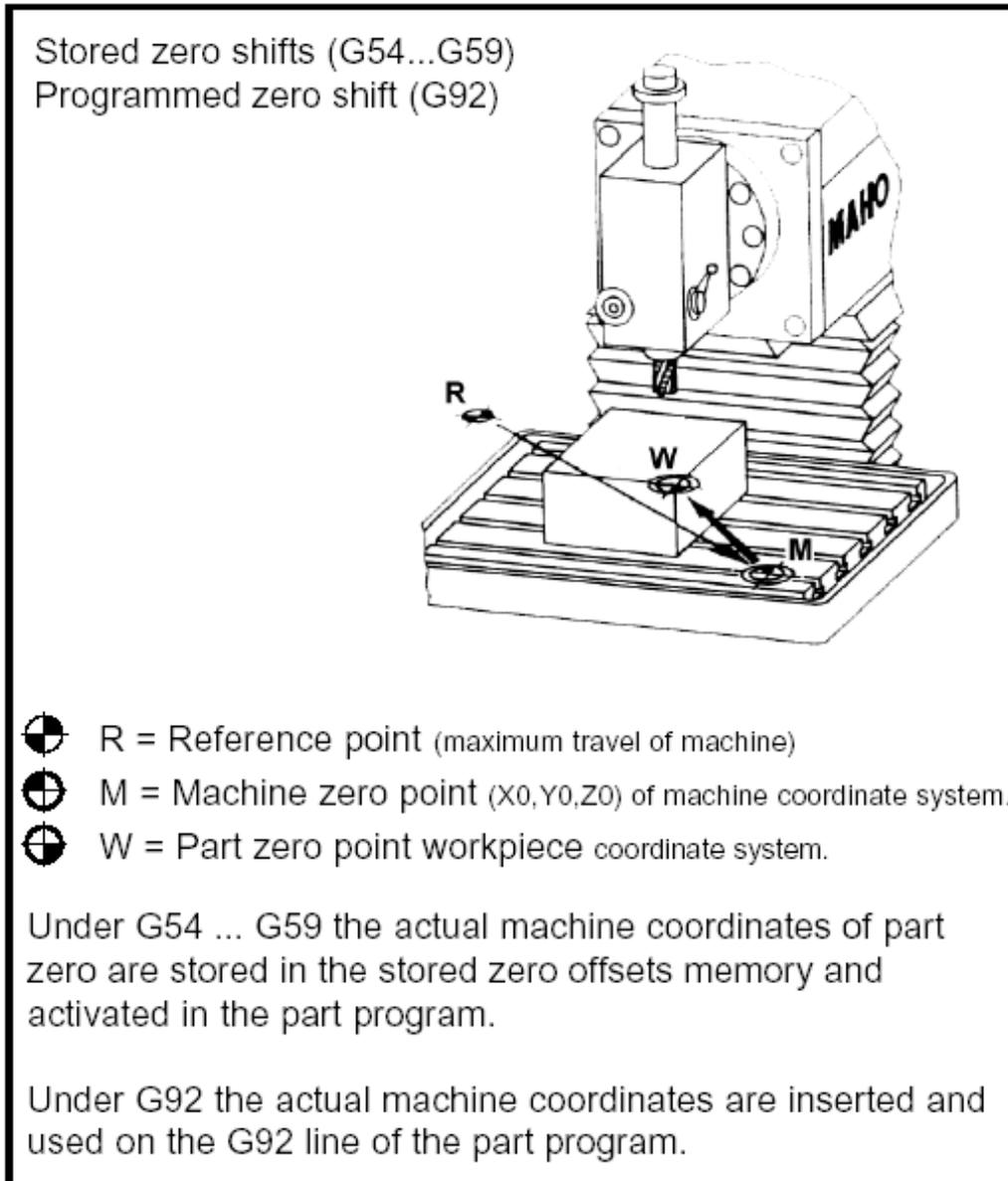


Fig. 19 The relationship between the part zero and the machine system of coordinates.

ABSOLUTE ZERO SHIFT - The absolute zero shift can change the position of the coordinate system by a command in the CNC program. The programmer first sends the machine spindle to home zero position by a G28 command in the program. Then another command (G92 for absolute zero shift) tells the MCU how far from the home zero location, the coordinate system origin is to be positioned, Fig. 19. The sample commands may be as follows:

N1 G28 X0 Y0 Z0 (sends spindle to home zero position)
 N2 G92 X4.000 Y5.000 Z6.000 (the position the machine will reference as part zero)

■ Work Settings and Offsets:

All CNC machine tools require some form of work setting, tool setting, and offsets (compensation) to place the cutter and work in the proper relationship. Compensation allows the programmer to make adjustments for unexpected tooling and setup conditions.

Work Coordinates

In absolute positioning, work coordinates are generally set on one edge or corner of a part and all programming is generally taken from this position. In Fig. 20, the part zero is used for all positioning for hole locations 1, 2, and 3.

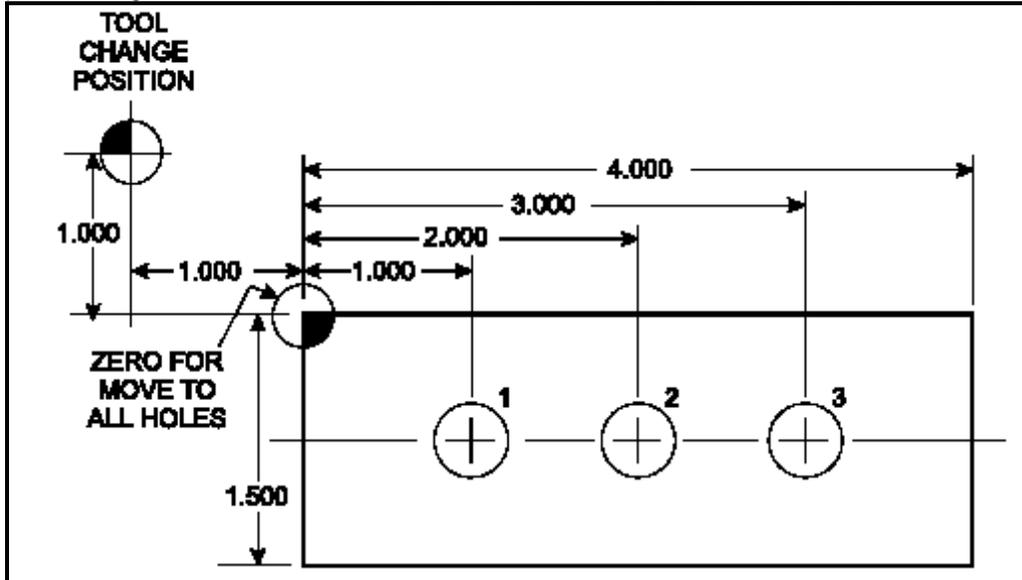


Fig. 20 In absolute programming, all dimensions must be taken from the XY zero at the top left-hand corner of the part.

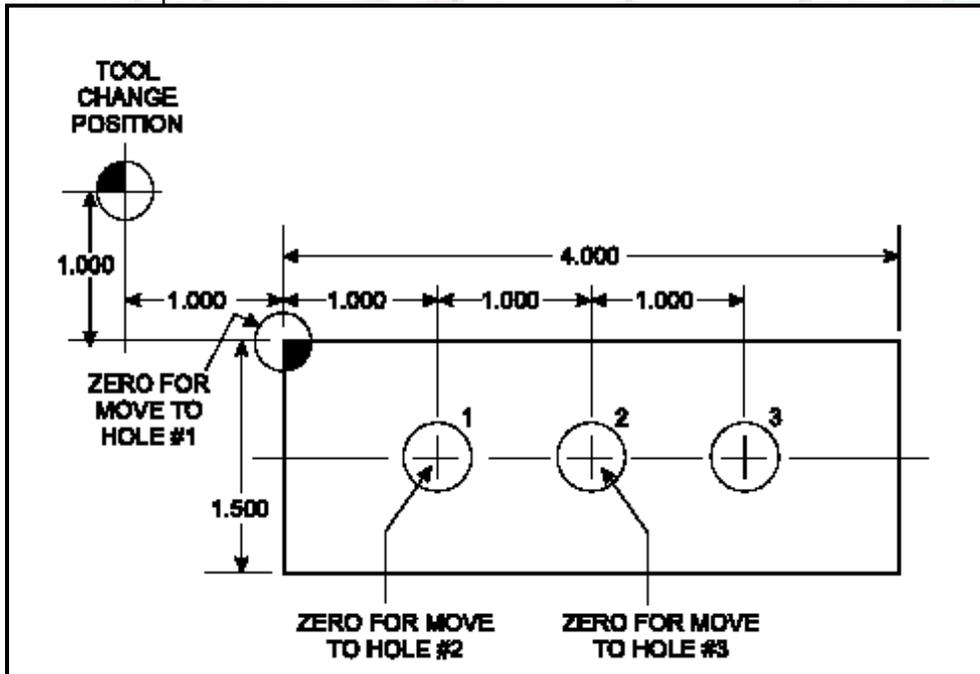


Fig. 21 In incremental programming, all dimensions are taken from the previous point.

In incremental positioning, the work coordinates change because each location is the zero point for the move to the next location, Fig. 21.

On some parts, it may be desirable to change from absolute to incremental, or vice versa, at certain points in the job. Inserting the G90 (absolute) or the G91 (incremental) command into the program at the point where the change is to be made can do this.

R Plane or Gage Height

The word-address letter R refers to a partial retraction point in the Z axis to which the end of the cutter retracts above the work surface to allow safe table movement in the X Y axes. It is often called the rapid-traverse distance, gage height, retract or work plane. The R distance is a specific height or distance above the work surface and is generally .100 in. above the highest surface of the workpiece, Fig. 22, which is also known as gage height. Some manufacturers build a gage height distance of .100 in. into the MCU (machine control unit) and whenever the feed motion in the Z axis is called for, .100 in. will automatically be added to the depth programmed. When setting up cutting tools, the operator generally places a .100 in. thick gage on top of the highest surface of the workpiece. Each tool is lowered until it just touches the gage surface and then its length is recorded on the tool list. Once the gage height has been set, it is not generally necessary to add the .100 in. to any future depth dimensions since most MCUs do this automatically.

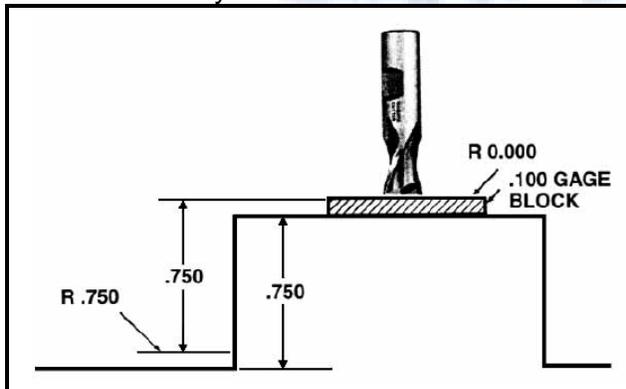


Fig. 22 Using a .100 in. gage block to set the gage height or R0 on the work surface.

Cutter Diameter Compensation

Cutter diameter compensation (CDC) changes a milling cutter's programmed centerline path to compensate for small differences in cutter diameter. On most MCUs, it is effective for most cuts made using either linear or circular interpolation in the X-Y axis, but does not affect the programmed Z-axis moves. Usually compensation is in increments of .0001 in. up to +1.0000 in., and usually most controls have as many CDCs available as there are tool pockets in the tool storage matrix.

The advantage of the CDC feature is that it:

1. Allows the use of cutters that have been sharpened to a smaller diameter.
2. Permits the use of a larger or smaller tool already in the machine's storage matrix.
3. Allows backing the tool away when roughing cuts are required due to excessive material present.
4. Permits compensation for unexpected tool or part deflection, if the deflection is constant throughout the programmed path.

The basic reference point of the machine tool is never at the cutting edge of a milling cutter, but at some point on its periphery. If a 1.000 in. diameter end mill is used to machine the edges of a workpiece, the programmer would have to keep a .500 in. offset from the work surface in order to cut the edges accurately, Fig. 23. The .500 offset represents the distance from the centerline of the cutter or machine spindle to the edge of the part. Whenever a part is being machined, the programmer must calculate an offset path, which is usually half the cutter diameter.

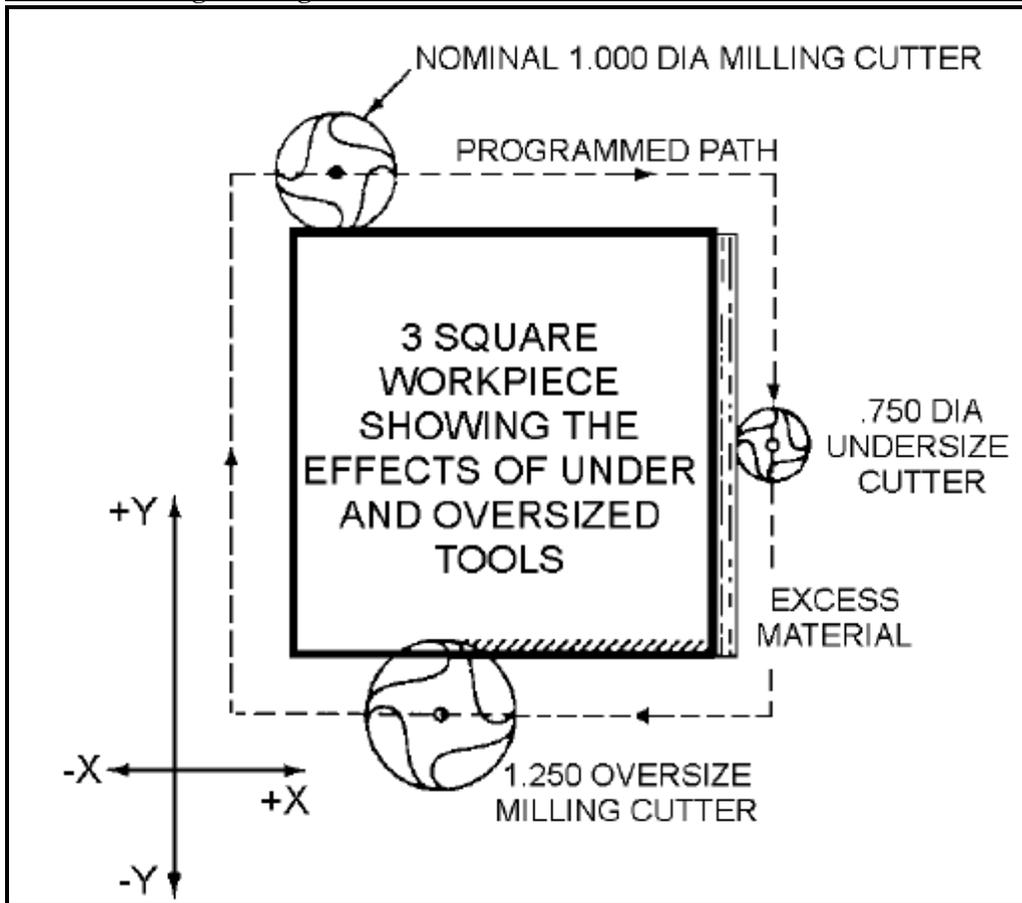


Fig. 23 Cutter-diameter compensation must be used when machining with various size cutters.

Modern MCUs, which have part surface programming, automatically calculate centerline offsets once the diameter of the cutter for each operation is programmed. Many MCUs have operator-entry capabilities which can compensate for differences in cutter diameters; therefore an oversize cutter, or one that has been sharpened, can be used as long as the compensation value for oversize or undersize cutters is entered.

■ CNC Programming Hints - MILLING

- ⊕ Machine reference point (maximum travel of machine)
- ⊕ Machine X Y zero point (could be tool change point)
- ⊕ Part X Y zero point (programming start point)
- + Indicates the tool change position. A G92 code will reset the axis register position coordinates to this position.

For a program to run on a machine, it must contain the following codes:

M03 To start the spindle/cutter revolving.
Sxxx The spindle speed code to set the r/min.
Fxx The feed rate code to move the cutting tool or workpiece to the desired position.

ANGLES:

The X Y coordinates of the start point and end point of the angular surface plus a feed rate (F) are required.

Z CODES:

- A Z dimension raises the cutter above the work surface.
- A Z- dimension feeds the cutter into the work surface.
- Z.100 is the recommended retract distance above the work surface before a rapid move (G00) is made to another location.

RADII / CONTOUR Requirements:

- The *start* point of the arc (XY coordinates)
- The *direction* of cutter travel (G02 or G03)
- The *end* point of the arc (XY coordinates)
- The *center* point of the arc (IJ coordinates) or the arc radius)

■ CNC Programming Hints – TURNING



Indicates the X Z 0 (zero) location which is the starting point for programming.



Indicates the tool-change position.

A G92 code will reset the axis register position coordinates to this position.

For a program to run on a machine, it must contain the following codes:

M03 To start the spindle/cutter revolving.
Sxxx The spindle speed code to set the r/min.
Fxx The feedrate code to move the cutting tool or workpiece to the desired position.

TAPERS/BEVELS/ANGLES

- The X Z coordinates of the small diameter, the large diameter, and a feedrate must be programmed.
- Z moves the cutting tool longitudinally away from the end of the workpiece.
- Z- moves the cutting tool along the length of the workpiece towards the chuck (headstock).
- X moves the cutting tool away from the work diameter.
- X- moves the cutting tool into the work diameter.

■ CNC:

The abbreviation CNC stands for *Computer(ized) Numerical(ly) Control(led)*, and refers specifically to the computer control of machine tools for the purpose of (repeatedly) manufacturing complex parts in metal as well as other materials, using a program written in a notation conforming to the EIA-274-D standard and commonly called *G-code*.

CNC was developed in the late 1940s and early 1950s by the MIT Servomechanisms Laboratory. CNC machines were relatively briefly preceded by the less advanced NC, or Numerical(ly) Control(led), machines. The introduction of CNC machines radically changed the manufacturing industry. Curves are as easy to cut as straight lines, complex 3-D structures are relatively easy to produce, and the number of machining steps that required human action have been dramatically reduced. With the increased automation of manufacturing processes with CNC machining, considerable improvements in consistency and quality have been achieved. CNC automation reduced the frequency of errors and provided CNC operators with time to perform additional tasks.

CNC automation also allows for more flexibility in the way parts are held in the manufacturing process and the time required to change the machine to produce different components. In a production environment, a series of CNC machines may be combined into one station, commonly called a "cell", to progressively machine a part requiring several operations. CNC machines today are controlled directly from files created by CAM software packages, so that a part or assembly can go directly from design to manufacturing without the need of producing a drafted paper drawing of the manufactured component. In a sense, the CNC machines represent a special segment of industrial robot systems, as they are programmable to perform many kinds of machining operations (within their designed physical limits, like other robotic systems).

■ G~code:

G-code is a common name for the programming language that is used for NC and CNC machine tools. It is defined in EIA RS-274-D. G-code is also the name of any word in a CNC program that begins with the letter *G*, and generally is a code telling the machine tool what type of action to perform, such as:

- rapid move
- controlled feed move in straight line or arc
- series of controlled feed moves that would result in a hole being drilled.
- change a pallet
- Set tool information such as offset.

There are other codes; the type codes can be thought of like registers in a computer

X position
Y position
Z position
M code (another "action" register)
F feed rate
S spindle speed
N line number
R Radius
T Tool selection

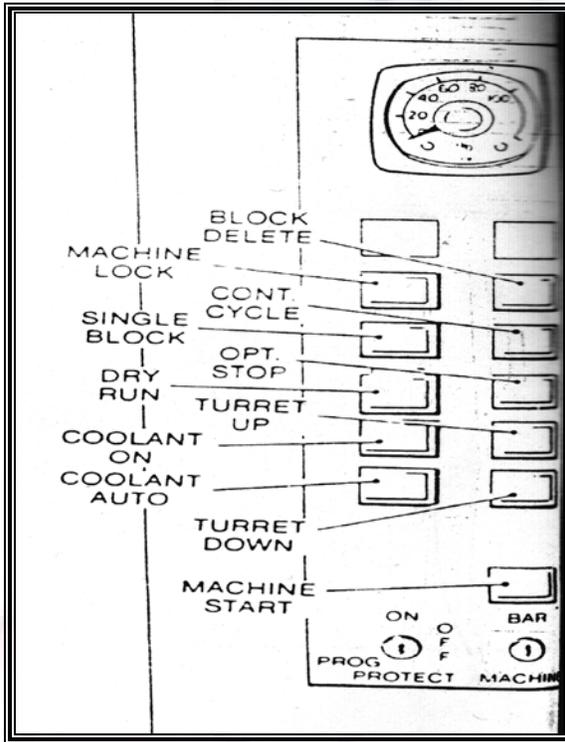
I Arc data X axis
J Arc data Y axis.

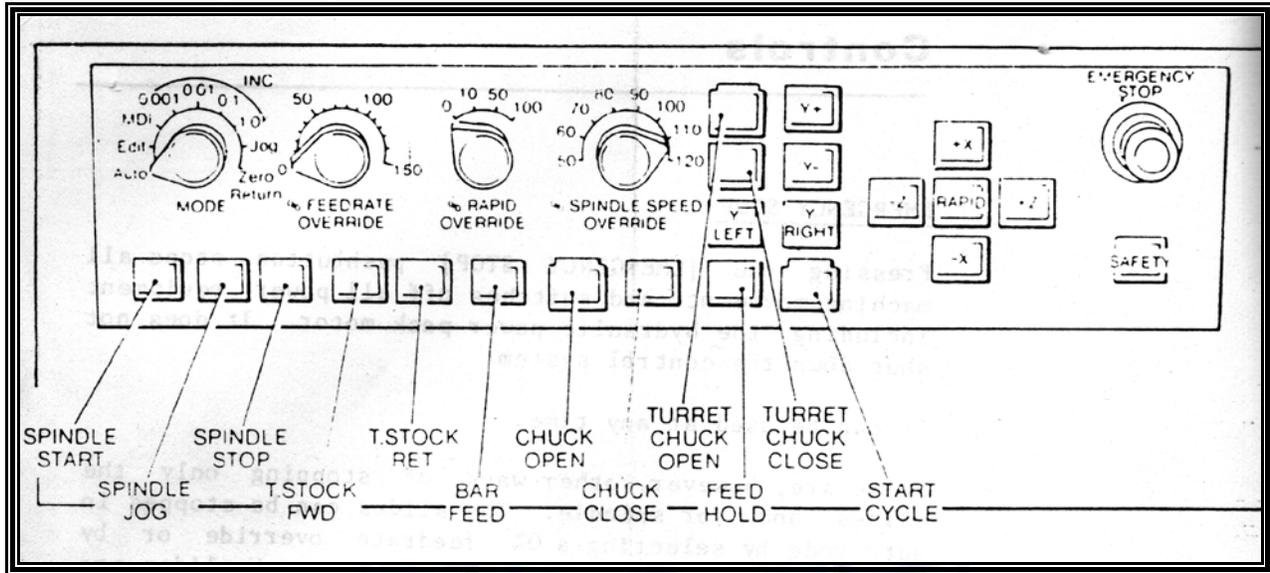
G-code files are output by CAM software such as Smartcam, Gibbscam, Featurecam, Edgcam, Mastercam, etc. G-code is also output by specialized CAD systems used to design printed circuit boards. Such software must be customized for each type of machine tool that it will be used to program.



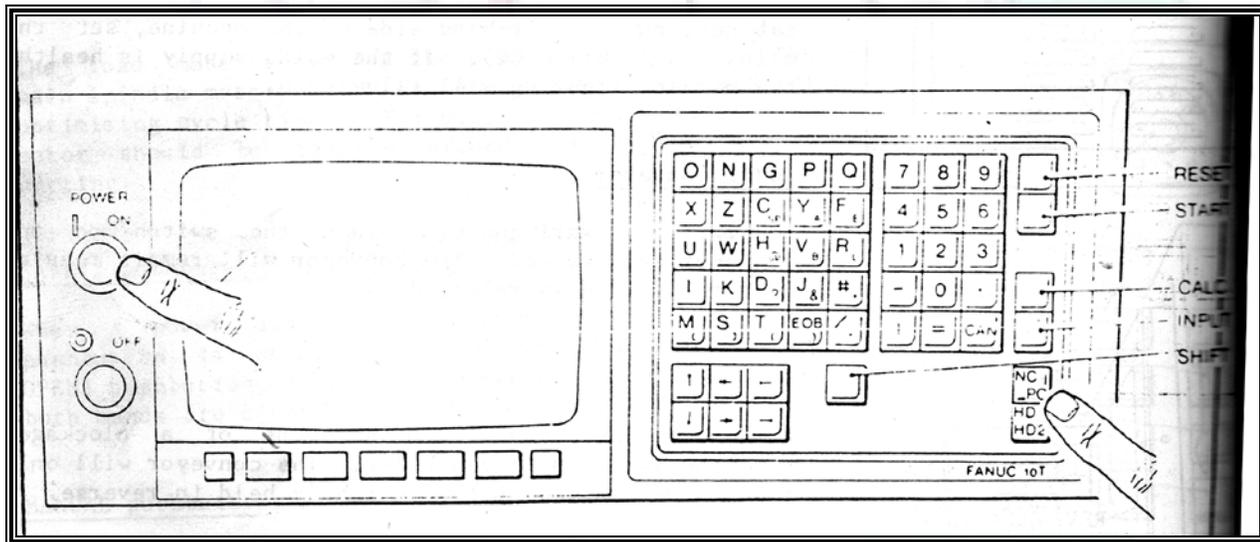
عمل الأزرار الرئيسية في ماكينة الخراطة المبرمجة (نصر 2):

- 1 - **SINGLE BLOCK** : تقوم بتنفيذ البرنامج خطوة خطوة (للتأكد من صحة البرنامج).
- 2 - **BLOCK DELETE** : يستخدم لألغاء بلوك معين من الأوامر ، ويعمل بجانب /* .
- 3 - **MACHINE LOCK** : يستخدم لوقف حركة المحاور الرئيسية مع استمرار البرنامج في العمل، (وهذا مهم للتأكد من صحة البرنامج قبل عملية التنفيذ النهائي) .
- 4 - **CONT. CYCLE** : تستخدم لأستئناف عملية التشغيل عند إيقافها لاي سبب كان او عند إيقافها بسبب (FEED HOLD) او في حالة استخدام (BAR FEEDER) .
- 5 - **DRY RUN** : حركة سريعة للبرنامج ، وتستخدم لفحص البرنامج بعد ازالة الشغلة .
- 6 - **TURRET UP & DOWN** : حركة حامل العدة (TURRET) الذي يحمل مجموعة من العدد ، حركة دورانية عكس عقرب الساعة او مع عقرب الساعة ، وهذا الزر يعمل مع الزر (SAFTY) .
- 7 - **COOLANT ON or OFF** : تشغيل سائل التبريد او إيقافه.
- 8 - **COOLANT AUTO** : تشغيل سائل التبريد او إيقافه من خلال الدوال المساعدة في البرنامج .
- 9 - **MACHINE START** : يستخدم لتشغيل هيدروليك الماكينة .





- 10- **CHUCK CLOSE OR OPEN**: تستخدم لفتح وغلق العينة.
- 11- **T, STOCK FWD, RET**: حركة الغراب المتحرك ذهاباً وإياباً.
- 12- **SPINDLE STOP**: تعمل على ايقاف حركة دوران العينة
- 13- **SPINDLE JOG**: تعمل على تدوير العينة مع زر SAFTY .
- 14- **OPT SAFTY**: تعمل عمل التوقف المشروط حيث تعمل مع الدالة (M1) دالة التوقف المشروط , حيث لا يتوقف البرنامج في حالة وجود دالة (M1) فقط الا اذا كان هذا الزر مضغو ط . لغرض تحرير زر (START CYCLE) يتم الضغط على زر (SAFTY + SPINDLE) في نفس الوقت بعد الضغط على زر (START CYCLE) مرتين.



- 15- **NC / PC**: يستخدم هذا الزر عند ربط CNC مع جهاز آخر حيث يستخدم لنقل المعلومات بينهما .
- 16- **CALCU**: وهو الزر الخاص بالحاسبة مع زر (INPUT) .
- 17- **HD1/ HD2**: يستخدم لاختيار العدة مع المخرطة ذات المحاور الأربعة .
- 18- **START**: تستخدم لبدء عملية التشغيل.

- 19 - FEED HOLD: تعمل على إيقاف عمل أي G موجودة مع بقاء العينة متحركة .
- 20 - M D I : يتم استخدامه للإدخال أو تشغيل البرنامج BLOCK BY BLOCK وتعمل عند إدخال البرنامج بواسطة الشريط المتقّب أو أي وسيلة إدخال أخرى أو في عمليات التصفير .
- 21 - REST : تستخدم لإعادة تشغيل الماكينة أو مسح الإنذار الذي يظهر على الشاشة .
- 22 - EMERGENCY: تستخدم عادةً لإيقاف الماكينة في حالة الطوارئ فقط.

FUNCTION MENU KEY	POSITION	PROGRAM	OFFSET	PRG- CMK	CHAPTER	OPERATION MENU KEY
-------------------------	----------	---------	--------	-------------	---------	--------------------------

- 23 - POSITION : لاظهار الموقع الموجود على الشاشة.
- 24 - PROGRAM : لإظهار برنامج معين واختياره.
- 25 - OFFSET : لاختيار سماحات العدة أو سماحات نقطة المرجع.
- 26 - PRG- CHK : تستخدم لفحص البرنامج على الشاشة .
- 27 - SETTING : لاختيار الثوابت على الشاشة .
- 28 - SERVICE : تستخدم لاستخراج المعاملات والثوابت .
- 29 - MESSAGE : تستخدم لأظهار الأخطاء الناتجة عن البرنامج أو اخطاء التشغيل .
- 30 - OVERALL : لاختيار المواقع .
- 31 - RELATIVE \ MODLINE : تعوض عن POSITION .
- 32 - DIR .MEM : يستخدم لأظهار ملف البرامج المخزونة في الذاكرة وعندما نضغط هذا الزر فإن رقم واسم البرنامج سوف يظهر على الشاشة .

أدناه الشفرات الرئيسية التي تعمل على الماكينة المبرمجة نصر 2 :

G-CODE	DESCRIPTION
G00	Rapid traverse (linear)
G01	Linear Interpolation at feed rate
G02	Circular interpolation (CW) R+
G03	Circular interpolation (CCW) R+
G10	Zero point tools
G20	Imperial data input
G21	Metric data input
G27	Reference return check
G28	Reference return
G29	Return from reference

G33	Thread cutting
G40	Cancel cutter radius compensation
G41	Cutter compensation left
G42	Cutter compensation right
G54	Work co-ordinate system no.1
G70	Finishing cycle
G71	Stock removal turning
G72	Stock removal facing
G74	Face grooving
G75	Diameter grooving
G76	Threading cycle
G90	Absolute data input
G91	Incremental data input
G94	Feed /minute mode
G95	Feed /rev mode
G96	Constant surface speed mode
G97	Cancel Constant surface speed

أدناه الشفرات الثانوية التي تعمل على الماكينة المبرمجة (نصر 2)

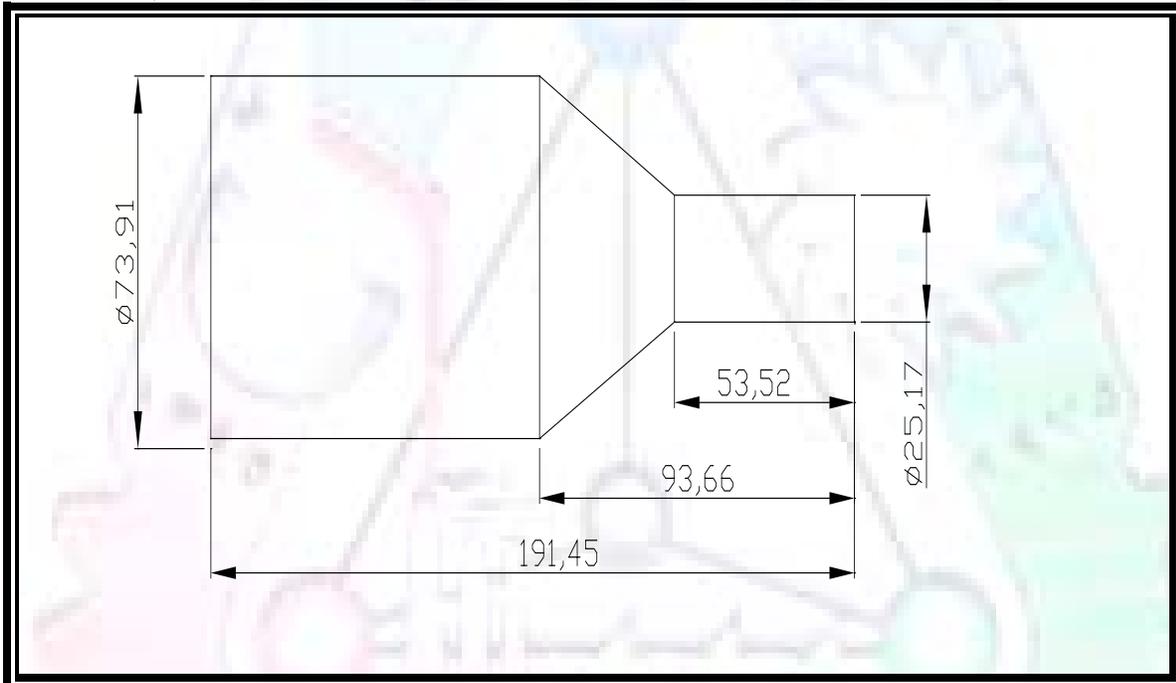
M-code	FUNCTION
M00	Program stop
M03	Spindle forward (Tool forward – M94 mode)
M04	Spindle reverse (Tool forward – M94 mode)
M05	Spindle stop (Tool stop – M94 mode)
M08	Coolant on
M09	Coolant off
M30	Program end (with rewind)
M48	Turret shortest path
M52	Turret direction up (CW)
M53	Turret direction down (CCW)
M73	Chuck open
M74	Chuck close

مثال على استخدام الدالة G71 :

```

T0100 M48;
G0 X26. Z2. M08 T0101;
G90 G95;
G97 S500 M03;
G71 P1 Q2 U0.3 W0.0 D500 F0.1;
N1 G0 G42 X0.0;
      G01      Z0.0;
      G01 X25.17;
      G01      Z-53.52;
      G01 X73.91 Z-93.66;
      G01      Z-191.45;
N2;
G28 M30;

```



الجانب العملي :

الاجهزة والادوات المستخدمة :

1. ماكينة خراطة مبرمجة CNC
2. قضيب من الالمنيوم قطر 80 ملم
3. فيرنج
4. هولدر كاربيدى ذواتجاه ايمن

طريقة العمل :

1. يتم تثبيت قضيب من الالمنيوم على العينة التابعة للماكينة مع مراعاة الطول المراد استعماله في عملية الخراطة .
2. تثبيت عدة القطع في المكان الصحيح في (turret) .

3. يتم تصفير الماكينة للمحاور X,Z بالنسبة الى القطعة المثبتة على العينة
- تصفير للمحور Z تكون ملاسمة العدة الى وجه الشغلة
 - تصفير للمحور X تكون ملاسمة العدة الى سطح الشغلة الموازي لمحور X .
4. يتم كتابة البرنامج أما بطريقة يدوية أو بالحاسوب مع وضع كل الظروف اللازمة للتشغيل (عمق القطع ، التغذية ، السرعة المطلوبة ، نوع المعدن).
5. ادخال البرنامج الى الحاسوب الموجود داخل الماكينة مع تهيئة الماكينة للتشغيل .

المطلوب :

1. عمل برنامج يدوي على الدالة G71 .
2. رسم الشكل النهائي المطلوب مع وضع الأبعاد .

G70 G01 X0.0 Z0.0 F0.1 ;

الغرض من التجربة : عمل انهاء سطحي للمشغولة بواسطة الدالة G70

هذه الدالة تعمل شوط واحد فقط وتكون معتمدة على عمليات القطع في الدالة G71 . من الضروري إعطاء دالة G42 لالغاء مقدمة الاداة (أي لكي تتم عملية الانهاء السطحي بدون حسابات نصف قطر مقدمة الاداة) .

الصيغة المستخدمة للدالة G70 :

G70 P1 Q2 F___;

حيث ان :

P1 = بداية الخطوة الاولى

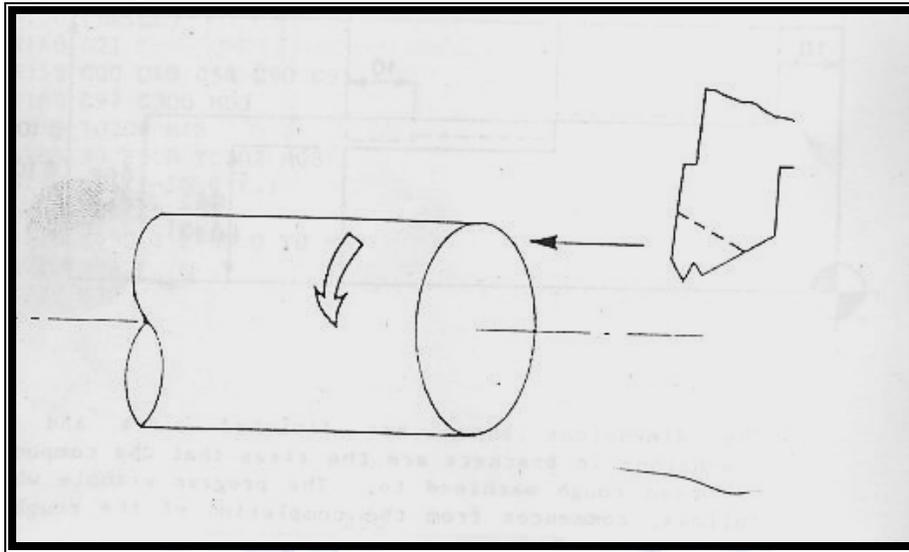
Q2 = نهاية الخطوة

F = التغذية ، والمرتبطة بالدالة G95

ملاحظة : الدالة لاتحتاج لكتابة السماحات التابعة للمحورين X, Z ، وذلك لأن دالة الانهاء السطحي تحقق المسافات الحقيقية المراد انتاجها للمشغولة.

مثال على الدالة G70 و G71 :

T0100 M48;
 G0 X26. Z2. M08 T0101;
 G90 G95;
 G97 S500 M03;
 G71 P1 Q2 U0.3 W0.0 D500 F0.1;
 N1 G0 G42 X0.0;
 G01 Z0.0;
 G01 X25.17;
 G01 Z-53.52;
 G01 X73.91 Z-93.66;
 G01 Z-191.45;
 N2;
 G97 S750 M03;



الخطوة : هي عبارة عن المسافة ما بين قمة سن إلى قمة سن آخر .
عمق السن : هو العمق المطلوب تكوينه للأسنان .

الصيغة المستخدمة للدالة G76 :

G76 X___ Z___ K___ F___ D___ A___ ;

حيث أن :

X = القطر الداخلي للسن

Z = طول السن

K = عمق السن، وتستخرج من المعادلة التالية : $K=0.65 * F$

F = الخطوة

D = عمق القطع بالمايكرون فقط

A = زاوية السن

مثال على استخدام الدالة G76 :

T0300 M48 ;

G00 X30. Z4. T0303 M08 ;

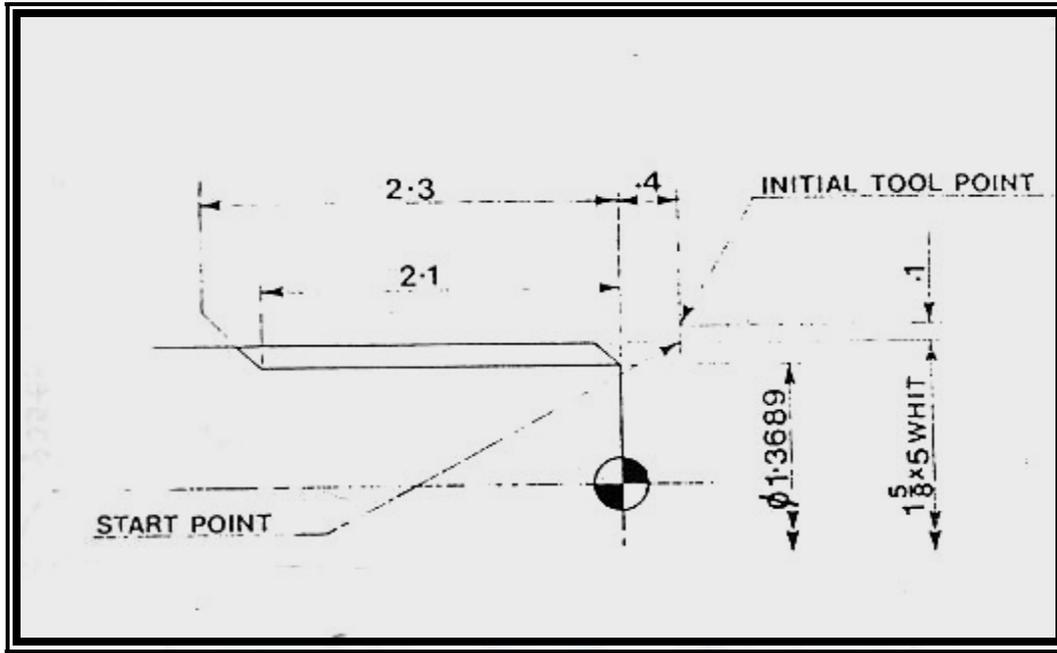
G97 S500 M03 ;

G76 X28.375 Z-20. K0.8125 F1.25 D120 ;

G00 X40. Z4. ;

G28 ;

M30 ;



الجانب العملي :
لاحظ التجربة الأولى.

المطلوب :

1. عمل برنامج يدوي على الدالة G76 .
2. رسم الشكل النهائي المطلوب مع وضع الأبعاد

VI X% = 0% = VI XI% (% =) FC% = 7% XIV% Pts : £K% = 5% = £% 5% Rs% =
: G75 £% = 0% = £™ 3 = VII (Groove)

الغرض من التجربة : تكوين groove بواسطة الدالة G75 :

تستعمل هذه الدالة لتكوين الحفر الخارجي والداخلي (groove) للمشغولة وتعتمد شكل الحفر على طريقة التشغيل وعلى الشكل المراد إنتاجه ، وتعتبر هذه الطريقة من عمليات التشكيل للمعدن .
في هذه التجربة سنقوم بعمل الحفر الخارجي بواسطة الدالة G75 ، مع ملاحظة إن هذه الدالة هي دالة تكرارية وغير تكرارية حسب نوع التشغيل .

الصيغة المستخدمة للدالة G 75 :

G75 X___ Z___ I___ K___ ;

حيث أن :

X = العمق النهائي لقطر الشغلة

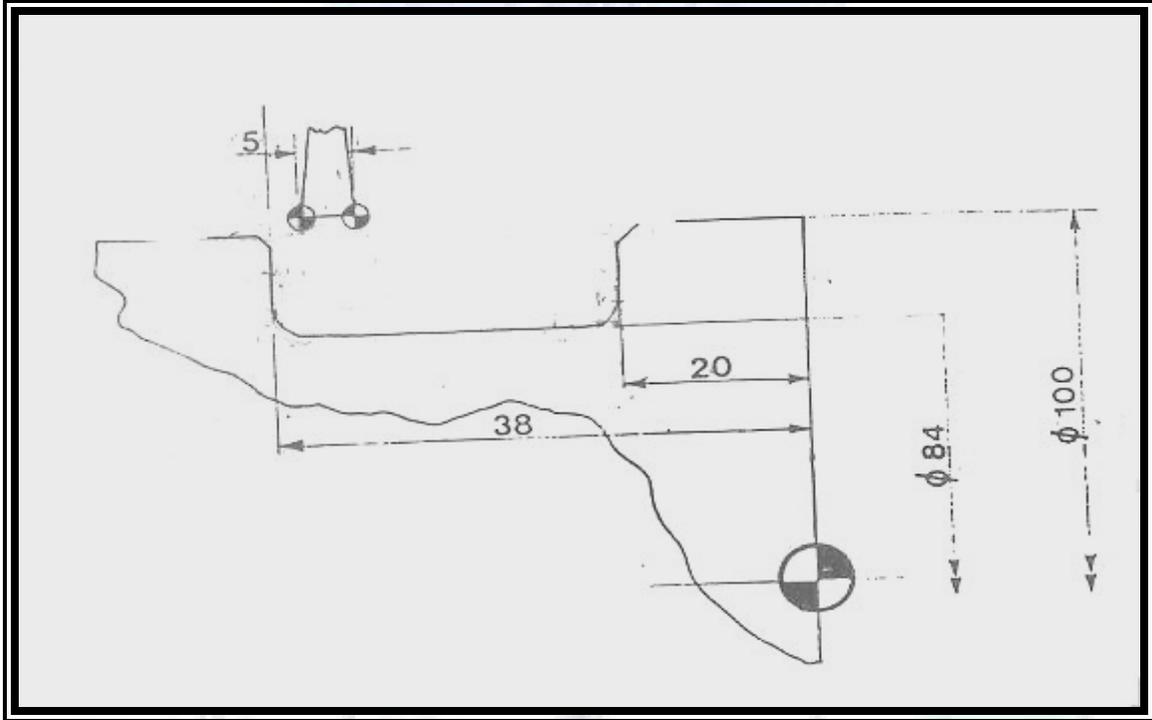
Z = عرض الحفر

I = الدخول في كل مرة

K = الترحيف على محور z بالنسبة للعدة

مثال على استخدام الدالة G75 :

T0500 M48 ;
 G00 X100. Z -20. T0505 M08 ;
 G97 S700 MO3 ;
 G75 X84. Z -38. I 0.5 K4. ;



الجانب العملي :
 لاحظ التجربة الأولى.

المطلوب :

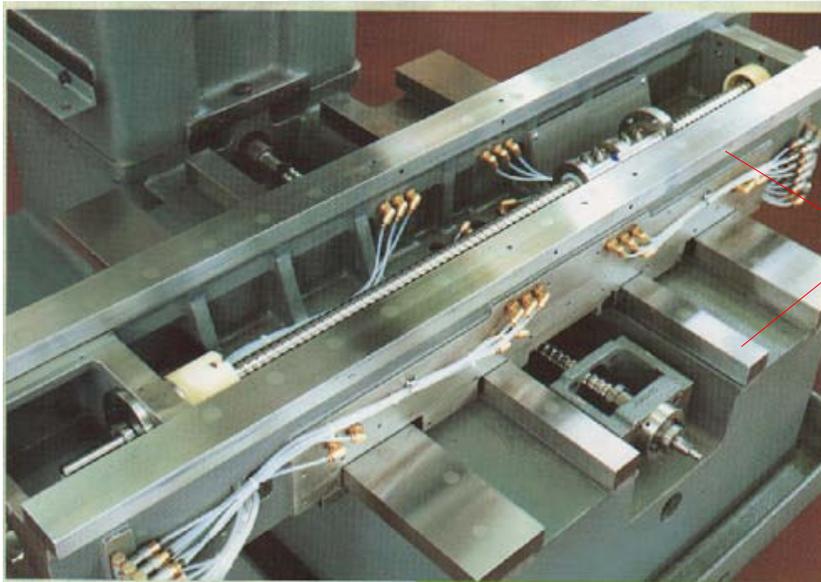
1. عمل برنامج يدوي على الدالة G76 .
2. رسم الشكل النهائي المطلوب مع وضع الأبعاد .



CNC MILLING MACHINE MECHANICAL CONSTRUCTION

■ Guideways

The guideways of CNC machines are made of steel. To reduce the frictional forces, they are coated with a strip of PTFE (Poly Tetra Fluoro Ethylene).



Guideway

■ Ballscrews

On conventional lathes a leadscrew is used to convert the rotary motion of the hand wheel into linear motion of the slides. Leadscrews have a high coefficient of friction and backlash. On CNC machines a ballscrew is used instead of a leadscrew. Friction and backlash are much less than on leadscrews.



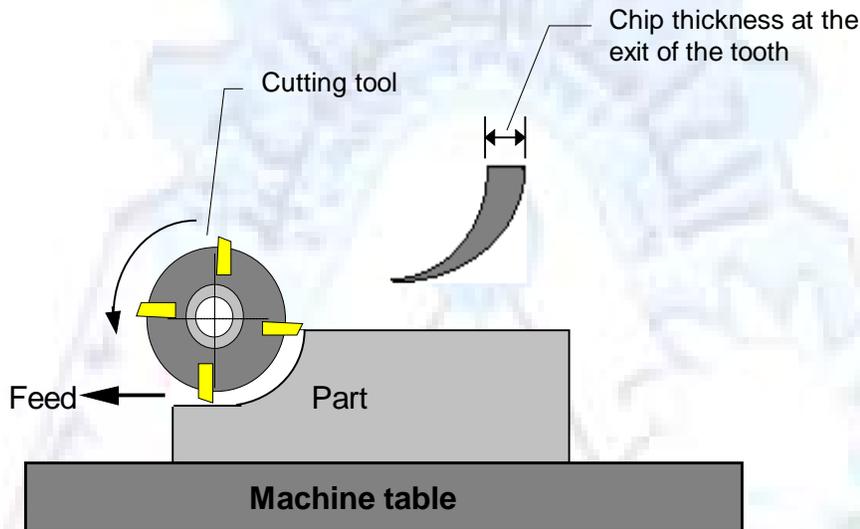
●Machining operations:

Direction of cutting:

Based on the direction of the cutter rotation and work piece feed, milling operations are classified into two types.

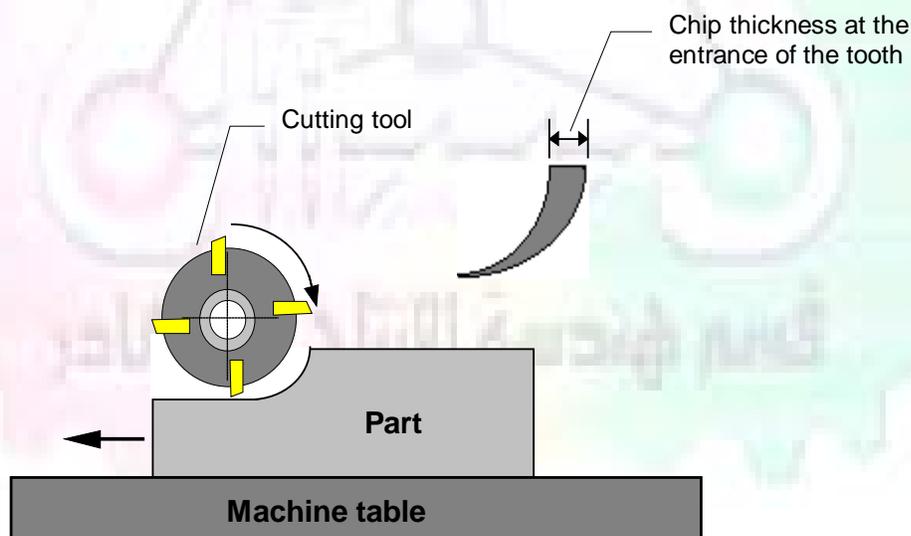
1-Up milling (Convention milling)

In up milling, the metal is removed by rotating the cutter against the direction of travel of the part.



2-Down milling (Climb milling)

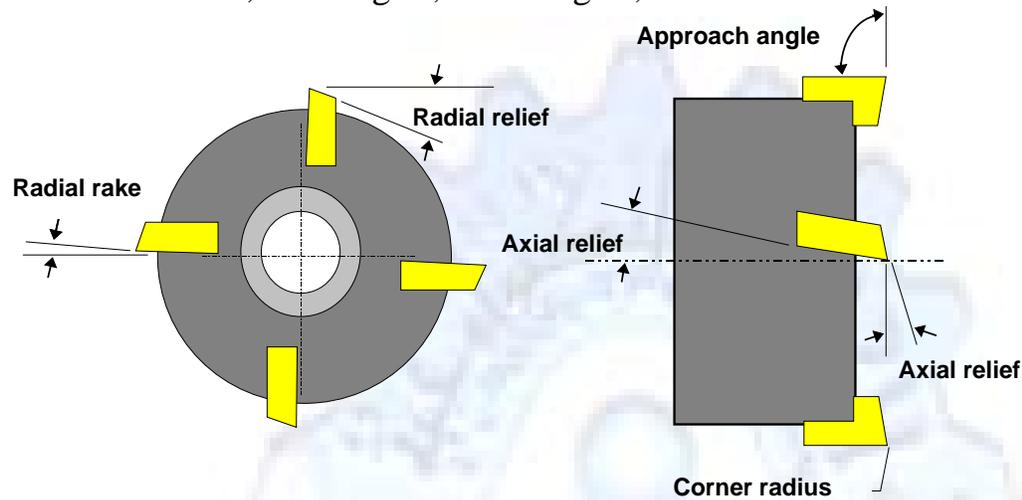
In climb milling, the metal is removed by rotating the cutter with the direction of travel of the part.



•Tooling

Tool geometry

The geometry of the tool tip affects the way that the metal cutting is performed. The nose radius, rake angles, relief angles, etc. must be selected with care.



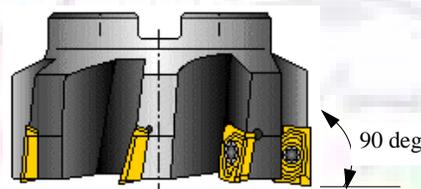
Face mill

Face milling cutters are used to mill flat surfaces. Commonly used face milling cutters:

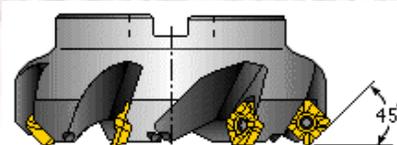
Cutter with 75 degree approach



Cutter with 90 degree approach (Square shoulder face mill)



Cutter with 45 degree approach

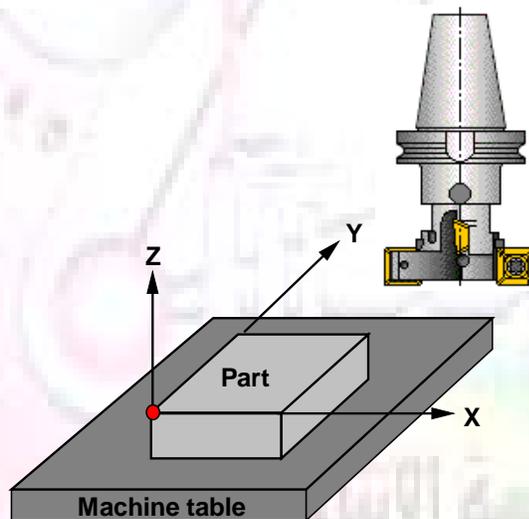
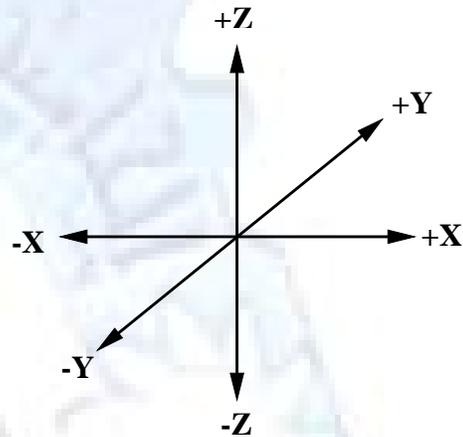
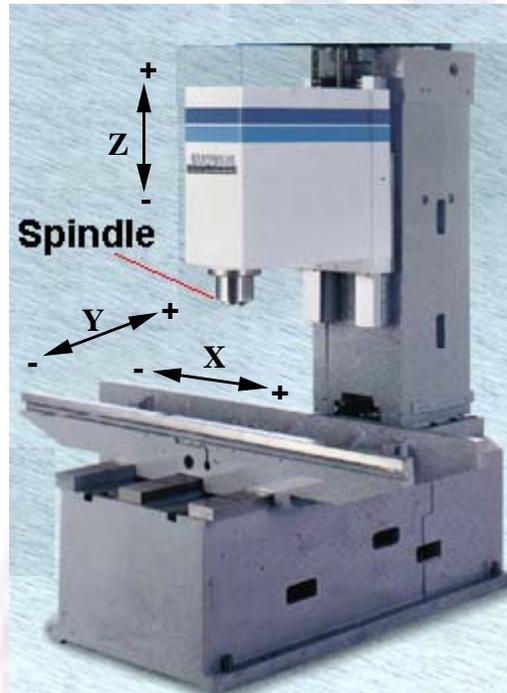


•CNC Programming

Axes convention

The Z axis is along the spindle axis. The X and Y axes are perpendicular to Z.

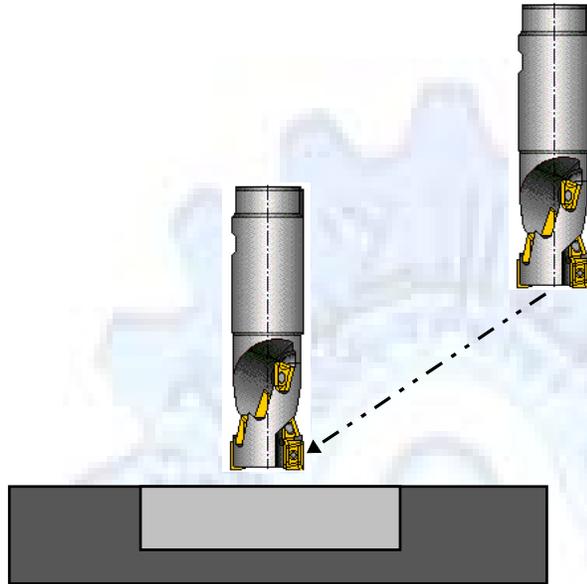
VMC



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.

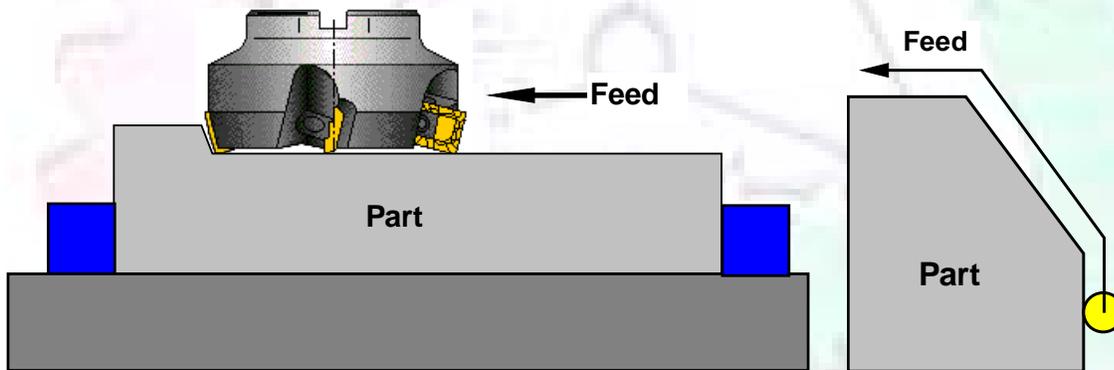
Format : N_ G00 X__ Y__ Z__



G01 Linear interpolation (feed traverse)

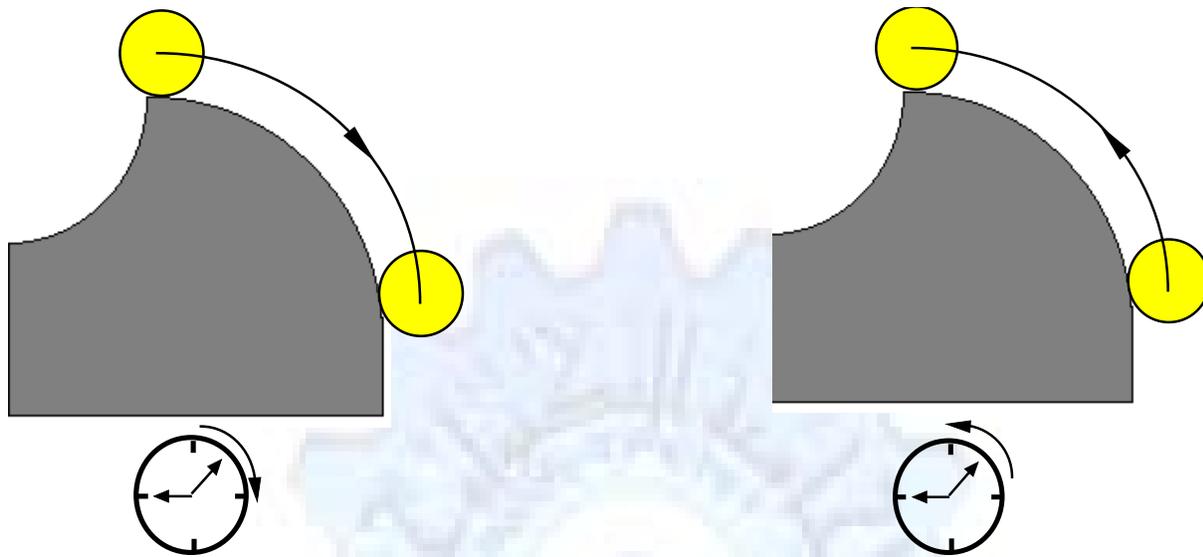
The tool moves along a straight line in one or two axis simultaneously at a programmed linear speed, the *feed rate*.

Format : N__ G01 X__ Y__ Z__ F__



G02/G03 Circular interpolation

The tool moves along an arc in two axes simultaneously at a programmed linear speed, the *feed rate*.



Format

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

OR

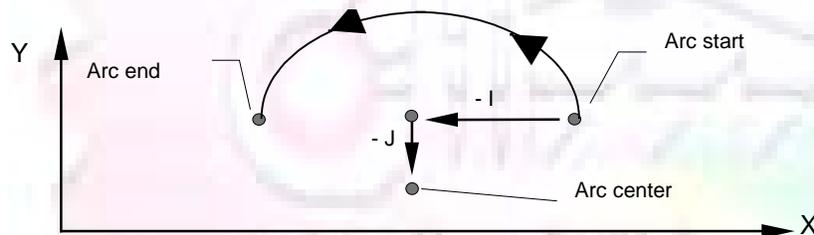
N__ G02/03 X__ Y__ Z__ R__ F__ using the arc radius

G02 moves along a CW arc

G03 moves along a CCW arc

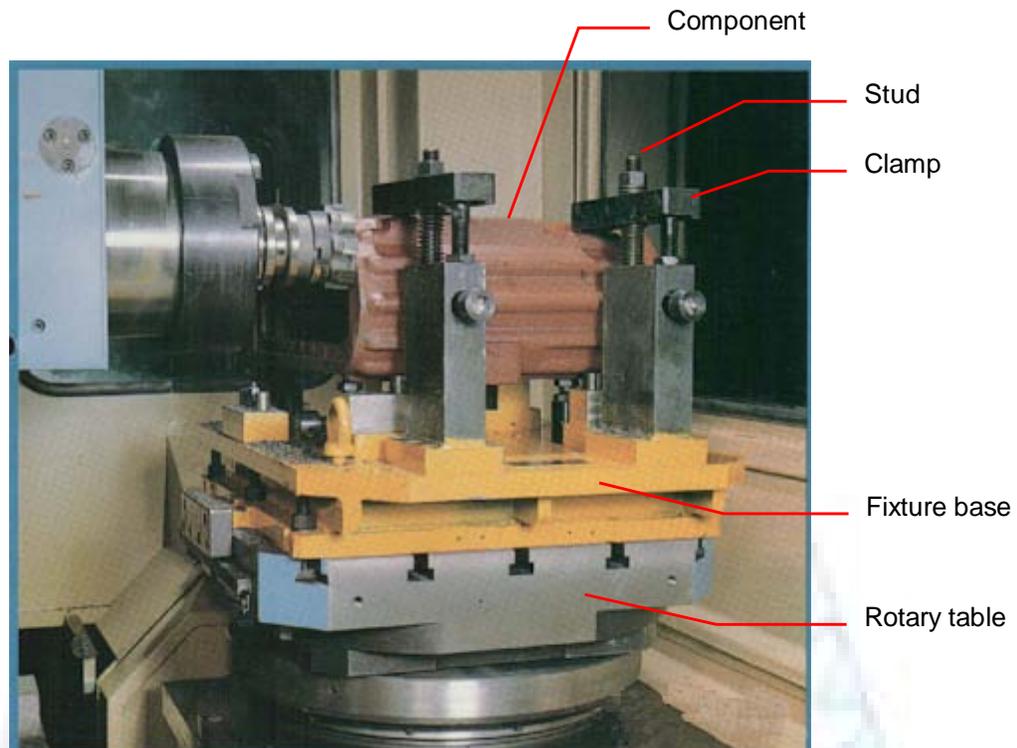
Arc center

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.



- I = X coord. of center - X coord. of start point
 - J = Y coord. of center - Y coord. of start point
 - K = Z coord. of center - Z coord. of start point
- I, J and K must be written with their signs.

•Work holding:



Fixtures

The use of fixtures

1. Improves repeatability of dimensions.
2. Improves working conditions.
3. Cuts down requirement of skilled labour.
4. Reduces part loading and unloading time.
5. Improves safety in the working area.
6. Reduces the cost of machining.

There are two types fixtures commonly used

1. Special fixture:

Special fixtures are used to hold and locate the typical components which are having complex shape and to be located with the reference hole (dowel hole). Special fixtures are required for components like gear case cover, cylinder heads, etc.

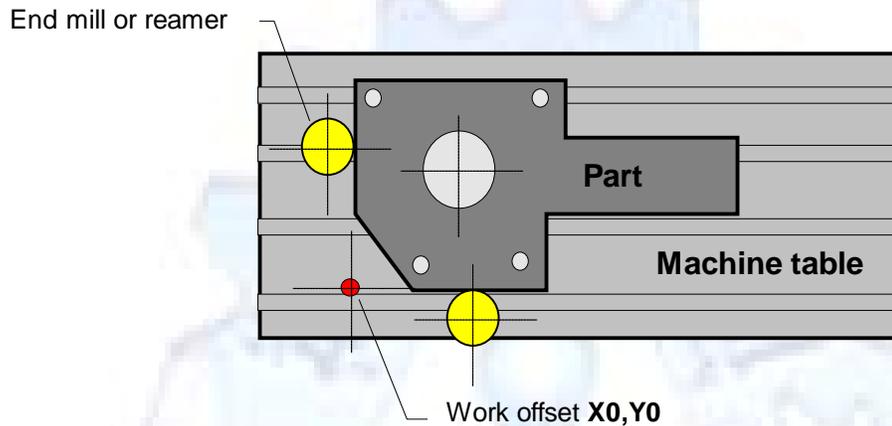
2. Modular fixture:

Modular fixture is a set different kind of clamping and locating elements. These fixtures are used to hold identical components in different sizes and components of box type. Example: Manifolds, head stocks, connecting rods, and flanges.

•Machine setting:

Determining X and Y work zero offset

Case 1



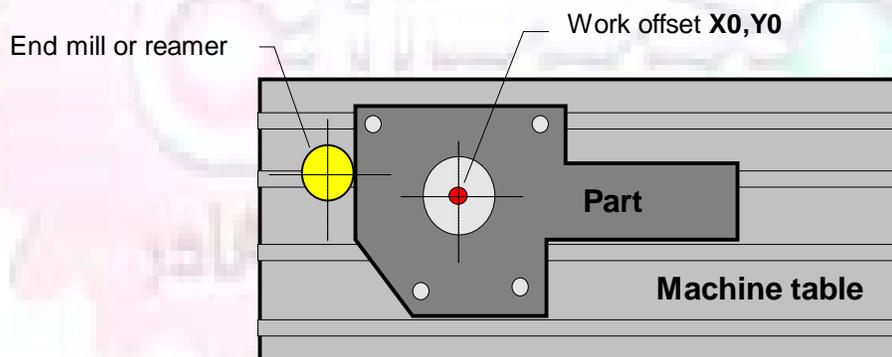
For determining X work zero offset

- Touch the periphery of the end mill or reamer to the part.
- Enter current machine **X** value + cutter radius as X work zero offset.

For determining Y work zero offset

- Touch the periphery of the end mill or reamer to the part.
- Enter current machine **Y** value - cutter radius as Y work zero offset.

Case 2



- Mount the dial indicator on machine spindle, true the bore.
- Enter current machine **X** and **Y** value as X and Y work zero offset.

■ How to Select a Cutting Fluid:

One part of the metal cutting equation that is sometimes overlooked, however, is the choice of an appropriate cutting fluid. With today's system approach to metal cutting, use of the right metalworking fluid has become just as much a part of the solution as the other elements. In fact, at least one company is now integrating cutting fluids and cutting tools into effective metal cutting systems. Following are some things to consider when choosing your next cutting fluid.

■ The Four Key Questions

There are four vital pieces of information that are necessary when choosing a cutting fluid. They are:

1. What are you cutting?
2. What is the chemistry of your water?
3. What are you using now?
4. What are your options?
5. What do you prefer?

■ What Are You Cutting?

The first question – “What are you cutting?” – is often the most difficult to answer, because most shops handle a variety of workpiece materials on a regular basis. If the answer is an unequivocal “aluminum,” or “stainless steel,” or “cast iron,” the selection of a cutting fluid can be quite simple and straightforward. Unfortunately, that is seldom the case in anything but a high-volume plant. Where aluminum or other nonferrous metals are a significant part of the mix, a non-staining cutting fluid is a must. Generally speaking, this will be a semi-synthetic with special ingredients to prevent bi-metallic corrosion and staining of non-ferrous workpieces. Where the ferrous metals predominate, however, a more general purpose semi-synthetic or hybrid fluid might be a more economical choice.

■What is The Chemistry of Your Water?

The second question – “What is the chemistry of your water?” – is probably the single most common reason for the success or failure of a cutting fluid. Very few plants invest the money necessary to install effective water purification systems using deionization or reverse osmosis technology. Unfortunately, the chemicals present, or not present, in the water used to re-constitute a cutting fluid make a tremendous difference in its performance and sump life. Most manufacturers of cutting fluids will perform the necessary water testing to determine exactly what minerals are dissolved in your water and make a fluid recommendation based on the results.

■What are you using now?

The third question – “What are you using now?” – helps narrow the range of choices. When coupled with an objective analysis of the strengths and weaknesses of the current fluid, the answer to this question can quickly point you toward a relatively small number of competitive alternatives that have the attributes required to meet your cutting needs. Many cutting fluids are in use today simply because they were in use yesterday. It is a hard reality of today’s marketplace that if one isn’t constantly striving to improve, they will certainly be overtaken by a competitor who is.

■What Are Your Options?

The fourth question – “What are your options?” – leads to the final selection of a cutting fluid. Cutting fluids can be divided into four general groups, based on their chemistry.

They are:

- Soluble Oils – quite literally, oil dispersed in water.
- Synthetics – completely oil free, man-made coolants in solution.

- **Semi-Synthetics** – a mixture of natural and synthetic lubricants with up to 30% mineral oil.
- **Hybrids** – a special class of semi-synthetics with a mineral oil content of about 15%. Each fluid type has advantages and disadvantages which make it suitable for a particular group of applications. In addition, there are cost/performance trade-offs to be made between the groups within a specific set of applications. For example, where a hybrid and a semi-synthetic may both be suitable for a particular use, the hybrid might provide longer sump life at a slightly higher initial cost, while some semi synthetics may be more economical to recycle or dispose of while not giving as long a tank life. The final decision in this case would have to be made on the detailed economics of the shop where the fluids were to be used, and on the owner's individual preferences.

■ Soluble Oils

Soluble oils are oils that disperse in water when emulsifiers are added. They are generally suitable for moderate-to heavy-duty machining and grinding of all non-ferrous metals, carbon and cast steels, and in applications where dissimilar metals such as aluminium and steel are present. They can also be used in turning, drilling, tapping, reaming, gear cutting, broaching, as well as internal and centerless types of grinding operations. Soluble oils provide excellent aluminium and copper corrosion control; good rancidity control, which extends fluid for long lasting, trouble-free performance; and good concentrate and mix stability, making it ideal for mixing in water with a minimal amount of agitation. An example of this is Valenite's VNT®-650 premium heavy-duty soluble oil.

■ Synthetics

Synthetics are completely oil free solutions made of polymers, organic and inorganic materials that are mixed with water. These clear, low foaming and bio-

stable coolants are ideal for machining and grinding of ferrous materials, carbon steels or cast iron. Their complex lubricant packages make synthetics well-suited for tough-to-machine materials such as stainless steels and high-temperature alloys. Certain synthetics are specifically designed to address the primary failure modes of flank wear and nose wear. Although there are many benefits, some synthetics may cause staining of non-ferrous materials.

■Semi-Synthetics

Semi-synthetic fluids are a mixture of oils, synthetic emulsifiers and water. They were primarily developed for the aircraft, nuclear and related industries. Consequently, they work extremely well on all non-ferrous metals such as titanium, aluminium, copper, brass, bronze and stainless steel in both machining and grinding. Semi-synthetics can be used on ferrous metals, too. They are chlorine-free to reduce bi-metallic corrosion and staining of metals. They should be used at a 5-8% dilution with water, depending on the severity of the operation.

■Hybrids

Hybrids are cutting fluids custom engineered to solve a particular problem. For example, Valenite's VNT®-800 was developed to be very clean in the machining and grinding of cast iron. It provides better stability in hard water and rejects tramp oil very well, which adds to its bio-stable chemistry. Hybrids work extremely well on all metals in both machining and grinding. They should be mixed at a 5-7% dilution with water, depending on the condition of the water.

■What Do You Prefer?

The final question – “What do you prefer” – is usually the “tie breaker” where there is more than one appropriate cutting fluid available. Here, again, as in the case of what is being used now, the key is to examine your preferences objectively

and specifically to identify the real reasons behind a preference. If you cannot point to specific reasons why one class of cutting fluid is preferred over another, you are well advised to follow the recommendations of the professional who has analyzed your requirements. Choosing the right fluid today is easier than ever, as the system approach to metal cutting productivity continues to grow in acceptance. The day is not far off when choosing a complete solution will be as easy as choosing the right insert and chip breaker today. The answers are available, and the results will more than justify the time you spend choosing the right metal cutting fluid.

Useful Links:

www.ball-screws.net

www.lubelink.com

www.machinerylubrication.com

www.advancedmanufacturing.com

www.mmsonline.com

www.cuttingtoolengineering.com

www.geartechnology.com

www.engineersedge.com

www.agma.org

www.efunda.com

www.sae.org

www.sme.org

www.astm.org

www.iss.org

www.machinetools.com

www.ansi.org

www.asme.org

www.cncci.com

www.matls.com

www.thefreedictionary.com

www.engineersupply.com

SurfCAM Tutorial – Beginning with the basics

Background

SurfCAM is a computer aided manufacturing (CAM) software which allows you to create, manage and modify tool paths for computer aided designed (CAD) part. The software requires a CAD model, or requires that you create some geometry within SurfCAM with some rudimentary tools provided. The tool paths are created or stored internally in SurfCAM in a generic programming language called APT. Once the tool paths are created, the APT paths are translated using a post processor into typical machine G-code.

Opening SurfCAM

SurfCAM can be opened from the Novell Applications Launcher on **most** Industrial Engineering computer labs workstations. Search under ‘all’ applications and double click on SurfCAM 2005. Once the program opens, a license must be selected from the list (not the Demo license) to enable saving the file later.

Choose one of the 30 licenses available. If for some reason the licenses are not listed, on the license manager form, select server list and type “bonnie.eos.ncsu.edu”. A list of available licenses should appear. Select a license and click OK.

After the license is selected an Add Stock/Fixture form (Figure 1) opens allows you to add a model which represents the pre-machined stock or a fixture. Click on OK without adding either. Both of these items can be added later under a more appropriate dialog.

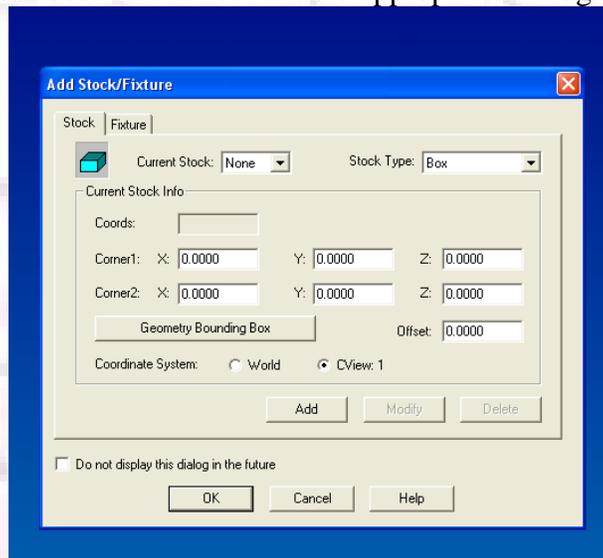


Figure 1 Add Stock or Fixture dialog box

SurfCAM Interface

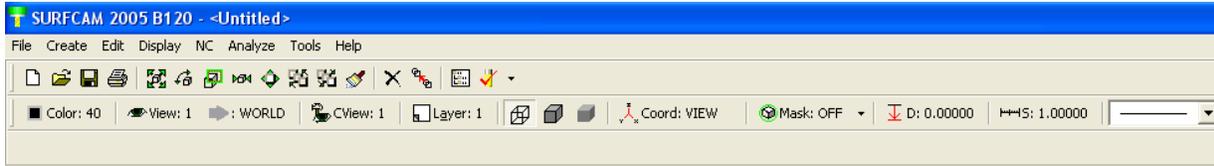


Figure 2 SurfCAM Windows based interface

SurfCAM is a windows based software system, so most of the typical windows functions under **File** are applicable. This is where you can open, save, exit, print etc.

The **Create** menu allows the user to create different geometry forms. Here you can create a point, line, surface, chamfer, circle and many other geometry types. For the most part, these functions are not used by beginners, but are useful when creating bounding curves where you don't want the tool to move outside (constraining the tool).

The **Edit** menu allows you to change or modify many different entities such as changing lines, altering surfaces, joining surfaces, trimming or breaking elements, add control points etc. Again these features are helpful to advanced users and are not covered in detail in this tutorial.

The **Display** menu offers the ability to change the orientation of the part on the screen. The typical rotate, pan, fit to screen, and zoom functions are available, as well as several functions which allow the user to shift between views or call up a named view (i.e. top, bottom, etc).

The **NC** menu is the main machining interface menu. In this menu you select the general form of machining (2 axis, 3 axis, 4 axis, 5 axis, turning etc) and under the general form a more specific machining function is selected. This menu is the most used and the options contained must be fully understood to get the most out of SurfCAM. Each function will be described in some detail in a further chapter.

The **Analyze** menu contains options such as measuring the distance between points, angles, tangency, elements, etc. These functions are auxiliary functions meant to help you understand the part better, but basically provide no functionality for making tool paths for CNC machines. Thus these will not be covered.

The **Tools** menu allows the user to change the system options, modify keyboard shortcuts, change the model units etc. These options are again not used by beginners.

Finally, the **Help** menu provides access to the online users manual. In lieu of clicking the Help menu, the F1 key is the shortcut key for help (as it is in most windows based programs).

Importing a CAD model

Importing a CAD model that is stored locally is done by selecting **File->Open**. On the Files of type: dropdown box, choose the type of file you wish to open (Figure 3). Do this prior to negotiating your way through the maze of folders to prevent having to go through the folder tree again. Once the file type is selected, find the file you are trying to open and click the **Open**

button. SurfCAM will then attempt to read the file you have selected. SurfCAM may provide a dialog box describing what it has imported and identify if it was successful. If this dialog is provided, simply click close without saving.

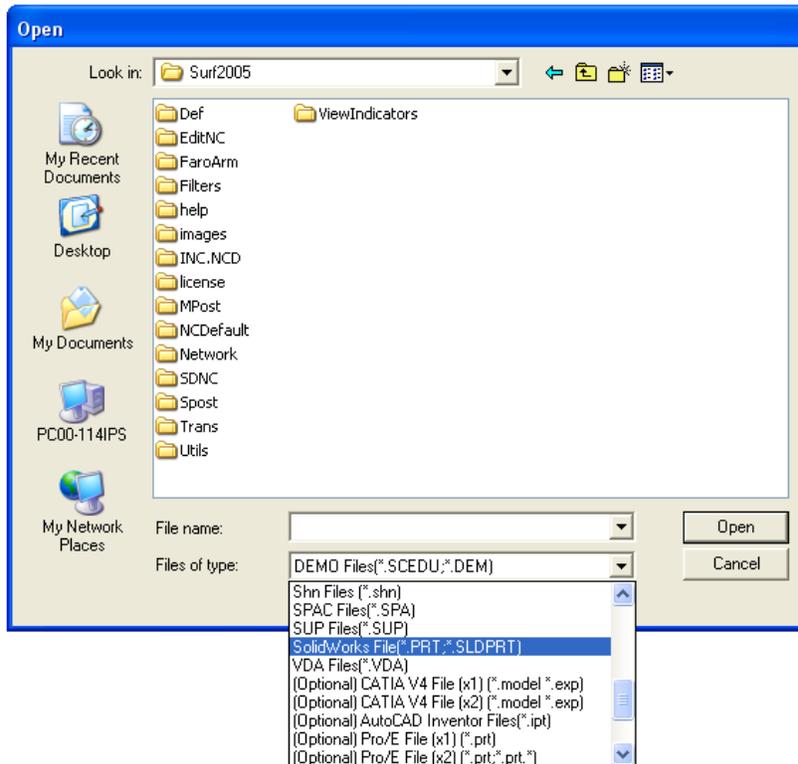


Figure 3 Open Dialog. Note - Change type of file prior to finding the file you wish to open.

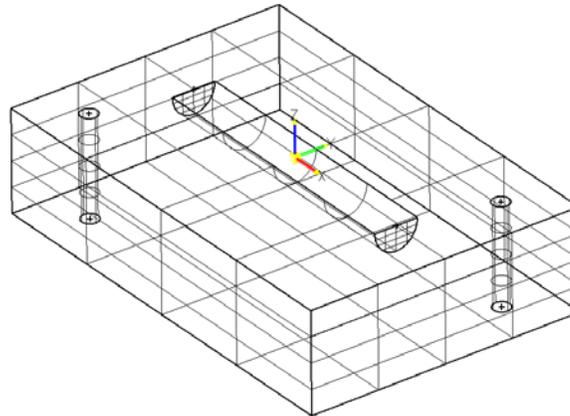
SurfCAM allows a user to import several different types of CAD files, the most common types which you will likely import are SolidWorks part files (*.sldprt or *.prt), AutoCAD files (*.dxf) or IGES (*.igs or *.iges). Native file import (such as is the case with SolidWorks) provides the greatest information transfer, and generic file imports (such as is the case with IGES) provide the least. Normally it is to the advantage of the user to use native file importing procedures when available.

Recently (2005) SurfCAM added some compatibility with SolidWorks. This allows the user to create tool paths for a part, and if an engineering change is needed for the part, update the part in SolidWorks. When the part is then opened in SurfCAM, the changes are noted and a simple update operation command is needed. This however does not operate with complete transparency and/or accuracy, and thus users need to be cognizant of changing CAD models.

SurfCAM Geometry

In SurfCAM, two basic types of geometry are used, the surface and the point. A surface (Figure 4) is a so called spline (NURBS or B-Spline are the two most common) surface (parameterized by two values, u and v) such that every location on the surface can be defined as a function of these parameters. In other words, SurfCAM can calculate the location of any point on the

surface. Surfaces are used primarily in 3, 4 and 5 axis machining and require a significant



amount of processing power and time.

Figure 4 Surfaces model of a mold half. Note the difference between this and Figure 2, a wireframe model

Point or line based models are often referred to as wireframe (Figure 5) models, because the models consist of connected lines and resemble the part only at the intersection of two surfaces. Wireframe models were made popular ‘back in the day’ when computer processing was expensive. There are times when each of the models are important, and selecting model geometry when applying a SurfCAM operation will be much simpler if you understand the differences between the two. When importing a SolidWorks model, the default is both the surfaces and the wireframe geometry are imported.

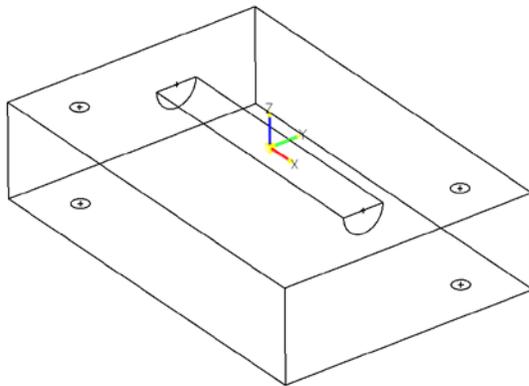


Figure 5 Wireframe model of the mold in Figure 1

Steps to Creating Tool Paths

Once a CAD model is imported into SurfCAM, tool paths can be created. There is no one correct way to create tool paths, however there is a set of general steps to take to ensure the tool paths are created with a minimum amount of hardship (note the wording...Hardship is inevitable when creating tool paths, it is the intent of this manual to minimize this). The following is a list of steps which should be considered when making tool paths:

1. Plan ahead.

- Orient the part in SolidWorks such that if only one side is to be machined, that side is the Top view. This can be done early in SolidWorks by choosing a plane parallel the X-Y plane as the sketch plane for the machined geometry.
- Orient the part in SolidWorks such that the origin is somewhere of relative importance and someplace in space that can be easily identified when placed on the machine. This is extremely important to minimize set up time at the machine (note – my personal preference for a part that is to be fully machined [i.e. stock exists on all sides] is to place the origin at the center and top of the part...just my preference).
- Orient the part in SolidWorks such that the X and Y axes are aligned as you want them on the machine...
- Planning ahead will save you many hours of headaches later...trust me.

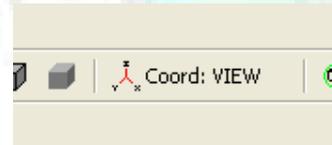
2. Import the model

3. Choose a construction view such that the tool approaches the side to be machined from the positive Z axis direction (as defined by part origin).

- This is done by clicking on the CView: icon



- From the list of named views chose the view which orients the origin as you would like.
- If one of the views does not fit your criteria, you have two choices
 - Perform a transformation of the part in SurfCAM or SolidWorks
 - Add a new construction view by selecting new. This option requires a relatively good understanding of right hand coordinate frames, but is very useful...Normally used by more advanced users.
- Be sure the Coord:VIEW selection is made vs. Coord:WORLD. This ensures the tools will be controlled from the coordinate frame selected by the view (i.e. the origin on the part) instead of the origin on the lower left corner of the screen. This will be especially important when we get into 4 and 5 axis machining.



4. In the Operations Manager (Figure 6) select Setup One and change the name to something meaningful – like Front Side Mold Cavity

- Access the Operations manager by right clicking or by selecting the Operations Manager icon in the tool bar
- Operations Manager is where the tool paths are managed once they are created, as well as where individual tool paths can be verified.



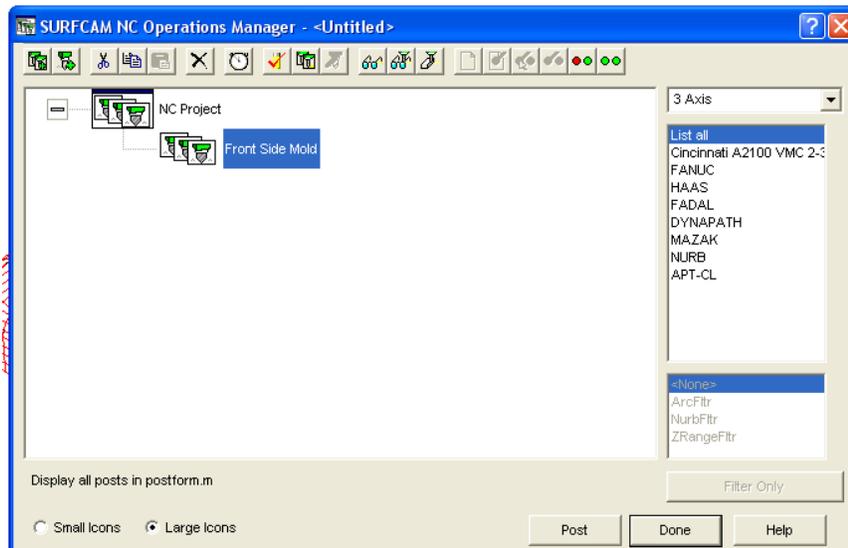


Figure 6 Operations Manager view showing setup one re-labeled as Front Side Mold

5. Roughing Tool Paths
 - a. Create the tool paths to remove the majority of the material (if applicable)
 - b. Use NC->3Axis->Z rough or Plunge Rough
 - c. Described in next section
6. Create Finishing Tool Paths
 - a. Described in next section
7. Add another setup if necessary
8. Perform steps 3 through 6 as necessary
9. Verify the tool paths are correct
10. Use a post processor for the tool paths to create G-Code for machine.

Creating Tool Paths

The following is a list of operations, what they are intended for, how they generally work, and some parameters that may be adjusted to modify the tool paths.

2 Axis tool paths

Unless otherwise noted, 2 axis tool path functions require lines or arc segments as input parameters (as opposed to surfaces). There are several options as to how these segments are chosen, the most prevalent is the chain option and the second most prevalent is the single.

When choosing the line segments using the chain option, SurfCAM will try to link a complete path of segments based on an initial segment chosen by the user. If the operation is planar (i.e. pocket or contour) the segments must lie on the same plane. If the operation is non-planar (i.e. contour 3D) SurfCAM will prompt the user to specify segments when it encounters a branch point.

When using the single element option, the user must manually choose each segment desired for the operation. Note that most operations do not require a closed loop of segments; you may specify, for instance, an open contour where the tool is supposed to move.

The order of choosing segments is important. This order determines the path of the tool in the sense that if three segments are chosen such that the middle segment is chosen last, there will be three different plunge and machine operations, one on each segment. If the operation was to make a continuous contour around the outside of the part, the segments should be chosen in order of the desired tool trajectory.

Operations:

Pocket

Pocket operation in SurfCAM is used to cut an enclosed space, from the inside to the outside, at one particular depth. The operation starts in the center of an enclosed area (with a plunge move in the $-Z$ direction) and makes incremental closed paths towards the pocket boundaries in user defined path offsets (Figure 7). The user chooses the boundary of the pocket (line segments), and if more than one pocket is chosen, SurfCAM will ask the user to click inside a pocket.

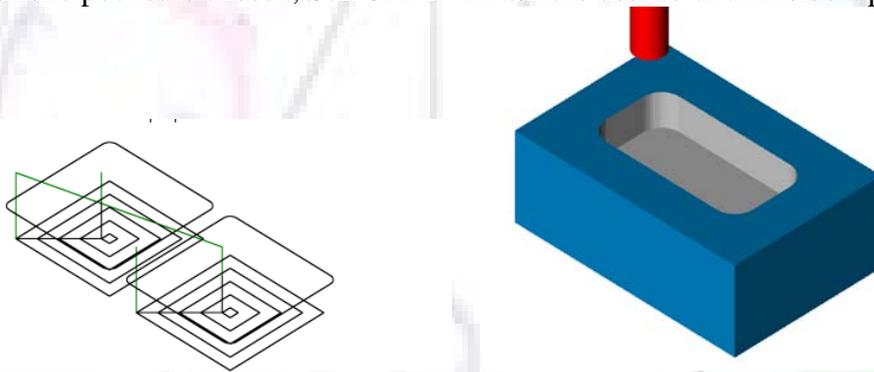


Figure 7 Two examples of using the pocketing operation

The pocket operation allows for creating a projection of the pocket paths on multiple planes. This is useful if you have a deep pocket which will not allow the tool to cut directly at the depth of the pocket, but would be easily done using the same paths at several different Z heights. On the cut control (Figure 8) tab the user specifies where the chosen geometry (pocket segments) lie (i.e. bottom or top). Then under the amount to remove, specify the total material height to be removed and the rough spacing (amount you wish to advance the Z axis with each pocket step). You may also specify a finishing pass and as well as stock to leave.

The screenshot displays the '2Axis Options' dialog box in SurfCAM. The 'Cutting Method' is set to 'Climb'. The 'Curve Tolerance' is 0.00100. The 'Rapid Plane' is 1.00000. The 'Plunge Clearance' is 0.10000. The 'Geometry' is 'Bottom' with a value of 0.00000. The 'Taper Angle' is 0.00000. The 'Cutter Compensation' is 'Offset' with a value of 0.00055. The 'Amount To Remove' is 0.00000. The 'Rough Spacing' is 0.00440. The 'Finish Passes' is 1. The 'Finish Spacing' is 0.00044. The 'Stock To Leave' is 0.00000. The 'On Sides' parameters are 0.00000. The 'In Z' parameters are 0.04500. The 'Leadin Move' and 'Leadout Move' are both set to 'None'. The 'Plunge Type' is 'Helical'. The 'Machine Left-over Material' checkbox is unchecked. The 'Enable High Speed Machining' checkbox is unchecked. The 'Minimum Radius' is 0.00200. The 'Maximum Radius' is 1.00000. The 'Disable for finish passes' checkbox is unchecked. The dialog box has 'OK', 'Cancel', and 'Help' buttons at the bottom.

Figure 8 Example of Contour Operation with multiple passes

Contour

Contouring in SurfCAM is used to perform peripheral milling on the inside or outside of a line segment (Figure 9). The user is to identify the line segments which define a contour (does not have to be a closed loop of segments). SurfCAM may or may not require the user to specify a side to contour (i.e. which side of the line is the tool to cut). For contouring each segment must lie on the same plane.

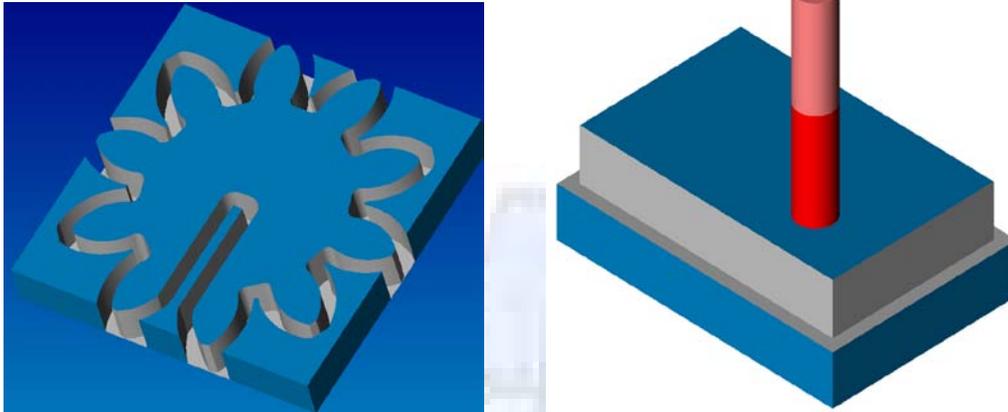


Figure 9 Examples of parts cut with 2-Axis Contour

Contour 3D

Contour 3D is the same operation type as contour, but allows the user to specify segments on different planes for the tool to cut along.

Drill

Drilling operations are any rotating tool operation meant to engage the part from a single direction – namely from the positive Z direction. Drilling operations can be specified using circle, point or line segments. After choosing NC->2-Axis->Drill, SurfCAM allows the user to choose multiple circles or points defining holes to be drilled. Drilling operations also allow the user to incorporate more than 1 operation, such as a center drill, a drill and a reaming operation.

After selecting the locations for the drilling operation, a dialog box (Figure 10) allows the user to add, change or delete tools to be used in the operation. As displayed in the following figure, the spotdrill operation is followed by a drilling operation followed by a tapping operation. The depth of each tool can be controlled individually by double clicking the total depth cell in and manually entering. Tools can be added before or after operations which the cells are highlighted by selecting either of the insert buttons at the top, or by the insert menu. Select tool button allows the user to change the tool size or type.

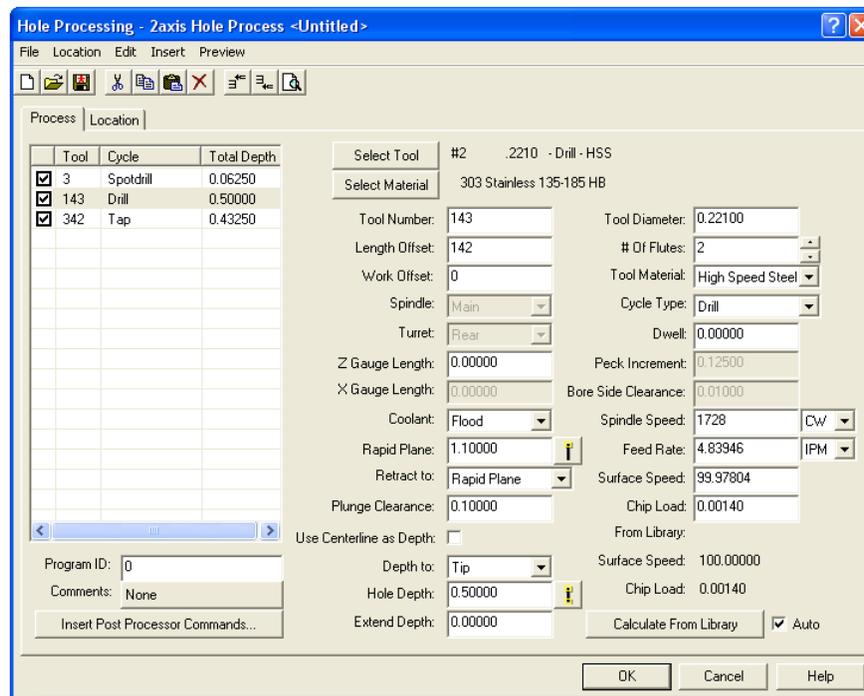


Figure 10 Drill operation manager. Several operations can be completed at one time

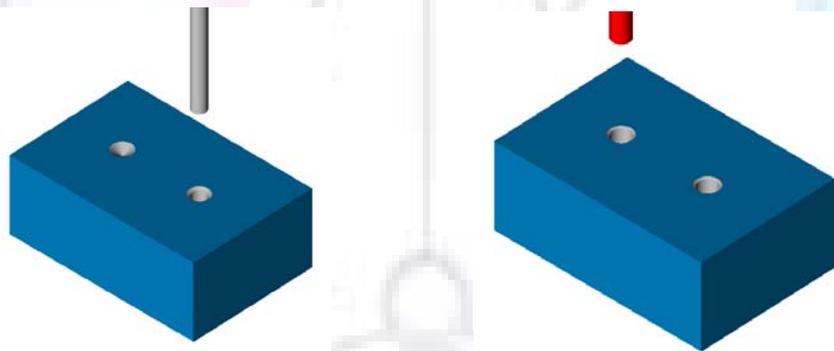


Figure 11 Drill operation including the spotdrill, drill and tap as outlined in Fig. 10

3 Axis tool paths

SurfCAM requires surfaces as inputs for 3 Axis tool paths. Surfaces can be created in SurfCAM, but are most easily created in a CAD package and imported into SurfCAM. Surfaces allow SurfCAM to identify every point on the surface in 3 dimensions, unlike 2D geometry (wireframes) which only identify positions on the plane. Creating tool paths with complex surfaces takes a very significant amount of processing power so expect 3 Axis tool path generation to take longer.

Operations:

Cut

3 Axis cut operation takes a surface and using the flow lines creates a rastering path over the surface, with the tool tip riding on the surface. The primary direction will be defined by the flow lines of the surface (which can be edited by Edit->Surfaces->Arrow->Direction). If you intend to cross surface boundaries, you must merge the surfaces (Edit->Join and choose the two surfaces).

Z-Rough

The Z-rough operation is used to remove a large amount of material prior to finish machining. Roughing operation is typically the first operation you perform for each setup.

The Z-Rough operation cuts the part in user defined intervals. For each of these intervals a surface intersection between the part and the plane is performed and a 2D pocket or contour operation is performed on that plane. This is performed on each plane.

When choosing NC->3 Axis->Z rough, the first dialog box you will encounter asks for a bounding box to be defined. Choose the Extents Box and a second dialog box will ask for the coordinates which define the box (Figure 12). This is the stock size you expect to machine the part from. Input the coordinates, or change the additional offset amounts (in XY or Z) as appropriate and click Calculate Bounding Box.

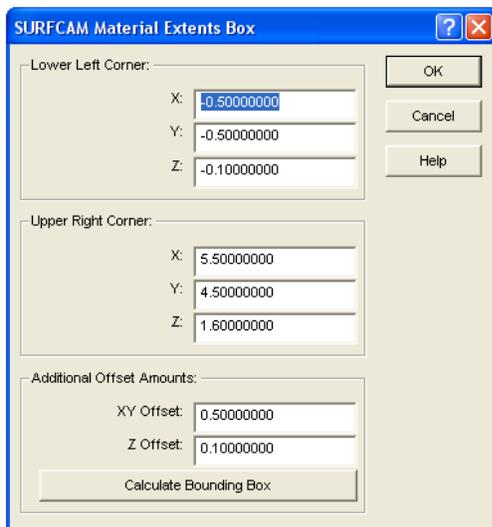


Figure 12 Material Extents Box Example

Next choose the tool and cutting parameters that you desire and click on the Cut Control Tab (Figure 13). Here you will find the following parameters which are often changed: Stock to

leave – this is an offset amount from the surfaces which leaves a small amount of material to finish machine; Minimum and Maximum Z values – Defines area to machine; Z Step Size – This identifies the plane intervals for the surface intersection planes (the first interval will be at the top of the selected area and successive increments down in Z will be cut until the Minimum Z value is reached)

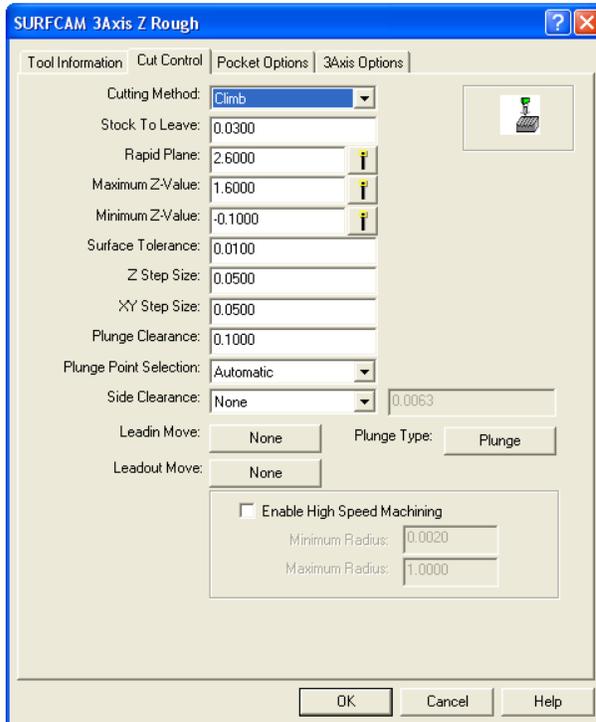


Figure 13 3Axis Z Rough Cut Control

The 3Axis Z-Finish operation works in exactly the same way. 3 Axis Z-Finish is most appropriate for finishing vertical or near vertical walls.

3D Offset

The 3D Offset operation creates a tool path using a 2D or 3D boundary by offsetting the boundary in concentric offset cutting passes. Each pass calculates the tool engagement and maintains the engagement somewhat constant. The advantage (significant) is the scallop height – height of the material between passes – is controlled, thus the surface finish is predictable.

When choosing 3D Offset, surfaces are selected as well as 2D or 3D boundaries. The boundaries define the concentric offset paths shape, as well as limit the tool path area. Choose NC->3Axis->3D Offset from the drop down menus. SurfCAM will prompt you to select the surface or surfaces to cut across. Select the surfaces you wish to machine and then select the tool and cutting parameters you wish use. Next select the Cut Control Tab and check the Use Boundary Curves option. Set the Step Type to Scallop and set the maximum scallop height. If you like,

you can also add stock to leave, if this is a roughing operation. You can also set the Direction and Start-End parameters from this tab. Click OK and the paths will be generated.

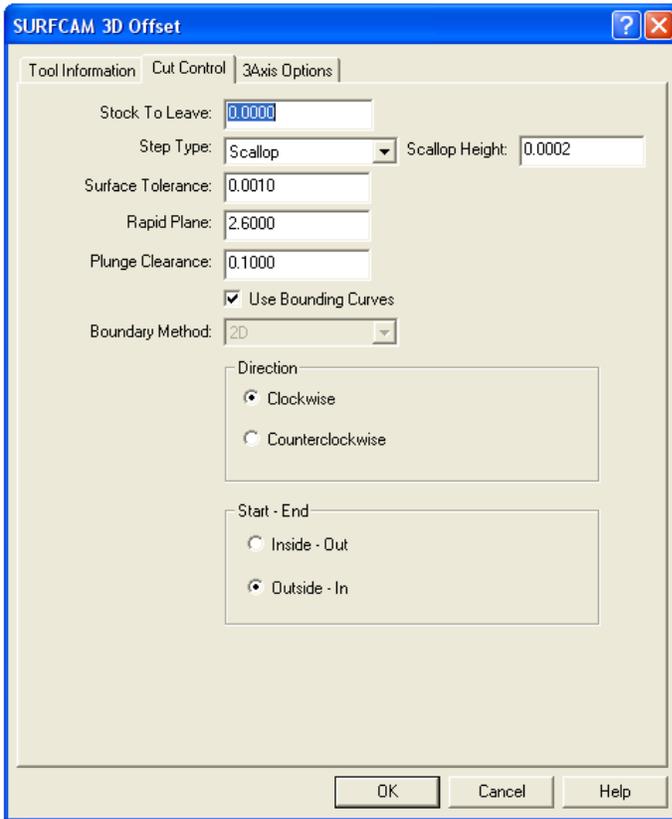
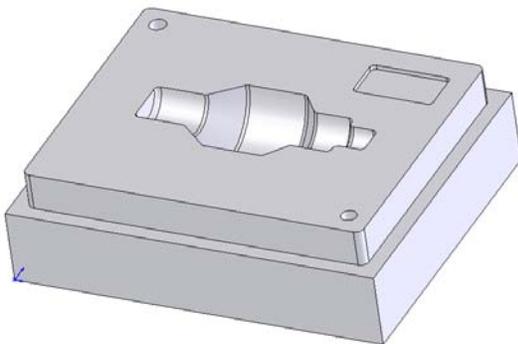


Figure 14 Cut Control Tab for 3D Offset

Example part using both 2D and 3D geometry



The tool paths for this mold will be generated in SurfCAM using 5 operations. The first will be a roughing operation using a ¼ inch flat end mill with the Z step size of 0.0625in. Choose NC-

>3Axis->Z-Rough from the drop down menu. Select all visible surfaces and create an extents box that has 0.25inch XY offsets and 0.1inch Z offset. Next select the 1/4 inch flat end mill from the tools menu. Under Cut Control tab set the Z step size to be 0.0625inch. The tool paths will look something like Figure 15.

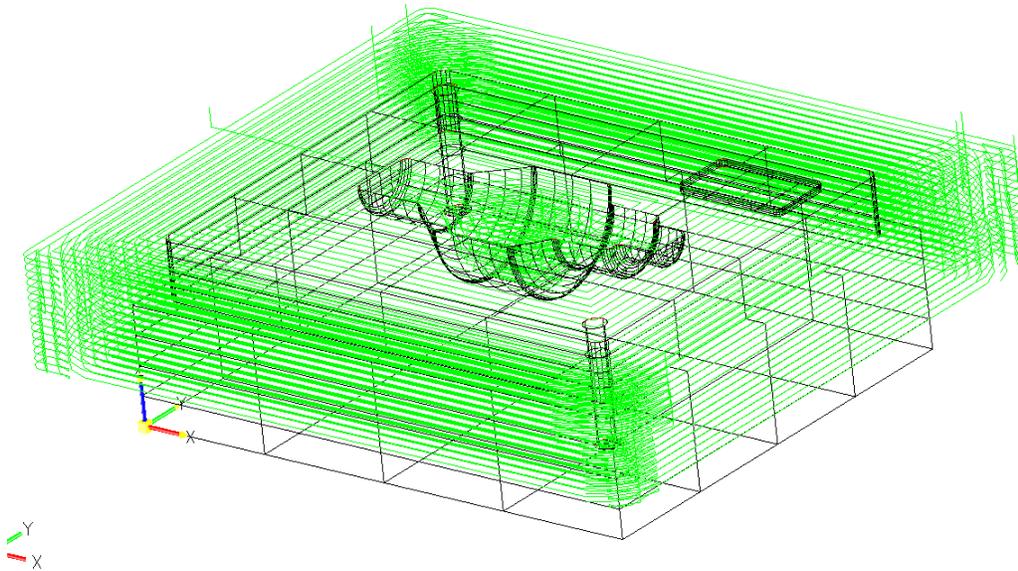


Figure 15 Z-Rough Tool paths for Example Part

Next a flat surface operation will be performed to cut the flat surface on the top. Select NC->3 Axis->Flat Surface from the drop down menu. Select all visible surfaces again. Choose the 1/2inch flat end mill and set the cutting parameters as desired. Under Cut Control Tab change the Minimum Area to 5.0 square inches (Figure 16). The tool paths should look something like Figure 17.

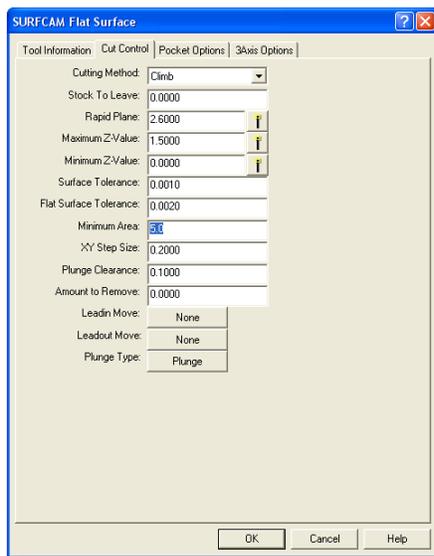


Figure 16 Cut Control for Flat Surface

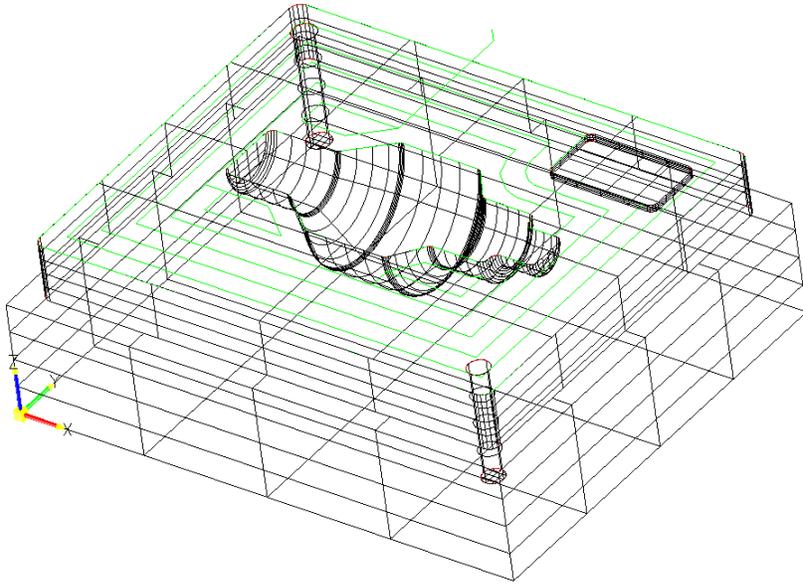


Figure 17 Flat Surface Tool Paths

Next we will use a 3D offset operation to cut the mold cavity. Select NC->3 Axis->3D Offset from the dropdown menu. Change the view to the top view and select within to capture the surfaces of the mold cavity. Draw a rectangle around the mold cavity. Select a 1/8 inch ball endmill from the tool library and set the cutting parameters as appropriate. Next set the Cut Control tab parameters as in Figure 14. The paths created should be similar to Figure 18.

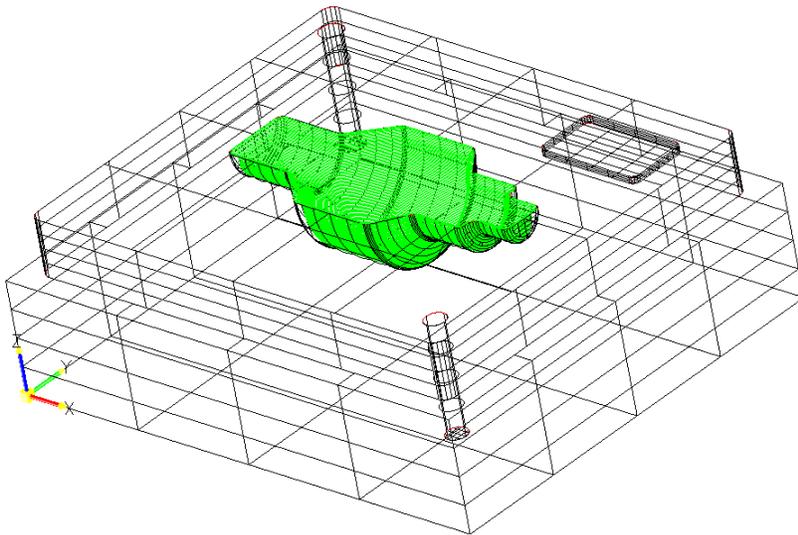


Figure 18 3D Offset Paths

Next we will perform a 2 Axis contouring of the top section. Select NC->2 Axis->Contour from the dropdown menu. Using the chain option, select the bottom contour of the top extrusion. Select the 1/2 inch flat end mill from the tool library and set the cutting parameters as required. In the Cut Control Tab change the Geometry to Bottom and the amount to remove in Z to 0.5.

Change the Rough Spacing in Z to 0.25 and add 1 finish pass of 0.02 (Figure 19). Click OK and SurfCAM will instruct you to choose a side to contour. Click anywhere on the screen OUTSIDE the geometry you selected for the contouring operation. Failure to click outside the geometry will result in the tool being on the wrong side of the contour. Your paths should look something like those in Figure 20

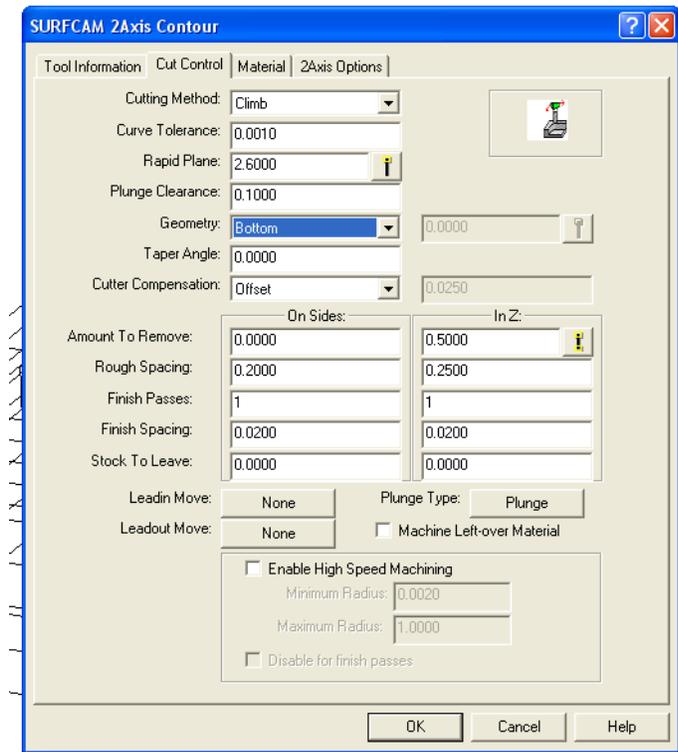


Figure 19 Cut Control Tab for the Contouring Operation

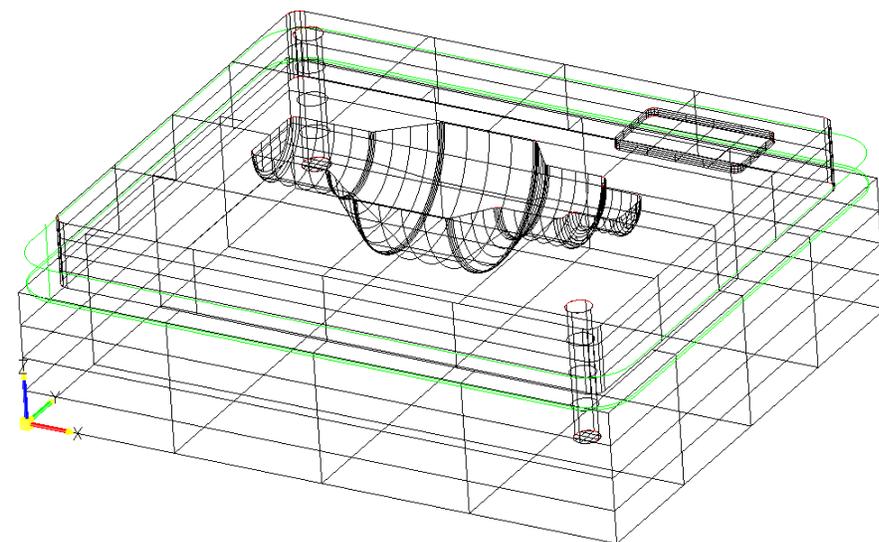


Figure 20 Contouring operation paths

Next we will perform a 2 axis pocket operation for the small pocket on the top face of the mold. Select NC->2Axis->Pocket and using the Auto select feature, select the boundary curve at the bottom of the pocket. Click Done twice and select the ¼ inch flat end mill from the tool library. Set the appropriate cutting parameters and click OK. SurfCAM will prompt you to click inside the pocket. Click anywhere inside the geometry you chose to define the pocket. The resulting paths should look something like those in Figure 21.

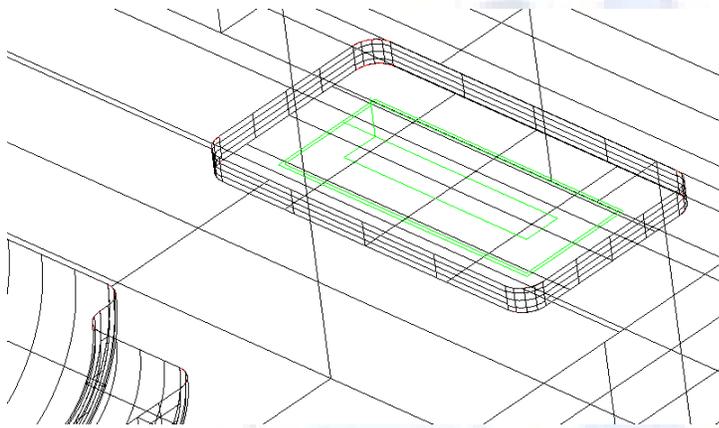


Figure 21 2 Axis Pocket Operation Paths

Finally we will add a drilling operation for the 2 holes at the corners of the mold. Select NC->2 Axis->Drill from the dropdown menu. Select the top circles defining the two holes. A menu box will appear with a default tool (Center drill) already programmed. Add a second tool to the list by selecting Insert->After and select the tool from the tool library. Change the drill depth to 1.0inches and click OK. The paths should look something similar to Figure 23

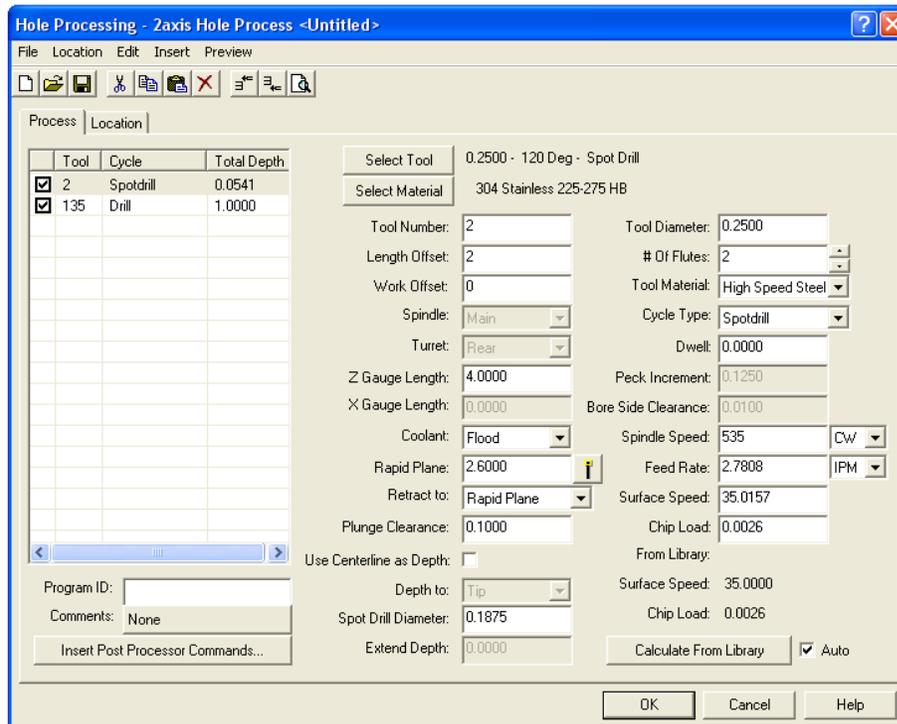


Figure 22 Drill Menu

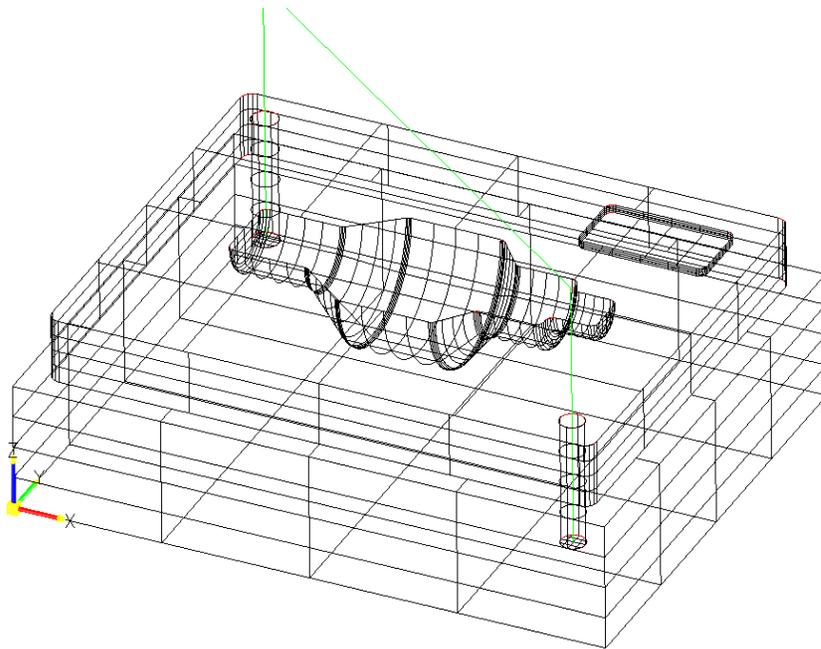


Figure 23 Drilling Operation Tool Paths

Now you can watch a simulation of the cutting by choosing the setup in the operations manager and clicking the simulate icon. Be sure to set the stock in the set up by right clicking the set up and selecting Edit Setup Information. In the Stock Tab you can create a bounding box meeting your needs and click Add. Your simulation should result in something like the following:

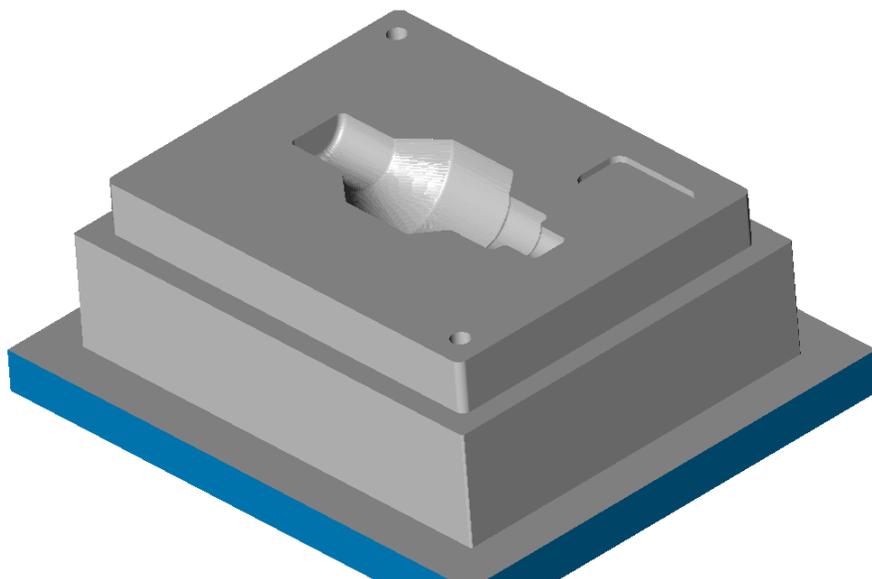


Figure 24 Final Simulation of the example Mold