CENTURION 7 CNC

Operation Manual

Version 3.2

June 2003

MILLTRONICS MANUFACTURING COMPANY
1400 Mill Lane
Waconia, MN 55387

952- 442-1410
952-442-1401 Technical Support
952-442-1418 Parts
http://www.milltronics.net/

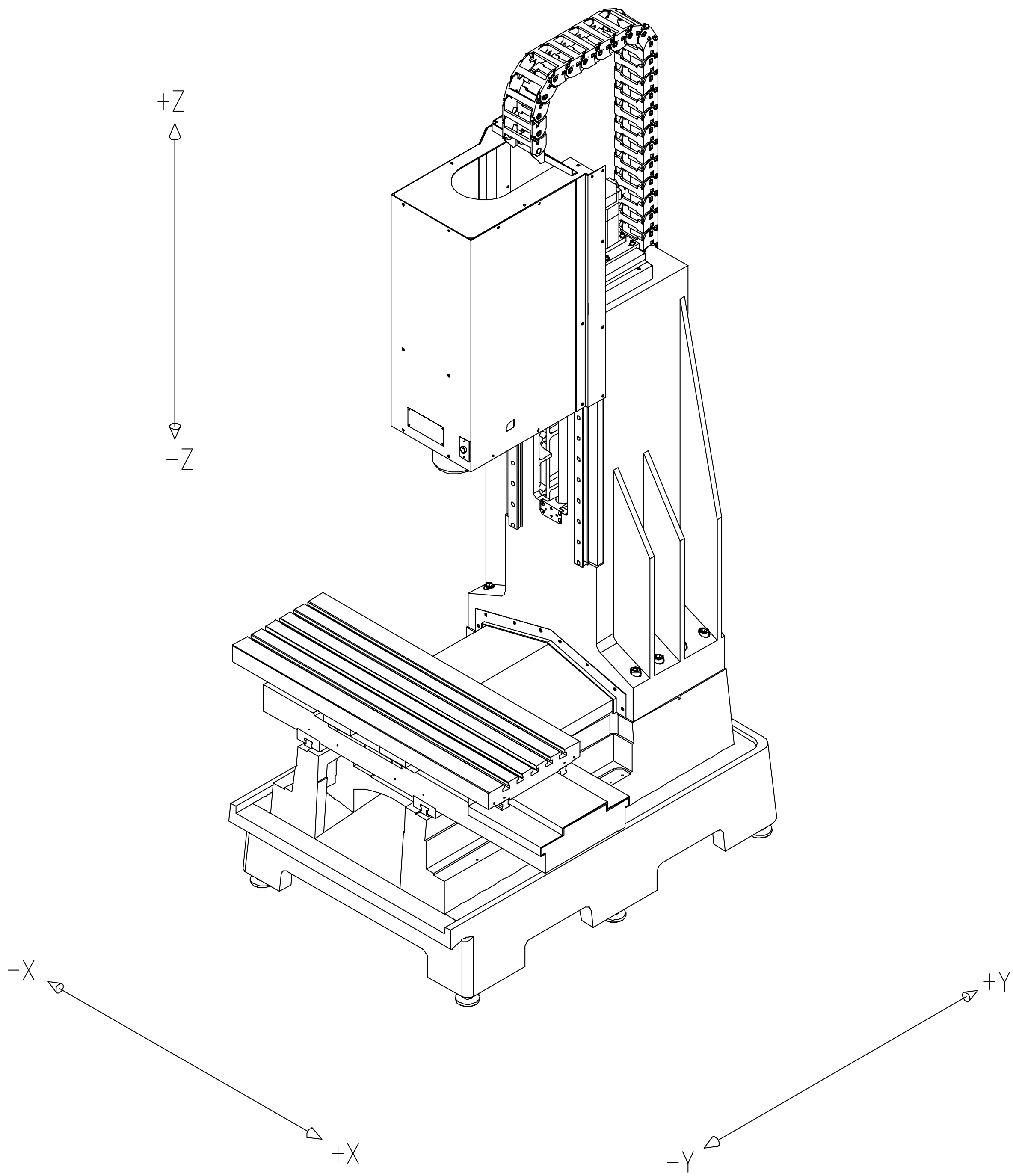
PREFACE

This manual describes the operation of the Centurion 5, 6 and 7 CNC controls. From the operator's standpoint there is no visible difference. Functionality is the same in all controls. The Centurion 7 hardware offers enhanced performance, larger memory, and faster processing. When this manual makes reference to Centurion 7, it implies Centurion 5 and 6 also. The Centurion 7 has five controllable axes in its basic configuration: X, Y, Z, A, and B. This manual assumes that the tool moves with respect to the workpiece.

The programming portion of this manual is divided into two sections: text programming and conversational programming. The conversational programming section is designed primarily to explain the various menus, screen entries, and the general flow from one screen to another. Detailed explanations of each function are described in the M and G code sections and should be referenced there.

TABLE OF CONTENTS

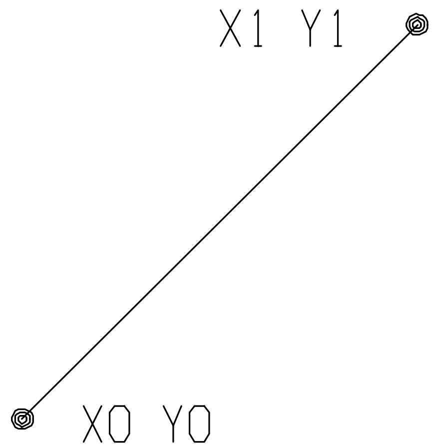
All directions are referenced with respect to the tool. The following illustrates the X, Y, and Z directions.



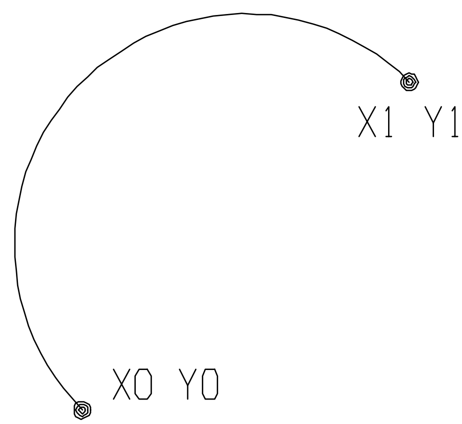
A group of commands given to the CNC for operating the machine is called a program. By specifying commands the tool is moved along a straight line or an arc, and machine functions such as coolant on/off, tool change, or spindle on/off are performed.

The function of moving the tool along straight lines and arcs is called interpolation.

Linear Interpolation

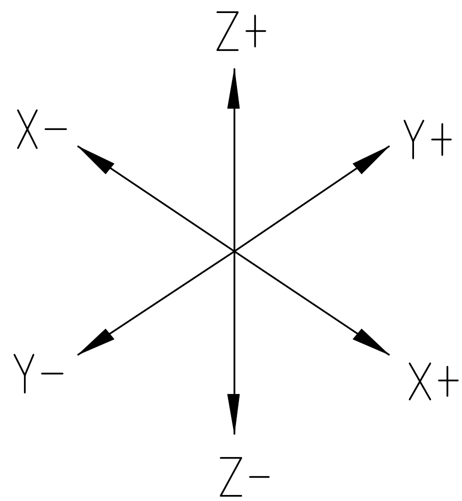


Circular Interpolation



When the commanded position to be reached by the tool is executed, the CNC moves the tool to that position via the circular or linear interpolation modes. The position is given as a coordinate value in a rectangular Cartesian coordinate system.

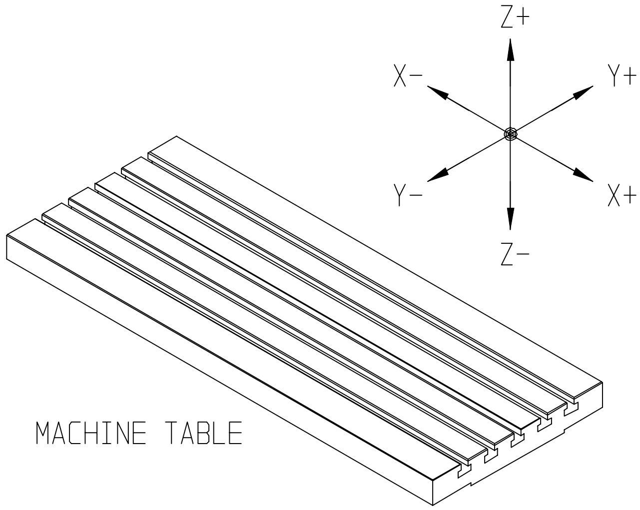
Coordinates



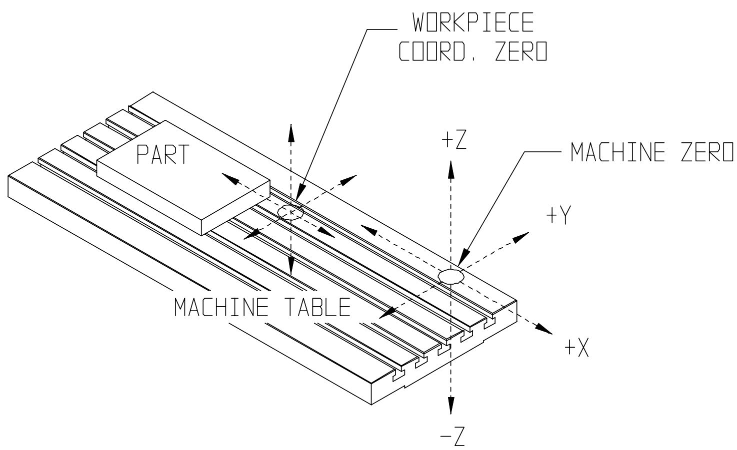
INTRODUCTION

The following types of coordinate systems are available.

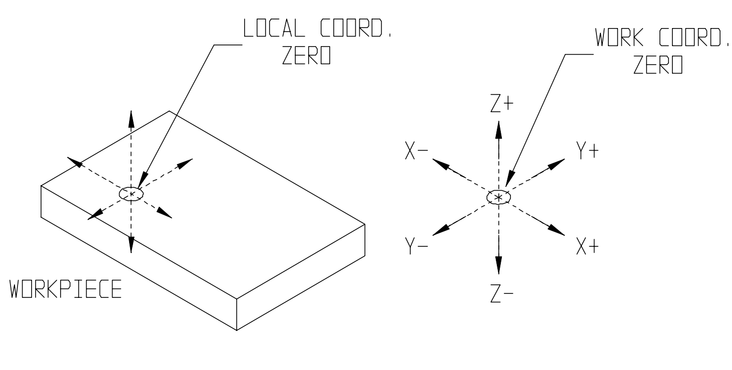
1. Machine system



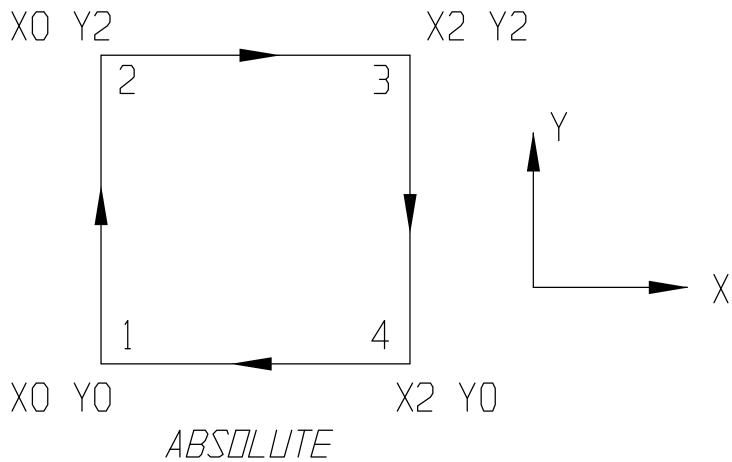
2. Work coordinate system

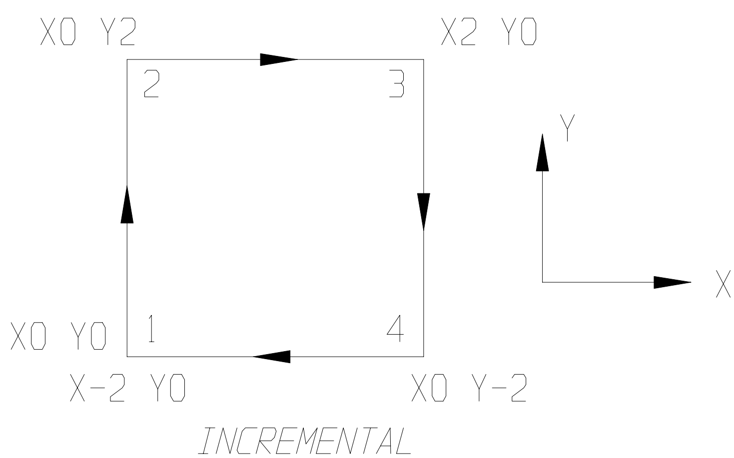


3. Local coordinate system

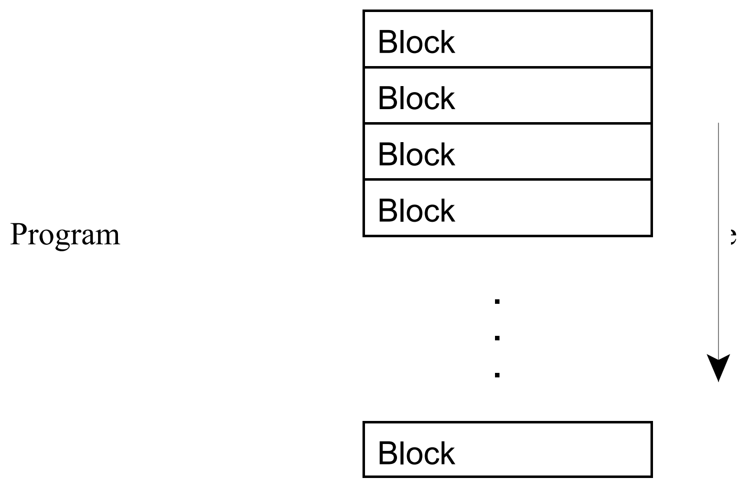


The position to be reached by the tool is commanded with a coordinate value referenced to one of the above coordinate systems. The coordinate value consists of one component for each axis, X, Y, and Z. Coordinate values may be given in either **absolute** or **incremental** mode. In absolute mode, the tool moves to a point **the programmed distance from the <u>zero</u> point** of the coordinate system. In incremental mode, the tool moves to a point **the programmed distance from the <u>current</u> tool position.**





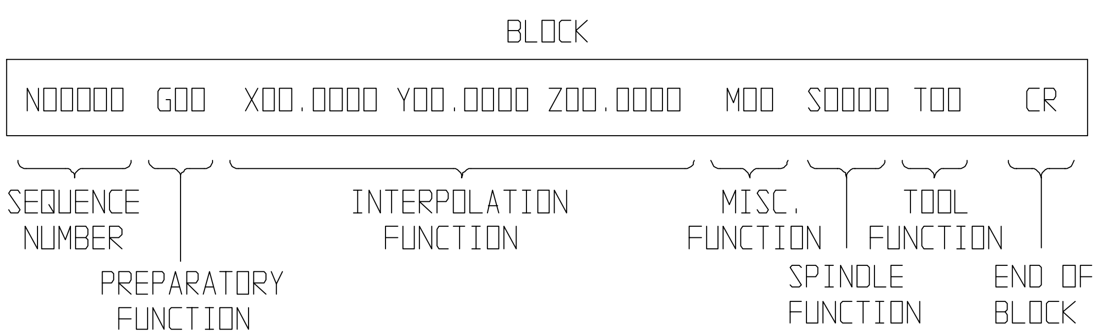
By definition, a program is a group of commands given to the CNC for operating a machine. By specifying commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off. In a program, specify the commands in the sequence of actual tool movements.



A group of commands at each step of the sequence is called the **block**. The program consists of a group of blocks for a series of machine moves. An optional number for definition of each move is called the **block number**, and the number for naming each program is called the **program number**.

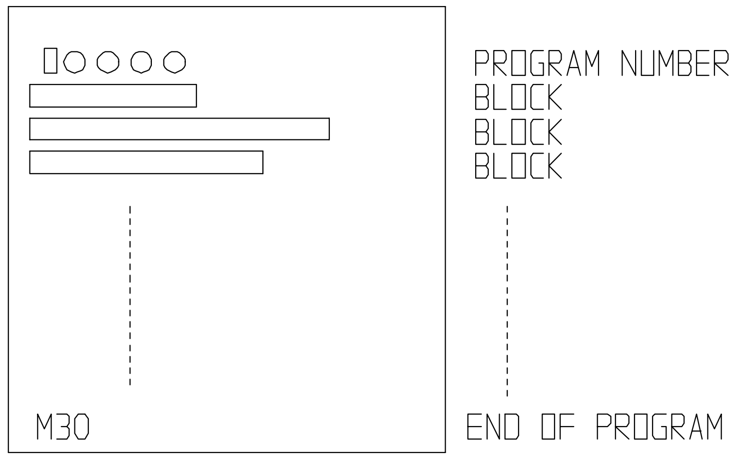
The block and the program have the following configurations.

Block



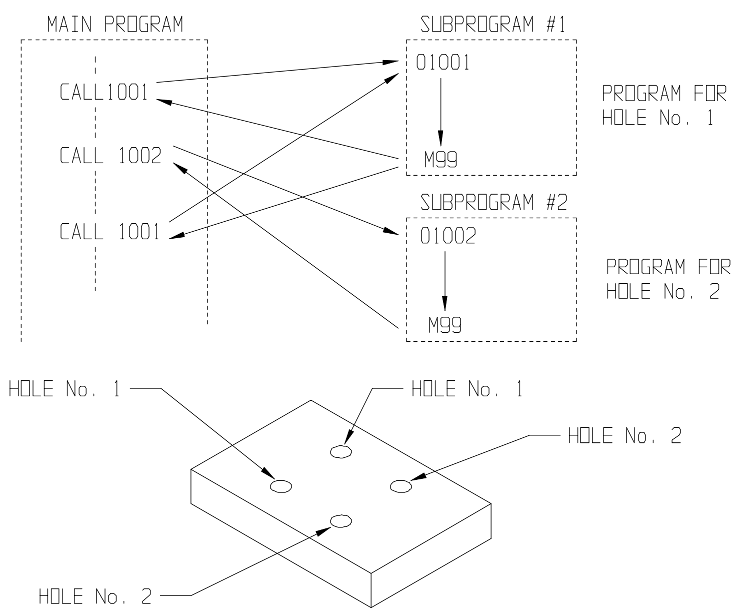
Each block begins with an optional number and ends with a <CR> carriage return.

Program



Normally a program number is specified at the beginning of a program, and a program end code (M99, M02, M30) is specified at the end of the program. Neither is required; however, it may be advantageous to omit the program end code from programs that are used as subprograms. An end program code is assumed when the end of the main program is encountered.

Main program, subprogram, and subroutines



When it is necessary to machine the same pattern at many places on a part, a program for the pattern should be created. This is called a **subprogram**. When a “M98" or “Call” (subprogram call) appears in the main program, the commands of the subprogram are performed before execution of the next block of the main program.

Subprograms can be used to build part libraries of commonly used patterns and can reside anywhere in memory.

Command format ranges

The basic address and command value ranges are listed in the table below. Note these figures give the maximum numerical limit for the control. These limits will always be greater than or equal to the physical limits of the machine. The machine limits are set via parameters in the machine setup section of the control.

Command Format Ranges

FUNCTIONS	COMMAND LETTER	INCH INPUT	METRIC INPUT
Subprogram # and Program #	O	1 - 9999	1 - 9999
Sequence #	N	1 - 99999	1 - 99999
Preparatory function	G	0 - 999	0 - 999
Dimension * words	XYZUVWQ ABCIJKRP	0 ± 999.9999	0 ± 9999.9999
Dwell	P	.01 - 9999.99	.01 - 9999.99
Feedrates *	F	.1 - 999.9	.1 - 9999
Spindle speed *	S	1 - 99999	1 - 99999
Tools	T	0 - 99	0 - 99
Misc. function	M	0 - 999	0 - 255
Repeat or loop	L	0 - 999	0 - 999

**These functions have selectable decimal positions. There may be any number of leading or trailing places as long as the total number of digits fits in the field.*

Command formats for axes: M and G codes

Axis commands can be programmed in a calculator format. No leading or trailing zeros are necessary. Whole numbers may be programmed without the decimal point. A decimal point may be used with mm, inches, or second values. The location of the decimal point is as follows.

Z15.0	Z15 millimeters or Z15 inches	(same as Z15)
F10.0	10 mm/min., or 10 inch/min.	(same as F10)
G04 P1	Dwell for one second	(same as G4 P1)

The following addresses can be used with a decimal point: X, Y, Z, U, V, W, A, B, C, I, J, K, R, F, P, Q, AA, AB, XC, YC, E, H, L, N, O, S, and T.

Axis Min/Max Values for Standard Systems

	Least Increment	Maximum Value
Metric	0.001 mm	99999.999 mm
English	0.0001 inch	99999.9999 inch
Degrees	0.001 deg	99999.999 deg

Axis positions are stored in floating point; therefore, digit commands greater than 8 will be accepted.

The Centurion 6 front panel has two 16-key keypads and 12 function keys. The keypads are used to enter the alphanumeric data requested by the CNC. The upper keypad is used primarily to enter alpha characters. To enter one of the shifted characters simply press and release the **Shift** key then the character. After the character has been entered, the control automatically returns to the non-shifted character set. **Shift** also works in the same manner on the lower or numeric keypad. Spaces between commands are optional when data is entered, but the **Enter** must be pushed to end a line of data or to go to the next function. The operation of the 12 function keys changes as different menus are displayed on the display. The following sections detail each function key.

The lower section of the panel is dedicated to manual machine cycles. Located on the far left of the panel is the electronic handwheel, which when turned in the handwheel mode will cause the selected axis to move. Next to the handwheel are the manual feed controls for the machine's axes. Turning the feedrate override will modify the current machine feedrate by the indicated percentage. Pressing the **Feedhold** button will cause axis motion to stop. To restart axis motion, press **Feedhold** again and press the **Cycle Start** button. The Cycle Start Button needs to be pressed anytime a machine command is to be executed. **Cycle Start** will blink when it needs to be depressed.

The next section of the panel deals with the spindle and coolant controls. The spindle override switch will modify the current spindle rpm by the selected percentage. If the machine is not equipped with a variable speed spindle option, the override switch has no effect on the spindle. The spindle **CCW**, **CW** and **Stop** buttons will override the current control commands giving the operator full manual override capabilities. The active state of the spindle will be represented by the illuminated button. The coolant buttons **(Mist** and **Flood**) work identically to **CW** and **CCW**. When they are lit, the function is active; however, the coolant will not turn on until the spindle is started. The **Tool Reset** button is only active during an M6 command. This button is a safety interlock, which prevents the spindle from starting during a manual tool change. The button will start flashing during a tool change and will need to be depressed after the tool change is completed before program operation can be resumed. The **Emergency Stop** button, when pushed, will stop all machine actions instantly. Once **Emergency Stop** is depressed, the **Reset** button will flash indicating that it must be pushed before any machine motion can be performed. The control is always in an Emergency Stop state after power-on. The following diagram shows the layout of a Centurion 6 front panel.

Centurion 7 Front Panel

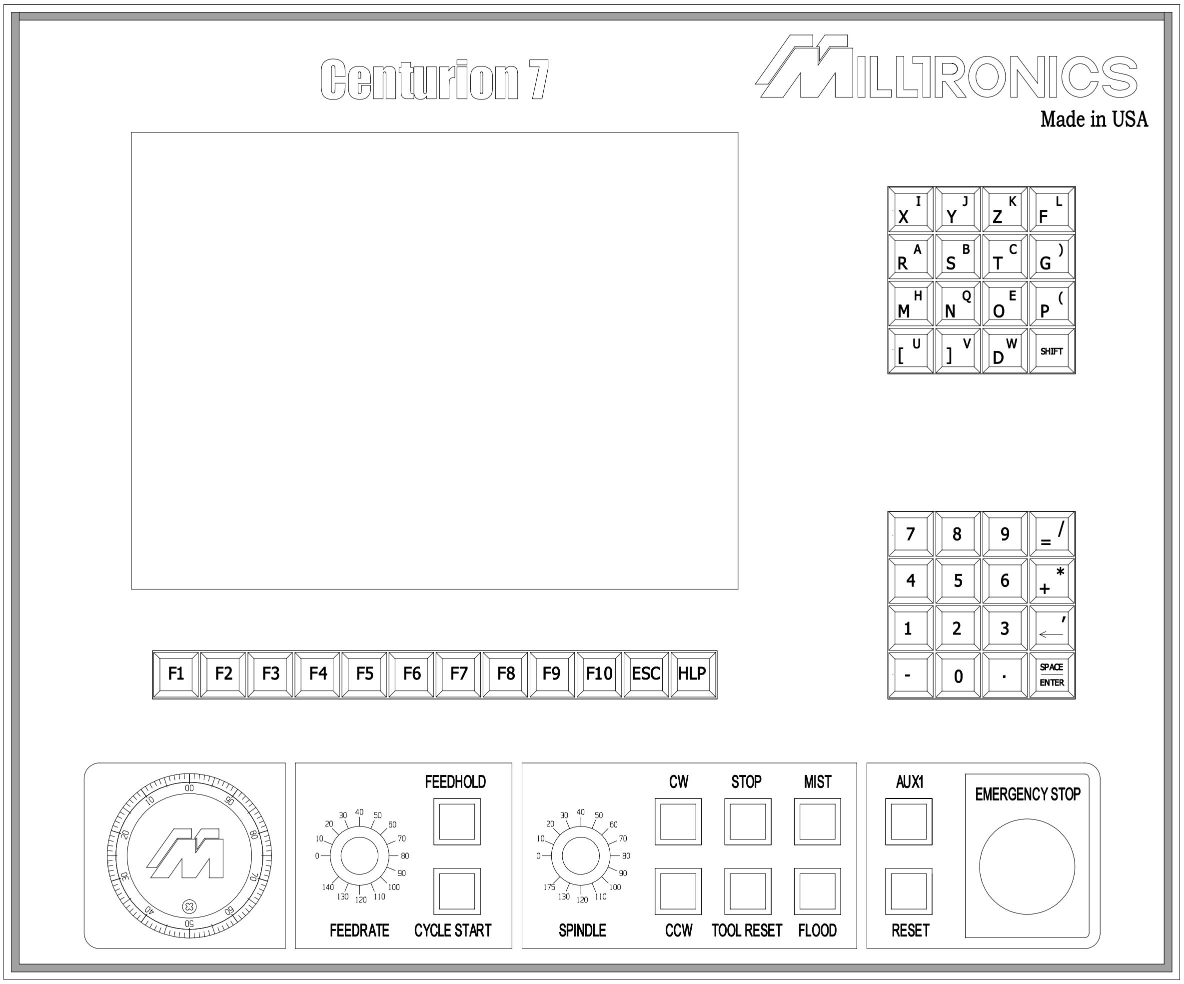
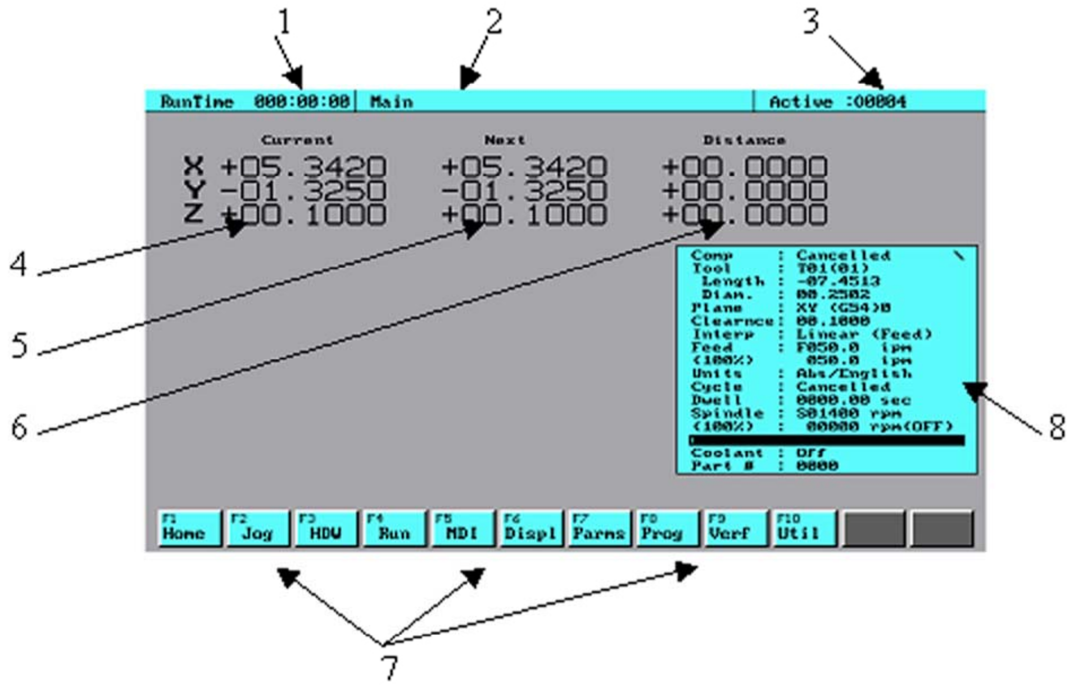


Diagram of Main Screen



1 RunTime
When you are verifying a program the runtime displays the calculated time to make the parts. When you are running a program it shows the elapsed time since the program was started. The total of all program run times are kept in the "Job Time" parameter (F7 Parms – F9 Ctrl).

2 History Line
History Line shows you where you are in the software and where you came from. If you are modifying the tool table, the history line would say Main-Parms-Tool-Edit.

3 Active Program
Active Program displays the program that you are running or verifying. If you are editing a program it will show the active edit program. If you are sending a file over the RS-232 it will show the program being sent.

4 Current Position
Current Position is the position relative to the work offset zero.

5 Next Position
If you are running or verifying, the Next Position is the position that the machine is going to.

6 Distance
Distance is the remaining distance the machine has left to go to finish the move being executed.

7 The Function Keys
Highlighted Function Keys are active or available.

8 Status Window
Status Window displays detailed information on the state of the control. A detailed description is given below.

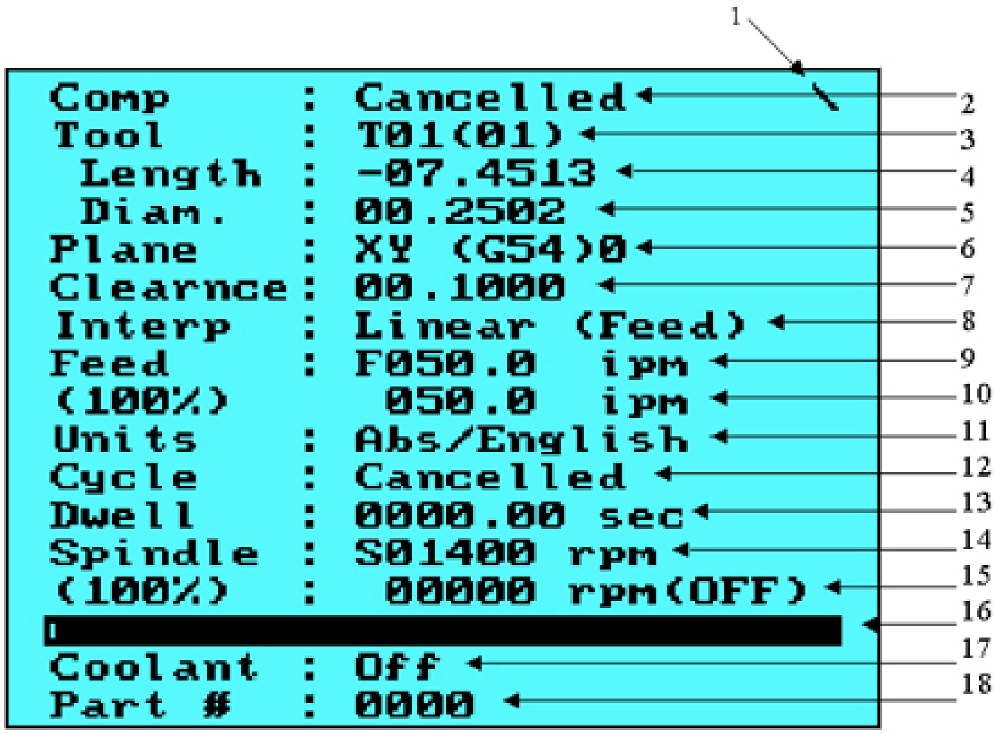


Diagram of Status Window

1 The \ changes back and forth to / and \ each time the status window is updated.
2 Comp: Tool Radius Compensation (Left, Right or Cancelled)
3 Tool: The first two digits indicate the active tool number. The second two digits in parentheses indicate the pending tool number. If you execute a T14 without the M6 the pending tool number will be 14.
4 The tool length.
5 The tool radius.
6 Plane and work offset: The plane is XY (G17), ZX (G18), or YZ (G19). The work offset shows the current work offset G54 (0)…G59 (9).
7 Clearance or R-Plane.
8 Interpolation, Linear (Feed), Linear (Rapid), Circular (CW), or Circular (CCW)
9 Feedrate: The programmed feedrate and its units (English feed per minute) ipm, (Metric feed per minute) mmpm, (English feed per revolution) ipr, (Metric feed per revolution) mmpr or (inverse feed) /min or /sec (If the machine is rapid mode the units will always be ipm or mmpm.)
10 Feedrate override: The position of the feedrate override and the resulting feedrate. If the machine is programmed to move faster than its maximum, the clamped feedrate will be displayed, and a an "*" will be displayed next to the %.
11 Units: (Absolute) Abs or (Incremental) Inc and English or Metric
12 Cycle: If there is a canned cycle or autoroutine active it will be displayed on this line.
13 Dwell: When a dwell is executed the dwell time will count down to zero.
14 Spindle: The programmed revolution per minute.

15 Spindle override and direction: The position of the spindle override and the resulting rpm (if there is no spindle encoder) or the actual rpm (if there is a spindle encoder). This line also displays whether the spindle is off or running CW or CCW.

16 The spindle load meter will show the load on the spindle. It becomes longer and changes color from blue to green to yellow to red as the load increases.

17 Coolant, Off, Mist, Flood or Mist/Flood

18 The Parts Counter increments each time a program ends normally. It will not increment if a program is aborted, or if there is an error in the program. It does not increment in dry run, verify or MDI. It does increment on RUN, DNC-Run and DNC-Fast. The parameter number used for the counter is P699 (you can use this for engraving or what ever you choose). You can zero, increment or decrement the counter using:
Shift-F1 (zero/reset)
Shift-F2 (decrement) (it will not decrement below zero)
Shift-F3 (increment)
You have to push and hold the Shift while pressing the F key.

The remainder of this section will explain each function that can be executed from the front panel.

F1 (Home) Main-Home

Following a power off sequence, the control will always have to be homed after the machine has been reset. Each axis will seek a home limit switch and a marker pulse on the encoder. After this procedure is finished, the machine's reference position will be established and will be recalled until another power off. To initiate a home sequence, push ESC until the main screen is reached and then push F1 (Home). A message requesting that the **Cycle Start** button be depressed will appear on the screen and will start flashing. Pressing **Cycle Start** will start the home sequence, and when it is finished the main screen will return. Homing parameters may be adjusted in the F4 (Axis) section.

F10 (Here) Home-Here

F10 (Here) is only active if the correct password has been entered in the setup parameters.

The machine can be homed without moving the axis. Toggle the F10 (Here) key when the key is lit and push **Cycle Start**. The position of the machine will be assumed as home zero. This action is useful in that it allows jogging, handwheeling, or MDI’ing without physically homing first. It is not recommended you run programs unless the machine is homed in the normal fashion as the software limits will not be valid after F10 (Here).

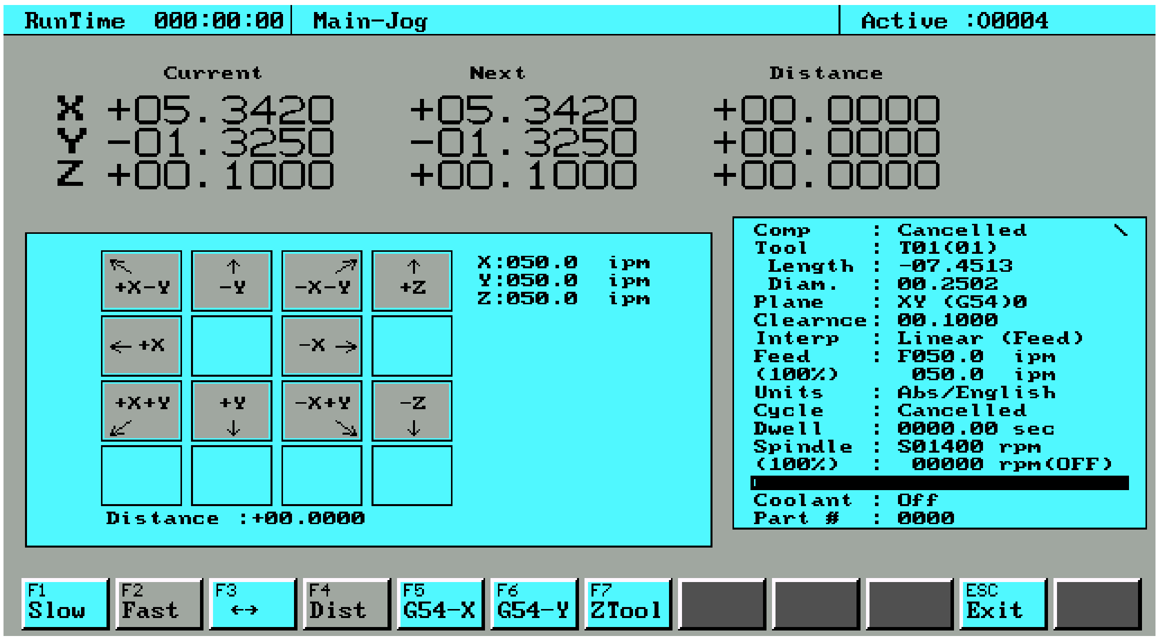
Note: If the machine has a glass scale used for quill feedback, the F1 (Quill) will zero the quill position . See page 50, Section 2 for more information on the quill scale.

F2 (JOG) Main-Jog

The machine must be homed prior to jogging.

F2 (Jog) is used to move the machine around in a manual mode to pick up zeros and align parts. Upon pressing F2 (Jog) the following screen appears.

Note: F5 and F6 can be changed to store the positions in the G92 floating zero offset.

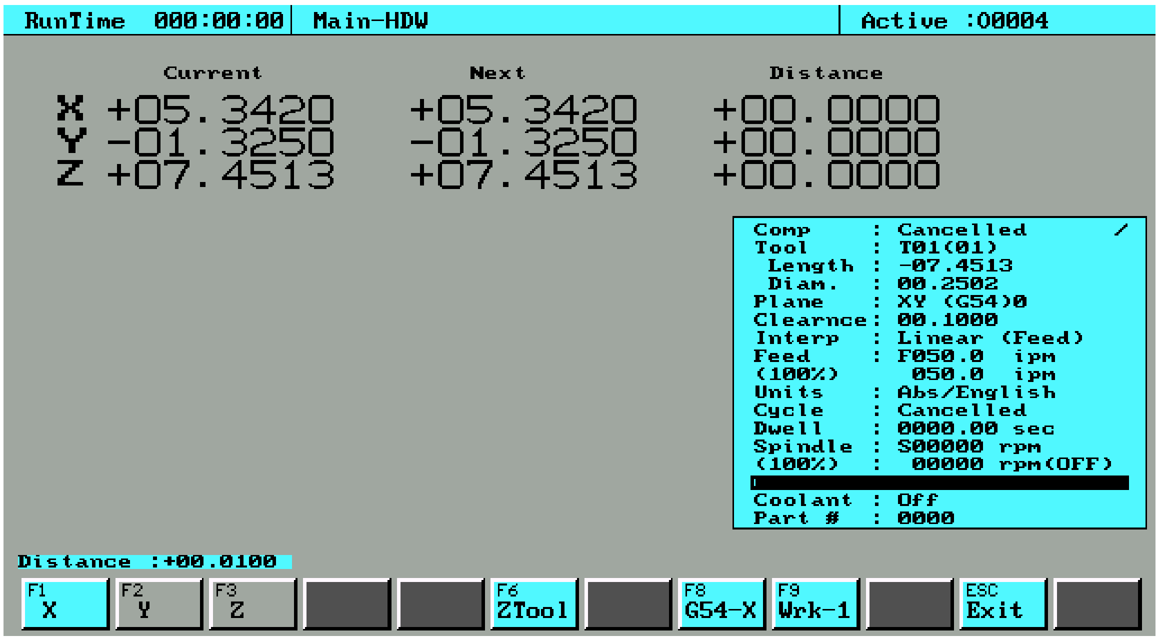


The function keys across the bottom of the screen select the desired jogging mode. The F1 (Slow) key selects slow jog. The feed override is active and speeds up or slows down the jog speed. The F2 (Fast) key selects rapid jog. The machine tool builder determines the slow and rapid feedrates and has the ability to change the values for each in the axis parameters. The F3 (↔) key selects the continuous jog mode. The jog defaults to continuous each time F3 (↔) is depressed. In continuous jog, the selected axis continues moving until the user releases the axis key or encounters the software limits. In incremental jog, the axis moves the selected increment and then stops each time the user presses and releases an axis key. After the operator presses the F4 (Dist) key, the control prompts the user to enter the desired amount of increment. The keyboard diagram displays the direction in which an axis moves when the corresponding key on the numeric keypad is pressed. To exit the incremental jog mode, the user depresses F3 (↔). Depending upon the value of the miscellaneous parameter "use FLZ instead of G54", the F5 and F6 keys perform either a G54 or a G92 for X or Y at the current machine position. This can be thought of as call this position "##.####". The F7 key is used to set tool lengths; the operator is prompted for the Z-axis position. If the user desires to leave the jog mode, pushing the ESC (Exit) key exits the jog mode and returns the control to the main screen.

F3 (HDW) Main-HDW

The machine must be homed prior to handwheeling.

The handwheel mode is used to move the machine around using the electronic handwheel. Its main use is for setting tool length offsets, setting work offsets, and aligning parts. Upon pushing F3 (HDW) the following screen will appear.



Note: F8 can be changed to store the positions in the G92 offset.

The F keys across the bottom of the screen are used to select which axis will move when the handwheel is turned. The feedrate override switch will determine the distance each axis will move for one click of the handwheel. The highlighted axis key determines which axis will move.

F6 (ZTool) is used to set a Z tool length offset into the tool table or H parameter table. In the handwheel mode, a tool could be put into the spindle and moved to its Z zero point. The F6 key can then be pushed indicating that we wish to enter the current Z position as a tool length offset *H parameter*. The CNC will prompt for a Z position. When the **Enter** key is pushed, the current Z position will be used for the tool length.

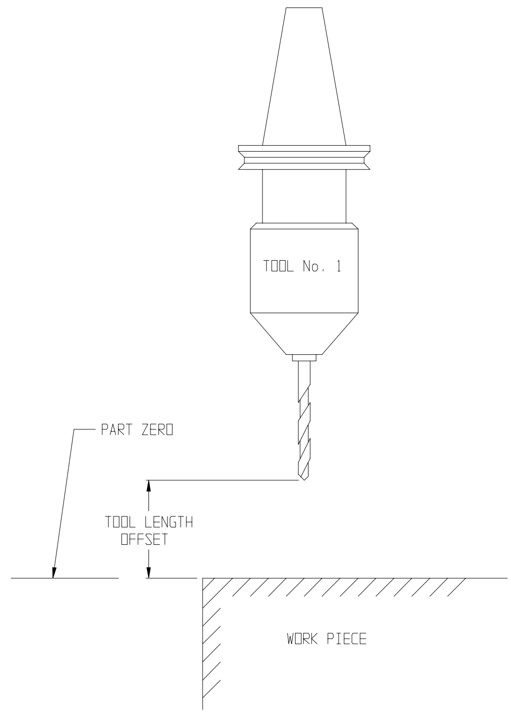
Depending on the value of the miscellaneous parameter "use FlZ instead of G54". The F8 key performs either a G54 or a G92 for X or Y or Z at the current machine position. The operator is prompted for the axis position. This can be thought of as a "call this position '##.####'".

Note: The distance shown is how far the machine will move per click of the handwheel.

Procedure for Setting Tool Length Offset

Note: there is an alternative method for setting tool lengths on 31.

A tool length offset is used to compensate for the difference between Z axis home and part surface (part zero). Setting floating zero in Z axis is not recommended.



To set tool length offset, load tool #1 in the spindle by doing an MDI→ T1M6. Use handwheel or jog to touch the tool on the part at a known location. Select F6 (ZTool). The control will prompt you to enter the Z position to be set. A tool length offset for tool #1 has been set. Now when tool #1 (H1) is programmed to a position, it will position in reference to part zero. Repeat this procedure for each tool.

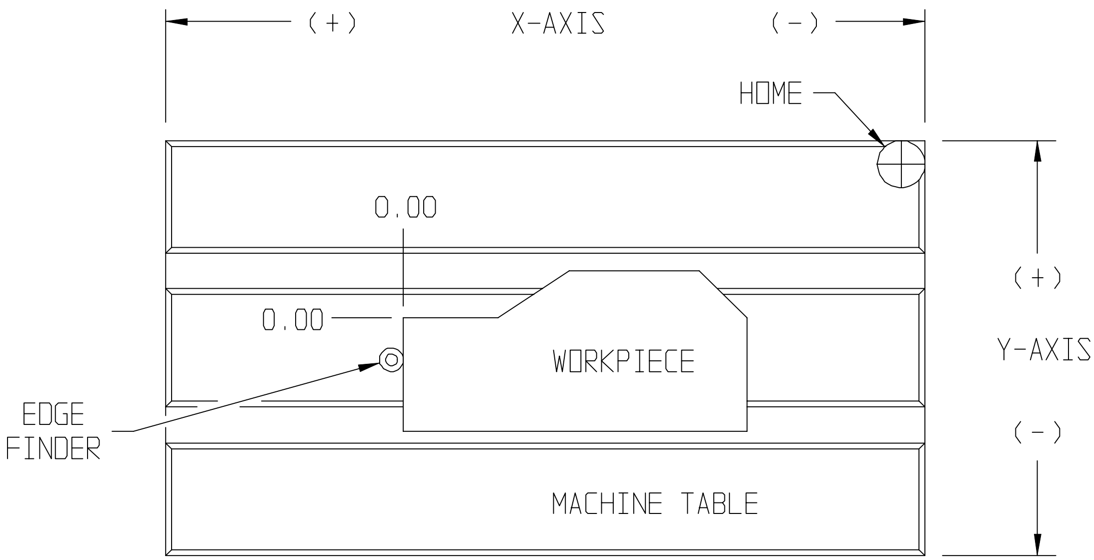
A tool length offset can also be set by entering a value into a tool offset table. The value can be measured by touching the part with the tool and reading the current position of the Z axis. If a shim is used between the tool and the part, the shim dimension should be added to the offset value. To enter the value select F7 (Parms) -F3 (Tool). Then enter the value as a negative number.

The F8 key on the handwheel screen is used to set a floating zero or work coordinate on the selected axis at the current machine position. F8 (G54-X), F8 (G54-Y), F8 (G54-Z) is the same as setting the G54 work coordinate to the current machine position.

The F9 key is used to select work coordinate systems. The F9 key will change the work coordinate on the F8 key. There are six different choices, G54-59.

Procedure for Setting a Work Offset

A work offset shifts the X and Y axis zero positions to a desired place (edge of the part). Thus a part can be programmed from its part zero. To find and set a work offset, refer to the example.



Using a ½" diameter edge finder in the X axis, handwheel or jog to the edge of the part and depress F8 (G54-X). Establish whether the edge finder is positive or negative from the desired zero. Enter -.25 for the X position. The current position will read X -.25. Repeat this procedure for the Y axis.

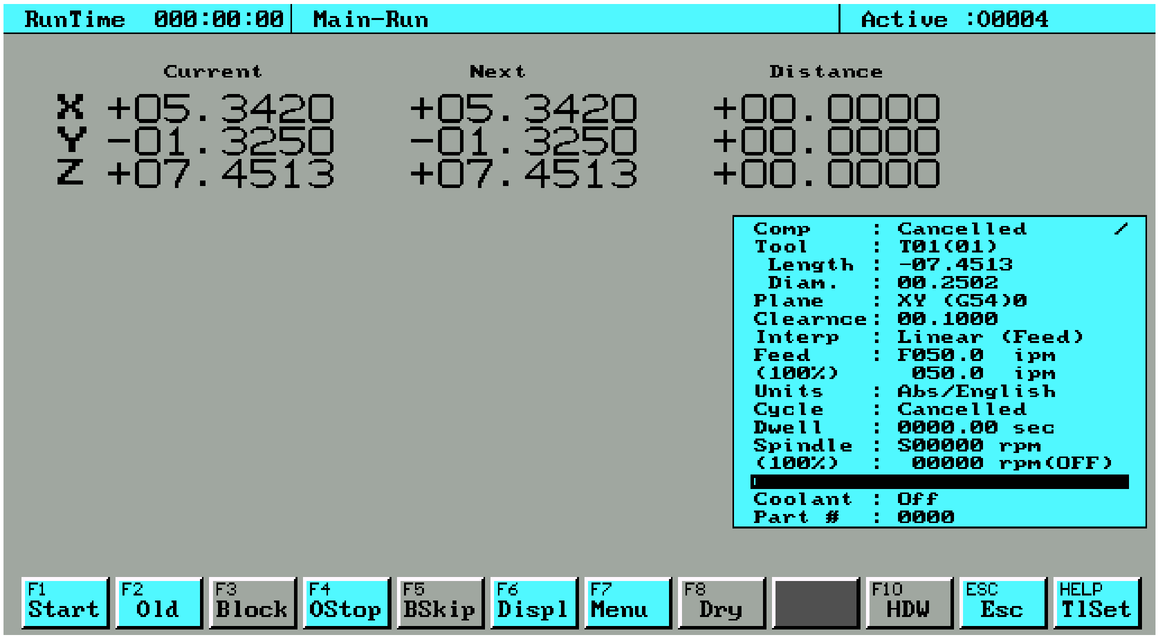
To check a work offset zero, select F5 (MDI), type G0 (rapid move) X0 Y0 (X and Y position to 0). Press **Enter**, then **Cycle Start**. Machine will position to the current work offset.

Caution: Machine will move in rapid mode. The tool should be above all parts, vises, etc.

For more information on floating zero, see page 294, Section 4.

F4 (Run) Main-Run

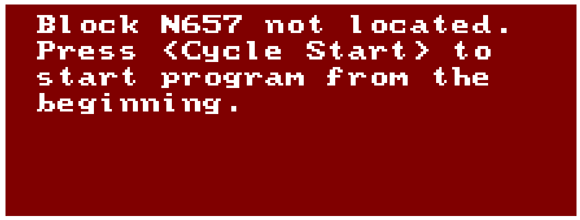
(The machine must be homed prior to running a program)
The F4 (Run) key is used to execute the active program. Upon depressing the F4 (Run) key, the following screen appears.



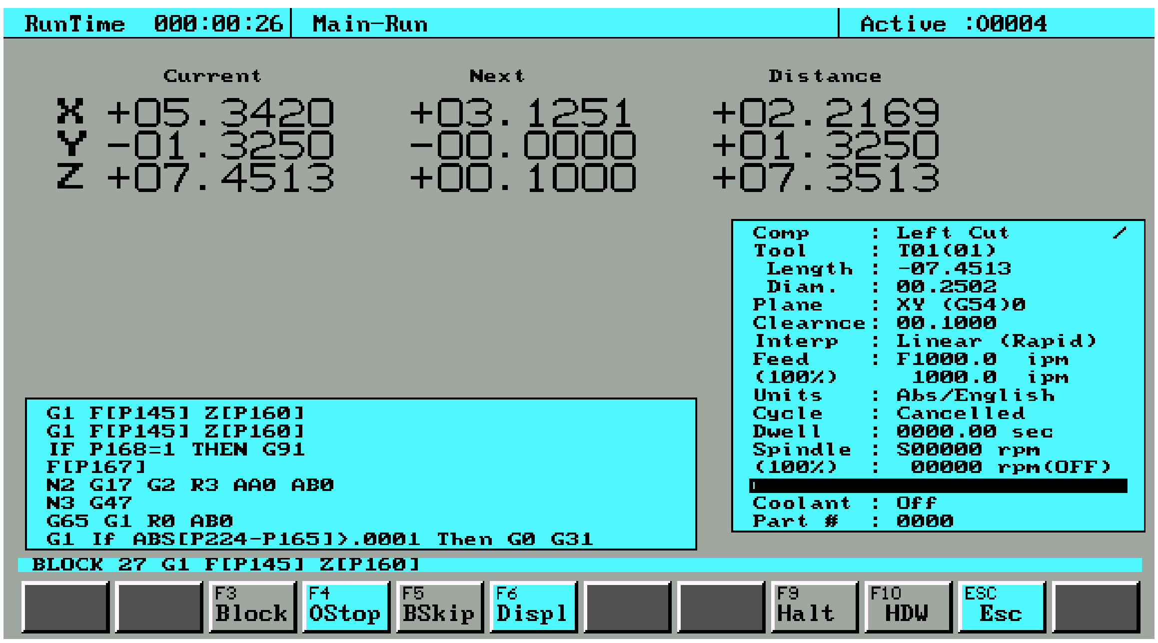
After the above screen appears, F1 (Start) must be pushed and the following screen will appear. *(Note: if the machine is equipt with an automatic tool changer, the operator will be requested to verify the tool number that is in the spindle)*

F1 (First) is automatically selected when this screen is displayed. Therefore, if one desires to run the active program from the beginning, one need only depress **Cycle Start**. If F2 (Block) is pushed, the control will request that the desired block or sequence number be typed in, followed by **Enter**. If **Cycle Start** is depressed, the active program will start running from the selected block number. If F3 (Tool) is depressed, the control will request a tool number. After typing the tool number followed by an **Enter**, **Cycle Start** is pressed, and the active program will start running at the desired tool number and the following screen will appear.

Note: If the block number or tool number requested is not found in the active program, the following window will appear.



Cycle Start *will start the program from the beginning.*



This screen is the basic run screen with two new additions: a block number display and F9 (Halt). The block number shows the current line being executed as the program runs. The F9 (Halt) key is similar to **Feedhold** in that when it is pushed the machine will stop. However, unlike **Feedhold**, F9 (Halt) also can exit the Run mode and allow a new program to be started.

F9 (Halt/Resum) Main-Run-Halt/Resume

If a program has been halted, the resume feature of the control becomes active. The F9 (Resum) key will now be displayed on the Run screen. A program can be resumed as long as one of the following functions is not performed: F9 (Verf), F5 (MDI), F1 (Home), or Emergency Stop. F9 (Resum) is also canceled if any parameters are modified. If no other function is performed, the axes can be jogged or handwheeled away from the work, the spindle may be turned on/off, and F9 (Resum) remains active. As long as the Resume is active, the F9 key on the Run screen will show a Resume function. If the Resume function is selected, the active program will be resumed at the halted point. First, Z will retract to the tool change position–all the way up–when a Resume Cycle Start is executed. Second, X and Y will rapid to the halted point. When X and Y are in position a Cycle Start will be requested. When **Cycle Start** is pressed, the Z axis will rapid to the R plane; it will then feed to its previous depth. The program will then start running as if nothing happened. **Feedhold** simply stops axes motion until it is pressed again and **Cycle Start** is pushed.

F2 (Old) Main-Run-Old

F2 (Old) will allow entry – from the message window – to an existing text program. After the number has been entered, the control will check the text programs to see if a program by that number is there. If one exists, that program will become the active program. If not, a message stating that the program was not found will appear. After pressing the ESC (Exit) key, another number may be entered.

The F3, F4, F5, and F8 keys on the Run screen set the mode of operation for the program. When these keys are in a highlighted state, the functions will be active in any program currently running or in any program to be executed.

F3 (Block) Main-Run-Block

When the F3 (Block) switch is activated, the program will stop at the end of each block. Each time **Cycle Start** is pushed, one more block will be run.

F4 (OStop) Main-Run-OStop

When the optional stop switch F4 (OStop) is activated, the program will stop at each M01 command. When **Cycle Start** is depressed, the program will continue to run.

F5 (BSkip) Main-Run-BSkip

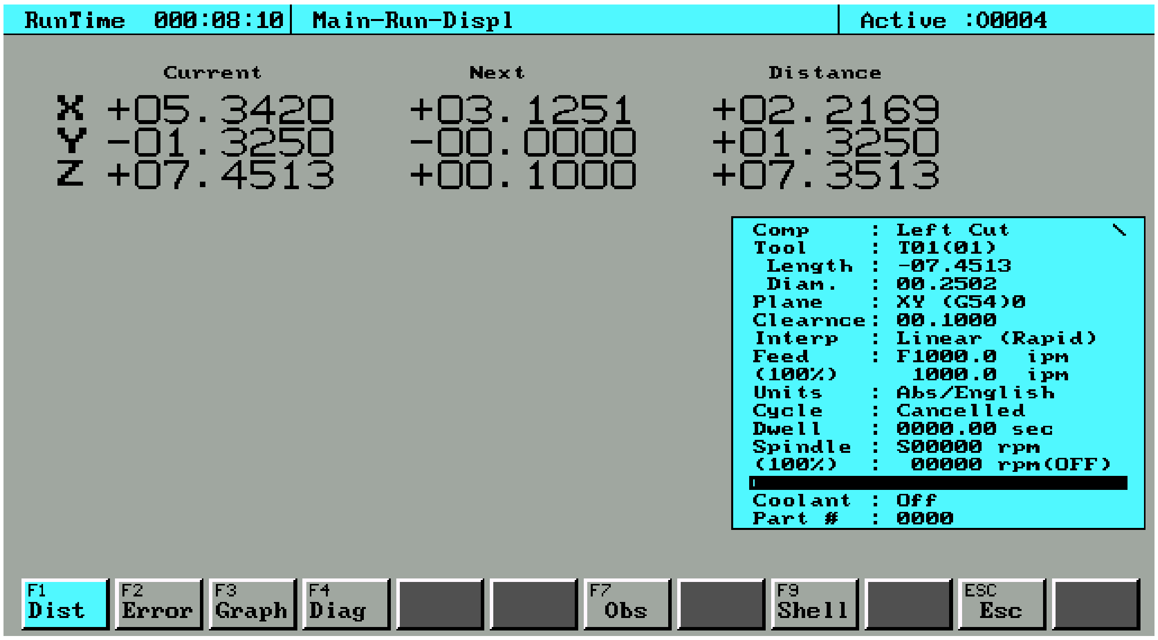
When the block skip switch F5 (BSkip) is activated, the program will skip all blocks starting with a / (slash).

Example: / M5

When the block skip switch is active, M5 (spindle off command) will not be executed.

F6 (Displ) Main-Run-Displ

The F6 (Displ) key can be accessed from a number of screens. The following screen is shown as though the F6 (Displ) was entered from the RUN screen. All the display functions and screens are identical, independent of the entry point. Only the return point differs based on the original entry point. When the F6 (Displ) key is depressed, the following screen will appear.



Note: F7 and F9 only come up in protected modes.

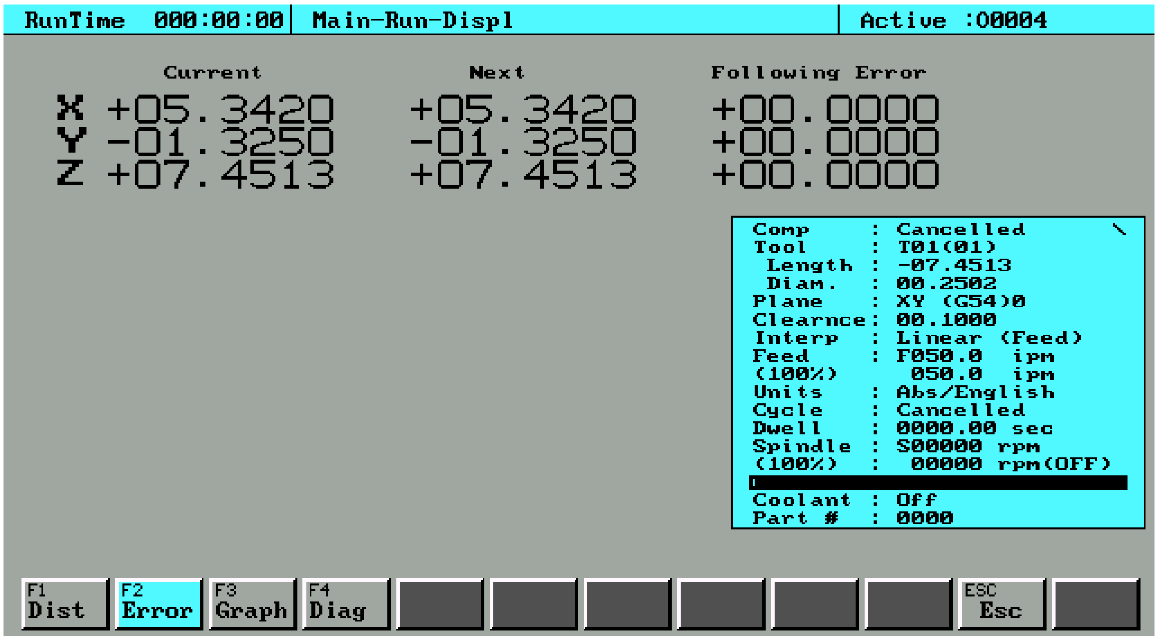
F1 (Dist) Main-Run-Displ-Dist

When the F1 (Dist) key is activated, the display shows the current position, next position, and the distance to go.

F2 (Error) Main-Run-Displ-Error

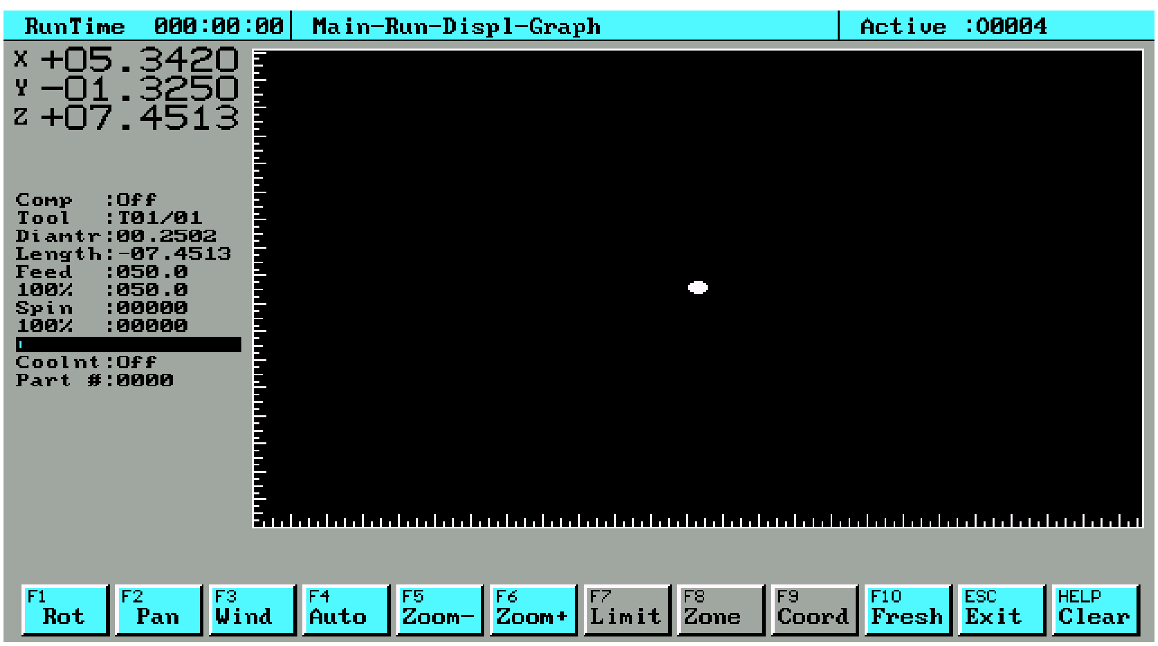
The Following Error refers to the lag in the servo system.

The F2 (Error) key changes the display to read current position, next position, and Following Error. The Following Error display is intended to help in machine setup or troubleshooting an axis problem. When F2 (Error) is pressed, the following screen will appear.



F3 (Graph) Main-Run-Displ-Graph

If F3 (Graph) is pressed, the control switches from displaying text to a graphic display of the active part program. The following screen will appear.



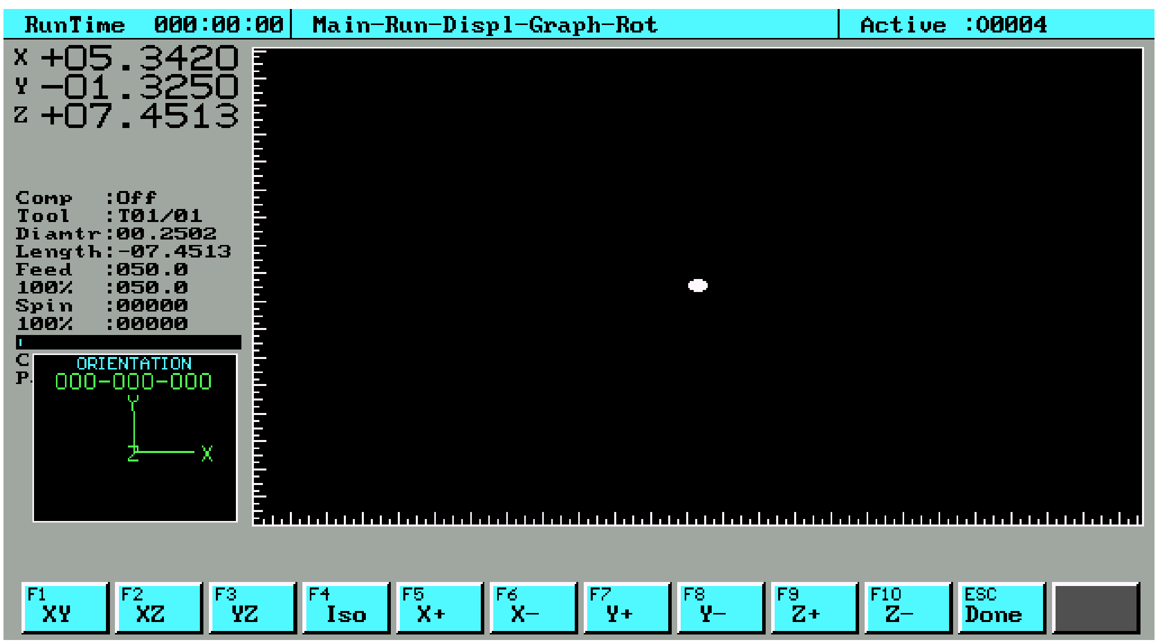
The graphics on this control are full 3D and will be displayed in the graphics area as long as the control remains in the Graph mode. When other displays are requested, windows will appear in the graphics area showing the requested data. When these functions are finished, the windows will disappear and the graphic display will be reinstated. The scales at the bottom and left side of the screen are to be used as a reference for the part size. As the screen scale is changed, the graduations on the rulers will change accordingly. The ruler graduations are in machine units; on inch system the largest graduations equal approximately one inch. In metric the largest graduations are 10mm. The Runtime at the top of the screen is basically a stopwatch that starts when a Run Program command is executed and stops at the end of program or when the program is aborted. The total job time is kept in program parameter #369 “Job Time”.

The next section will explain how to manipulate the part displayed in the graphics area. All the following functions are accessible through the Displ-Graph screen.

Note: The graphics are cleared from the display at the start of running or verifying a program.

F1 (Rot) Main-Run-Displ-Graph-Rot

When the display rotate function F1 (Rot) is selected the following screen is displayed.



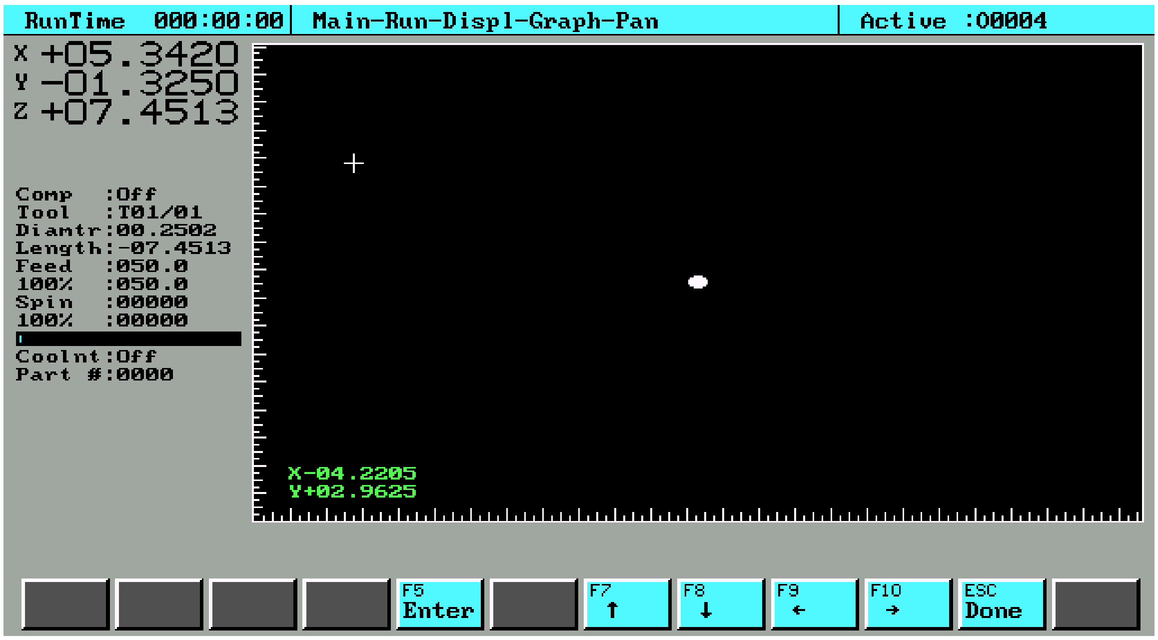
(F1 -XY, F2 -XZ, F3 -Y2, F4 -ISO)

The F1, F2, F3, and F4 keys give the four standard rotations of a part: XY top, XZ front, YZ side, and isometric views. The orientation index in the lower left corner of the screen shows the current part orientation. Depressing the F1 thru F4 keys moves the orientation index to its new position. The F5 thru F10 keys are used to rotate any of the selected axes in 5° increments. Again, as the key is pushed, the orientation index rotates indicating the orientation of the part display. F5 and F6 rotate X axis ±, F7 and F8 rotate Y axis ±, and F9 and F10 rotate Z axis ±.

Note: The display is auto-scaled when the new orientation is displayed.

F2 (Pan) Main-Run-Displ-Graph-Pan

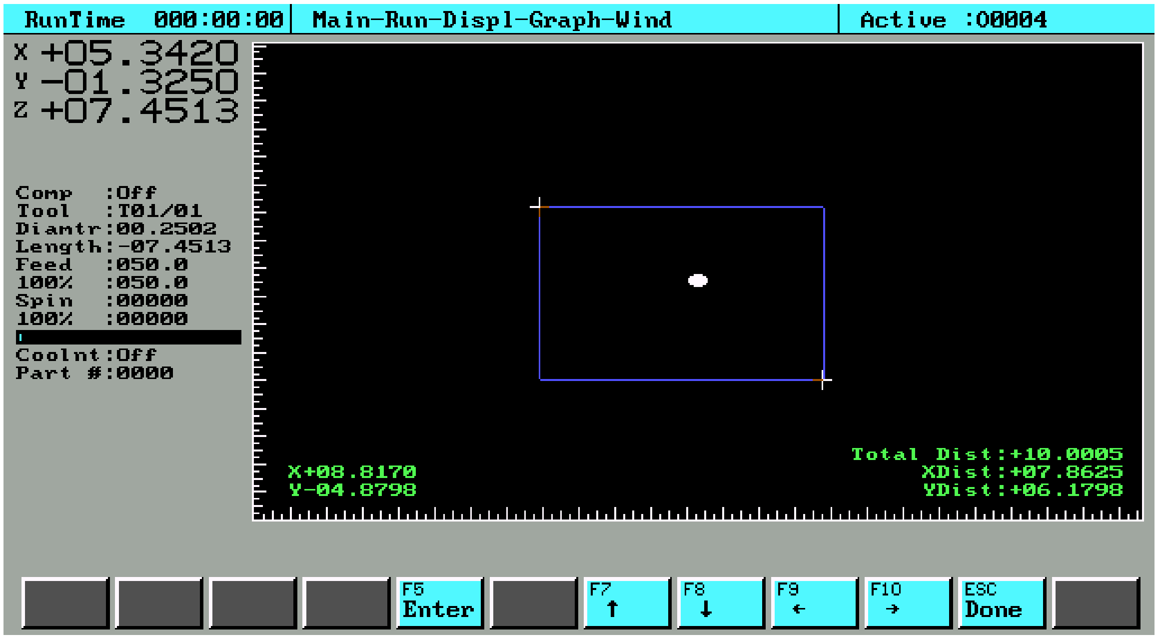
The F2 (Pan) key selects the pan function, which allows the operator to pan around a part. The following display will appear.



The crosshair which appears on the screen can be moved around using the numerical keys or arrow keys F7 thru F10. To pan, move the cursor to the point on the display that is the desired center of screen and push F5 (Enter), or **Enter** on the keyboard. The display will shift to its new position and panning may resume. The ESC key will cancel the pan function and return to the Graph screen.

F3 (Wind) Main-Run-Displ-Graph-Wind

The F3 (Wind) key selects the window function which allows the operator to window in on a particular area of the part. The following display will appear when F3 (Wind) is selected.



The crosshair which appears on the screen can be moved around using the numeric keys or arrow keys F7 thru F10 (the same as pan). To zoom in using a window, move the cursor to the first corner of the window area and depress F5 (Enter) or **Enter**. Then move the cursor around until the desired area of the part to be viewed is enclosed in the rectangular box being drawn on the screen. Next, press **Enter**. The area enclosed in the box will now be displayed on the entire screen. After depressing **Enter** to window on the part, window cancels itself and returns to the graphic screen.

The X Y coordinates on the lower left side of the screen indicate the position of the cursor, relative to part zero. The X Y coordinates on the lower right side of the screen indicate the size of the window. These numbers can be used as a quick reference of the part size and dimensions.

F4 (Auto) Main-Run-Displ-Graph-Auto

The F4 (Auto) key selects the auto zoom function. This function automatically scales and centers all machine parts on the screen.

F5 (Zoom-) Main-Run-Displ-Graph-Zoom-

The F5 (Zoom) key selects the zoom- function which decreases the size of the part currently being displayed on the screen by half. Generally, this function is used to view a larger portion of the part.

F6 (Zoom+) Main-Run-Displ-Graph-Zoom+

The F6 (Zoom) key selects the zoom+ function which doubles the size of the part being displayed on the screen. Generally, this function is used to enlarge a specific area of a part enabling the operator to see greater detail.

F7 (Limit) Main-Run-Displ-Graph-Limit

The F7 (Limit) key draws a box on the screen which corresponds to the axis limits of the machine. This allows viewing of the part in relation to the machine's overtravels. If the part extends beyond this box, it cannot be run on the machine unless some corrective action is taken to change the work offsets. The axis overtravel limits are set from the parameter screens. If the tool is programmed outside this box, an *"axis software limit overtravel"* error will result.

Note: Part zero and tool length must be set to show true position within limits.

F8 (Zone) Main-Run-Displ-Graph-Zone

The F8 (Zone) key draws a box on the screen which corresponds to an axis safe zone. This is a zone which the tool cannot enter. If the tool is programmed into this box with the safe zone turned on (G23), an *"attempted to move into safe zone"* error will be generated. G22 shuts off the safe zone.

F9 (Coord) Main-Run-Displ-Graph-Coord

The F9 (Coord) key draws an axis coordinate through X0, Y0, Z0. This give a visual reference to where the zero is on the part.

F10 (Fresh) Main-Run-Displ-Graph-Fresh

The F9 (Fresh) key redraws the currently displayed part on the screen.

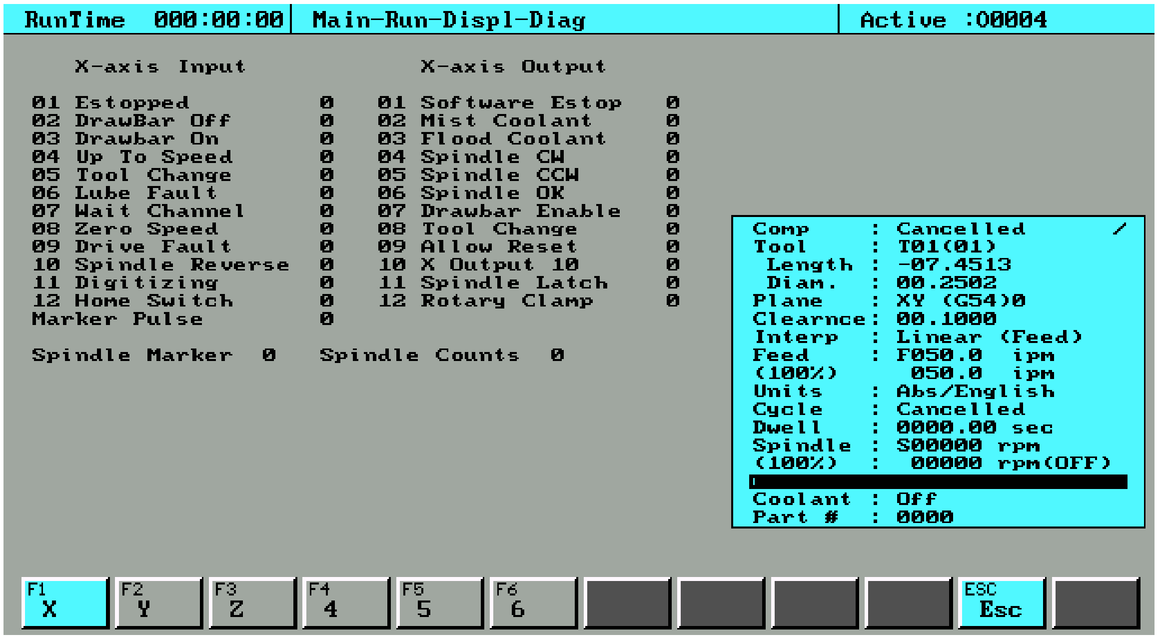
Help (Clear) Main-Run-Displ-Graph-Clear

The F10 (Clear) key clears the current display buffer. After the clear screen command is executed nothing will be displayed until either the program is verified or run again.

F4 (Diag) Main-Run-Displ-Diag

The F4 (Diag) key is mainly used for machine setup or troubleshooting machine functions. The diagnostic screens bring up all the external I/O bits connected to the CNC. The status of each bit is continuously displayed on the screen, and as they change on the machine, the screen will be updated. Function keys F1 thru F5 display the various axes enabled on the CNC. Selecting X, Y, Z, etc. brings up the appropriate axis I/O channels. A "0" displayed on the screen means the I/O channel is not active. A "1" indicates active.

The following screens represent the displayed information for the various axis selections.



Note: The diagnostic screens will differ for machines that have varying options. The text that shows up on the screen is from the files INP.XXX and OUT.XXX, where XXX is the extension for the desired language to display on the screen. The INP.XXX and OUT.XXX files for a basic machine with no options are present in the ROM directory. The files that correspond with the options that a machine might have will be present in the RAM directory. The control will first check the RAM directory. If the files are not found in the RAM directory, the control will use the default files in the ROM directory.

Each axis has inputs and outputs, pressing F1…F6 will show the inputs and outputs for the associated axis.

Note: Machines with tool changers, etc. will have different diagnostics screens.

F7 (OBS) Main-Run-Displ-Obs

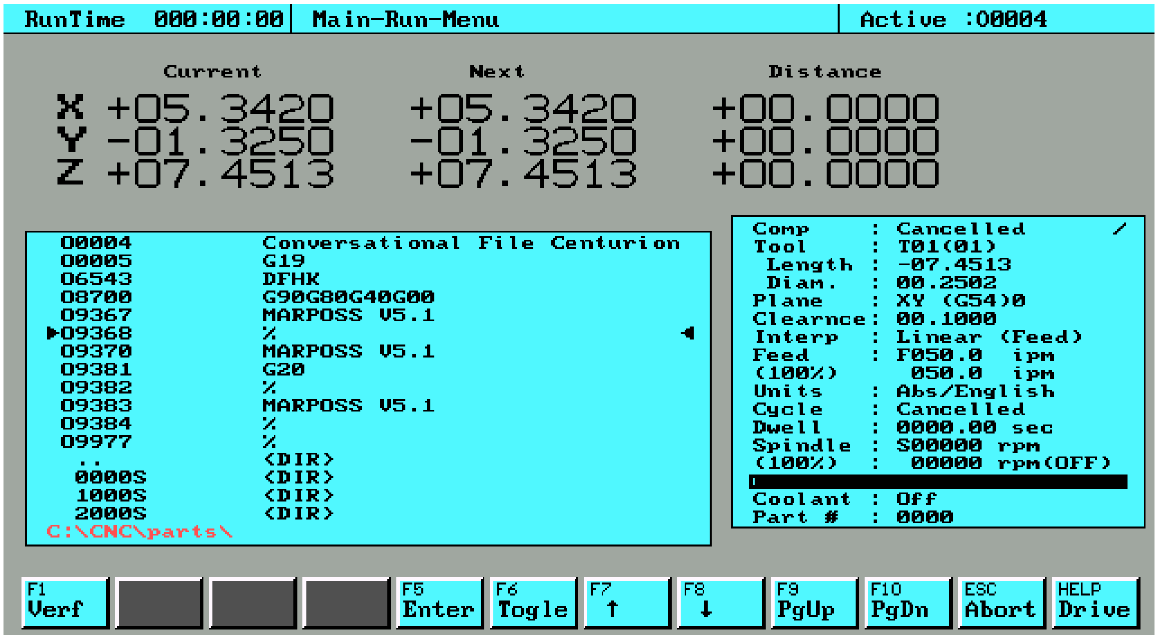
The F7 (OBS) key will only display in access level 2 or greater. F7 (OBS) will be highlighted and is only available if the current display is F1 (Next), F2 (Dist), or F5 (Error). It will display information on the motion control cards. F7 (Obs) comes up in protected modes only.

F9 (Shell) Main-Run-Displ-Shell

F9 (Shell) is available if the control is in access level 2 or greater. It will Shell out to DOS. To return to the CNC, type *exit*. F9 (Shell) comes up in protected modes only.

F7 (Menu) Main-Run-Menu

The F7 (Menu) key selected from the Run or Verify screen brings up a window containing a listing of all the available programs, which may be run. The screen shown below will be displayed when the program menu is requested.



To activate one of the programs listed in the window, use the arrow and page keys to move the cursor to the desired program and press F5 (Enter), or **Enter** on the keyboard. The Menu function can be called from other screens but works the same way from all. When called from the Run or Verify screen, the selected program becomes the active program. When called from an edit screen, the selected program becomes the current program being edited. F1 (Verf) will graphically verify the part the curser is on. F6 (Togle) will display the file size, last time and date it was edited, and seven characters for the file name. Subdirectories are specified by <DIR> as the file name. The ".." specifies the parent to the current directory. If you press the HELP (Drive) key, a list of the available drives is displayed.

Note: The file name is enclosed in parentheses. If there are no parentheses on the first block, the file name is the first block. Pressing the "D" key toggles the menu to and from "Full Dos Filenames". Pressing the "N" key toggles sorting files by filename and file description.

F8 (Dry) Main-Run-Dry

When the F8 (Dry) is active, all program feedrates will run at the dry run feedrate.

Note: The spindle commands can be ignored during dry run by setting a parameter.

Switching to or from Dry Run cannot be done if the program is running. The machine must first be F9 (Halt)ed, switched to or from F8 (Dry) Run, and then F9 (Resum)ed. The F8 (Dry) key will not be visible if the program is running.

F10 (HDW) Main-Run-HDW

When the F10 (HDW) key is activated, the axis moves in the program will relate to turning the electronic handwheel. The feedrates (feed/rev. or feed/min.) will effect the distance moved per click of the handwheel. The four miscellaneous parameters that effect the distance moved per click of the handwheel are listed below.

1. Handwheel Encoder PPU
   Pulses per rev of the electronic handwheel. This should be 400 for our current systems.

2. Cranking Minutes/Rev
   Total minutes of normal program execution time should correspond to one turn of the handwheel. Multiplying an IPM feed by this factor results in an IPR feed (inches per turn of the handwheel). This should be about 0.0010 for our current system.

3. Cranking Max IPM
   This limits the feedrate while hand-cranking to get reasonable response at slow programmed feedrates. Adjusting the other handwheel parameters can give excess errors on rapids above 100 IPM. It should be about 100 for our current system.

4. Cranking Factor
   The multiplier for each handwheel click should be about 100 for our current system.

Cranking Factor/Cranking Mins Per Rev is proportional to the max feedrate allowed while hand-cranking.

If handwheeling and dry running a program, the distance moved per click of the handwheel relates to the Dry Run Feed parameter.

When switching from the handwheel mode to automatic mode, the operator will be prompted to press cycle start.

If operator is handwheeling a program and a tapping cycle is started, the message *"Tapping cycle will not be in handwheel mode. Press any key to continue"* will appear. The machine will tap the hole and return to the handwheel mode.

Help (TlSet) Main-Run-TlSet

When no program is running, the Help (TlSet) key will be the tool setting key. Help (TlSet) is similar to F1 (Start). When pressed, a Cycle Start will be requested.

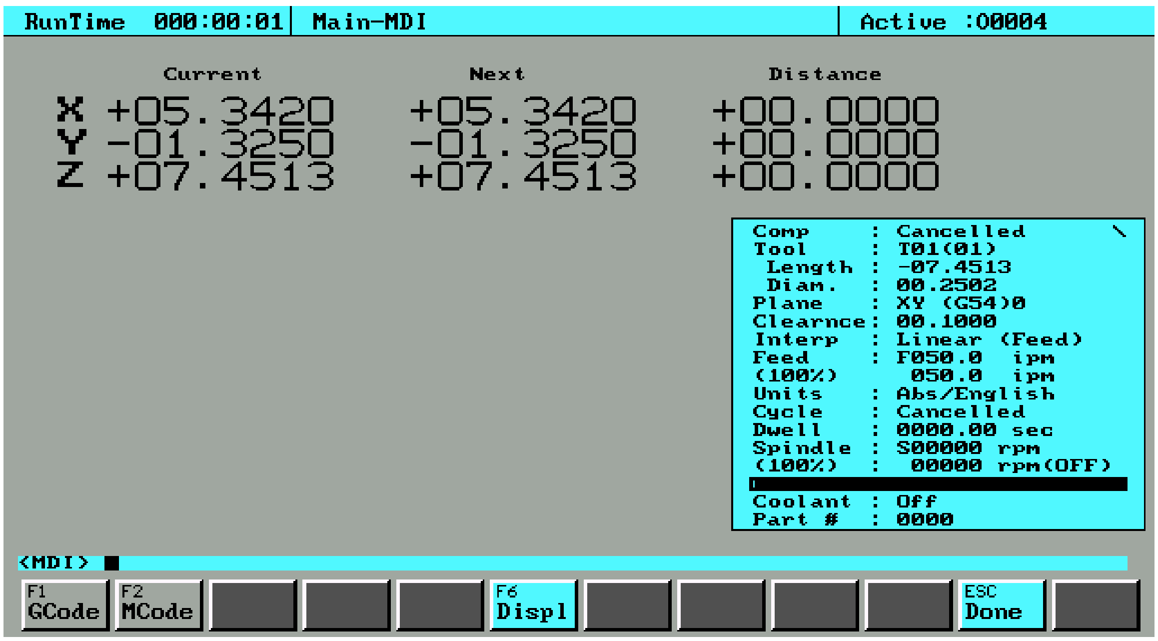
1. The operator is prompted for the height of the setting gauge.
2. The operator is prompted for the tool number to set the offset. The machine does a tool change to this tool number.
3. Operator handwheels the tool tip to the gauge or face of the part. He then presses **Enter**.
4. Operator is prompted for the tool radius.
5. Go to number 2.

During the tool setting routine, the tool table is loaded with the appropriate values. After all the tool offsets are loaded, the operator can press ESC (Halt) to exit the tool setting routine.

Note: This routine can be modified for specific applications (auto tool setters, different tool changers etc.).

F5 (MDI) Main-MDI

The F5 (MDI) key on the Main menu selects the MDI (manual data input) function. Through MDI any programmable machine function can be executed one function or one block at a time. When MDI is selected the following screen appears.



As the functions are typed in they will appear on the MDI line at the bottom of the screen. After the data has been typed in, **Enter** must be pressed to end the block. At this point a **Cycle Start** will execute the MDI line. During MDI the ESC (Esc) key will terminate any MDI command. F6 will bring up the normal graphic displays discussed earlier. If graphics are turned on during MDI, the graphic display will graph all the MDI moves as they are executed.

F1 (Gcode) Main-MDI-Gcode

F1 (Gcode) will bring up a list of legal G codes with a short description of each code. Pressing the F1 (Gcode) key will bring you to the next page of G codes.

00 Linear Rapid	01 Linear Feed
02 CW Arc	03 CCW Arc
04 Dwell	09 One-Shot Exact Stop
10 Set Data Mode	11 Clear Data Mode
12 Clear Floating Zero	17 XY Plane
18 XZ or ZX Plane	19 YZ Plane

20 English	21 Metric
22 Safe zone check off	23 Safe zone check on
24 Circ Pocket Clear	25 Circ Finish Inside
26 Circ Finish Outside	28 Reference Return
29 Return From Ref	30 2nd-4th Ref Return
31 Z to Clearance	32 Z to Tool Change
33 Facing Cycle	34 Rect Pocket Clear
35 Rect Finish Inside	36 Rect Finish Outside
39 Threading Cycle	40 CutterComp Off
41 Left CutterComp On	42 Right CutterComp On
43 + H Offset Dir	44 - H Offset Dir
45 Left Auto Comp On	46 Right Auto Comp On
47 Auto Comp Off	49 Cancel Tool Offsets
50 Scaling Off	51 Scaling On
52 Local Coordinate	53 Machine Coordinates
54 Worksystem 1	55 Worksystem 2
56 Worksystem 3	57 Worksystem 4
58 Worksystem 5	59 Worksystem 6
60 One-Shot Rapid Move	61 Exact Stop Mode
63 Tapping Mode On	64 Tapping Mode Off
65 Move Lockout Block	68 Rotation On
69 Rotation Off	70 Mirror Image Off
71 Mirror Image On	72 Bolt hole
73 Woodpecker Drill	74 Left Tap
76 Fine Bore	78 Manual Bore
80 Canned Cycle Cancel	81 Drill
82 Drill/Dwell	83 Peck/Drill
84 Tap	85 Bore
86 Fast Bore	87 Back Bore
88 Hard Tap (option)	89 Bore/Dwell
90 Absolute	91 Incremental
92 Set Floating Zero	93 Inverse Time Feed
94 Inches per Minute	95 Inches per Revolution
98 Ret to Initial Level	99 Return to R-plane

Note: The text file that displays the legal G codes on the screen is Gcodes.XXX, where XXX is the extension for the corresponding language. The control first looks for Gcodes.XXX in the RAM directory. If not found, it then searches the ROM directory for the file.

F2 (Mcode) Main-MDI-Mcode

F2 (Mcode) will bring up a list of legal M codes with a short description of each code.

00 Program Stop	01 Optional Stop
02 End of Program	03 Spindle On CW
04 Spindle On CCW	05 Spindle Off
06 Tool Change	07 Mist On

08 Flood On	09 Coolant Off
30 Spindle Off, End of Program	90 Graph Off
91 Graph On	93 3D Sweep Off
94 3D Sweep On	95 Tapered Wall
96 Rounded Wall	97 Pocket Clear
98 Call Jump	99 End of Program

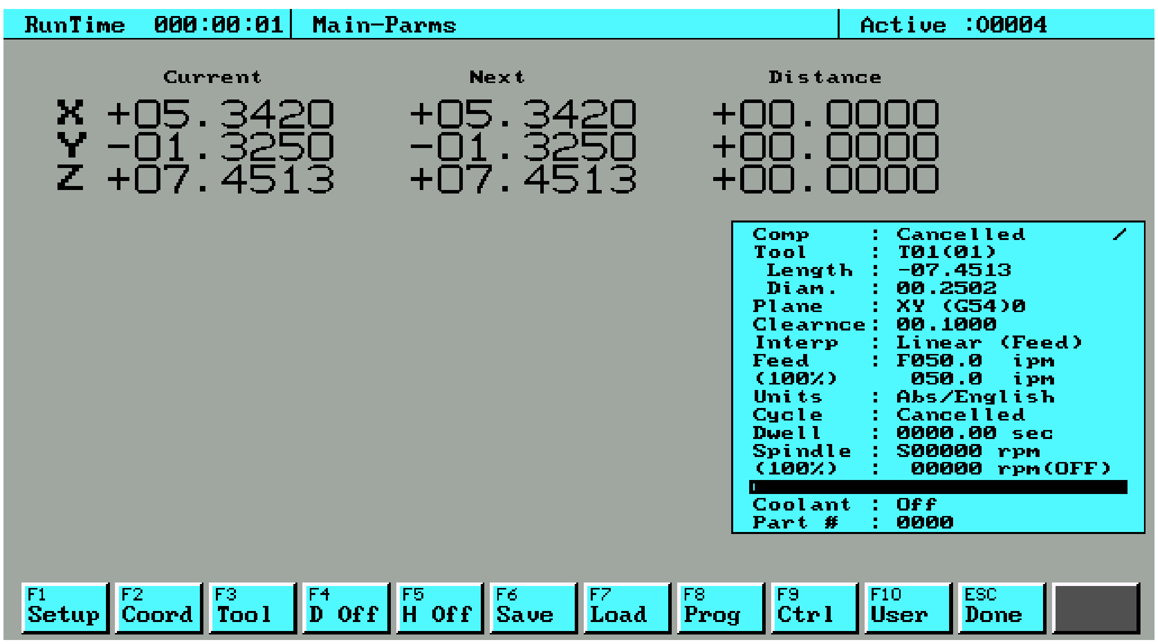
Note: The text file that displays the legal M codes on the screen is Mcodes.XXX, where XXX is the extension for the corresponding language. The control first looks for Mcodes.XXX in the RAM directory. If not found, it then searches the ROM directory for the file. Depending upon the machine options, the legal M codes will vary. The Mcodes.XXX in the ROM directory is for a basic machine with no options.

F6 (Displ) Main-Displ

See explanation under F6 (Displ) Main-Run-Displ on page 23, Section 2. This function can be entered from either screen.

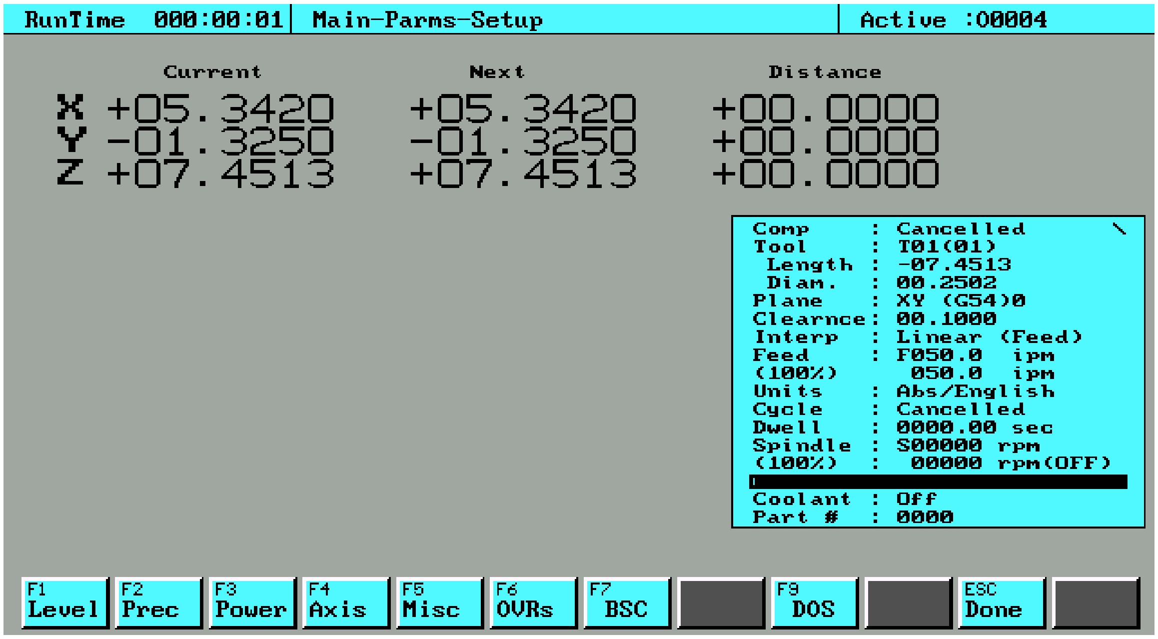
F7 (Parms) Main-Parms

The F7 (Parms) key from the main screen brings up the following parameter screen.



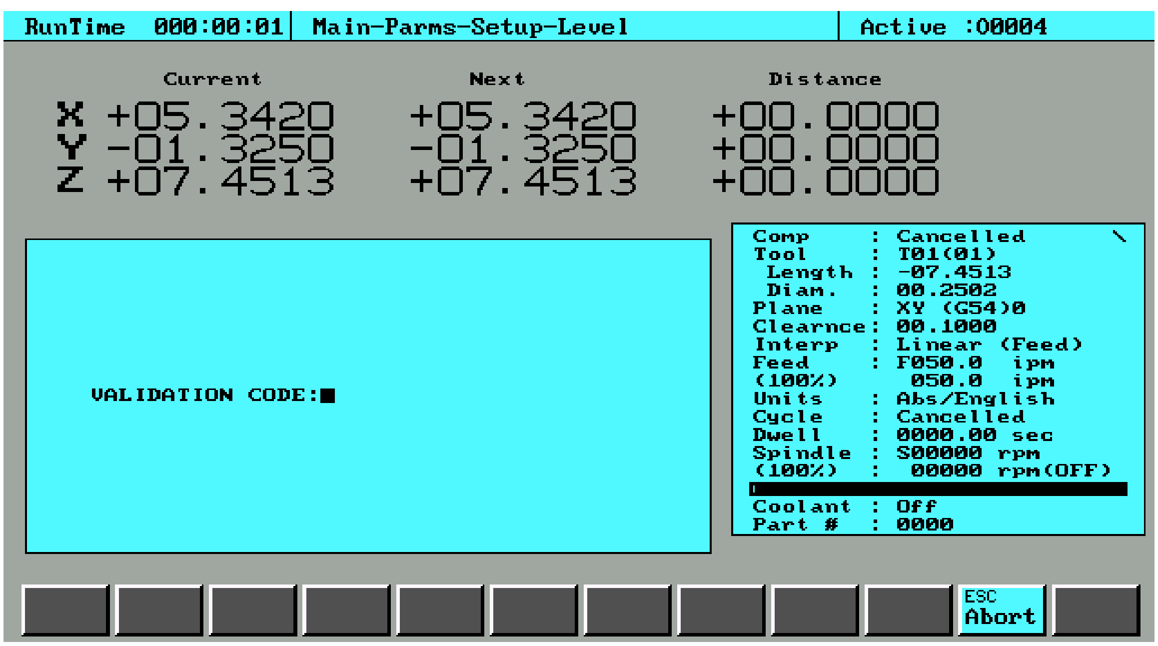
F1 (Setup) Main-Parms-Setup

The F1 (Setup) selection brings up the parameters, which make the control unique to a particular machine or application. When F1 (Setup) is selected the following screen appears.



Note: F2 (Prec) through F9 (DOS) are only displayed if the control is in the protected mode set by a validation code.

The CNC requires a VALIDATION CODE and an ACCESS LEVEL number to allow the machine setup parameters to be displayed or changed. The machine tool builder can supply the validation code and access level.

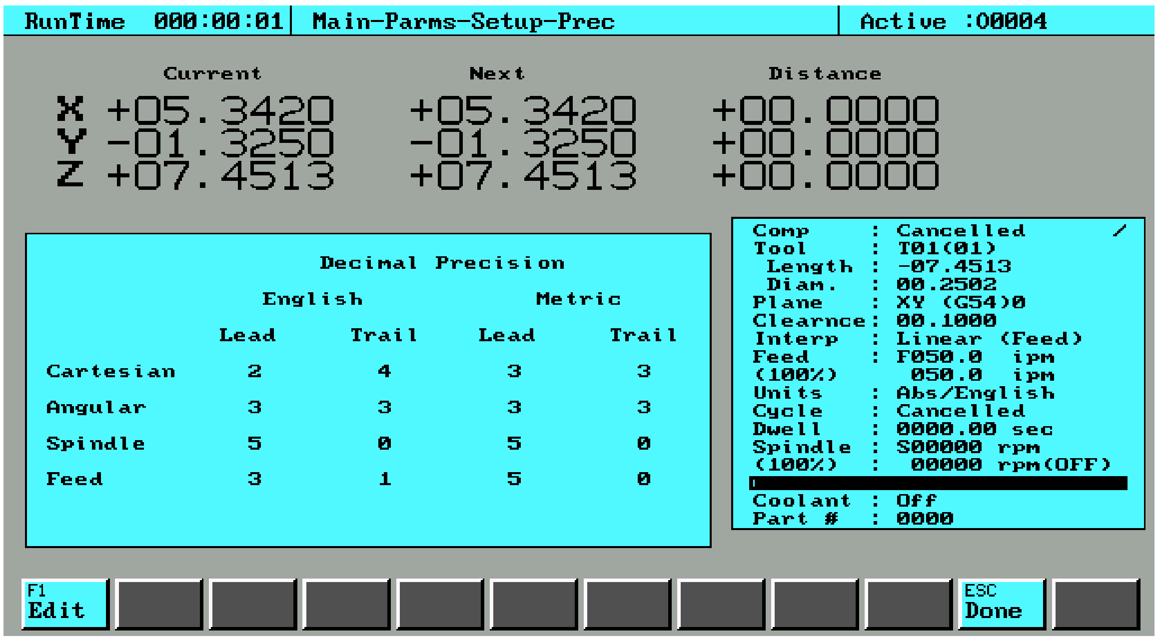


Note: The parameters in the setup sections are normally set by the machine tool builder. Changing these parameters can affect a large number of machine functions and machine performances and should only be modified by experienced service personnel.

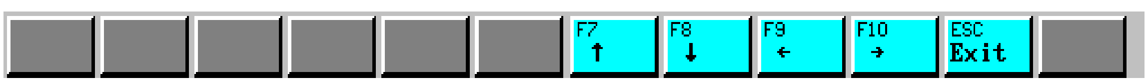
Assuming the proper codes have been entered, the following screens can be selected.

F2 (Prec) Main-Parms-Setup-Prec

If the F2 (Prec) selection for Machine Precision is made, the following screen will be displayed.



Keys displayed in the Edit Mode:



The above screen shows some typical settings for leading and trailing zeroes for the different coordinate systems. The number of leading and trailing zeroes is unlimited, but some practical limits do exist. If the numbers get too large, they will not fit on the screen in their allotted space. If they are smaller than the feedback units, they will not cause movement. To change a parameter press F1 (Edit). A series of arrow keys will be displayed; use them to move the cursor to the desired parameter and type in the new number. Once all the numbers have been edited, pressing the ESC (Exit) key will validate the new numbers and return to the previous screen. These parameters are for all numbers entered into the control except for the axis parameters, which are set separately in the F4 (Axis) parameters.

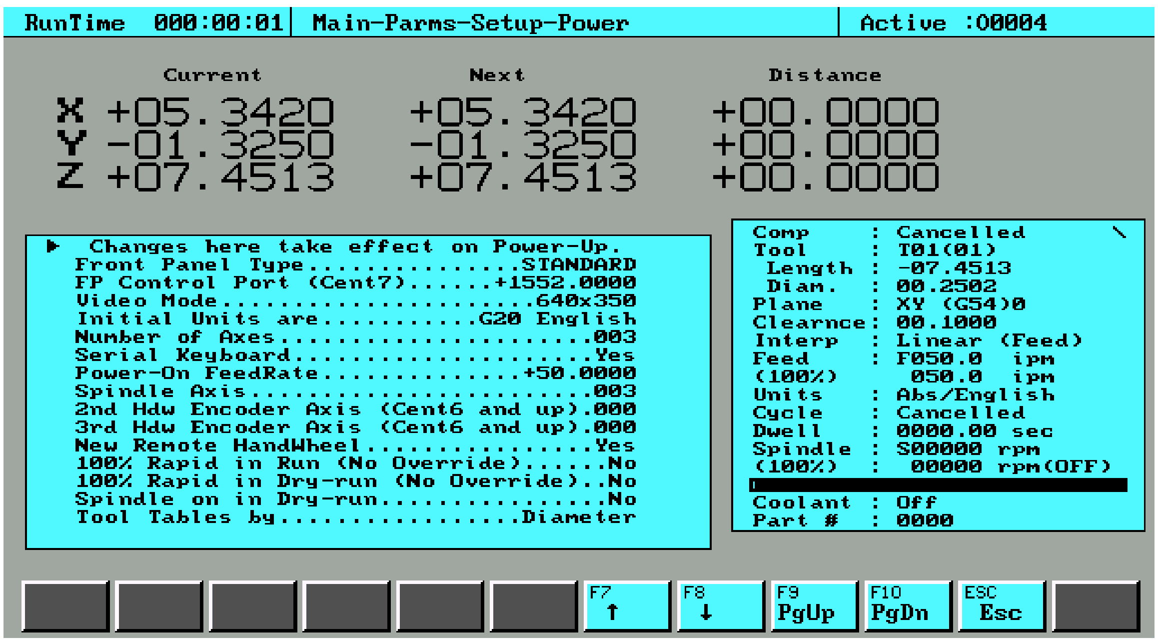
F3 (Power) Main-Parms-Setup-Power

Power parameters are parameters that directly relate to the configuration of the machine tool and will normally be set by the machine tool builder.

Note: When editing or entering parameter values (or any other numeric value on the control), you can use the built in calculator.

Example: *Instead of entering .3750 you may enter 3/8*
Instead of entering 1.3750, you may enter 1 + 3/8
If you want to modify the current value, you may use "." as the current value.
If the current value is .358 and you want to add .002, type .+.002 (instead of entering .360).

The F3 key brings up the power-on defaults as shown in the following screen.



Front Panel Type: Standard or Mini-Series Membrane

FP Control Port (Cent 7): Front Panel port 1552 for standard Cent 7 systems

Video Mode: Standard settings are 640 x 350 for CRTs and 800 x 600 for LCDs

Initial Units are: G20 English or G21 Metric to power up in inch or metric.

Number of Axes: Can be 1 to 5.

Serial Keyboard : Enables or disables serial keyboard

The keyboard will boot up transmitting over the CLK/DATA interface and not over the RS-232 interface. This means the PC will be able to recognize that a keyboard is present upon reset and not hang with a message such as, <PRESS F1 TO RESUME>. The keyboard will always recognize commands over either the RS-232 or CLK/DATA interface. Whenever a valid command with no error is received over the RS-232 interface, it will start transmitting over the RS-232 interface and not the CLK/DATA interface.

In like fashion, it is possible to switch back to the CLK/DATA interface. Whenever a valid command with no error is received over the CLK/DATA interface, it will start transmitting over the CLK/DATA interface and not the RS-232 interface.

Since the RS-232 interface is not as interlocked as the CLK/DATA interface, the following additional requirements are placed on the RS-232 protocol.

1. The system is allowed to send commands only in response to transmissions set by the keyboard.

2. When the keyboard transmits it will wait for the transmission to be complete and then delay another two milliseconds before again checking the keystroke buffer. This delay allows the system time to send a resend command or other command such as turning on a light. Since transmitting a character takes approximately one millisecond at 9600 baud, this allows about 1 millisecond processing time for the system.

3. To allow the system a chance to send out commands, the keyboard will send out a strobe signal periodically. The strobe signal is simply the character EAh. This character will be transmitted if the keyboard is otherwise idle for approximately 30 milliseconds. The system can then send out a command in response to the strobe instead of having to wait until the next keystroke.

Hardware and Software Requirements

The serial keyboard requires a REV E keyboard controller card with firmware version #1.21 or greater. Firmware for the REV E cards cannot be used with older revision cards and vice versa.

CNC software must be of version # *.86 or greater.

The serial keyboard uses COM2. Therefore, if the serial keyboard is used, the secondary serial port parameter should be set to *none*. The secondary serial port parameter may be removed in a future version.

On machines set up to use the serial keyboard there will be a revised cable in the mag box. Previously, the cable coming in was split in two and ran to the video card and to the controller card. Now the cable will split in three and have a 9-pin D-Sub connector to attach to the I/O card. The connector on the I/O card should be set up to be COM2 (as shipped to us, it is usually COM1).

Enabling the Serial Keyboard

The CNC program will use the serial keyboard interface if the power-up parameter Serial Keyboard is set to *Yes* and if the program is executed without any command line options.

Alternatively, the CNC program will use the serial keyboard interface if the serial keyboard parameter is set to *Yes* and the letter *"s"* appears as an option on the command line.

The serial keyboard is enabled in the following examples if the serial keyboard parameter is also set to *yes*.

CNC	
CNC s	(serial keyboard only)
CNC fs	(front panel lights, serial keyboard)
CNC as	(acroloops, serial keyboard)
CNC afs	(same as no options)

The serial keyboard is disabled in the following examples:

CNC n	(demo mode)
CNC f	(front panel lights)
CNC a	(acroloops)
CNC af	(acroloops and front panel lights)
CNC b	(NCB, Cent 6)
CNC bf	(NCB and front panel lights)

Power-On Feedrate:	Can be any number up to the maximum feedrate.
Spindle Axis:	1 to 5 1=X, 2=Y, 3=Z, 4=A . . . etc Normally set to 3 for Z axis.
Primary HDW Encoder Axis (Cent 6 and up)	The encoder to use for the primary handwheel. 7 for standard Cent 6 systems, 8 for standard Cent 7 systems.
2nd Handwheel Encoder Axis:	Used for optional dual handwheel systems (4 if it has the option).
3rd Handwheel Encoder Axis:	Used for optional dual handwheel systems (5 if it has the option).
New remote handwheel:	*Yes*, if it has the new style remote handwheel option.
Remote HDW Encoder:	The encoder to use for the remote handwheel.
100% Rapid in Run:	*No* means the feedrate override will affect rapid moves. *Yes* means the feedrate override will not affect rapid in the run mode. Rapid moves are 100%.

100% Rapid in Dry-Run: *No* means the feedrate override will affect rapid moves in the dry run mode.
Yes means the feedrate override will not affect rapid in the dry run mode. Rapid moves are 100%.

Spindle on in Dry-Run: *No* means the spindle will not come on in the dry run mode.
Yes means the spindle will come on in the dry run mode.

Tool Tables by: Radius or Diameter

Don't Load Canned Cycles: *No* means Canned Cycles will be loaded.
Yes means Canned Cycles will not be loaded; the canned cycles (drill, pockets, frame etc.) take approximately 39K of RAM.

Block Skip On: If *Yes* the block skip will be on at power up.

Optional Stop On: If *Yes* the optional stop will be on at power up.

Safe Zone On: If *Yes* the safe zone will be on at power up.
G22 turns the safe zone off.
G23 turns the safe zone on.

Foreign Extensions: Refers to the extension for data files. Current valid extensions are listed below.

CES	CZECH
CHN	CHINESE
NED	DUTCH
DAT	ENGLISH
FRN	FRENCH
GRM	GERMAN
ITA	ITALIAN
SPN	SPANISH
TRK	TURKISH

Tool Changer Information

Machine Type : Type in VMD 30....

ATC Type is: Manual/Avanti, Milltronics, or other

ATC tool (pocket count): Number of pockets on the tool changer

Spindle orient delay(MS): Time delay for orient pin (use 0 for roller pins, or electronic orient)

M6 (Tool Change) Macro: Path and Name of the program used for custom M6

Put Pot down on swing arm tool changers:	Yes means the pot will be down for the pending tool. No means it will be up.
Check drawbar switch:	Yes for newer machines with drawbar switches.
Milltronics ATC is:	Plunger, Geneva Two-Step, Geneva One-Step or Swing Arm
Milltronics CW Coast: **Milltronics CW Brake:** **Milltronics CCW Coast:** **Milltronics CCW Brake:**	Partner 1 Geneva tool changers use these Parameters to set the times to index and brake the tool changer in the CW and CCW directions. Suggested values are 110 for coast and 60 for brake.
Check Air Pressure:	If the parameter is set to *yes* and the pressure is low (Y input #1), the machine will emergency stop and error 448 *"Air Pressure Error. Check air hose connection"* will occur.
Drawbar Delay:	Used in the Partner tool changers as a delay (in seconds) after the drawbar comes on and before the head goes up to remove the tool to prevent pulling on the tool holder.

Custom M and G code Tables

Custom G code O9010	
Custom G code O9011	
Custom G code O9012	
Custom G code O9013	
Custom G code O9014	
Custom G code O9015	
Custom G code O9016	Cannot have a custom G code for G0
Custom G code O9017	
Custom G code O9018	
Custom G code O9019	

Custom M code O9020	
Custom M code O9021	
Custom M code O9022	
Custom M code O9023	
Custom M code O9024	
Custom M code O9025	
Custom M code O9026	Cannot have a custom M code for M0
Custom M code O9027	
Custom M code O9028	
Custom M code O9029	

Parameter file version: Should always be 1

Use Small Soft keys: Should be set to *no* for 12" CRT monitors

Notes on Parameters

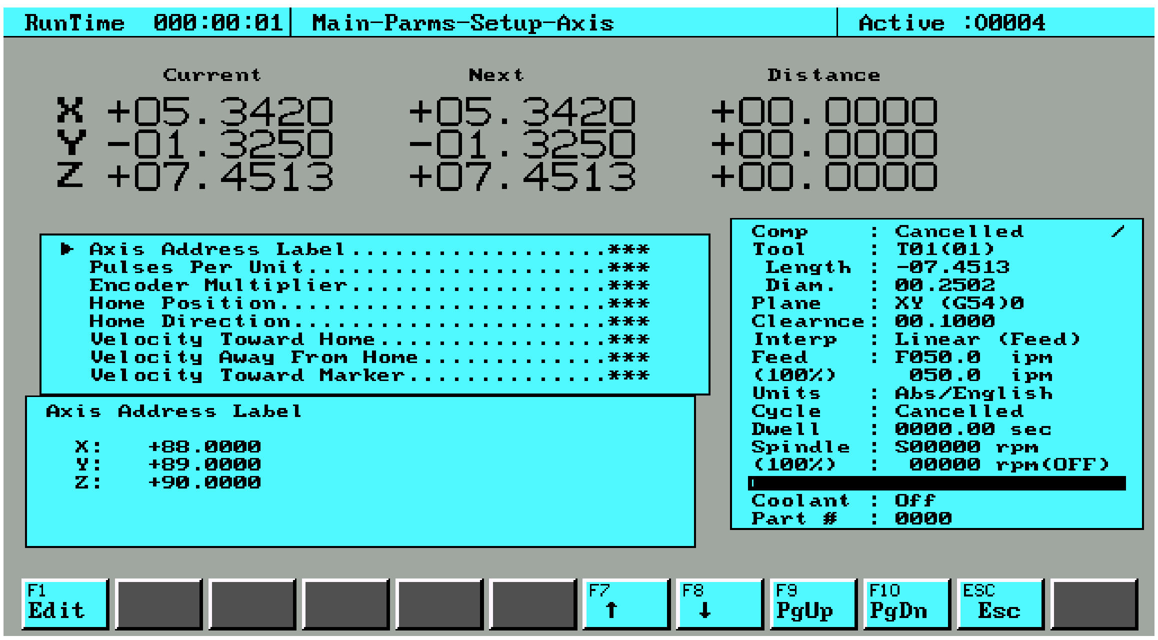
Note 1: To change a parameter, press F1 (Edit) and type in the new number.

Note 2: After changing any power parameter, return to the main menu before cycling power.

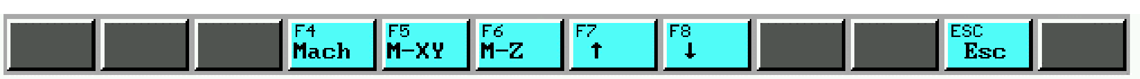
Note 3: After any power parameter is changed, the machine must be powered down, then up again. These parameters are only read on power up.

F4 (Axis) Main-Parms-Setup-Axis

If the F4 (Axis) key is pressed, the following screen will be displayed.



Keys displayed when F1 (Edit) is pressed:



The PgUp, PgDn, and arrow keys move through the tables on the upper display. The lower display changes to show the data associated with the cursor position of the upper display. To edit the values in these tables the cursor is positioned using the arrow keys to the desired parameter in the upper screen. When the F1 (Edit) key is pressed the cursor will move to the lower screen. Next, move the cursor using the arrow keys to the axis or parameter desired and type in the new values using the keypad. After the new values have been entered, press the **ESC** (**Esc**) key. The new value will be entered and the cursor will return to the upper screen; the selection process may then recommence. Some parameters can be related to the machine position. To edit or load these parameters, use F4 (Mach) to load the X, Y, and Z positions into the parameters. Use the F5 (M-XY) for the X and Y axis or the F6 (M-Z) key for the Z axis. The following is a list of all the selectable parameters displayed in this mode and a description of their functions.

Axis Address Label	ASCII code assigned to each axis
X 88.0000	
Y 89.0000	
Z 90.0000	
Pulses Per Unit	The number of pulses the feedback
X 10000.0000	gives per unit of travel
Y 10000.0000	
Z 10000.0000	
Encoder Multiplier	Sets an internal multiplier on the
X 02.0000	number of pulses coming from the
Y 01.0000	encoder. 0 is times 1, 1 is times 2, and 2 is times 4
Z 01.0000	
Home Position	The dimension assigned to the
X 00.0000	machine zero or home position
Y 00.0000	
Z 00.0000	
Home Direction	Defines the direction of rotation of the motor
X 00.0000	when a home is commanded
Y 00.0000	CW = 00.0000
Z 00.0000	CCW = 01.0000
Velocity Toward Home	Sets the feedrate at which an axis
X 60.0000	seeks the home limit switch
Y 60.0000	
Z 60.0000	
Velocity Away From Home	Sets the velocity at which an axis
X 12.0000	feeds off the home limit switch
Y 12.0000	
Z 12.0000	
Velocity Toward Marker	Sets the velocity at which an axis
X 02.0000	searches for the encoder marker pulse
Y 02.0000	
Z 02.0000	

Home Sequence X 02.0000 Y 02.0000 Z 01.0000	These numbers determine the order the axes will home in: #1 first, #2 next, etc. Axes with the same number home together. 0 will cause that axis to not home.
Positive Limit X 00.0000 Y 00.0000 Z 00.0000	Dimension from machine zero where the positive software limit occurs
Negative Limit X -31.0000 Y -18.0000 Z - 6.5000	Dimension from machine zero where the negative software limit occurs
Maximum Feed X 200.0000 Y 200.0000 Z 200.0000	Sets the maximum G01 feedrate in inches per minute or mm per minute
Dry Run Feed X 75.0000 Y 75.0000 Z 75.0000	Sets the dry run feedrate in inches per minute or mm per minute
Rapid Velocity X 800.0000 Y 800.0000 Z 800.0000	Set the maximum G00 feedrate in inches per minute or mm per minute
Feed S-Curve Acc (Cent 7 and up)	The units are identical to the (linear) acc/dec. These ramps are used when the s-curves are enabled in feed moves.
Rapid S-Curve Acc (Cent 7 and up)	The units are identical to the (linear) rapid acc/dec. These ramps are used when the s-curves are enabled in rapid moves.
Acc/Dec X 20.0000 Y 20.0000 Z 20.0000	The Acc/Dec is a number that determines the rate at which the axis velocity is increased or decreased for feed moves. The smaller the number the longer the Acc/Dec times will be. Acceleration and deceleration in this control are linear ramps. Units are in inches per sec^2.

Rapid Acc/Dec
X 20.0000
Y 20.0000
Z 20.0000

The Rapid Acc/Dec is a number that determines the rate at which the axis velocity is increased or decreased for rapid moves. The smaller the number the longer the Acc/Dec times will be. Acceleration and deceleration in this control are linear ramp units and are in inches per sec^2.

Feed S-Curve Acc
(Cent 7 and up)

The units are identical to the (linear) acc/dec. These ramps are used when the s-curves are enabled in feed moves.

Rapid S-Curve Acc
(Cent 7 and up)

The units are identical to the (linear) rapid acc/dec. These ramps are used when the s-curves are enabled in rapid moves.

Slow Jog Velocity
X 50.0000
Y 50.0000
Z 50.0000

Velocity in IPM or MMPM

Fast Jog Velocity
X 200.0000
Y 200.0000
Z 200.0000

Velocity in IPM or MMPM

Jog Key Direction
X 00.0000
Y 00.0000
Z 00.0000

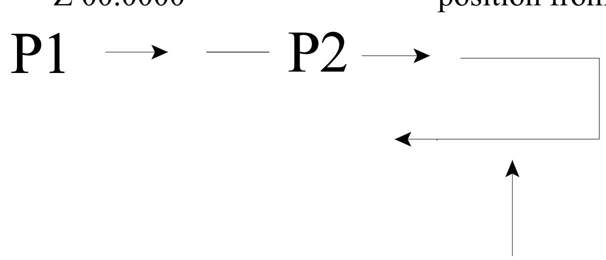
Sets whether the Jog is related to moving the table or moving the tool. A “1" reverses the direction

In Position
X 00.0050
Y 00.0050
Z 00.0050

After any rapid move, the machine will wait until all axes are within position to the destination before starting the next block

G00 Unidirectional
X 00.0000
Y 00.0000
Z 00.0000

Sets the distance in inches or mm which an axis will go past the destination point in one direction before reversing direction so that the machine will always position from the same direction. Active only in G00 mode.



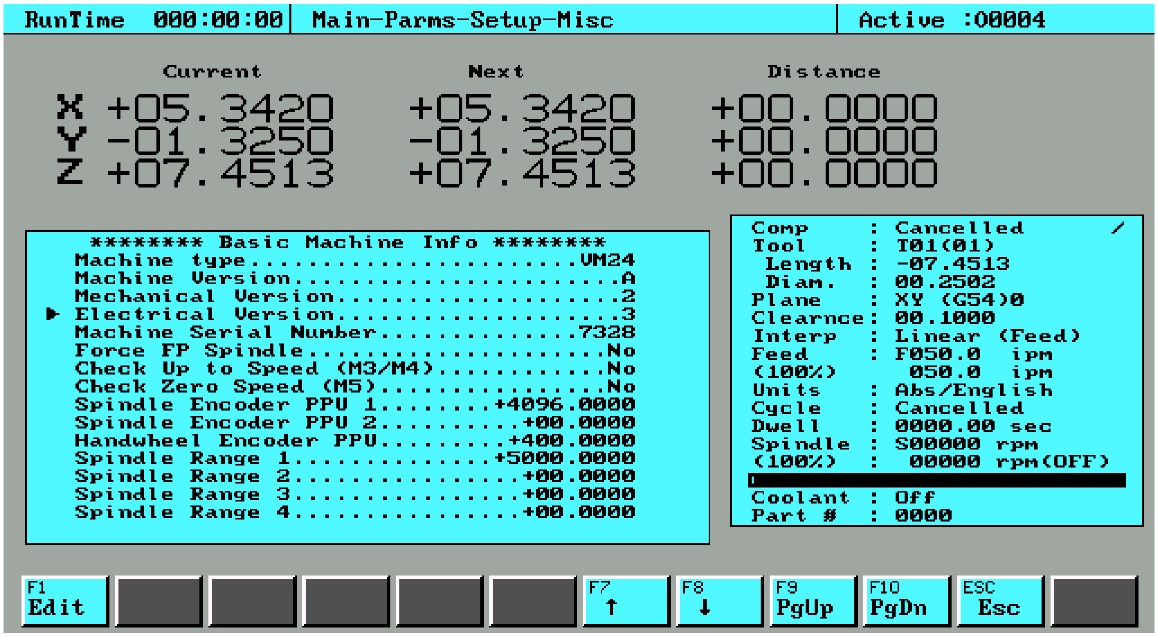
Uni-distance

G60 Unidirectional X 00.0000 Y 00.0000 Z 00.0000	Same as G00 unidirectional except only active in a G60 block
Backlash X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in inches or mm. The control will compensate for lost motion whenever an axis reversal takes place. Active in all modes.
Excess Error X 00.2500 Y 00.2500 Z 00.2500	Sets the distance in inches or mm. The machine can lag behind the CNC. The CNC will shut the system down due to an excess following error. 00.0000 = will never force an excess error.
Rotary=0 Linear=1 X 01.0000 Y 01.0000 Z 01.0000	Sets whether an axis should be treated as rotary or linear. In rotary, the feedrate is interpreted as degrees per minute rather than inches per minute. There is no conversion between inch and metric for rotary axis.
Handwheel Normal=0 Invert=1 X +00.0000 Y +00.0000 Z +00.0000	A “1" will reverse the direction of the handwheel.
English Leading X 02.0000 Y 02.0000 Z 02.0000	Sets the number of characters to the left of the decimal point for the inch system, for the specified axis only
English Trailing X 04.0000 Y 04.0000 Z 04.0000	Sets the number of characters to the right of the decimal point for the inch system, for the specified axis only
Metric Leading X 03.0000 Y 03.0000 Z 03.0000	Same as English Leading except for the metric case
Metric Trailing X 03.0000 Y 03.0000 Z 03.0000	Same as English Trailing except for the metric case

Home Switch=0 Marker=1 X 00.0000 Y 00.0000 Z 00.0000	Sets whether an axis will seek a home limit switch and then the marker pulse, or just seek the nearest marker pulse.
Max Handwheel Error X 01.0000 Y 01.0000 Z 01.0000	When the excess error reaches this value, pulses from the handwheel are ignored. Error is specified in inches or mm.
Tool Change X + 00.0000 Y + 00.0000 Z + 00.0000	Z moves to this location on a G32 (Z to toolchange) or M6 (toolchange) command. This position is specified in inches or mm and is relative to the home zero. The Y axis tool change position is used on bridge mills. X and Y are used on Pick-n-Place Tool Changers
Tool Changer Spacing (arm style)	Use to avoid large tools being place next to each other on arm style tool changers
Gain Proportional (KP) X + 08.0000 Y + 08.0000 Z + 08.0000	KP is in units of DAC steps per count of following error.
Gain Velocity (KV) X + 250.0000 Y + 250.0000 Z + 250.0000	KV is in units of DAC steps per unit of velocity. These units assume a 16 bit (1 bit sign, 15 bits magnitude) DAC, although the lower four bits are not actually significant. The units of velocity in this case are counts/heartbeat, one heartbeat being 3/4096th of a second. Larger values reduce the following error. Reasonable values are between 50 and 327.
Handwheel Gain (KP) X 04.0000 Y 04.0000 Z 04.0000	Same as proportional. Smaller values are used to soften the handwheel motion. Reasonable values are between 2 and 8.

F5 (Misc) Main-Parms-Setup-Misc

The F5 (Misc) key brings up various miscellaneous setup parameters dealing with the spindle and M codes. When F5 (Misc) is selected, the following screen appears.



Miscellaneous parameters are edited similarly to the Power parameters. The following is a list and short description of the miscellaneous parameters.

Basic Machine Info	
Machine Type	Type in VMD30, etc.
Machine Version	The current model
Mechanical Version	The current mechanical version
Electrical version	The current electrical version
Machine serial number	The machine serial number
Force FP Spindle	*Yes* for older style Centurion Vs that output the spindle analog from the front panel.
Check Up to Speed (M3/M4)	Control will wait for up-to-speed signal from the spindle controller before continuing on M3 (spindle CW) or M4 (spindle CCW)
Check Zero Speed (M5)	Control will wait for zero speed signal from the spindle controller before continuing on M5 (spindle off)

Spindle Encoder PPU1	Pulses per rev of spindle, used for hard tapping option and displaying the RPM in gear 1
Spindle encoder PPU2	Pulses per rev of spindle, used for hard tapping option and displaying the RPM in gear 2
Handwheel Encoder PPU	Pulses per rev of the handwheel, should be 400 for current systems
Spindle Range 1	Max spindle speed for gear 1
Spindle Range 2	Max spindle speed for gear 2
	.
	.
	.
Spindle Range 8	Max spindle speed for gear 8
Spindle Range	Which range the spindle is in: 1 to 8. Scaled assumes the range based on the programmed spindle speed and puts out the appropriate voltage.
Maximum Spindle Speed	This will clamp the overall spindle speed. It is used to limit the RPM's in high gear while still getting a large range of speeds in lower gears.
Spindle Ramp Time 1 (sec)	The tapping deceleration point is adjusted based on spindle RPM, pitch, and spindle ramp time so as not to overshoot the programmed depth. It relates to the time in seconds to decelerate the spindle to 0 RPM’s from full RPM’s in gear 1.
Spindle Ramp Time 2 (sec)	The tapping deceleration point is adjusted based on spindle RPM, pitch, and spindle ramp time so as not to overshoot the programmed depth. It relates to the time in seconds to decelerate the spindle to 0 RPM’s from full RPM’s in gear 2.
2 Gear Yaskawa M5 Spindle Motor	Yes for newer M5 spindle drives with 2 gear spindle motors. The spindle does not need to stop to shift.
Electronic Spindle Gear	Yes enables the shifting of electronic geared spindle drives.
Spindle Gear Box	Yes for RW30 with a mechanical spindle gear box.

Hard Tap Fudge Factor	Used to adjust the depth of rigid tapping cycle. Higher numbers will decrease the amount of overshoot at the bottom of the hole.
Feed Back on Mitutoyo Scale	Yes will enable the Z axis feedback from the Mitutoyo scale options.
Feed Back on Quill Scale	Yes enables the glass scale for Z axis quill options.
Quill Epsilon	If the value is zero no error checking is done for quill movement. If the value is none zero, quill motion is monitored while a program is running. If the quill moves, the control goes into block mode and an error 666 "Quill motion detected, return the quill to it's original position – 12.3456" will be displayed. The program will not continue until the quill is returned to it's original position. The position of the quill at the start of a program (and during tool changes) is saved and used to compensate Z positions during the program.
Quill Scale Encoder	The encoder to use for the glass scale for Z axis quill options.
Quill Scale PPU	The pulses per inch for the quill scale encoder.
Back Gear Reverse	If the back gear input is sensed and the parameter is Yes, the outputs for CW and CCW are reversed.
Keyboard Code	015 for systems with a CAT900 card, 0 for others
Yaskawa Drives	*Yes*, if it has Yaskawa drives (Yaskawa drives give a drive OK signal instead of a drive fault signal)
Door Open Switch	See notes on European code (page 55) at the end of this section.
European Code	See notes on European code (page 55) at the end of this section.
Auger Aux (Cent 7 and up) **Setup Aux (Cent 7 and up)**	The front panel Aux buttons can be assigned to specific functions. To assign the Aux1 button to the auger enter "001", to assign the Aux2 button to the setup button enter "002" in the parameter
Door Open Axis **Door Open Input**	See notes on European code (page 55) at the end of this section.

Door Open Override Axis **Door Open Override Input**	See notes on European code (page 55) at the end of this section.
Check Tool Door Open	If *Yes*, Z input 10 is checked before tool changer arm is commanded. If the input is not seen in 15 seconds, a timeout error is displayed.
Check Processor Temperature	If *Yes* and the over temperature Z input 4 is made, an over temp message will be displayed.
Cool-Down Time (Min) (Cent 6 and up)	Used on spindle air-purge systems.
Lube Cycle Time (Cent 6 and up)	Used on spindle air-oil lubricating systems.
Recycle Pump Time (Secs) (Cent 6 and up)	Used on coolant through the spindle systems.
Max Feed w/Door Open	See notes on European code (page 55) at the end of this section.
Soft Start Delay (Secs)	The time delayed before allowing axis movement after the machine is reset.
Digital Readout in E-Stop	If *Yes* the axis position display will be updated even when emergency-stopped.
Probe Axis and Probe Input	Designates which axis and input number for the Probe 1 and Probe 2 commands to use.
Cranking Minutes/Rev	How many minutes of normal program execution time correspond to one turn of the handwheel. Multiplying an IPM feed time, this factor results in an IPR feed (inches per turn of the handwheel). This parameter should be approximately 0.0010 for our current systems.
Cranking Factor	Multiplier for each handwheel click is approximately 100 for our current systems. Cranking/Factor/Cranking MinsPerRev is proportional to the max feedrate allowed while hand-cranking.
Cranking Max IPM	This limits the feedrate while hand-cranking to get reasonable response at slow programmed feedrates. Adjusting the other parameters can give excess errors on

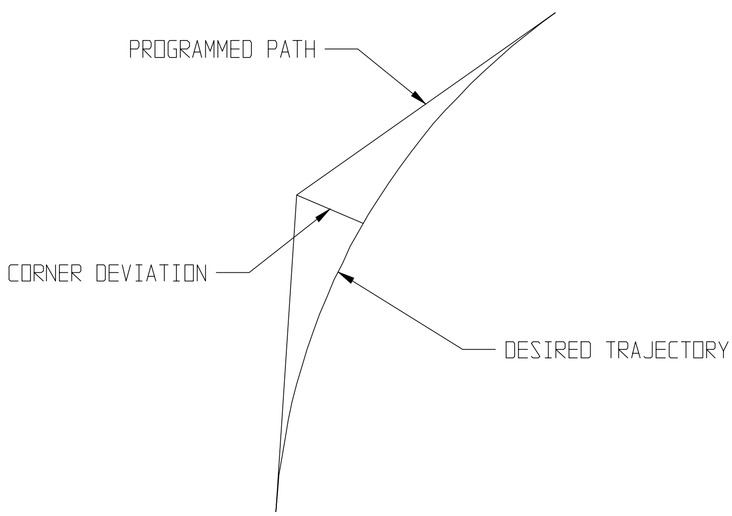
rapids. This parameter should be approximately 100 for our current systems.

Sharp Corners

Yes will cause all corners to be rounded to a maximum specified by the max corner deviation parameter. *No* will round the corners proportionally to the feed rate. Full feed rate will round the corners by the max corner deviation parameters. Slower federates will reduce the deviation.

Max Corner Deviation

This number sets the maximum deviation allowed on a corner. A value of .001 means the machine will slow down when approaching a corner by more than .001". (Centurion 6 only.)



Min Block Time (sec)

If the time to execute a move is less than this value, the control will slow the feedrate down in an effort to forestall jerky motions from data starvation.

End-of-Cycle Axis and End-of-Cycle Outputs

These two parameters define which output will be used for the End-of-Cycle Light.

The end of cycle light does not need to be programmed. The light will come on anytime there is no motion and a program is running and at the end of a program. This would be the case for block stops, waiting for tool change reset, etc. The light goes off when any key is pressed or motion recommences. The light may illuminate during dwells, automatic tool changes, orienting, and etcetera, but it will shut off when the machine recommences motion.

Acc Settling Time (Msec)
Dec Settling Time (Msec)
(Cent 6 and up)

The amount of time to keep the velocity constant after accelerating or decelerating.

Software Options	
Security Code # 0	Secret code to enable S-curves
Use S-curves (Cent 7 and up)	Yes to enable S-curves for acceleration and deceleration.
Look Ahead (Cent 6 and up)	Specifies the number of axis moves the control can see ahead, which enables the control to prepare for sharp corners or features that could otherwise be rounded off when feedrates are increased.

Note: Look Ahead is only active in DNC-fast mode. Valid values are between 10 and 255.

Max graph file size	Limits the amount of graphics on the screen. 30,000 for standard systems
Minimum Parts Space	If the parts space is less than the parameter value in KB, the system will go to the erase screen each time a key is pressed on the main menu.
G18 is	XZ plane (X is the primary axis) or ZX plane (Z is the primary axis)
G93 is	1/min or 1/sec for inverse feedrates
Special Flags	Normally set to 0

Bit 1 (#1) will place a dot on the graphics screen when plotting the tool.

Bit 2 (#2) will shut off trig help.

Bit 3 (#4) will shut off cutter compensation.

Bit 5 (#16) draws a circular tool in XY plane same diameter as active tool.

Bit 6 (#32) draws a 3-D tool on graphics screen.

Ignore Rapids in Autoscale	If *Yes*, rapid moves will be ignored when auto-scaling the graphics.
Remove Length from Graphics	If *Yes*, the graphics will treat all tool lengths as 0.
Screen Blank Time (min)	Time in minutes to blank the screen if no key is pressed
Flat Screen Blank	*Yes* for CRTs. Some flat screens take too long to refresh if the parameter is set to *Yes*.

Use FLZ instead of G54	Used for setting work offset in jog and handwheel mode. *Yes* means use FLZ (G92 offsets). *No* means use G54 offsets.
Tool Setting	If set to any tool the jog and handwheel tool setting routines will prompt the operator for the tool # being set. If it is set to current tool the control will assume the active tool # is the one that is being set.
Tool Setting Use Work Offsets	If set to *No*, the position given by the operator is relative to home. If set to *Yes*, the position given is relative to the work offset (this is useful for tool setting off the table and using a Z work offset to the top of the part).
Extract Input Programs	If set to *Yes*, the control will extract programs based on 0###'s in file loaded from the floppy drive or received in RS-232. Duplicate programs are overwritten without prompting the operator.
P899	Consult factory for special options.
Resolve DNC programs	If *Yes*, goto's can be used in DNC, Run, or DNC Verf.
Full Dos File Names	If *Yes*, eleven character DOS file names [FILENAME.EXT] may be used. Pressing the "D" key on any menu toggles the Full Dos File Names parameter.
Disable 417 Errors	If *Yes*, parameters may be modified while a program is running or verifying. If *No*, an error 417 will be given when attempting to modify a parameter while a program is running.
Load Tool Offsets	Set to *Yes* each time a program is started or when entering MDI. The tool offsets will default to the values from the tool table for the current tool. If set to *No*, the tool length and radius will default to zero.
Multiple Block Display	If *Yes*, the current block being executed plus the next seven blocks will be displayed when programs are running.

CAD Parameters

CAD Type is, DXF, CDL	CAD file type to import in conversational programming.
CAD Epsilon	Tolerance for geometry intersections of imported CAD files.

Post M codes Table

Post M code #0	
Post M code #1	
Post M code #2	
Post M code #3	
Post M code #4	M codes listed here will be executed
Post M code #5	after all other operations within
Post M code #6	the block.
Post M code #7	
Post M code #8	
Post M code #9	

Report File	DOS file names to write DPRINT text to when using POPEN P0
Command Name	Any string in this parameter will show up on the F10 (Util) key. Pressing F10 will execute the DOS file associated with the command name. See page 112 for more information on this feature.
PULSEX pulse delay (MS)	Msec pulse time used with PULSE0 and PULSE1
Spindle Power Raw	Value read from ADC, 0 to 255
Spindle Power Scale	Multiplier to obtain Spindle Power Value
Spindle Power Value (AMPS)	Calculated from the spindle power raw and scale
Spindle Power Limit (AMPS)	Not used

European Code Parameter and Operation Descriptions

Door Open Switch	Enables the door open switch
European Code	Enables the door open button
Door-Open axis	The axis # that the Door-Input-Switch goes into
Door-Open-Input	The input # that the Door-Input-Switch goes into
Door-Open-Override Axis	The axis # that the Door-Open-Override Button goes into
Door-Open-Override Input	The input # that the Door-Open-Override Button goes into

Max Feed with Door Open	The maximum speed the machine can move with the door open with the Door-Override Button pressed
Soft Start Delay (secs)	Time delay before allowing axis movement after the machine is reset.

The software that relates to European codes concerns a safety door open switch. Below is a description of how the software operates relating to the Set Up button and the software parameters.

If only the door open switch is enabled
When the door opens, it shuts the spindle off and feedholds the machine. When the door is open, the spindle will not start. When the door is open, the machine will not do an MDI move, will not jog, and will not do a move in run mode; however, it will handwheel. The machine will do I/O related functions other than turning on the spindle (i.e. arm-in, drawbar, etc.) If the machine is tapping and the doors open, the machine will finish the tap, feedhold, and stop the spindle.

If the doors are opened while the machine is running:

1 The spindle will shut off.
2 The axis will stop moving.
3 The feedhold light will come on.
4 The spindle will not restart.

To continue operation:

1 Close the doors (**Cycle Start** will flash).
2 Push the **CW** button on the front panel (the spindle will restart).
3 Push the **Cycle Start** Button on the front panel (**Cycle Start** will stop flashing and the **Feedhold** light will go out).

If the machine is not running and the spindle is not running, opening the doors should have no effect on the machine. However, the **Feedhold** light will come on.

If the door open switch is enabled and the European code is enabled
When the door opens, which does so by pressing the **door open** button, the spindle shuts off as well as feedholds the machine. When the door is open, the spindle will not start. When the door is open, the machine will not do an MDI command; it will not jog nor will it start running a program; however, it will handwheel. If the machine is tapping and the doors open, the machine will finish the tap and then **feedhold** and stop the spindle.

If the door is open, the machine will allow the tool changer to index only one position at a time using the tool changer utility or the tool setting utility.

If the door is open and **Setup** is held in, the machine will jog up to the Max Feed with Door Open (ipm) parameter.

If the door is open and **Setup** is held in, the machine will handwheel up to the 70% rate on the feedrate override switch, which is 2mm per click of the handwheel. (It is difficult to generate speeds greater than 1000 mmpm in this mode). Modifying the distance per click of the handwheel also requires a password available only to the machine builder.

If the doors are opened while the machine is running:

1 The spindle will shut off.
2 The axis will stop moving.
3 The feed-hold light will come on.
4 The spindle will not restart.

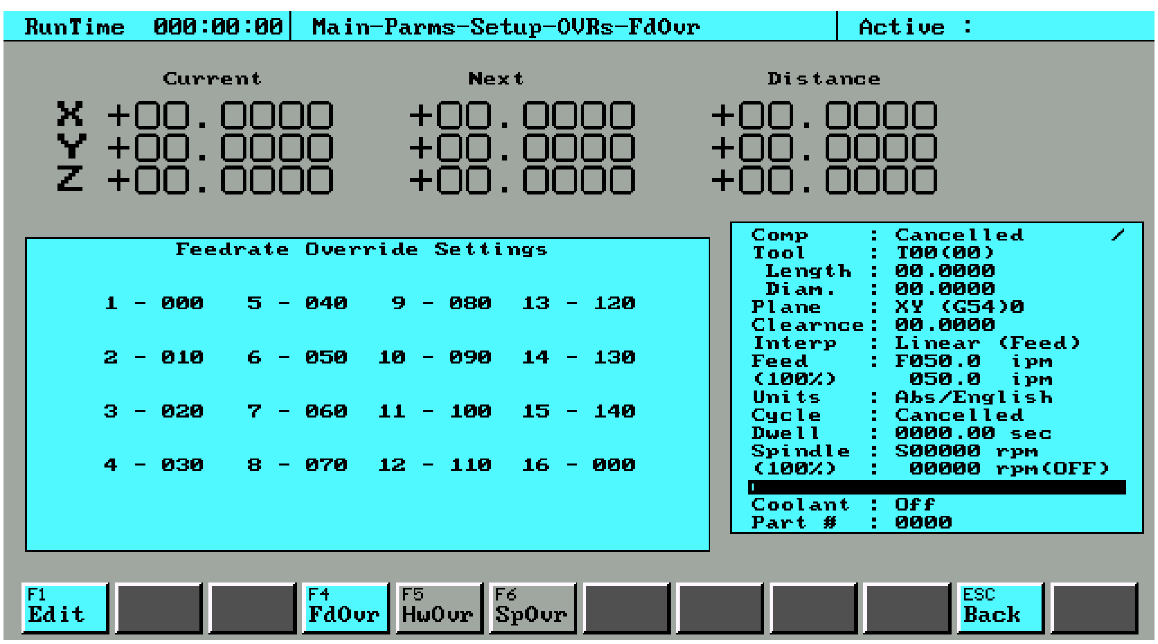
If **Setup** is pressed, the machine will move at the clamped feedrate while the button is held in.

To continue operation:

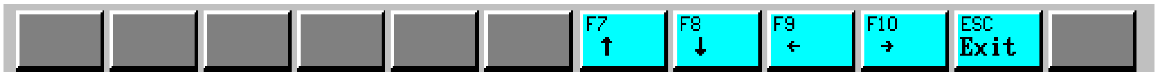
1 Close the doors (the **Cycle Start** Button will flash).
2 Push the CW button on the front panel (the spindle will restart).
3 Push the Cycle Start Button on the front panel (the **Cycle Start** Button will stop flashing and the **feedhold** light will go out).

F4 (FdOvr) Main-Parms-Setup-OVRs-FdOvr

The F4 (FdOvr) key brings up the feedrate override parameter settings. These settings determine which percentage will be used for each of the 16 feedrate override switch positions. (Entry 5-040 means 40% at 5th position, 1 is 0%, 2 is 10%, 3 is 20% . . . 16 is 0%.) The following screen is displayed when F4 (FdOvr) is selected. To edit the parameters, select F1 (Edit) and move the cursor to the value to be changed. Type in the desired change and press **Enter**.

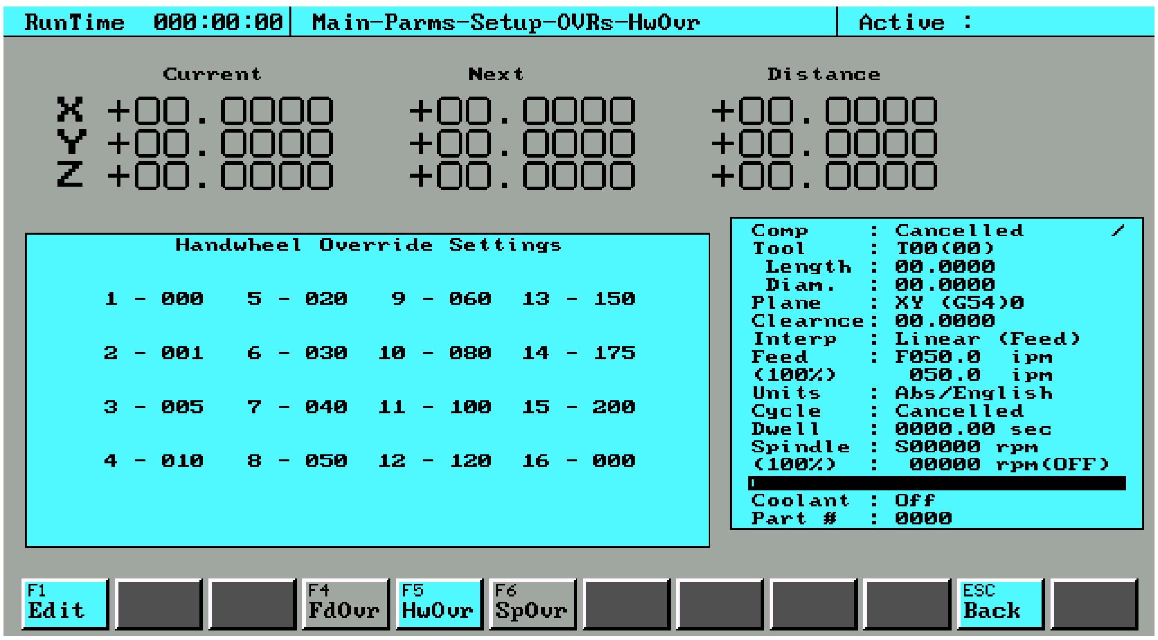


Keys displayed in the Edit Mode:

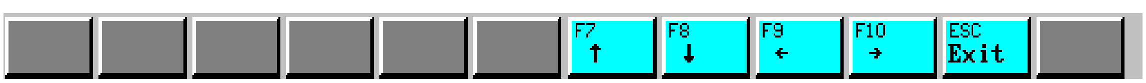


F5 (HwOvr) Main-Parms-Setup-OVRs- HwOvr

The F5 (HwOvr) key brings up the handwheel switch settings for the feedrate override switch. These settings determine how far an axis will move for one increment of the handwheel (001=1 pulse). Editing is performed in the same fashion as feedrate override parameters. The following screen displays the handwheel override settings.

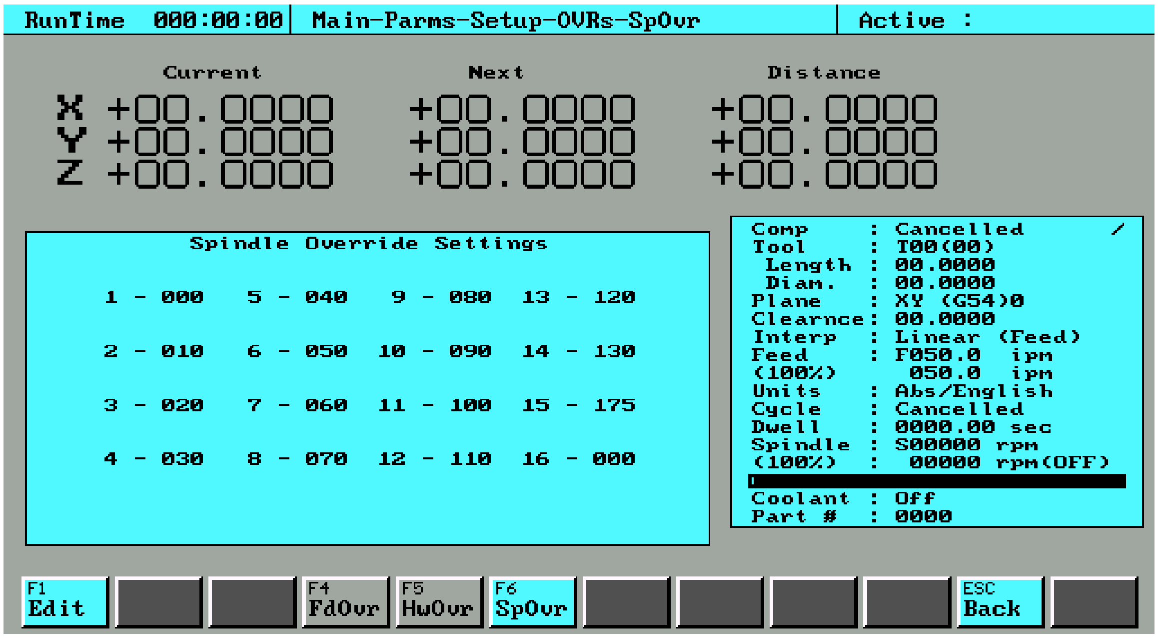


Keys displayed in the Edit Mode:

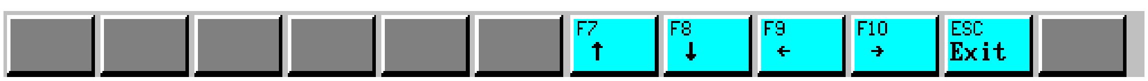


F6 (SpOvr) Main-Parms-Setup-OVRs-SpOvr

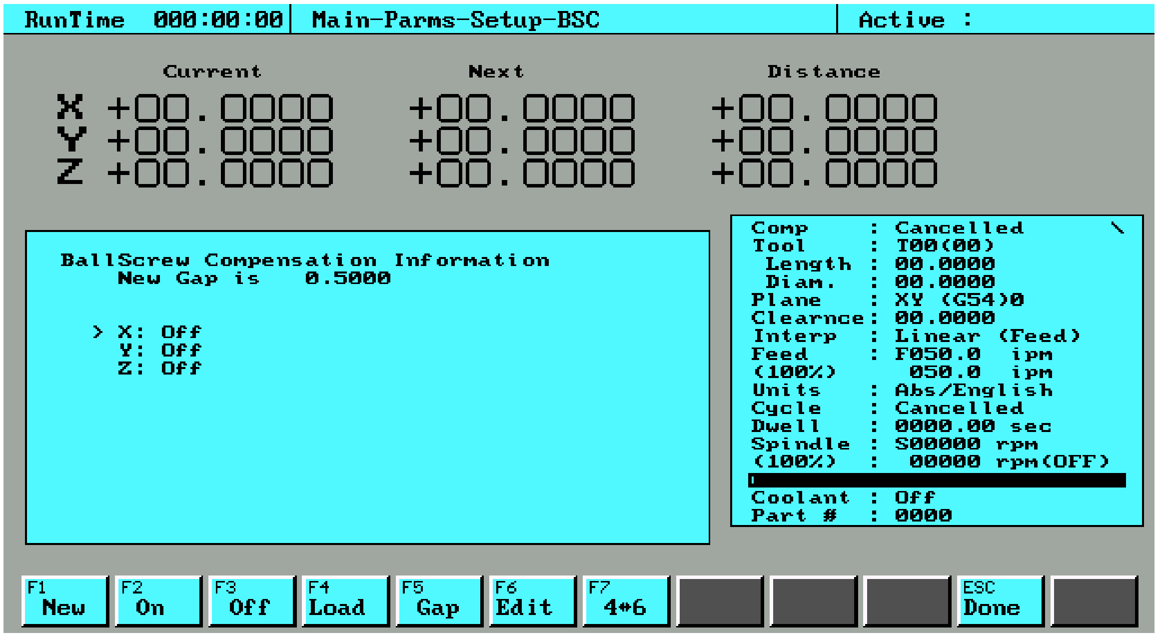
The F6 (SpOvr) key brings up the 16 spindle override switch settings. These settings are the percentages a spindle command will be overridden at each switch position. The spindle override parameters are changed in the same fashion as the feedrate override parameters. The spindle override screen is displayed below.



Keys displayed in the Edit Mode:



F7 (BSC) Main-Parms-Setup-BSC



<u>Ballscrew Compensation Table Creation Help</u>

Type X, Y, Z (A,B,C) to select the axis.
F1 (New) creates a new, zero ballscrew table.
F2 (On) turns ballscrew comp on for given axis.
F3 (Off) turns ballscrew comp off for given axis.
F4 (Load) loads the ballscrew table into the axis.
F5 (Gap) changes the spacing in the ballscrew file generated from F1 (New).
F6 (Edit) jumps into editor with ballscrew table.
F7 (4<>6) chooses 4 or 6 decimal places in the ballscrew file generated from F1 (New).
ESC (Done)

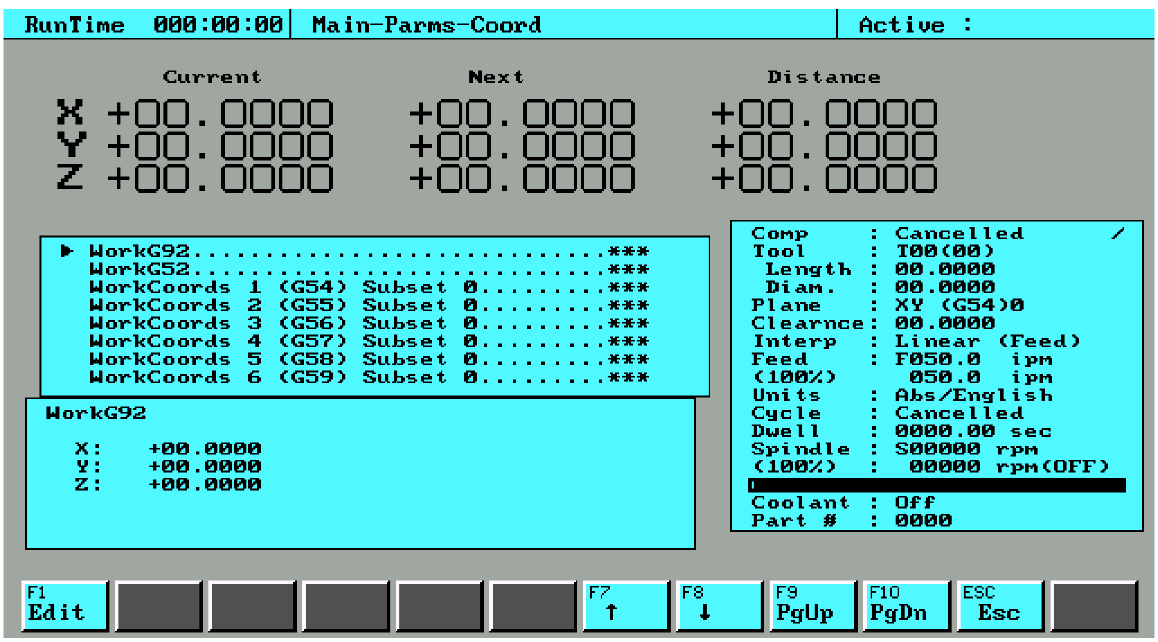
F9 (DOS) Main-Parms-Setup-DOS

F9 (DOS) will exit the CNC software and return to the DOS prompt.

F2 (Coord) Main-Parms-Coord

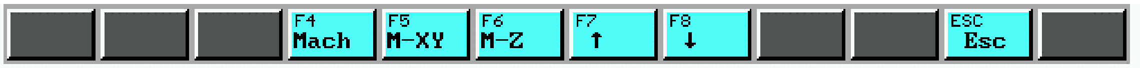
The F2 (Coord) key of the parameter screen brings up the parameters dealing with the various coordinate systems in the control. To edit the work coordinate parameters, use the PgUp, PgDn, and arrow keys to position the cursor to the correct parameter, then push the F1 (Edit) key and arrow to the desired axis. Type in the new values and press **Enter**, then ESC (Exit). The following screen shows the G92, G52, and Work System parameters G92/G52/54-59 described in the G code section. The next page of this display contains parameters for the following.

Positive Safe Zone X 00.0000 Y 00.0000 Z 00.0000	A position relative to machine zero which, along with the negative safe zone position, describes a cube that the tool cannot enter. If the tool is programmed into this cube an error will be displayed.
Negative Safe Zone X 00.0000 Y 00.0000 Z 00.0000	G22 turns the safe zone off G23 turns the safe zone on
G28 Reference Point 1 G30 Reference Point 2 G30 Reference Point 3 G30 Reference Point 4	Are described in Section 4, page 222.



Notice G54 through G59 work coordinates have subsets. To view/edit, for example, G573 work offset, move cursor to the work coordinate 4 (G57), then toggle (F3) until the subset reads 3.

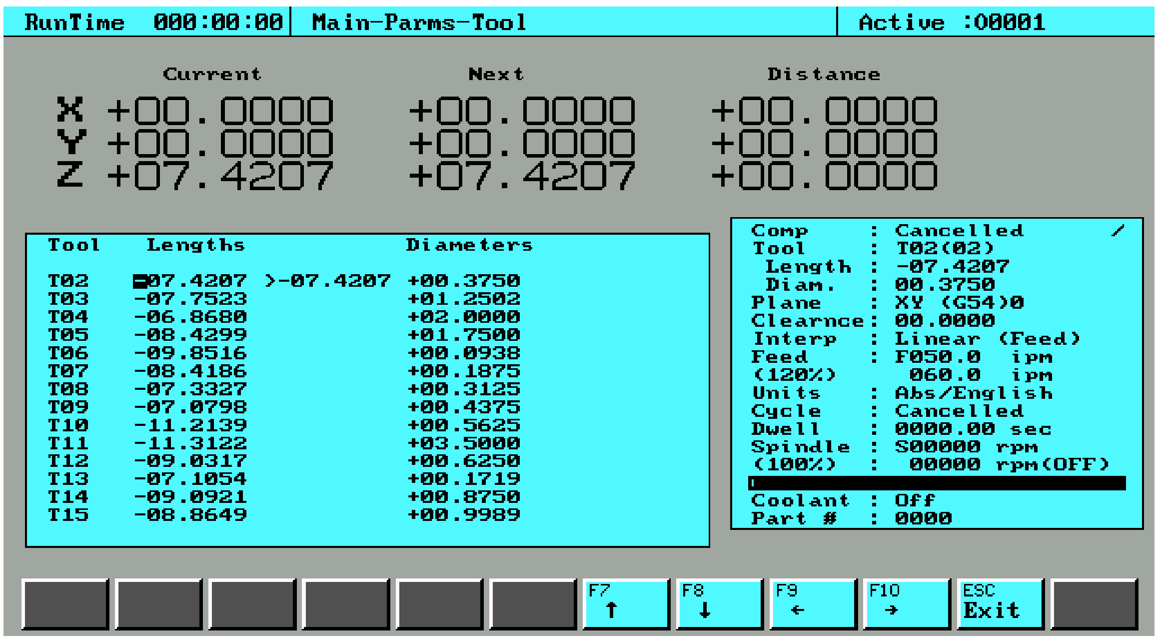
Keys displayed in the Edit Mode:



Operation of the work coordinate systems G92 and G52 are discussed in Section 4 page 294 (G92) and page 257 (G52). These parameters are positions relative to the machine zero and will become the new zero point when they are used. The F4 (Mach) key in the edit mode enters the current machine position as the work coordinate zero point for X, Y, and Z axes. The F5 (M-XY) enters the coordinates for X and Y axis only. The F6 (M-Z) enters the coordinate for the Z axis only.

F3 (TOOL) Main-Parms-Tool

The F3 (Tool) key brings up the following screen.



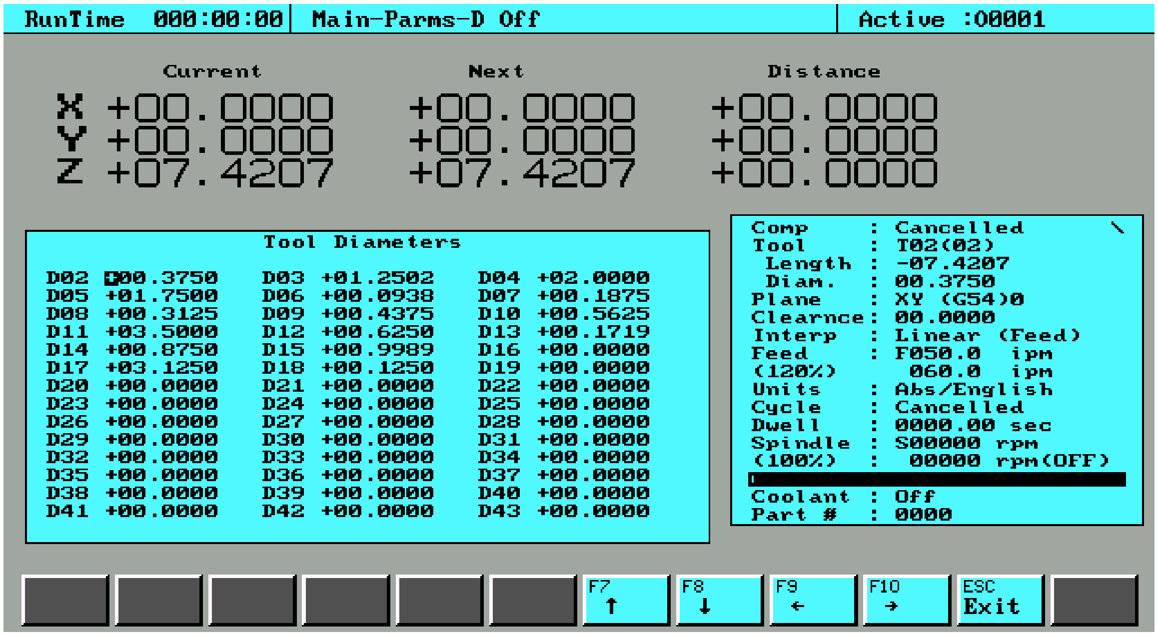
The cursor will default to the active tool number.

The process of editing on this screen is the same as on all the other Parms (parameters) screens. The F3 tool selection brings up both the H tool length table and the D tool radius table. A value typed into T04 length is entered into H04 as well. A value typed into T04 radius is likewise entered into D04. This entry screen exists mostly for convenience and is helpful if H and D offsets are associated with their corresponding T numbers.

Note: The control can be changed to use tool radii instead of diameters. When editing a tool offset, the current value is displayed to the right of the value being edited, so small values can easily be added or subtracted from the offsets.

F4 (D Off) Main-Parms-D Off

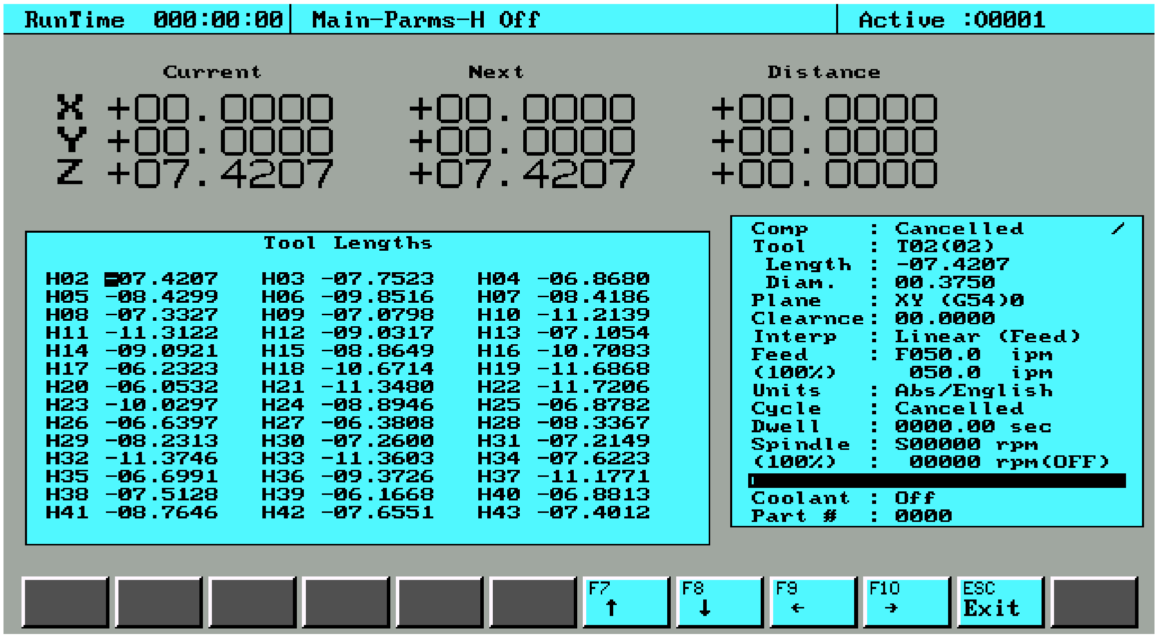
The F4 (D Off) key displays the 99 D radius or diameter offsets available on the CNC. These offsets are accessed and edited in the same manner as all other parameters. Following is the D offset screen.



The cursor will default to the active tool number.

F5 (H Off) Main-Parms-H Off

The F5 (H Off) key displays the 99 H tool length offsets available on the control. These offsets are accessed and edited in the same manner as all other parameters. The H offset screen follows.



The cursor will default to the active tool number.

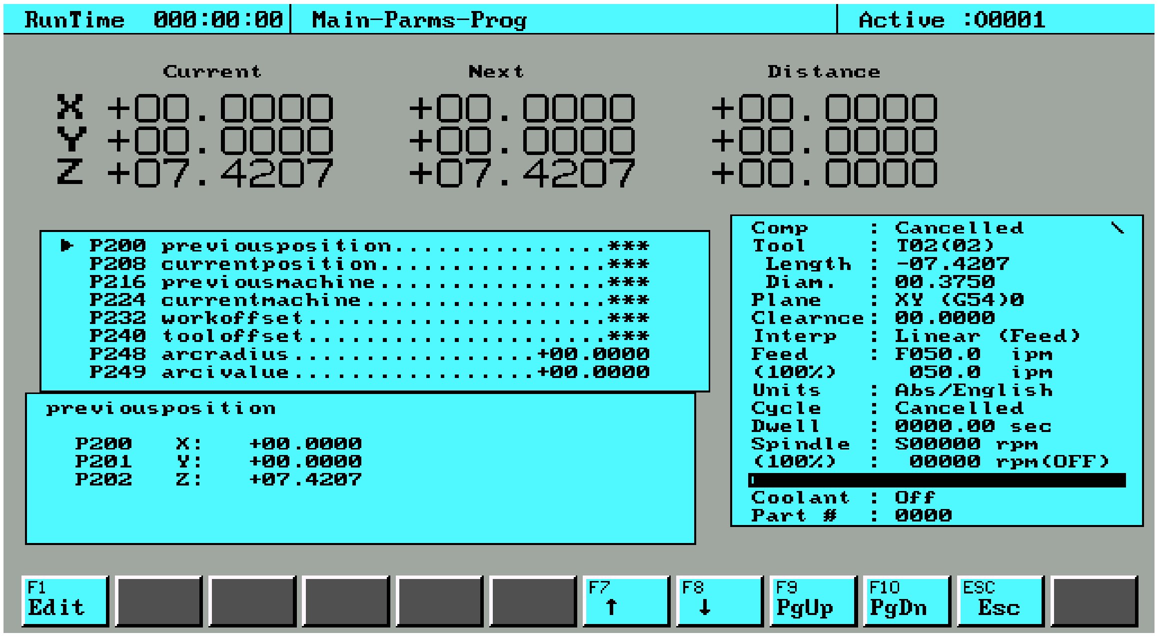
F6 (Save) Main-Parms-Save

F6 (Save) saves all files in the RAM directory to a floppy. You will be informed that duplicate files on the floppy will be overwritten. You will then be asked to verify by pressing F1 (Ok).

F7 (Load) Main-Parms-Load

Loads files from a floppy disk into the RAM directory. You will be informed that the current parameters will be overwritten. You will then be asked to verify by pressing F1 (Ok).

F8 (Prog) Main-Parms-Prog



This set of parameters gives the machine programmer access to all the internal parameters the CNC is using to execute a program. Normally these parameters would be used for display purposes only as an aid to program debugging. However, it is possible to read and change these parameters in a parametric program. **Great care must be taken when doing this because these parameters are used directly by the CNC to produce the next machine movement or function.** These parameters are displayed and edited in the same manner as the coordinate parameters. The following is a list and description of these parameters.

P200 thru P207	Contains the previous programmed position relative to the current work offsets of the enabled axis P200=X P201=Y P202=Z . . . etc
P208 thru P215	Contains the current programmed position relative to the current work offsets of the enabled axis P208=X P209=Y P210=Z . . . etc
P216 thru P223	Contains the previous machine position relative to the machine zero of the enabled axis P216=X P217=Y P218=Z . . . etc
P224 thru P231	Contains the current machine position relative to the machine zero of the enabled axis P224=X P225=Y P226=Z . . . etc

P232 thru P239	Contains the work coordinate offset relative to the machine zero of the enabled axis P232=X P233=Y P234=Z . . . etc
P240 thru P247	Contains the active tool length (H) parameter for the enabled axis P240=X P241=Y P242=Z . . . etc
P248	Contains the current arc radius
P249	Contains the current arc I or J or K value
P250	Contains the current arc I or J or K value
P251	Current feedrate in inches per minute or mm per minute
P252	Current dwell time
P253	Current spindle speed
P254 thru P258	Last X, Y, Z, A, B, C values executed
P260	Contains active tool number
P261	Contains active D tool radius
P262	Contains active H tool length
P263	Contains active D offset number
P264	Active H offset number
P266	Contains safe zone and overtravel status (0=off, 1=on)
P267	Contains ratio of feedback pulses to program units (1 is inch, .03937 for metric)
P268	Pending tool number
P270 thru P274	temp i, temp j, temp k
P273 thru P299	Used as temporary storage
P300 thru 303	Modal 00-Modal 03

P304	Status if control is in data mode or normal programming (o=off, 1=on)
P305	H offset direction or sign
P306	Status of 0=G0, 1=G1, 2=G2, 3=G3 mode
P307	Not used
P308	Number of active plane 0=G17 (X4), 1=G18 (ZX), 2=G19 (YZ), 3=G18 (XZ)
P309	Cutter comp. status 0=G40, 1=G41, 2=G42, 5=G45, 6=G46
P310	Current active canned cycle number: 0=Canceled, 1=Drill, 2=Drill/Dwl, 3=Peck I, 4=Tap, 5=Bore, 6=Bore II, 7=Peck II, 8=Hard Tap, 9=Bore/Dwl, 10=Left Tap, 11=Cir Pck Clear, 12=Cir Fin Inside, 13=Cir Fin Outside, 14=Rec Pck Clear, 15=Rec Fin Inside, 16=Rec Fin Outside, 17= Threading,18=XZ sweep, 19=YZ sweep
P311	0 = Absolute, 1= Incremental mode
P312	Feed unit (0 is feed/min, 1 is feed/rev, 2 is inverse feed)
P313	Spindle unit (0 is RPM, 1 is constant surface speed)
P314	CW or CCW spindle direction
P315	0 = Inches, 1 = Metric dimensions
P316	Scaling (0 = off, 1 = on)
P317	Rotation (0 = off, 1 = on)
P318	Mirror image (0 = off, 1 = on)
P319	Current work coordinate number (G54=1, G55=2…G59=6)

P320 thru P322	Gives the primary, secondary, and tertiary axis based on plane selection X=1 Y=2 Z=3 . . . etc For G17 XY pri=1 sec=2 ter=3 For G18 ZX pri=3 sec=1 ter=2 For G19 YZ pri=2 sec=3 ter=1 For G18 XZ pri=1 sec=3 ter=2
P323	0 = Return to R-plane (G99), 1 = initial level (G98)
P324	Tapping mode (0 = off, 1 = on)
P369	Job time (sum of run times)
P370	True tool number used for tool changers
Block Rate	Block per second for the last program executed.

F9 (CTRL) Main-Parms-CTRL

This set of parameters is an extension of the F8 (Prog) parameters. This group deals primarily with parameters used to create the auto-routine and canned cycles. These parameters can be accessed and changed in the same fashion as all other parameters in the control. The following is a list and description of these parameters.

Load Text Cycles	*Yes* to load the text engraving cycles. If it is set to *No* and a text command is executed, an error 578 *“Undefined text cycle”* will occur.
Offset Round Tapered Walls	See description of M95 and M96 (pages 308 and 312 in Section 5).
Spindle Range	Which gear range the spindle is in
Chip Remove On (Min)	If the machine is equipped with a chip auger removal system, this specifies the time that the auger will run after an M38 or after pushing the front panel **Auger** button.
Wash Down On (Sec) **Wash Down Off (Sec)**	While the auger is running, the wash down cycles on and off using these times.
Sort Directories by Name	If yes, it sorts the menus alphabetical by the 1st line of the program.
Drive Key on Menus	If yes, the help key allows the operator to select a new drive for the menu.

Auto Rotary Brake — *Yes* will control the rotary brake on A and B axis automatically. This parameter shuts off before rotary moves, and it turns the brake back on when the move is complete.

Rotary Brake Delay (Secs) — This is the delay in seconds after an M11 (A axis brake release) and M13 (B axis brake release). It is also the delay time after autobrake release in Home, Jog, and Handwheel. The default is .25 seconds. Max is 2.55 seconds.

Ignore Tool too Large Errors — If yes, the control will ignore 569 errors and cut a corner when compensated arcs are too small.

Pre Check Software Limits — Never, Run Only, or Always. Never: the control never checks the soft limits. Previous versions of the software acted this way. The soft limit error only occurs when the axis moves outside the limits. Run Only: checks are not made when verifying. When running it will check to see if any axis is going to move outside the limits before the move starts. The graphics will show the move but the machine will not make the move. This can prevent crashes where faster moves go beyond the limits. Prior versions would rapid to the limit and would be unable to stop before the limit switch and/or hard stop. Always: the control will check to see if any axis is going to move outside the limits when running or verifying. The graphics will show the move, but the machine will not make the move. It may be useful to verify a program before running it.

Tool Load Flag — Limit Exceeded or Limit not Exceeded

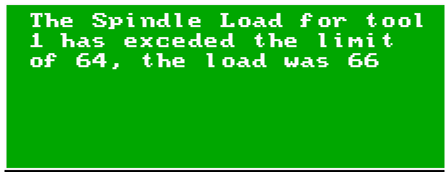
Tool Load Monitor — There are currently six options:

Off	No tool/spindle monitoring will be done.
E-Stop	If the limit is exceeded the machine will E-stop and display a message.
Feed Hold	If the Limit is exceeded the machine will Feed Hold and display a message.
Block Mode On	If the Limit is exceeded the machine will go into Block Mode and display a message.
Set Flag-No Message	If the Limit is exceeded the machine will set a flag. No message is displayed. The flag is PB418. The program can check this parameter to see if the Limit was exceeded

and perform an alternate operation, change to a different tool, etc. The operator can also view the state (or change the state) of the flag in the CTRL parameters. The parameter "Tool Load Flag" has two states: Limit Exceeded or Limit not Exceeded.

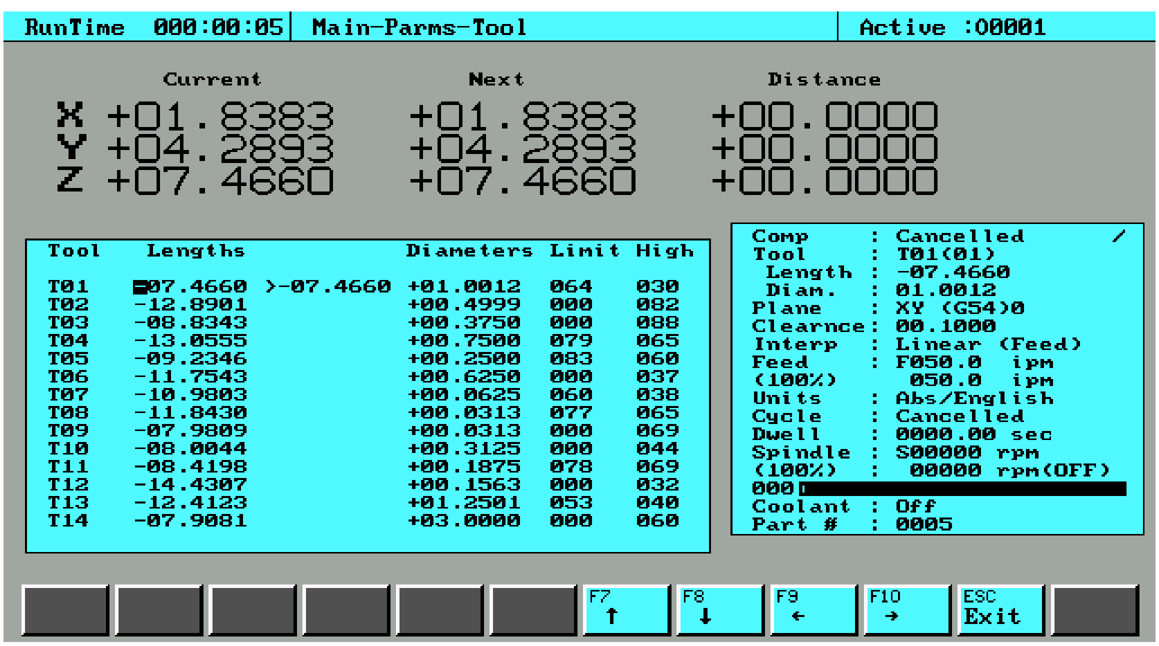
Message Only — If the Limit is exceeded only the message is displayed.

The message shown appears below:



Sampling is done only in feed moves (G1, G2 and G3). Sampling is not done in rapid moves (G0). Sampling is only done when the spindle drive is outputting up to speed, a program is running and the spindle is on. The spindle load needs to stabilize after any change in RPM. This requires that the software wait approximately 5 seconds after up to speed or any change in the spindle override. The time is hard-coded (no parameter). The time is based on an 8,000 RPM two-speed spindle configuration. Different drives, spindles and/or drive parameters may affect this delay time.

The Limits and the Max load for each tool are shown below in the tool table. If a Limit for a tool is zero then no tool/spindle monitoring will be done. The High values are not zeroed unless the operator zeros the values in the tool table. Initially the operator will run a program and then observe the High values for each tool. For the tools that he wishes to monitor he loads appropriate limits based on the High values that were obtained from running the program.



Only tools 1 through 25 are monitored and will have these parameters available. The spindle load bar will also have a number associated with it.

Serial Port Data

Note: Com port, baud rate, parity, data bits and stop bits are communications parameters. See page 100, Section 4 on F4 (RS232) for more information.

Primary Serial Port	NONE, COM1, or COM2
COM1 Baud Rate	110, 150, 300, 600, 1200, 2400, 4800, 9600, or 19200
COM1 Parity, Data and Stop Bits	N/7/1, N/7/2, N/8/1, N/8/2, E/7/1, E/7/2, E/8/1, E/8/2, O/7/1, O/7/2, O/8/1 or O/8/2
COM2 Baud Rate	110, 150, 300, 600, 1200, 2400, 4800, 9600 or 19200
COM2 Parity, Data and Stop Bits	N/7/1, N/7/2, N/8/1, N/8/2, E/7/1, E/7/2, E/8/1, E/8/2, O/7/1, O/7/2, O/8/1 or O/8/2
Secondary Serial Port	NONE, COM1, or COM2
Tape Start Character	ASCII value used to send when opening a com port
Tape Stop Character	ASCII value used to send when closing a com port
RS232 Buffer Size	1 for users not doing long RS-232 communications, up to 255 to improve RS-232 in DNC
LF in change to CR	*Yes* means change all carriage returns received on RS-232 to line feeds
End of Block Send CR/LF	*Yes* means send a carriage return/line feed at the end of blocks sent out on the RS-232
RS232 EOF Character	ASCII value of character sent at end file sent on RS-232

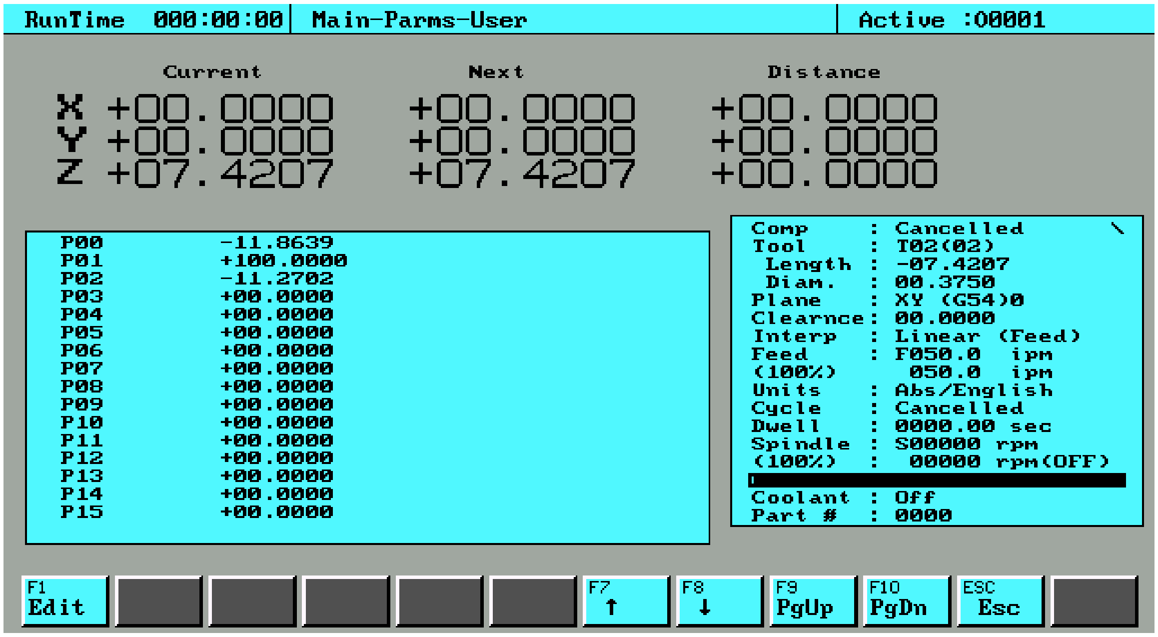
Digitizing Parameters

P100 Digitizing	Proportional gain
P101 Digitizing	Integral gain
P102 Digitizing	Differential gain
P103 Digitizing	Subscan increment
P104 Digitizing	Detail angle
P105 Digitizing	Probe backlash
P106 Digitizing	Probe radius
P107 Digitizing	Feed 1 - sampling
P108 Digitizing	Feed 2 - searching
P109 Digitizing	Feed 3 - retract
P110 Digitizing	Probe vibration
P111 Digitizing	Wall seek activate
P120 thru P139	Used by 3D pocket
P140	R plane dimension
P141	Final depth of canned cycle
P142	Initial level of canned cycle
P143	Increment of canned cycle
P144	First depth of canned cycle
P145	Tertiary axis feedrate for canned cycle
P146	Distance to retract chip break cycle
P147	R plane distance for each peck cycle
P148 and P149	Dwell times for canned cycles

P150	Circular auto-routines radius, rectangular auto-routine, corner radius
P151	X rectangular pocket dimension
P152	Y rectangular pocket dimension
P153	XY finish stock for autoroutines
P154	Z finish stock for autoroutines
P155	Cut width on pocket clearing autoroutines
P156	Radius of bolt hole cycles
P157	Bolt hole start angle
P158	Bolt hole number of holes in 360°
P159	Bolt hole number of holes to be drilled
P160 thru P171	Scratch
P172 thru P179	Coordinates of the center position for a mirror image command for the enabled axis P172=X P173=Y P174=Z . . . etc
P180 thru P187	Coordinates of the scaling center for the enabled axis P180=X P181=Y P182=Z . . . etc
P188 thru P195	Scale factor for each of the enabled axis P188=X P189=Y P190=Z . . . etc
P196	I, J, K position of primary axis center of rotation
P197	I, J, K position of secondary axis center of rotation
P198	Angle of rotation
P199	Auto routines plunge/ramp Z axis 0 = plunge, 1 = ramp
P346 **Bore Relief Angle**	Used by G76 (fine bore) and G87 (back bore) cycles when oriented and moving away from the bore
P347 **Bore Relief Distance**	Same as P346

F10 (User) Main-Parms-User

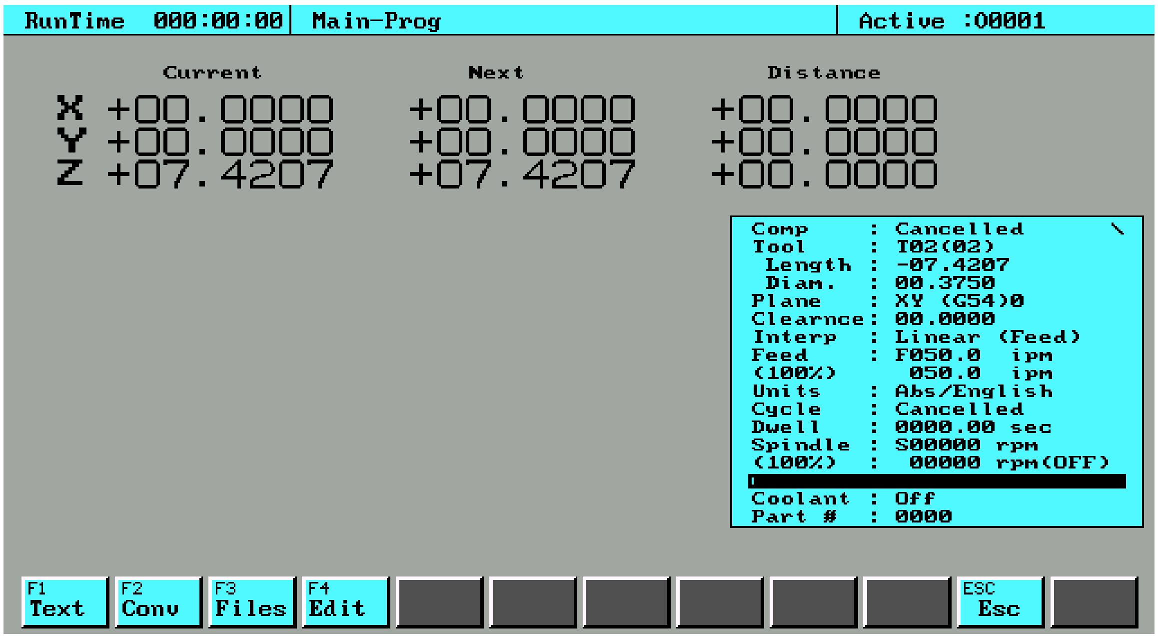
This set of 100 parameters is reserved for the parts programmer to use when writing parametric programs. These parameters are undefined and can be edited, displayed, or loaded from this screen. The editing and displaying formats are identical to the parameters discussed in this section. See Section 5 for information on parameter programming.



F8 (Prog) Main-Prog

There are two modes of program file creation/editing available on the Centurion 6 control: text and conversational.

Pressing the F8 (Prog) key will enable the soft keypad to allow selection of the type of programming desired. It also allows the transfer of programs to or from the floppy disk drive.

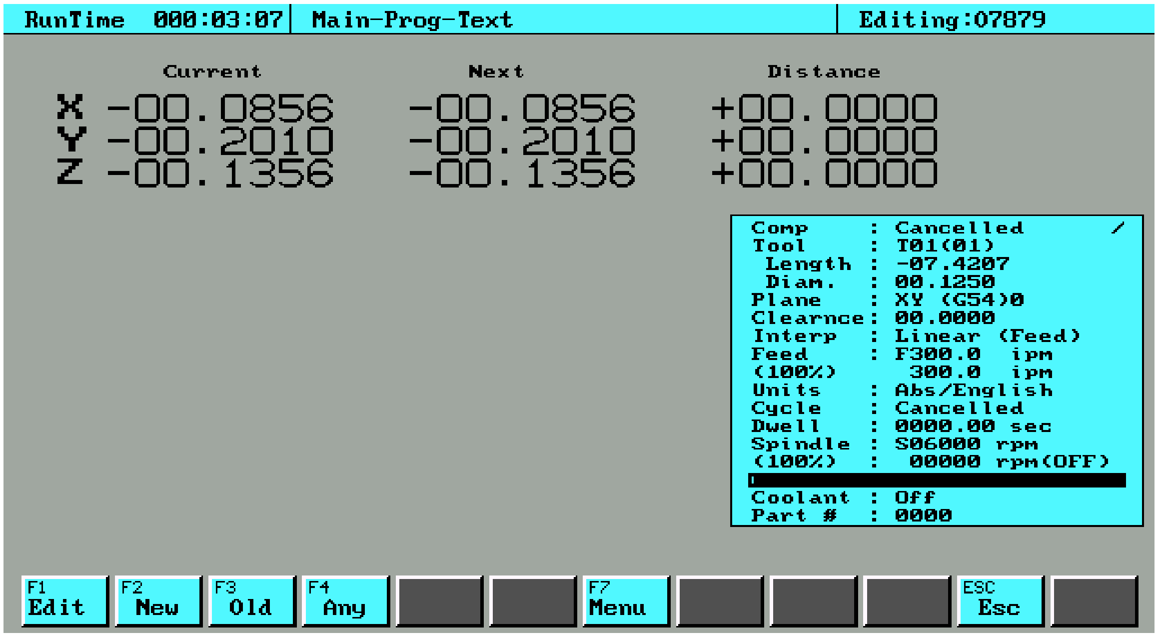


Text and conversational programs are stored in the control in different file formats and have different prefixes to distinguish them. Text programs are prefixed with the letter "O" and are kept in the parts directory in ASCII format. Conversational programs are kept in the parts directory in two formats: ASCII, prefixed with an "O"; and conversational, prefixed with a "P".

Only programs prefixed with an "O" may be run or verified.

F1 (Text) Main-Prog-Text

Upon entering the text programming mode, the upper right-hand box containing the active program number will display the last text program edited.



F1 (Edit) Main-Prog-Text-Edit

The F1 (Edit) key will select the program shown in the upper right corner as the active text edit program.

Text Editing Terms

Editor: A program that allows entry and modification of information, then stores it in a file for later retrieval and/or further modification

Text: Refers to a sequence of characters and/or lines being edited. The individual characters are manipulated using the widely accepted American Standard Code for Information Interchange (ASCII).

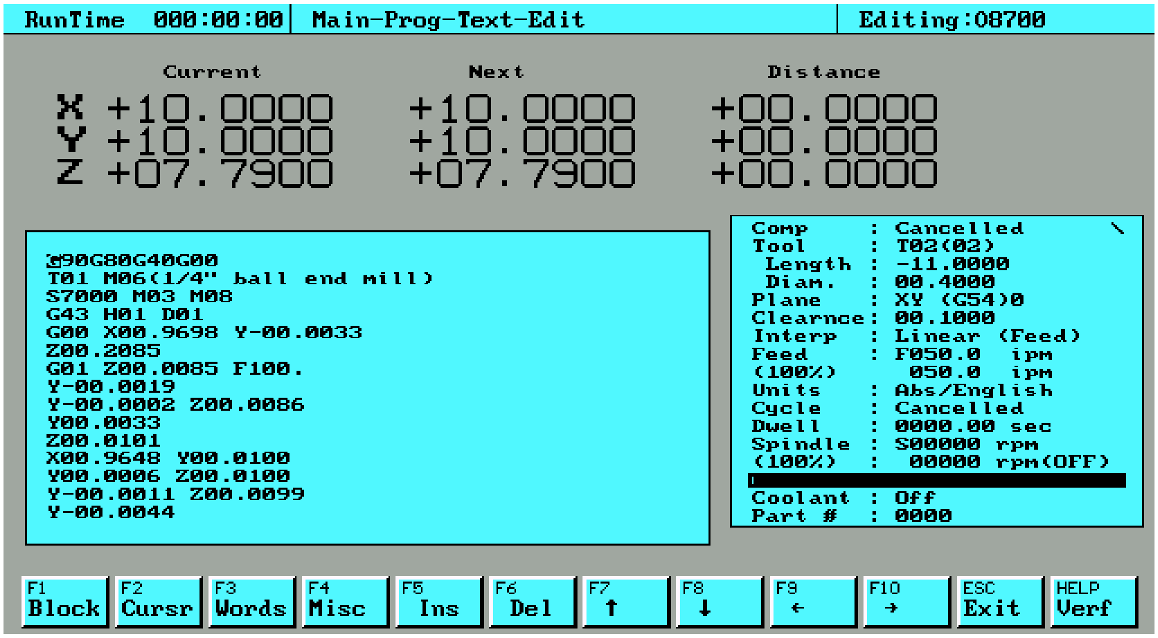
Cursor: A small box on the screen that marks where changes are being made to the text.

Entering and Editing Text

You enter text in much the same way as you enter text on a typewriter, and most of the keys on the keypads respond in the same fashion (pressing Enter terminates a block, for example). However, there are many important differences to note.

The cursor always indicates where the new text will be entered, and you can move the cursor in a number of ways. You may copy and move text with block commands. You may locate a particular string of text with the F8 (Find) command and optionally replace it with another string using the F7 (Chang) command. And, in most cases, you can even undo your last few changes with the F2 (Rest) restore line or F1 (UnDo) commands. These commands, and many more, are described briefly in the following sections.

The first screen you will see when entering the text editor is the edit screen with the first 16 lines of the program displayed. The main edit soft keys are located at the bottom of the screen.



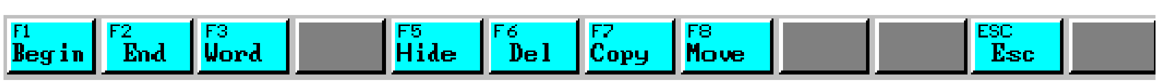
Text may now be entered at the current cursor position.

Key Definitions

F1 (Block) Main-Prog-Text-Edit-Block

A block is any arbitrarily defined, contiguous unit of text. A block can be as small as a single character or as large as an entire program. Mark a block by placing a begin-block marker at the first character in the desired block and an end-block marker just beyond the last character. Once marked, the block can be copied, moved, or deleted.

Although marked blocks are normally highlighted so you can see what you've marked, the block may be **hidden** (or made visible) with the F5 (Hide) hide block command.

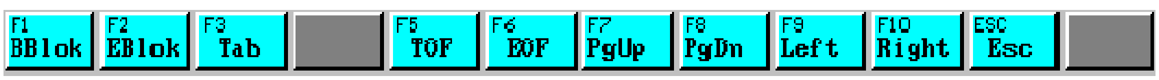


F1 (Begin) Marks the beginning of a block. The marker itself is not visible on the screen, and the block becomes visible only when the end-block marker is set.

F2 (End)	Marks the end of a block. Like the begin-block marker, the end-block marker is invisible, and the block itself will not be displayed unless both markers are set.
F3 (Word)	Marks a single word as a block, combining the functions of the begin-block and end-block commands. If the cursor is positioned within a word, that word will be marked. If it is not within a word, then the word to the right of the cursor will be marked. And if there is no word to the right of the cursor, then the word to the left will be marked.
F5 (Hide)	Toggles off and on the visual marking of a block.
F6 (Del)	Deletes a marked and displayed block. Although F1(UnDo) can usually restore portions of an accidentally deleted block, **there is no command** to restore a deleted block in its entirety, so use this command with care.
F7 (Copy)	Creates a copy of a marked and displayed block at the current cursor position. The original block is left unchanged, and the markers are placed around the new copy of the block.
F8 (Move)	Moves a marked and displayed block from its current position to the cursor's position. The markers remain around the block at its new position.

F2 (Cursr) Main-Prog-Text-Edit-Cursr

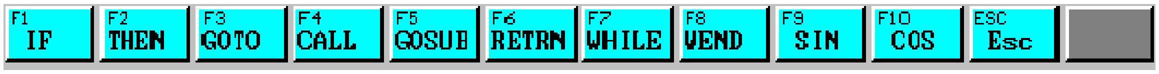
The F2 (Cursr) menu contains extended cursor movement commands:



F1 (BBlok)	Moves the cursor to the position of the block-begin marker.
F2 (EBlok)	Moves the cursor to the position of the block-end marker.
F3 (Tab)	Moves the cursor to the beginning of the next word.
F5 (TOF)	Moves the cursor to the first character of the program.
F6 (EOF)	Moves the cursor to the last line of the program.
F7 (PgUp)	Moves cursor up 15 lines.
F8 (PgDn)	Moves cursor down 15 lines.
F9 (Left)	Moves cursor to beginning of line.
F10 (Right)	Moves cursor to end of line.

F3 (Words) Main-Prog-Text-Edit-Words

The F3 (Words) soft key represents reserved words that may be used for programming the control.

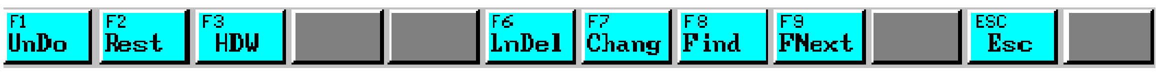


Pressing a key will cause that word to be printed on the screen. See Section 6 on parametric programming.

Note: F6 (RETRN) will print RETURN.

F4 (Misc) Main-Prog-Text-Edit-Misc

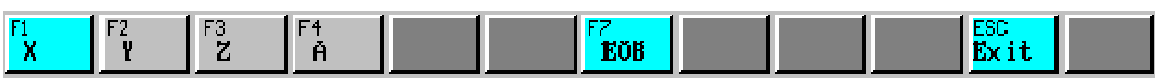
This section discusses a number of commands that do not readily fit into any of the other category.



F1 (UnDo) Restores whole lines deleted with the delete line command or the delete block command. It does not restore single characters or words.

F2 (Rest) Will undo any changes made to a line of text as long as you have not left the line. The line is restored to its previous contents regardless of the changes made.

F3 (HDW) Allows the use of the handwheel to move to a position. You may insert the axis position in the program by pressing the Enter key.



F1 - F4 (X/Y/Z/A) Choice of axis position to insert

F6 (LnDel) Deletes the line containing the cursor and moves any lines below it up one line. The cursor moves to column 1 of the next line.

F7 (Chang) This operation works the same as the Find command except that you can replace the found string with any other string of up to 67 characters. After entering the search string, you are asked to enter the replacement string. The last replacement string entered, if any, will be displayed; you may accept it, edit it, or enter a new string. Finally, you are prompted for options. The options you used last are displayed first. You may enter new options (canceling the old ones), edit the current options, or select them by pressing **Enter**. The options available are the same as those for the Find command.

F8 (Find) Lets you search for a string of up to 67 characters. When you enter this command you will be asked for a search string. The last search string entered, if any, will be displayed. You can select the string again by pressing **Enter**, or you may edit it or enter a new search string. After the search string is entered you must specify your search options. The options you used last, if any, are displayed. You can enter new options (canceling the old ones), edit the current options, or select them again by pressing **Enter**. The following options are available.

B Searches backwards from the current cursor position toward the beginning of the program.

G Searches globally. The entire program is scanned for the search string regardless of the current position of the cursor. The search starts at the beginning of the program if searching forwards and at the end if searching backwards.

L Limits searches to the currently marked block.

U Ignores case; treats all alphabetic characters as if they were uppercase.

W Searches for whole words only; skips matching patterns embedded in other words.

If the text contains a target matching the search string, the cursor is positioned just beyond it.

Note: An additional option is available for the change operation.

N: Without the "N" option, before changing each string, a prompt "Replace Y/N/A/Q" will allow the operator to

Y: Yes - change it

N: No - don't change it

A: REPLACE ALL without prompting (used with "G" global option)

Q: QUIT - stop (used with "G" global option)

ESC: does the same thing as quit

With the "N" option, no prompt is displayed before it changes the string or strings.

F9 (FNext) Repeats the last search operation. If the last search command called for a Find Operation, the same search string and options will be repeated; for a Find-and-Replace Operation, the replacement string will be reused as well.

F5 (Ins) Main-Prog-Text-Edit-Ins

If the F5 (Ins) key is on, the editor is in insert mode and characters will be inserted at the cursor position. If the F5 (Ins) key is off, the editor is in overwrite mode and characters will overwrite any previous character at the cursor position.

F6 (Del) Main-Prog-Text-Edit-Del

Deletes the character under the cursor and moves any characters to the right of the cursor one position to the left. This command does not work across line breaks.

Note: At any time while in the editor, pressing the ← (backspace) key on the numeric keypad will move the cursor one character to the left and delete the character positioned there. Any characters to the right of the cursor are moved one space to the left.

F7 (↑) Moves cursor up one line.

F8 (↓) Moves the cursor down one line.

F9 (←) Moves the cursor to the left.

F10 (→) Moves the cursor to the right.

ESC (Exit) Upon pressing the ESC (Exit) key to leave the editor, the active edit program is checked to determine if it was modified. If it was, a prompt will be displayed in the message window asking if the changes should be accepted and stored. Pressing "Y" will accept the changes and alter the program file. Pressing "N" will abort the changes and leave the file unchanged. ESC will return to the editor.

HELP (Verf) Main-Prog-Text-Edit-Verf

When editing a text file, you can verify the program you are editing.

The HELP (Verf) key will not be active if you are currently running or verifying a program. When you press the HELP (Verf) key, the program will dry run verify with a zero radius tool, just as if you had verified the program using the F5 (Part) option. Feed moves are shown as yellow. Rapid moves are not shown. After the program is verified, it will be auto-scaled. You will be looking at the F6 (Displ)-F3 (Graph) screen. You can then rotate, scale, zoom, etc. If you press ESC you will return to the text editor.

Notes: *When the program is being verified, it will ignore M6s, M0s, M1s, INPUT statements, etc. The program is copied to a file in the parts directory called "TEXTVER". No checks are done for out-of-parts space when the file is copied. The "TEXTVER" file is deleted when you return to the edit screen. This is done so that it will not automatically save the program you are working on (it will still prompt you with the* "Program was Modified. Accept changes? (Y/N)" *message.*

The HELP(Verf) key will not show up if you are in the next level down such as F1(Block), F2(Cursr), F3(Words), or F4(Misc).

If the program takes a long time to verify, you can press the ESC key while it is verifying. It will auto-scale what it has verified and put you at the F6(Displ)-F3(Graph) screen. The program will continue to verify until it ends or until you press ESC again.

F2 (New) Main-Prog-Text-New

The F2 (New) key will allow entry of a number for a new text program. After the number has been entered, the control will check the text programs to see if a text program by that number already exists. If one is found, a warning will be displayed and the operator will be allowed to okay the erasing of the text program prior to entering the text editor.

F3 (Old) Main-Prog-Text-Old

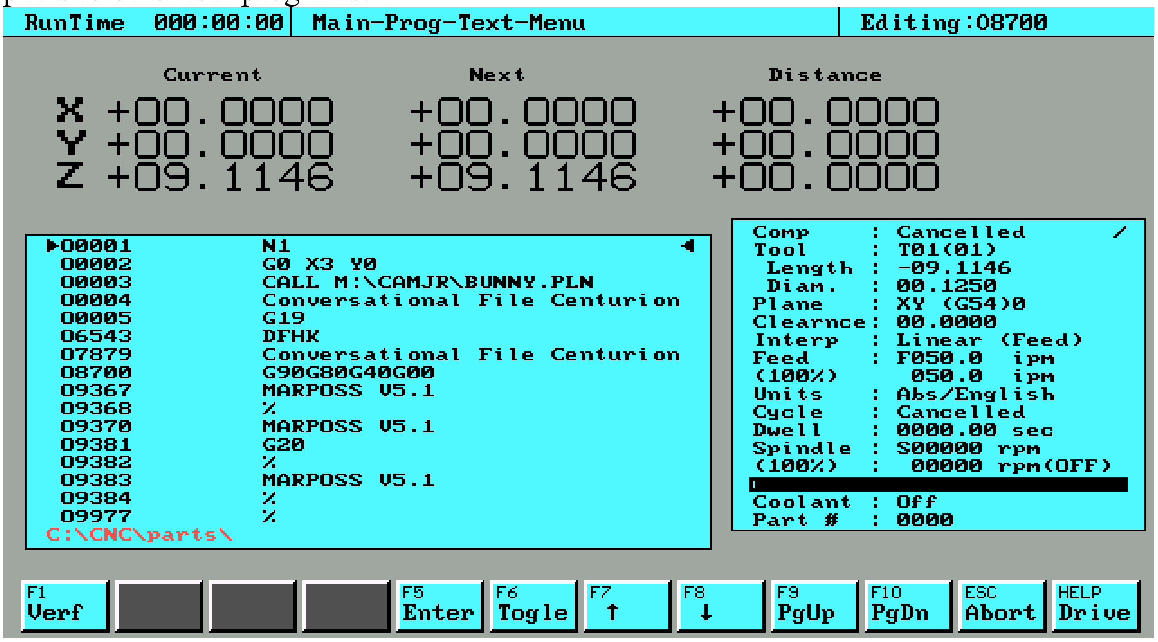
The F3 (Old) key will allow entry of the number of an existing text program. After the number has been entered, the control will check the text programs currently in the parts directory to see if a program by that number already exists. If one is found, the edit window will appear displaying the first 16 lines of the program. If one is not found, an error stating such will appear and, after pressing the ESC key, another number may be entered.

F4 (Any) Main-Prog-Text-Any

Pressing the F4 (Any) key will prompt the user to enter the file to be edited. Specify the drive, path, file name, and extension.

F7 (Menu) Main-Prog-Text-Menu

The F7 (Menu) key will display a list of all text programs currently in the parts directory. By using the F7 - F10 keys, file selection arrows are positioned at the program to edit, and the F5 (Enter) key is pressed to make a selection. See Section 4 for information on changing drives and paths to other text programs.



The edit window will appear displaying the first 16 programs. To select one of the programs listed in the window, use the arrow and page keys to move the cursor to the desired program and press F5 (Enter) – or Enter – on the keyboard. The Menu function can be called from other screens but works the same way from all. When called from the Verify screen, the selected program becomes the active program being edited. F1(Verf) will graphically verify the part the cursor is on. The graph shown will be for a 0 diameter tool. F6(Togle) will display the file size, last time and date it was edited, and seven characters for the file name. Subdirectories are specified by <DIR> as the file name. The ".." specifies the parent to the current directory. If you press the HELP(Drive) key, a list of the available drives is displayed.

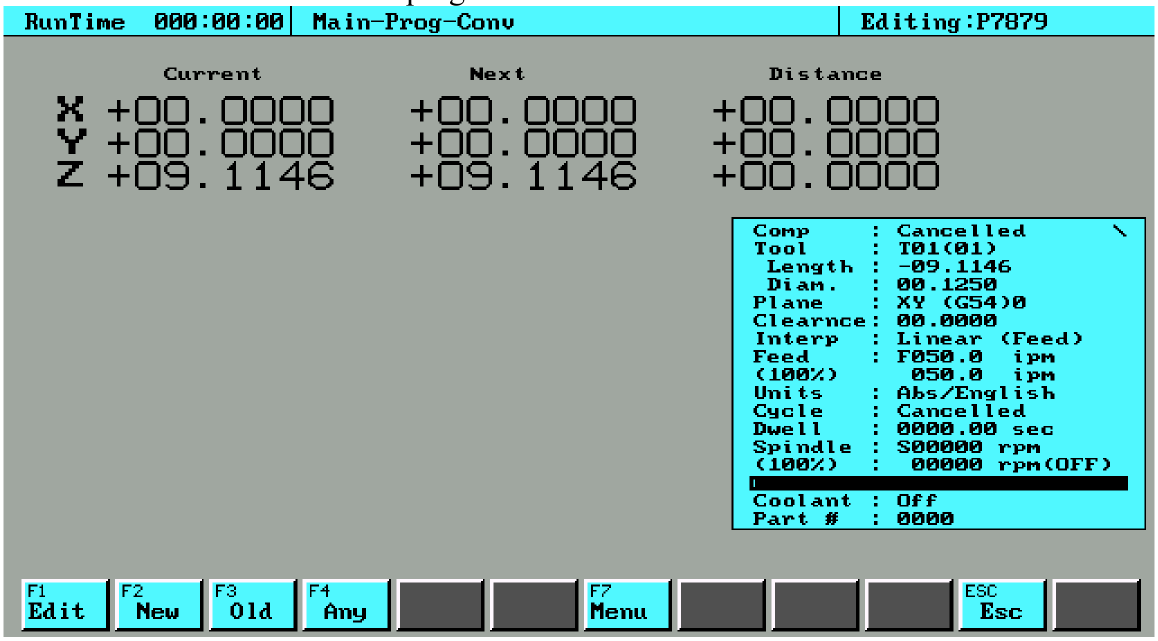
Note: The file name consists of the characters included in parentheses. If there are no parentheses on the first block, the name is the first block.

Pressing the "D" key toggles the menu to and from full DOS names. Pressing the "N" key toggles sorting by file name or file description.

Regardless of which selection mode (Edit, New, Old, Any, or Menu) is used, whenever the edit window is displayed, the number of the text program being edited will be shown in the active window.

F2 (Conv) Main-Prog-Conv

The following discussion will deal with selecting conversational programs. Upon entering the conversational programming mode, the active window in the upper right-hand corner will change to show the last conversational program edited.



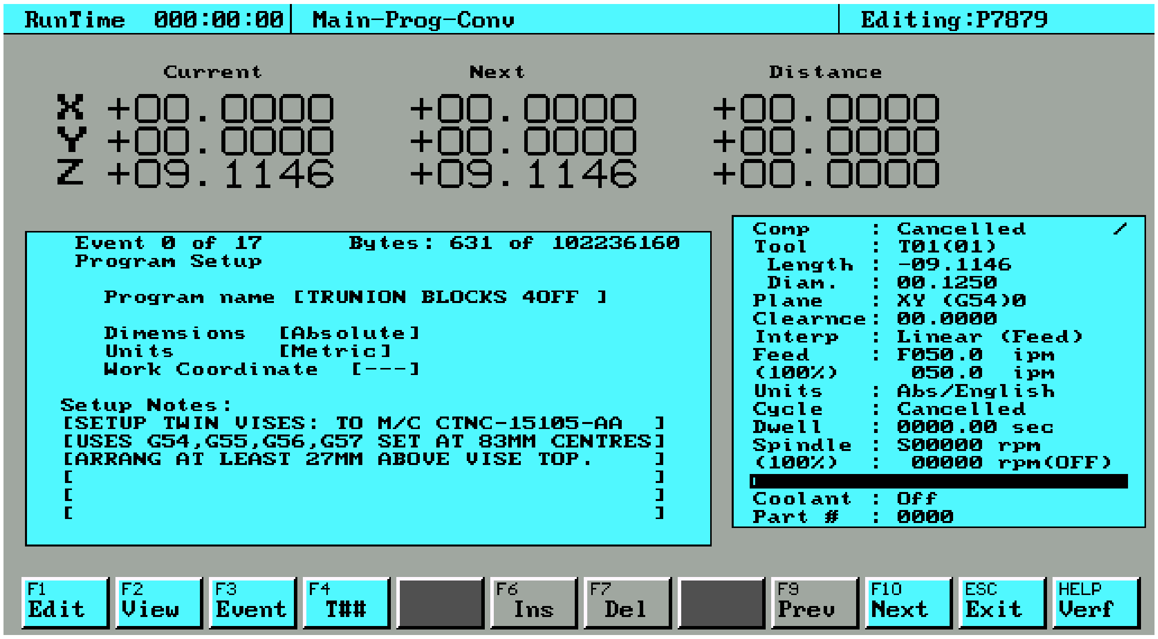
Five options are available from which to choose the conversational program that is to be edited.

F1 (Edit) Main-Prog-Conv-Edit

Pressing the F1(Edit) key will select the active conversational program and enter the conversational programming system.

While programming or editing in the conversational system, three types of soft key configurations will be encountered. They are:

- Soft key configuration 1: Edit Keys

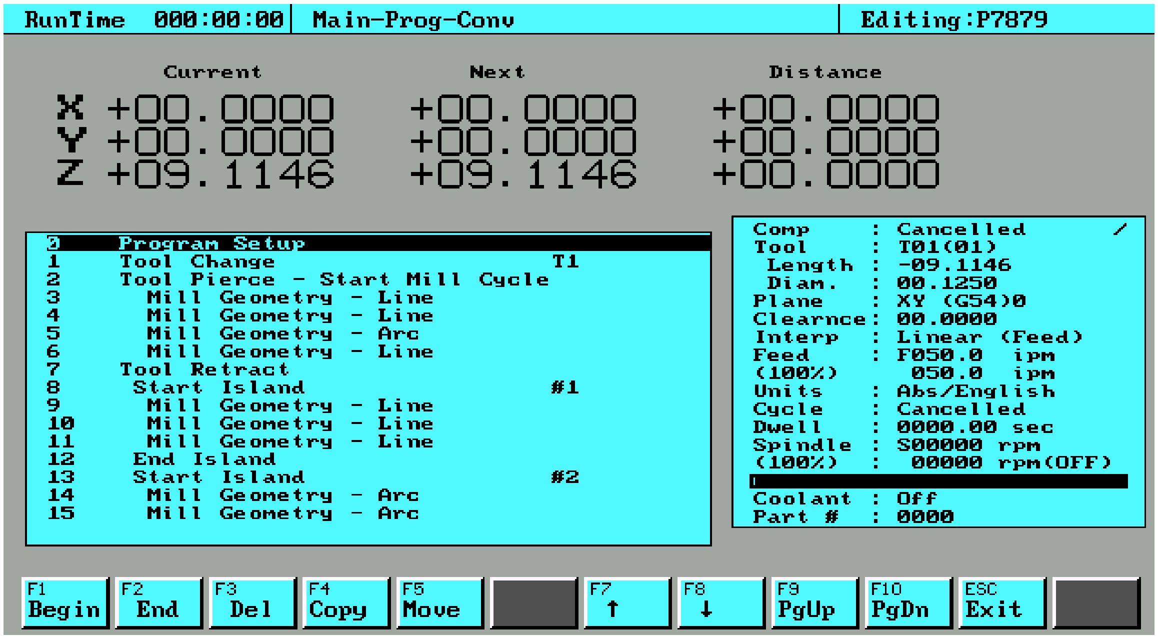


These function keys are available whenever the edit menu system has not been entered. It is possible to step through the program, edit events, and insert or delete events.

Definitions of the Edit Keys

F1 (Edit): Pressing the F1(Edit) key will position the cursor at the first field of the current event. The store/input keys will appear and the event may be edited.

F2 (View): Allows viewing of the entire program and lets the operator position to any of the events in the program. A window similar to the following will be displayed.



F7 (↑), F8 (↓), F9 (PgUp), and F10 (PgDn) can be used to move to the desired event. Then press ESC or Enter to display that event.

F1 (Begin): Marks the block beginning

F2 (End): Marks the block end

F3: Deletes the block of events

F4 Copies the block of events to the cursor position

F5 Moves the block of events to the cursor position

Note: You cannot move, delete, or copy the program setup screen or the end of program events.

F3 (Event) Allows entry of an event number for which to search. If the event number is not found, the end-of-program screen will be displayed.

F4 (T##) Allows entry to search for a tool number. If tool number is not found the event displayed will not change.

F6 (Ins) INS is used to insert events in a program. The new event(s) will be inserted before the event that is currently displayed. Inserting will continue until the F10 (Exit) soft key is pressed.

F7 (Del) Will delete the event currently being displayed.

F9 (Prev) Displays the previous event in the program file.

F10 (Next) Displays the next event in the program file.

Help (Verf) When editing a conversational file, you can verify the program you are editing. The Help key will show Verf when it is active.

The HELP (Verf) key will not be active if you are currently running or verifying a program. When you press the HELP (Verf) key, the program will dry run verify with a zero radius tools, just as if you had verified the program using the F5 (Part) option. Feed moves are shown as yellow. Rapid moves are not shown. After the program is verified it will be auto-scaled. You will be looking at the F6 (Displ) - F3 (Graph) screen. You can rotate, scale, zoom, etc. If you press ESC, you will return to the conversational editor.

Notes on HELP (Verf)

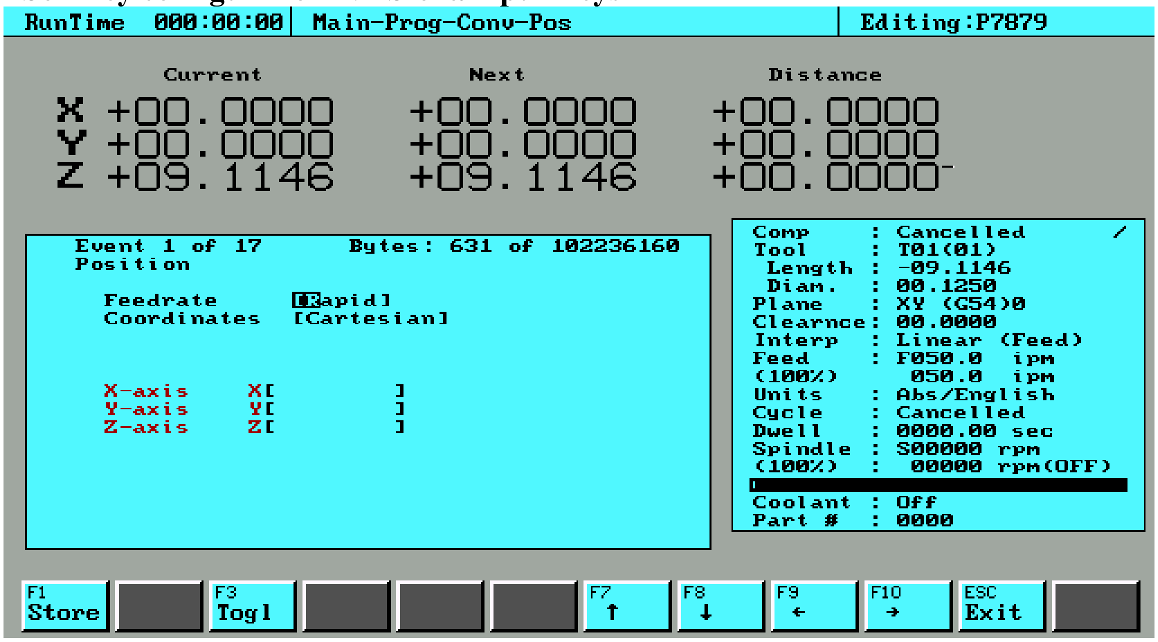
Note 1: *When the program is being verified, it will ignore M6s, M0s, M1s, INPUT statements, etc.*

Note 2. *If the program takes a long time to verify, you can press the ESC key while it is verifying. It will auto-scale what it has verified and put you at the F6(Displ)-F3(Graph) screen. It will then continue to verify until it ends or you press ESC again.*

Note 3. ***Whiles** without **wends** and **wends** without **whiles** will not give error messages.*

ESC (Exit) Exits the conversational system and automatically creates the executable text program with an O prefix.

- Soft key configuration 2: Store/Input Keys



These soft keys will be available whenever input is expected. At this time, a screen containing any number of fields will be displayed and the cursor will be positioned in one of the fields.

There are two types of fields, which may be displayed on an input screen: data and toggle. Data fields are fields in which data is entered using the keypad. Fields that are red require entries. Toggle fields are fields that have a limited number of possible input values and whose value may only be changed by pressing the F3 (Togl) key.

Definitions of the Store/Input Keys

F1 (Store) Accepts the entries and adds to the program file. If all required data has not been entered, the F1 (Store) key will not work and the cursor will position to the field requiring input. Each screen stored is called an event.

F3 (Togl) Pressing this key will result in the next toggle value being displayed in the field. It will not be present in a data field.

F5 (Del) Used to delete an entry from a data field. It will not be present in a toggle field.

F7 (↑) Moves cursor to the previous field.

F8 (↓) Moves cursor to the next field.

F9 (←) Moves cursor to the left. Has no effect in a toggle field.

F10 (→) Moves cursor to the right. Has no effect in a toggle field.

ESC (Exit) Will abort input and return to the menu keys.

F2 (New) Main-Prog-Conv-New

Pressing the F2 (New) key will allow entry of the number for a new conversational program. After the number has been entered, the control will check the conversational programs currently in the parts directory to see if a program by that number already exists. If it does, two warnings will be displayed and the operator will be allowed to okay the erasing of the conversational program and its associated text program prior to entering the conversational system. Remember, conversational programs are stored in both formats.

F3 (Old) Main-Prog-Conv-Old

The F3 (Old) key will allow entry of the number of an existing conversational program. After the number has been entered, the control will check the conversational programs currently in the parts directory to see if a program by that number exists. If it does, the edit window will appear displaying the program setup screen. If not, an error stating that the program was not found will appear and, after pressing the ESC (Exit) key, another number may be entered.

F4 (Any) Main-Prog-Conv-Any

The F4 (Any) key will allow entry of any file. Specify the drive, path, file name, and extension. If the name is of the correct format for a conversational program, you will enter the conversation program mode. If not, you will enter the text programming mode.

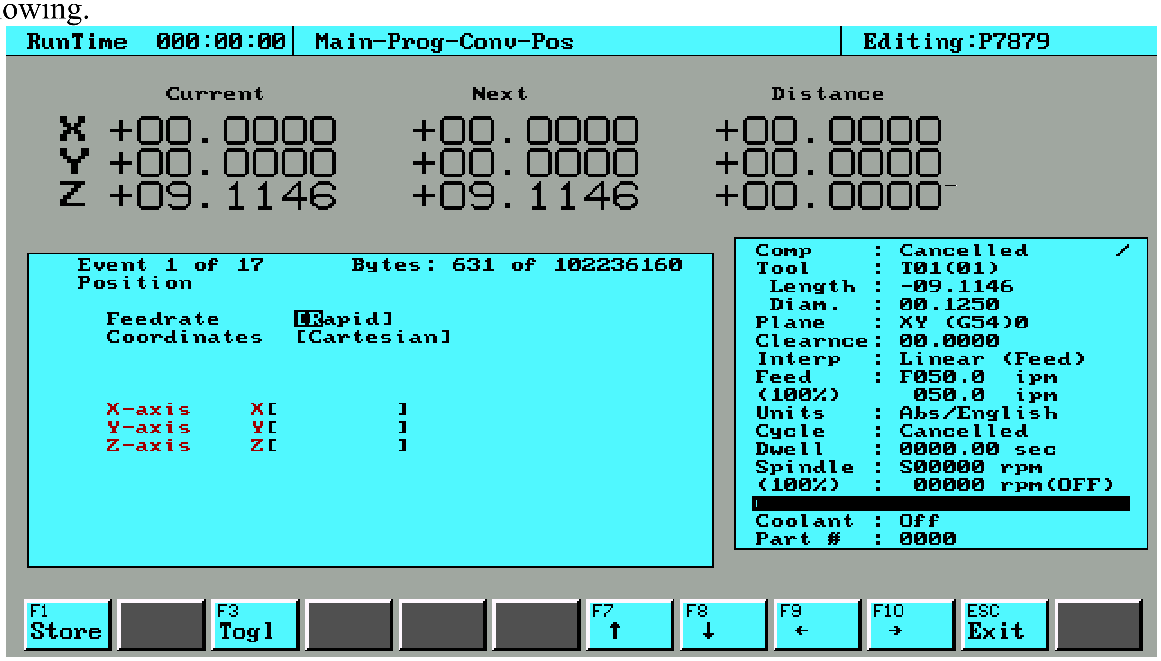
Main-Prog-Conv

The menu keys are used to move throughout the conversational system and reach the desired input screen. The menu key sequence is listed on the Conversational System Flowchart in Section 6.

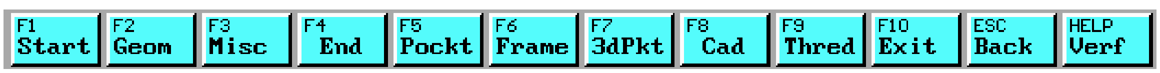
- Soft key configuration 3: Programming Keys

Keys displayed in the first level menu:

Pressing a function key will either bring up an input screen [e.g. F1 (Pos)] much like the following.



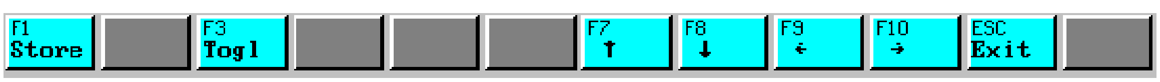
or another menu [e.g. F2 (Mill)] like this:

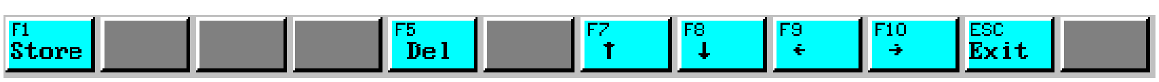


Notice that on all levels except level 1 there is an ESC (Back) key. This key will return you to the previous level menu keys.

Pressing the F10 (Exit) key will exit the menu subsystem and display the level 1 Edit keys.

When an input screen is encountered the following Store/Input keys appear.



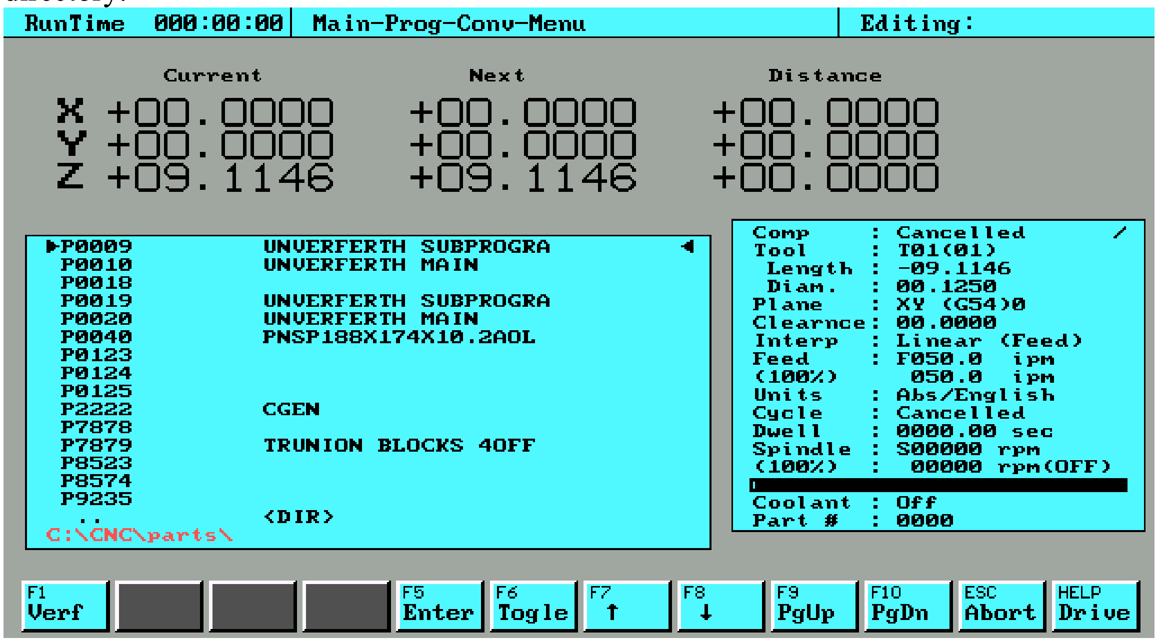


Note: F3 (Togl) and F5 (Del) cannot be on the same screen.

See SECTION THREE - CONVERSATIONAL INPUT SCREENS *for more detailed information on the conversational input screens.*

F7 (Menu) Main-Prog-Conv-Menu

The F7 (Menu) key will display a list of all conversational programs currently loaded in the parts directory.



By using F7 - F10 keys, file selection arrows are positioned at the program to edit, and the F5 (Enter) key is pressed to make the selection. The edit window will appear displaying the program setup screen.

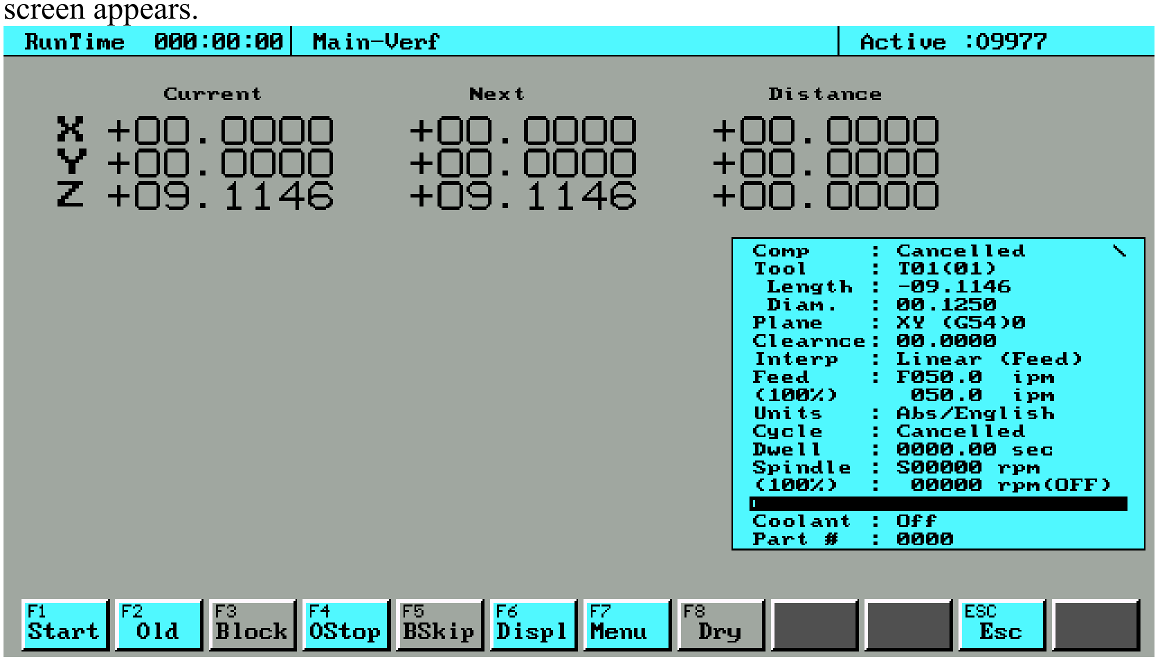
Regardless of which selection mode [F1 (Edit), F2 (New), F3 (Old), F4 (Any), or F7 (Menu)] is used, whenever the edit window is displayed the number of the conversation program being worked with will display in the editing window.

F6 (Togle) displays the time and date for each program.

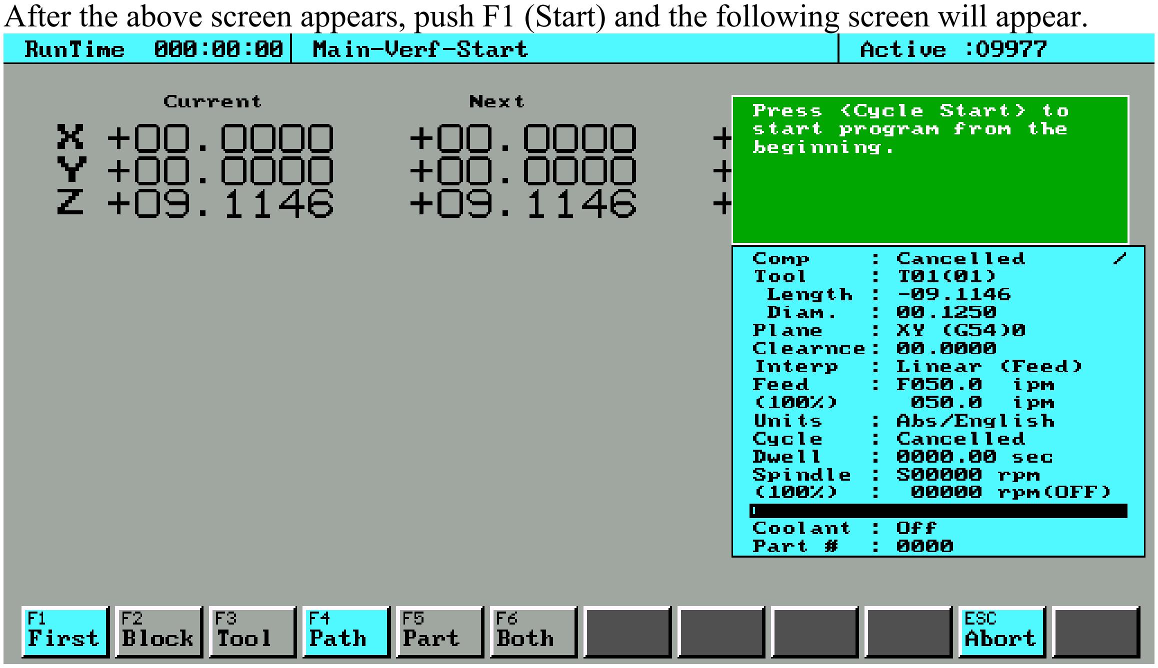
F9 (Verf) Main-Verf

The F9 (Verf) function is used to verify part programs. Most verification is done in the graphics mode, but it doesn't have to be. The times are valid during verify and can be used to estimate machining times. The program that is verified is the active program. To get coordinate information to compare against a print, put the control in block mode and step through the program. The cursor in the graphic mode will step around the part, and the X Y Z display will read out the coordinate values of each point.

Verify is used to verify the active program. Upon pressing the F9 (Verf) key, the following screen appears.



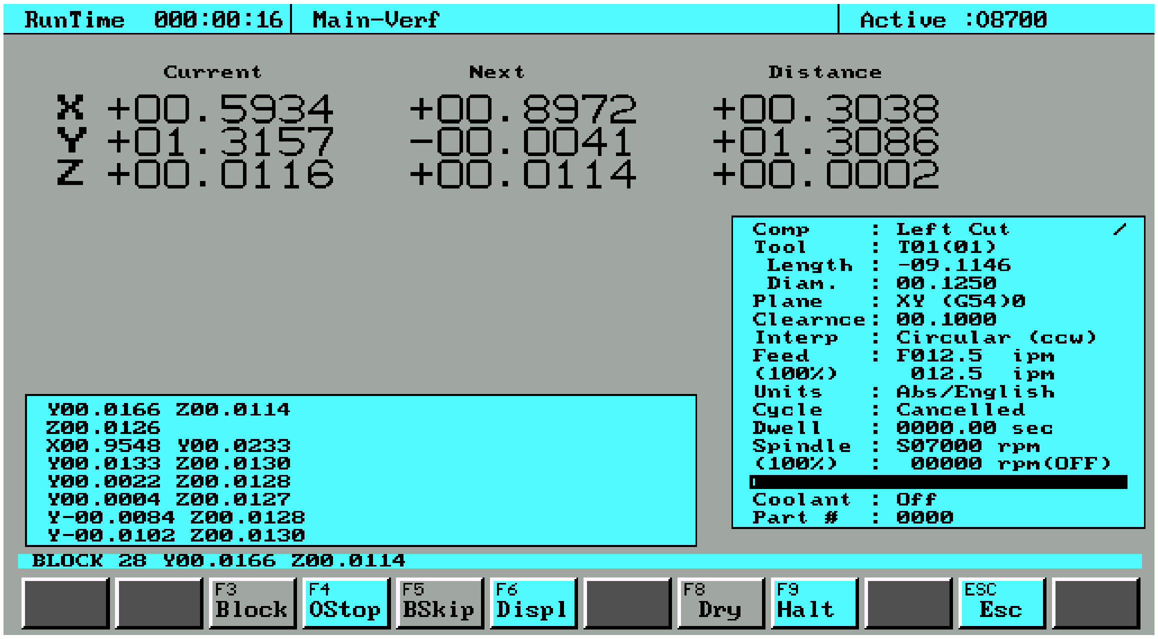
After the above screen appears, push F1 (Start) and the following screen will appear.



The F1 (First) key is automatically selected when entering this screen from the Verf screen. Therefore, if it is desired to verify the active program from the beginning, you need only to press the **Cycle Start** button. If F2 (Block) is pressed, the control will request that the desired block or sequence number be typed in followed by enter. If **Cycle Start** is pressed, the active program will start verifying from the selected block number. If F3 (Tool) is pressed, the control will request a tool number. After typing the tool number followed by depressing **Enter**, the

Cycle Start button is pressed. The active program will start verifying at the desired tool number and the following screen will appear. The F4 (Path) key will show the tool path on the graphics screen, which is the default. The F5 (Part) key will show the part path on the graphics screen. The F6 (Both) key will show both the part path and then the tool path on the graphics screen.

Note: If the block # or tool # typed in is not located within the program, the program will start from the beginning.



The screen above is the basic verify screen with two new additions: a block number display and F9 (Halt). The block number shows the current line being executed as the program verifies. The F9 (Halt) key is similar to **Feedhold** in that when pushed the machine will stop. However, unlike **Feedhold**, F9 (Halt) also allows a new program to be started.

F3 (Block) Main-Verf-Block

When the F3 (Block) switch is activated, the program will stop at the end of each block. Each time the **Cycle Start** button is pushed one more block will be verified.

F4 (OStop) Main-Verf-OStop

When the optional stop key F4 (OStop) is activated, the program will stop at each M01 command.

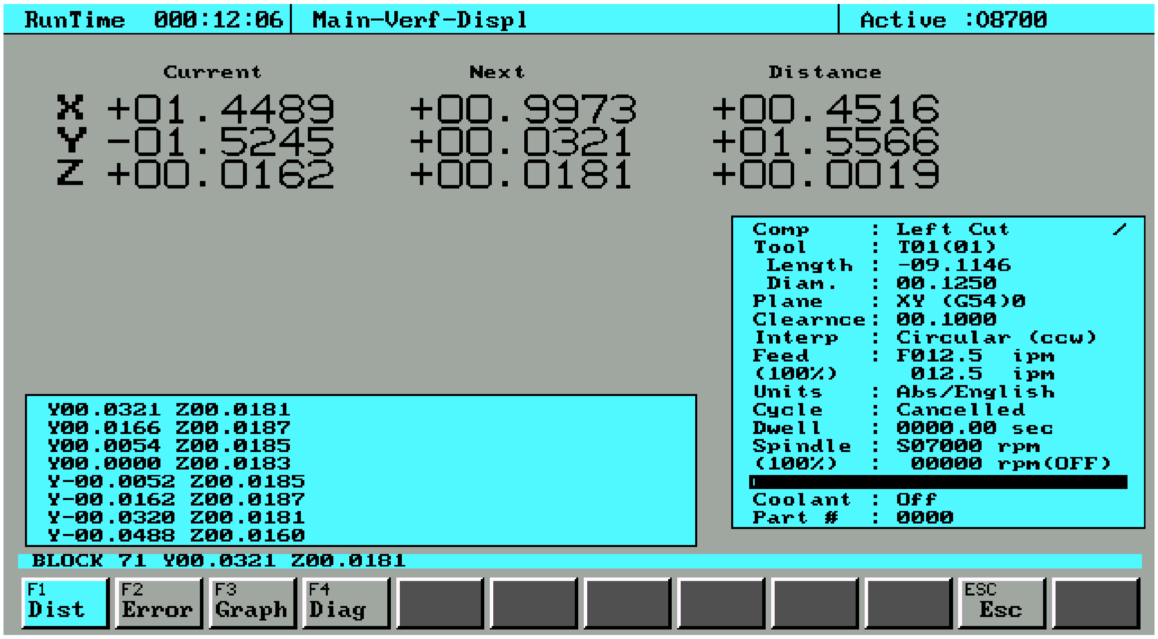
F5 (BSkip) Main-Verf-BSkip

When the block skip switch F5 (BSkip) is activated, the program will skip all blocks started with a / (slash).

/F5 When the block skip switch is active the Feedrate change command will not be executed.

F6 (Displ) Main-Verf-Displ

The F6 (Display) key can be accessed from a number of screens. The following screen is shown as if the F6 (Displ) was entered from the F9 (Verf) screen. All the display functions and screens are identical, independent of the entry point. Only the return point differs based on the original entry point. When the F6 (Disp) key is pressed, the following screen will appear.



F1 (Next) Main-Verf-Displ-Dist

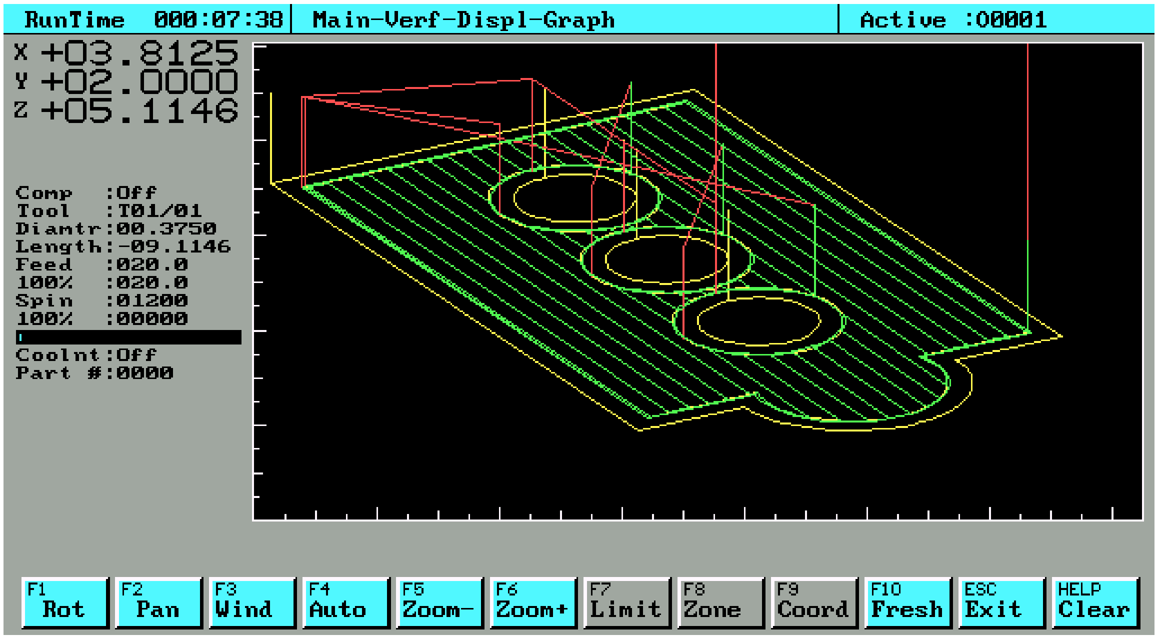
See page 23, Section 2.

F2 (Error) Main-Verf-Displ-Error

See page 24, Section 2.

F3 (Graph) Main-Verf-Displ-Graph

If the F3 (Graph) key is activated, the control switches from displaying text to a graphic display of the active part program. The following screen will appear.



The graphics on this control are full 3D and will be displayed in the graphics area as long as the control remains in the F3 (Graph) mode. When other displays are requested, windows will appear in the graphics area showing the requested data. When these functions are concluded, the windows will disappear and the graphic display will be reinstated. The scale at the top of the screen is to be used as a reference for the part size. As the screen scale is changed, the graduations on the ruler will change accordingly. The ruler graduations are in machine units, but on an English system. The largest graduations equal approximately one inch.

Note: The graphics are cleared from the display at the beginning of running or verifying a program.

The graphics functions used in the verify mode are the same functions used in the run mode. For a full explanation of these functions, see page 24, Section 2 on runtime graphics.

F4 (Diag) Main-Verf-Disp-Diag

See page 28, Section 2.

F8 (Dry) Main-Verf-Dry

F8 (Dry) run in the verify mode will run the program as fast as possible. For feedrate override positions 100% and greater – verify speeds are progressively slower for overrides 0-90%.

F9 (Halt) Main-Verf-Halt
F9 (Resum) Main-Verf-Resum

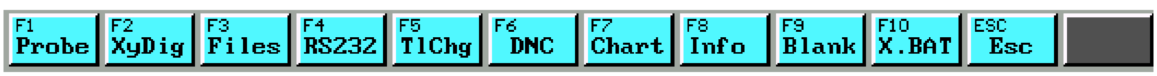
Once a program has been F9 (Halt)ed, the resume feature of the control becomes active. The F9 (Resum) key will now be displayed on the verify screen. A program can be resumed as long as the resume is active. If the resume function is selected, the active program will be resumed at the halted point.

The F3, F4, F5, and F8 keys on the verify screen set the mode of operation in which a program will verify. When these keys are in a highlighted state the functions will be active in any currently verifying program or program to be verified.

Note: You can switch to run from verifying. This is convenient for long DNC programs as it picks up on all feeds/speeds/tool offsets/cutter comp/etc. To switch to run from verifying, you must be in block mode waiting for cycle start. Press the F9 (Halt). The F10 key will no show F10 (Run). When the F10 (Run) is pressed, the control will be as if a halt resume was done while running a program. Be sure to take the control out of block and/or dry run if you are in it. <u>Do not try this in the middle of automatic tool changes or other I/O related routines.</u>

F10 (Util) Main-Util

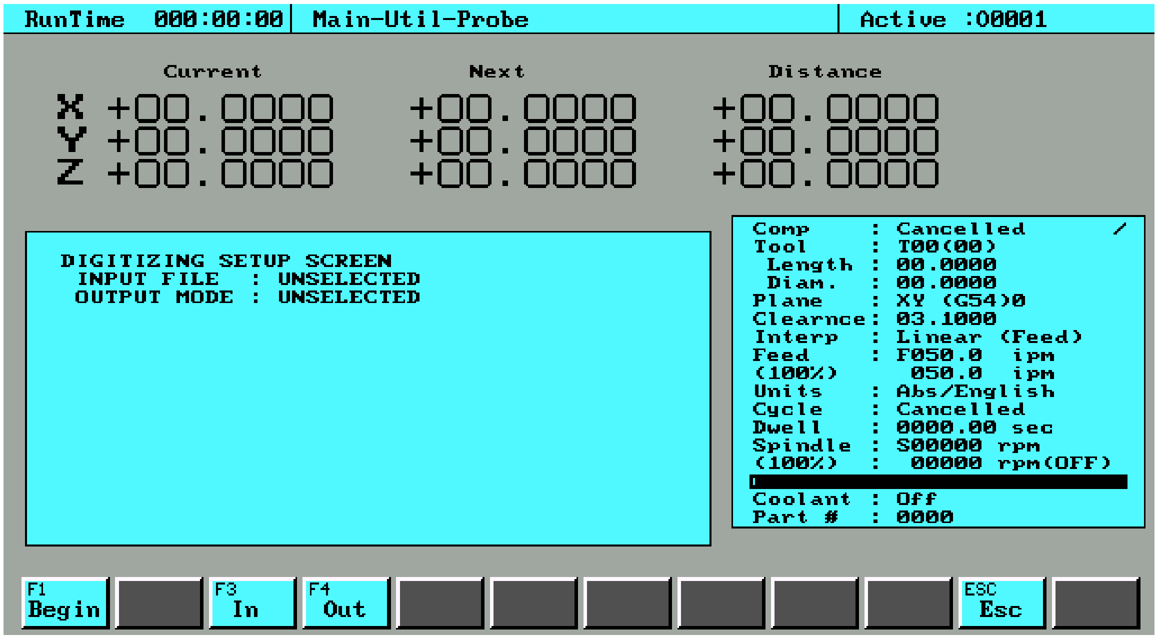
Keys and options displayed when F10 (Util) is pressed:



F1 (Probe) Main-Util-Probe

Note: For machines with the Digitizing option.

If Digitizing is an option and F1 (Probe) is pressed, the following screen will be displayed.



Note: F1 (Probe) is used for XZ or YZ digitizing only.

F1 (Begin) If the input file and output mode have been selected, digitizing will begin. If not, error message 808 *"set up not selected"* is displayed.

F3 (In) Displays a menu from which to select a file that contains the guidance information for digitizing.

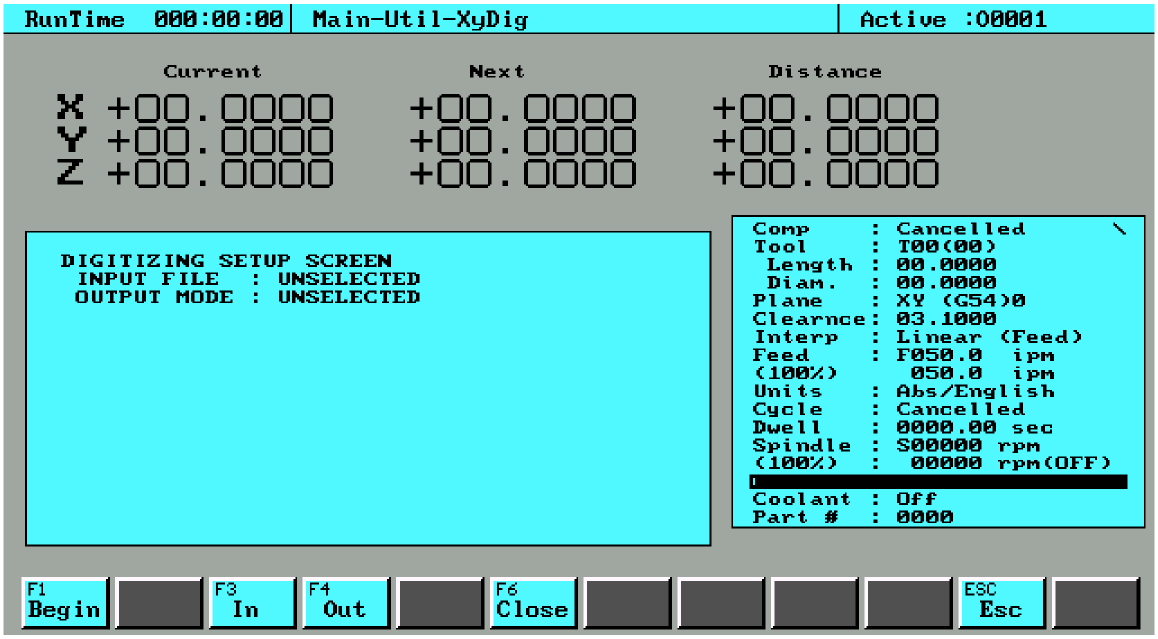
F4 (Out) Specifies the output mode for the digitized data. The options are as follows.

F1 - RS232	The primary serial port
F2 - File	A CNC file number
F3 - Disk	A floppy disk file name

F2 (XyDig) Main-Util-XyDig

Note: For machines with the Digitizing option.

If Digitizing is installed and F2 (XyDig) is pressed, the following will be displayed.



F1 (Begin) If the input file and output mode have been selected, digitizing will begin. If not, error message 808 *"set up not selected"* is displayed.

F3 (In) Displays a menu from which to select a file that contains the guidance information for digitizing.

F4 (Out) Specifies the output mode for the digitized data. The options are as follows.

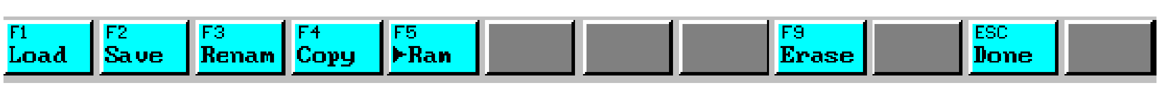
F1 - RS232	The primary serial port
F2 - File	A CNC file number
F3 - Disk	A floppy disk file name

F6 (Close) Will toggle between close and open.

Note: More information on digitizing is available in the digitizing manual.

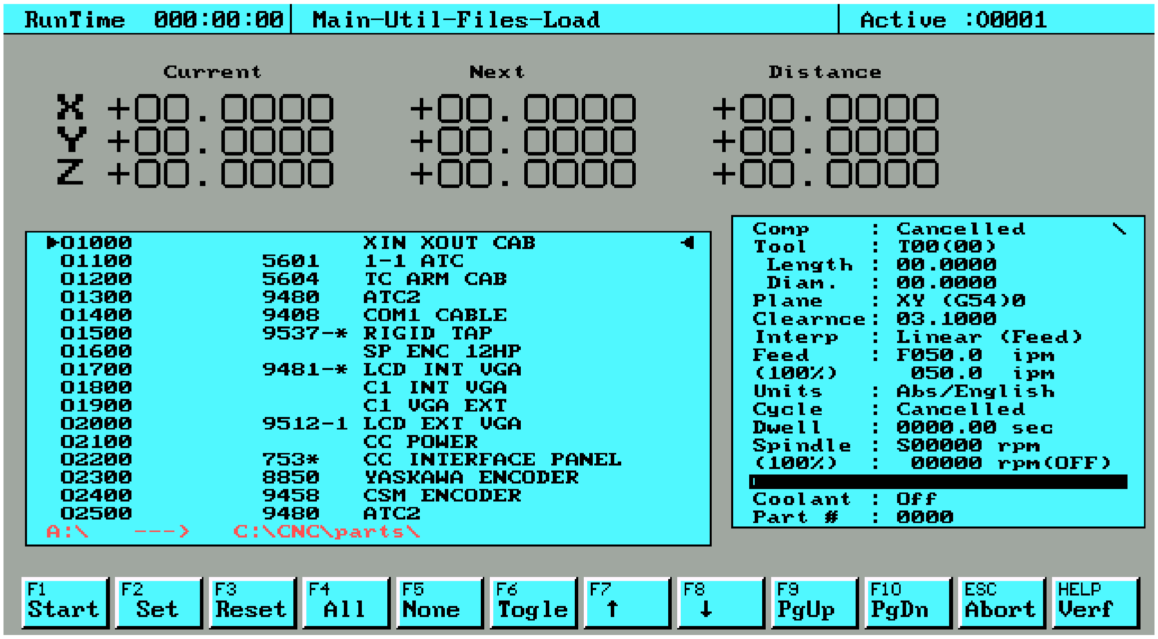
F3 (Files) Main-Util-Files

The file utilities contain basic program manipulation functions. They are:



F1 (Load) Main-Util-Files-Load

The F1 (Load) function is used to load programs from the floppy disk into the control's program memory. When this function is selected the following screen is displayed.



The edit window will display a list of the programs on the floppy drive. The selection cursor (> <) is positioned at the first program.

F1 (Start)	Pressing this key will begin the transfer of the selected programs from floppy disk to program memory. The operator will be prompted if the file already exists.
F2 (Set)	Selects the file at the cursor position to be loaded from floppy disk. It will be highlighted.
F3 (Reset)	Unselects the file at the cursor position. Unhighlights, highlighted files.
F4 (All)	Selects all programs on floppy disk to be loaded. Highlights all files.
F5 (None)	Unselects all selected programs. Unhighlights all highlighted files.
F6 (Togle)	Shows size and date of the file.
F7 (↑)	Moves selection cursor up one line.
F8 (↓)	Moves selection cursor down one line.
F9 (PgUp)	Moves selection cursor up 16 lines.
F10 (PgDn)	Moves selection cursor down 16 lines.

The help key will be used to either select a new drive or to verify a program (based on the control parameter)

Help (Drive) Displays a list of available drives for a new menu.

Help (Verf) Will graphically verify the part that the cursor is on.

Note: If the parameter to extract files is set, a single file on the floppy separated by 0#### can be loaded and separated into several files. When extracting files being loaded, the program does not check for files that already exist.

F2 (Save) Main-Util-Files-Save

The F2 (Save) function is used to save programs from the control's program memory to the floppy disk.

The F2 (Save) function operates the same as the F1 (Load) function with the exception that the transfer direction is changed.

F3 (Rename) Main-Util-Files-Rename

The F3 (Rename) function changes the name of a program. The operator will be prompted if the new filename already exits.

F4 (Copy) Main-Util-Files-Copy

The F4 (Copy) function makes a copy of a program under another name. The operator will be prompted if the new file already exists.

F5 (▸Ram) Main-Util-Files-▸Ram

This function is used to load programs from a floppy disk, zip drive or network to the RAM drive and later DNC'ed from the RAM drive.

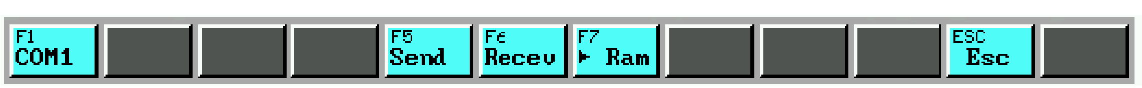
Note: Bigger RAM drives are available as an option.

F9 (Erase) Main-Util-Files-Erase

The F9 (Erase) function is used to erase programs from the control's program memory. The keys available are the same as those in the F2 (Save) option.

F4 (RS232) Main-Util-RS232

The F4 (RS232) utilities contain basic communications functions. They are:

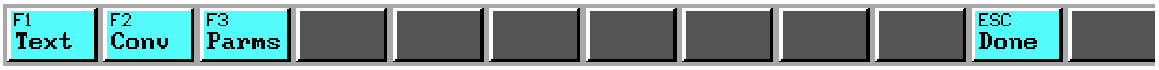


F1 (COM1 or COM 2)

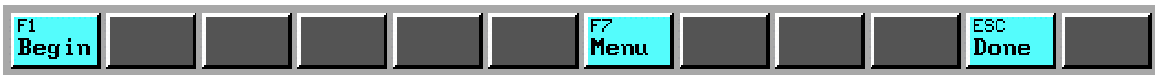
F1(COM1 or COM2) is the primary port.

F5 (Send) Main-Util-RS232-Send

When F5 (Send) is depressed, the following keys appear.



After selecting F1 (Text) or F2 (Conv), select which programs you want to send to the off-line computer from the menu. These actions will display the following menu.



F1 (Begin) starts transmission of the active send program that is shown in the upper right-hand corner.

F7 (Menu) allows selection of programs from a menu to send.

Notes on Send

Note 1: *If sending parameters, F7(Menu) is not an option.*

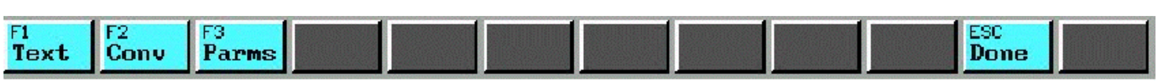
Note 2: *The RS-232 parameters must be set to the same values on both the control and the computer. Always check these parameter settings.*

Note 3: *The RS-232 parameters are found in the control parameters (Main-Parms-Ctrl). Normal parameter settings are:*

Primary Port	*COM1*
Baud rate	*1200 through 9600*
Parity	*Even*
Data bits	*7*
Stop bits	*2*

F6 (Recev) Main-Util-RS232-Recev

The F6 (Recev) option is used to receive programs or parameters from an off-line computer into the control program memory. Upon pressing the F6 (Recev) key, a new set of selection keys will appear as shown below. Then a new program number must be entered in the message window. Once a valid program number has been entered, the edit screen will appear and display the program as it is received. Receiving will continue until the ESC (Done) key is pressed.



Note: If the program number being received already exists, the operator will be prompted.

If the parameter to extract files is set, several files can be sent from an off-line computer; the control will extract them to the correct #### file if they are separated by 0####'s. When extracting, no checking is done for files that already exist.

F7 (▸Ram) Main-Util-RS232-▸Ram

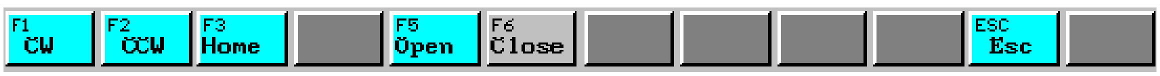
This function is used to load programs from RS-232 to the ram drive and later DNC'ed from the ram drive.

Note: Bigger ram drives are available as an option.

F5 (Tlchg) Main-Util-Tlchg

F5 (Tlchg) is an option for machines with automatic tool changers.

If the machine has a carousel tool changer, the following keys are displayed.

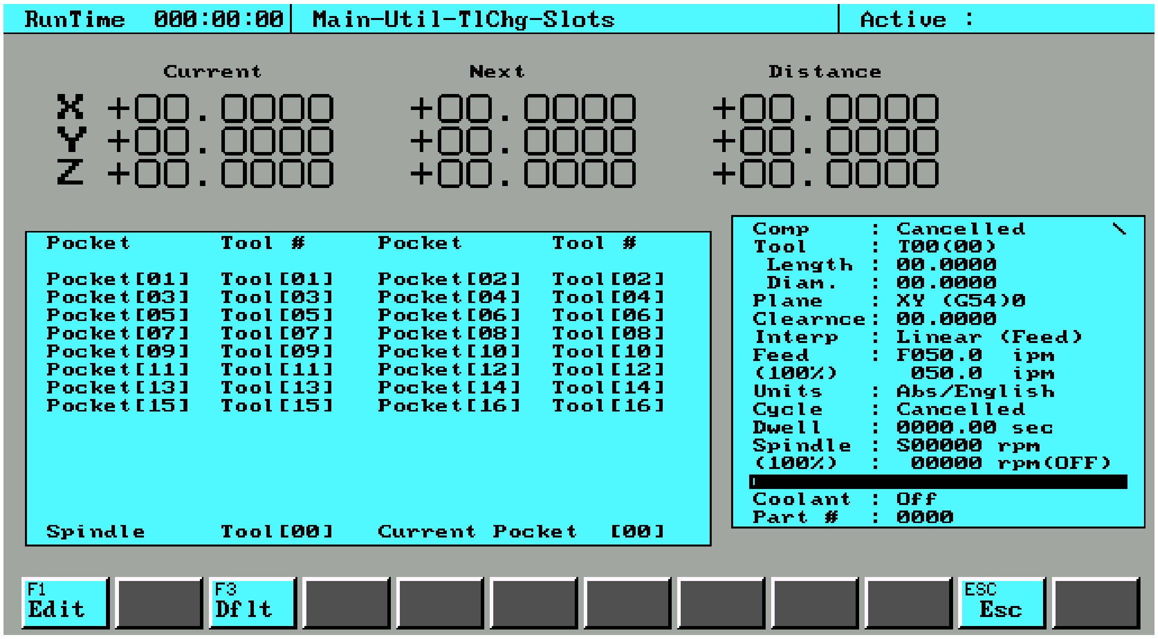


If the F1 (CW), F2 (CCW), or F3 (Home) is pressed, the active tool number is zeroed indicating there is no tool in the spindle.

F1 (CW)	moves the carousel CW one tool position
F2 (CCW)	moves move the carousel CCW one tool position
F3 (Home)	homes the carousel
F5 (Open)	opens the tool changer door
F6 (Close)	closes the tool changer door

If the machine has a swing arm tool changer, the following keys are displayed.

F1 (Home), F2 (CW), and F3 (CCW) perform the same functions as they do on a carousel tool changer. The F7 (Slots) key is used to edit the tool numbers in the tool changer pockets. If F7 (Slots) is pressed, the following screen is displayed.



F1 (Edit) allows you to modify the tool number in each pocket.

F3 (Dflt) loads the default tool numbers (tool 1 in pocket 1, tool 2 in pocket 2, etc). F7 also zeroes the tool number in the spindle.

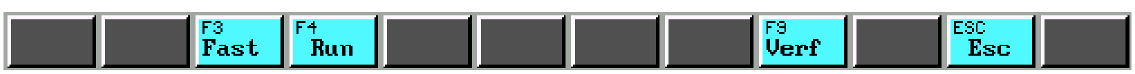
F6 (DNC) Main-Util-DNC

The F6 (DNC) mode is used for running large programs. These programs are not loaded into the control's memory, therefore they cannot have GO-TO's, WHILE-WEND loops, GO-SUB's, or CALLS.

Note: CALLS are allowed in fast DNC fast.

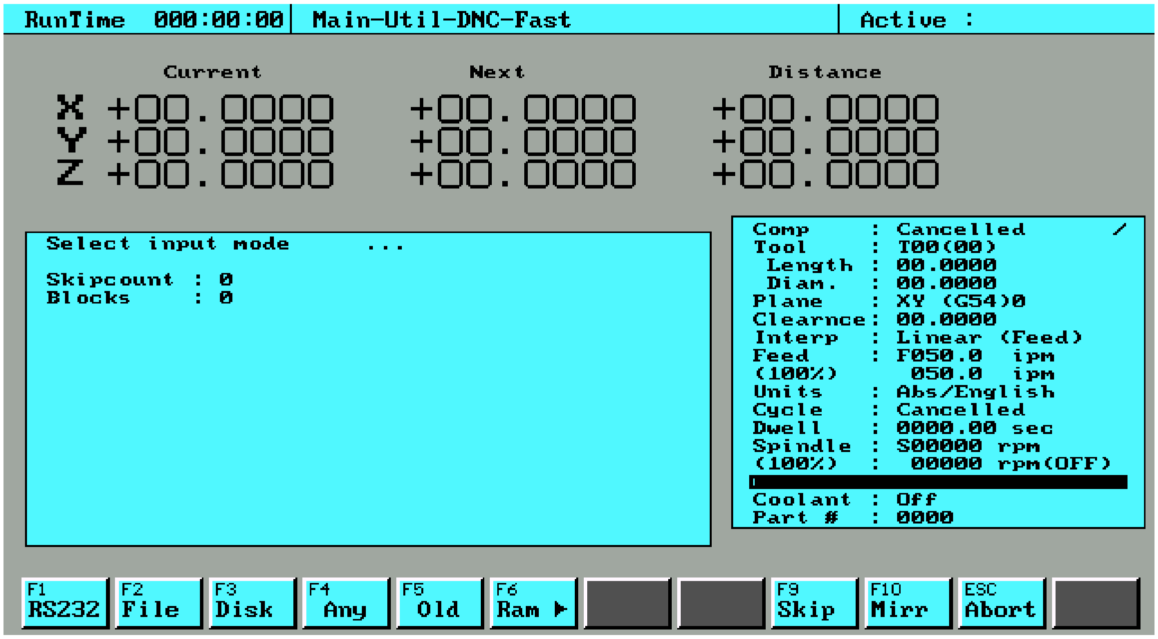
GO-TO's are allowed in run DNC and Verf DNC if the "resolve DNC" parameter is set to Yes.

When F6 (DNC) is pressed, the following DNC modes appear.



F3 (Fast) Main-Util-DNC-Fast

After pressing F3 (Fast), the following screen will appear.



F1 (RS232)

Depressing F1 (RS232) will wait for data from the Com port, then request a cycle start to begin the program.

F2 (File)

F2 (File) selects a program from a menu of the programs in the control.

F3 (Disk)

F3 (Disk) selects a program from a menu of the programs on the floppy disk or from local area network.

F4 (Any)

F4 (Any) allows you to type in the name of any drive, path, file, and extension.

F5 (Old)

F5 (Old) allows you to type in the number of the file you want to run.

F6 (Ram ▶)

F6 (Ram ▶) is used if a program has been loaded to the RAM drive (via RS-232 or floppy disk). F6 (Ram ▶) will select that program.

After selecting the INPUT mode, the control will request a cycle start to begin the program. All of the above input modes – except RS-232 – will allow the program to start at locations other than the beginning. The other options are as follows.

F1 (First)

F1 (First) starts from the beginning.

F2 (Block)

F2 (Block) starts from a sequence number.

F3 (Tool)

F3 (Tool) starts from a tool number.

F4 (Cont)

F4 (Cont) allows starting from a location where DNC was aborted earlier. When a file is aborted the file position is saved. No checking is performed if the operator has switched file numbers or modified the DNC file. F4 (Cont) will continue from the file position that was aborted on any file.

F9 (Skip)

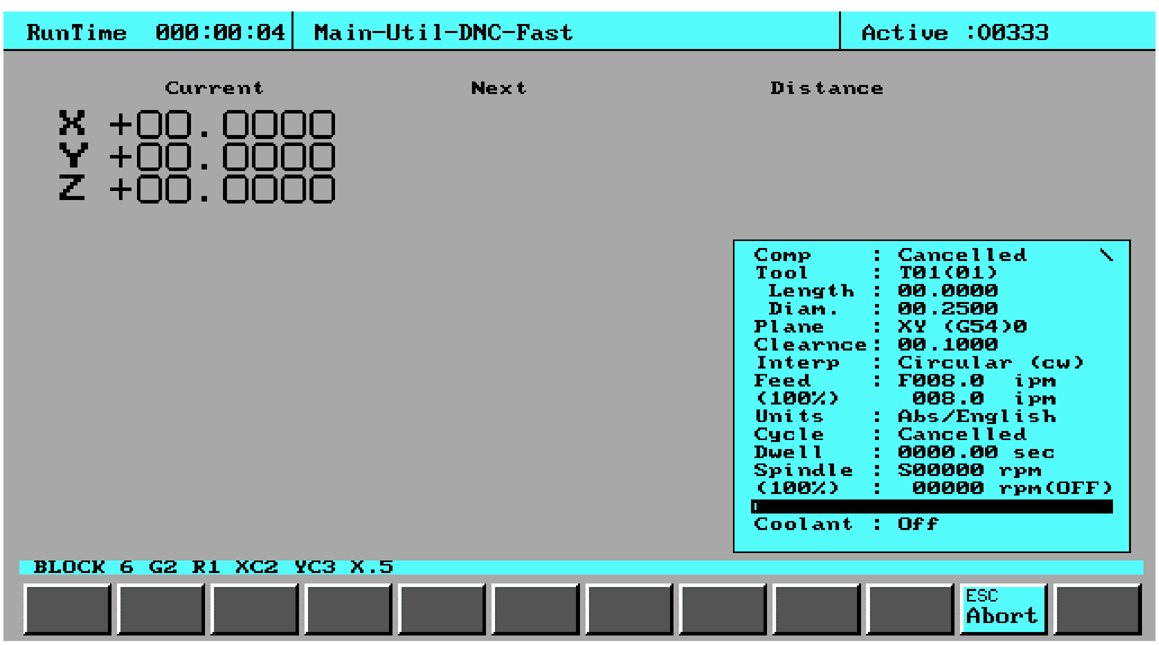
F9 (Skip) will ask for a skip count. If the skip count is non-zero, a cycle start will be requested for skipping (or ignoring) that number of blocks, and then a second cycle start will run from the next block.

F10 (Mirr)

F10 (Mirr) allows mirroring around X0 and Y0 in DNC-FAST.

F3 (Fast)

When F3 (Fast) is running, the following display appears.



When the program ends or ESC (Abort) is pressed, the display will show the total DNC time, total distance, average feedrate, average block time, and average blocks/seconds.

This mode should be used for large programs where a fast block rate is required, for example when making short moves at fast feedrates. No trig help, cutter comp, rotating, scaling, or other non-standard commands, can be done in this mode. Valid data for the fast mode are:

X, Y, Z, A, B, I, J, K, F and N

Valid G codes are:

G0, G1, G2, G3, G17, G18, G19, G70, G71, G90, G91

If a code that is not in these groups is executed, the block time for that block will slow down.

F4 (Run) Main-Util-DNC-Run

F4 (Run) allows the same selections as F3 (Fast) for input modes. F4 (Run) does not have the F10 mirror option. After the input mode has been selected and any blocks have been skipped, the control selects the run screen (Main-Run) and the DNC program can be run similar to any other program.

F9 (Verf) Main-Util-DNC-Verf

F9 (Verf) works identically as F4 (Run) mode. After the input mode has been selected and blocks have been skipped, the control moves to the verify screen (Main-Verf) and the DNC program can be verified similar to any other program.

Note: For F4 (Run) and F9 (Verf) RS-232 DNC mode, a search for a tool number or block number can be done to start a program from a desired location. The program cannot be reset to "first" after a search for a tool or block number..

The option for part/path/both is not available in DNC verify.

F7 (Chart) Main-Util-Chart

The F7 (Chart) key will display help charts created by the end user specific to their applications. If there is a file called charts.dat in the RAM directory, it will be displayed. The format of this file allows an indexing system to other files and data available to the operator.

The following is a sample listing of charts.dat.

Note: The first 16 lines of chart.dat are not displayed.

<u>Speeds and Feeds</u>
(updated on 10-May-94)
See Bill Jones for more information
or to make changes to these charts.

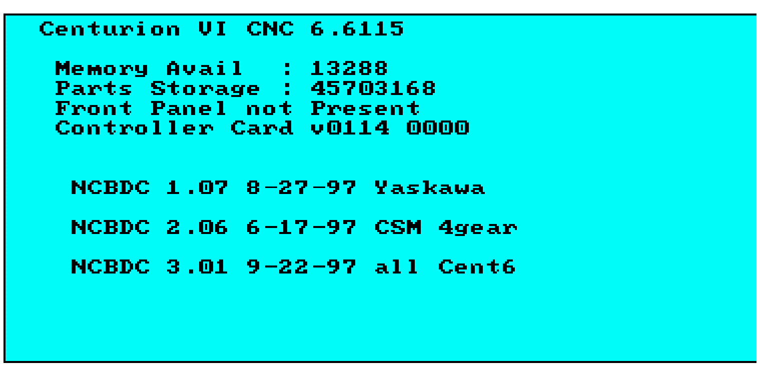
```
Carstl.dat      * Carbon Steel
AllSt.dat       * Alloy Steel
                .
                etc.
                .
                .
Allum.dat       * Aluminum
CstIron.dat     * Cast Iron
Acme.dat        * Jobs for ACME Inc.
```

F8 (Info) Main-Util-Info

F8 (Info) shows information about the software, hardware, etc. The following function keys will be displayed.

F1 (Std) Main-Util-Info-Std

F1 (Std) shows standard information as representative in the following window.



Memory Avail displays the system memory available in bytes.

Parts Storage displays the amount of parts storage in bytes.

Front Panel displays the front panel version code.

Controller Card displays the controller card version (v0206 is 2.06) and an error count (should be zero).

Acroloop information for X, Y, and Z axis for Centurion 5 systems is as follows.

> version, (v0214=2.14)
> error count is noted (should be zero)
> four bytes giving optional acroloop programs loaded (the .HEX files) formatted as:
>
> aamm ssrr, where:
>
> > aa=axis (01=X, 02=Y, etc)
> > mm=major version (00=manual, 01=P1ATC plunger, 07=P7ATC, 08=P8ATC, 11=P1 geneva)
> > ss=sub version; uniquely identifies .HEX file.
> > rr=reserved (always zero)

NCB information for X, Y, and Z axis for Centurion 6 systems is as follows.

> version number
> date of each NCB.HEX file
> a brief description of the file

F2 (Sys) Main-Util-Info-Sys

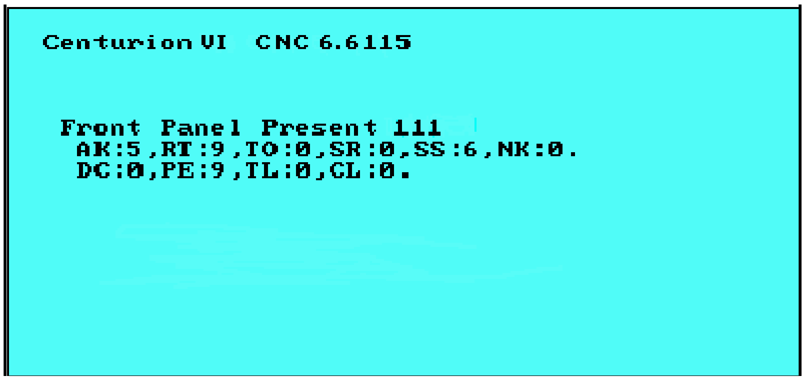
F2 (Sys) shows system information as representative in the following window.

```
Centurion VI CNC 6.6115

 PrefixSeg    = 1DFB:0000
 OvrHeapOrg   = 7073:0000  52780
 HeapOrg      = 7706:0000  06930
 Heap Now     = 8D89:0000
 Max Mem       = A000:0000  28FA0

 MS-DOS version 06.22   80386 CPU
 Bios Date: 08/12/96
 TP7.0
 Blocks =2126,59971,0
 BIOS: FFFF6801806800F0509CFA87EC8B4604
 DEML: FFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFF
 Unknown                  None
```

This screen gives internal information about the system. Lines 1 through 5 show memory allocations to DOS, CNC overlays, and the heaps. Line 6 shows the MS-DOS version and whether the CPU is an 80286, 80386, 80486, or 80586. Lines 11 & 12 show the compiler and blocks pre-allocated for canned cycles, text cycles, and custom M and G codes. Line 13 is a hex dump of the BIOS ROM area at F000:0. Line 14 is a hex dump of the Disk Emulator ROM at CA00:0. Line 15 gives a guess at the Bios type and Disk Emulator type.

F3 (Fp) Main-Util-Info-Fp

F3 (Fp) shows front panel information as illustrated in the figure below.

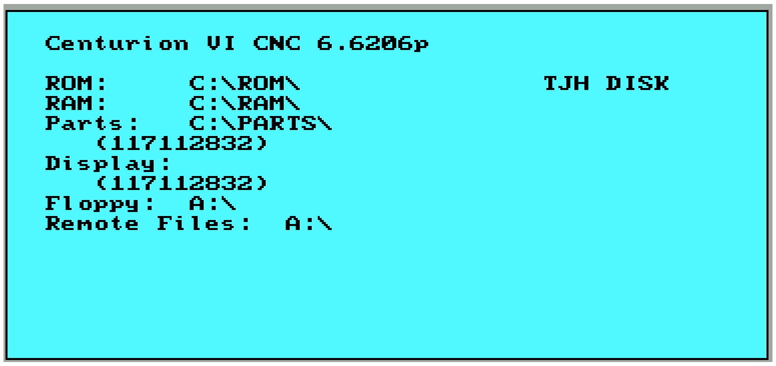


The first line gives the front panel version. The next two lines show several abbreviations and numbers:

AK	Acks received from keyboard encoder/controller
RT	ReTrans received from keyboard encoder/controller
TO	Timeouts on transmissions to keyboard encoder, detected by keyboard controller
SR	Slice resends - timeouts detected by CNC
SS	Slice sends - standard sends to keyboard encoder
NK	Nacks detected by keyboard controller
DC	Spindle DAC value send to keyboard encoder
PE	Parity Errors detected by keyboard controller
TL	Time Limits detected by keyboard controller
CL	Control pressed received by keyboard controller (used to indicate data/command mismatch)

F4 (Path) Main-Util-Info-Path

F4 (Path) displays the following screen, which shows the path file. In the standard order, these are the directories for ROM, RAM, Parts, Display, and Floppy. Below the Parts directory is shown the available parts space in bytes. For the ROM, Parts, and Display directories, the DOS volume ID for that drive is shown to the right.



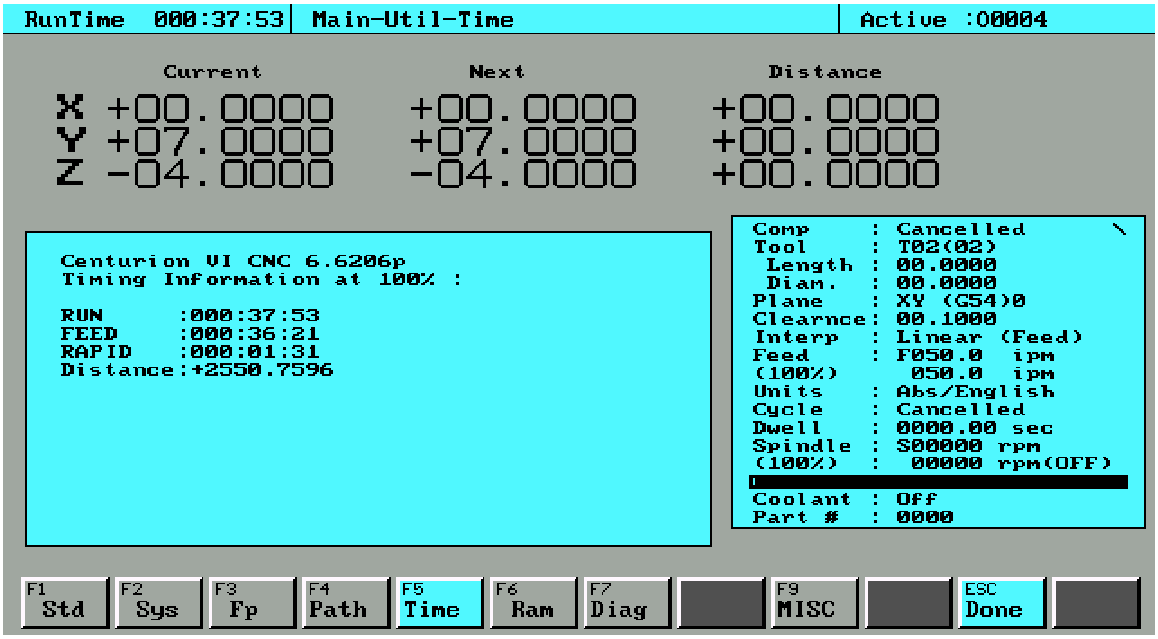
Note: The path file is reloaded when the above information is displayed. Therefore, one can edit the path file then reload it by using this command.

If the paths for parts and/or remote files have been changed from the menus, they will be stored back to the original paths from PATH.DAT.

Version greater than #.#169 have a "Floppy" path and a "Remote Files" path. The "Floppy" path is used specifically for saving and loading parameters.

F5 (Time) Main-Util-Info-Time

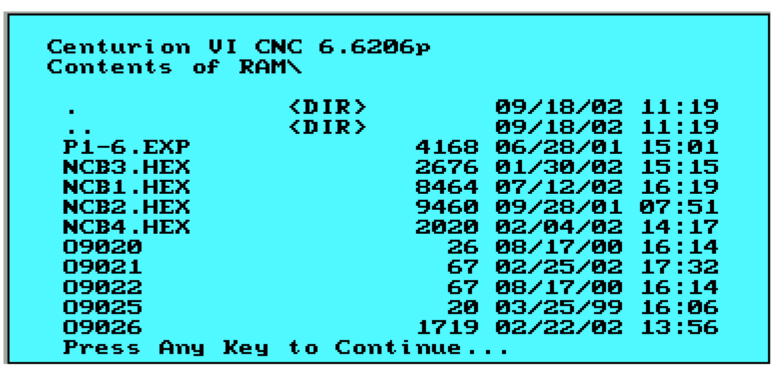
F5 (Time) displays the times and distances calculated when verifying a program. Timing information is for 100% on the feedrate override for tool changes, spindle up to speed, block stops, etc. Times are in hrs, min., sec. Distance is in inches or mm. Tool changes add 10 seconds to the feed time.



This above screen shows timing information for the last program verified. The times displayed assume the feedrate override is 100%. RUN gives the total time, FEED the milling time (G1, G2, G3), and RAPID the G0 time. All times are in Hours:Minutes:Seconds. Distance is the total distance moved in inches or millimeters.

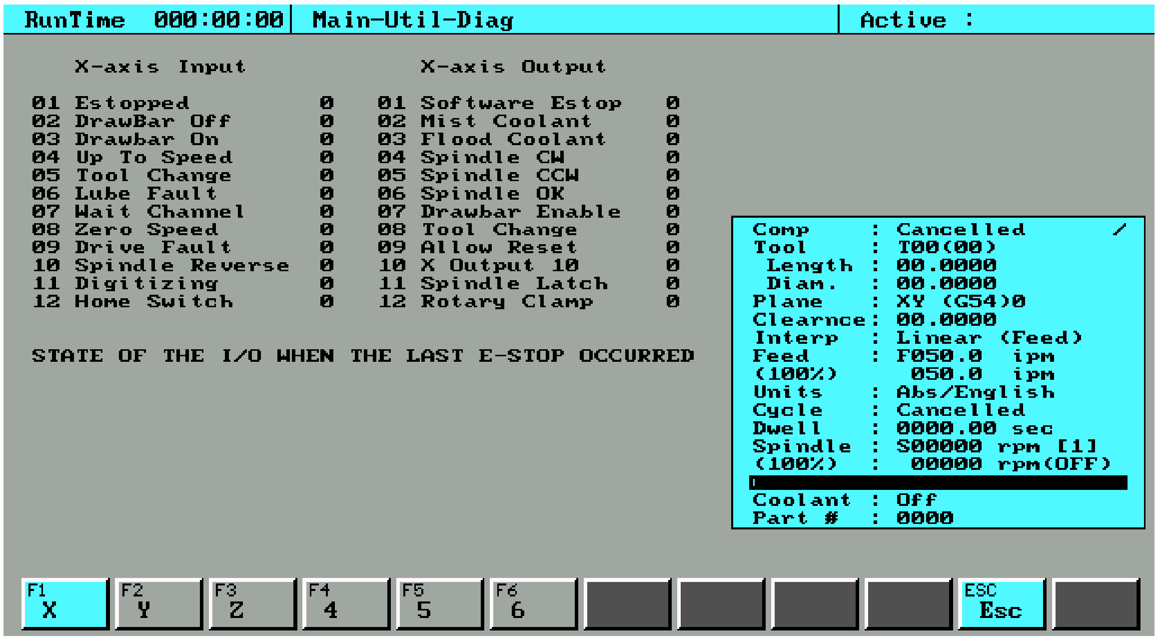
F6 (Ram) Main-Util-Info-Ram

F6 (Ram) displays the following screen, which shows the contents of the RAM directory. F6 is used mainly for diagnostic purposes. Programs that run special options on a machine are usually stored in this directory.



F7 (Diag) Main-Util-Info-Diag

F7 (Diag) displays the following screen, which shows the diagnostics of the machine just prior to the machine e-stopping. F7 is used mainly for diagnostic purposes to determine the source of the e-stop. The user is capable of viewing the states of the inputs and outputs for each axis.



F9 (Blank) Main-Util-Blank

F9 (Blank) will blank the screen. Using this function will reduce images being burned into the CRT. The screen will remain blanked until any key is pushed.

The screen will blank automatically and remain blank until any key is pressed. Block stops, tool changes, requests for cycle start, etc. will unblank the screen, prompt the operator, and then reblank the screen. There is also a timer to blank the screen if keys are not pressed in a set amount of time. See page 53, Section 4 concerning screen blank time parameter.

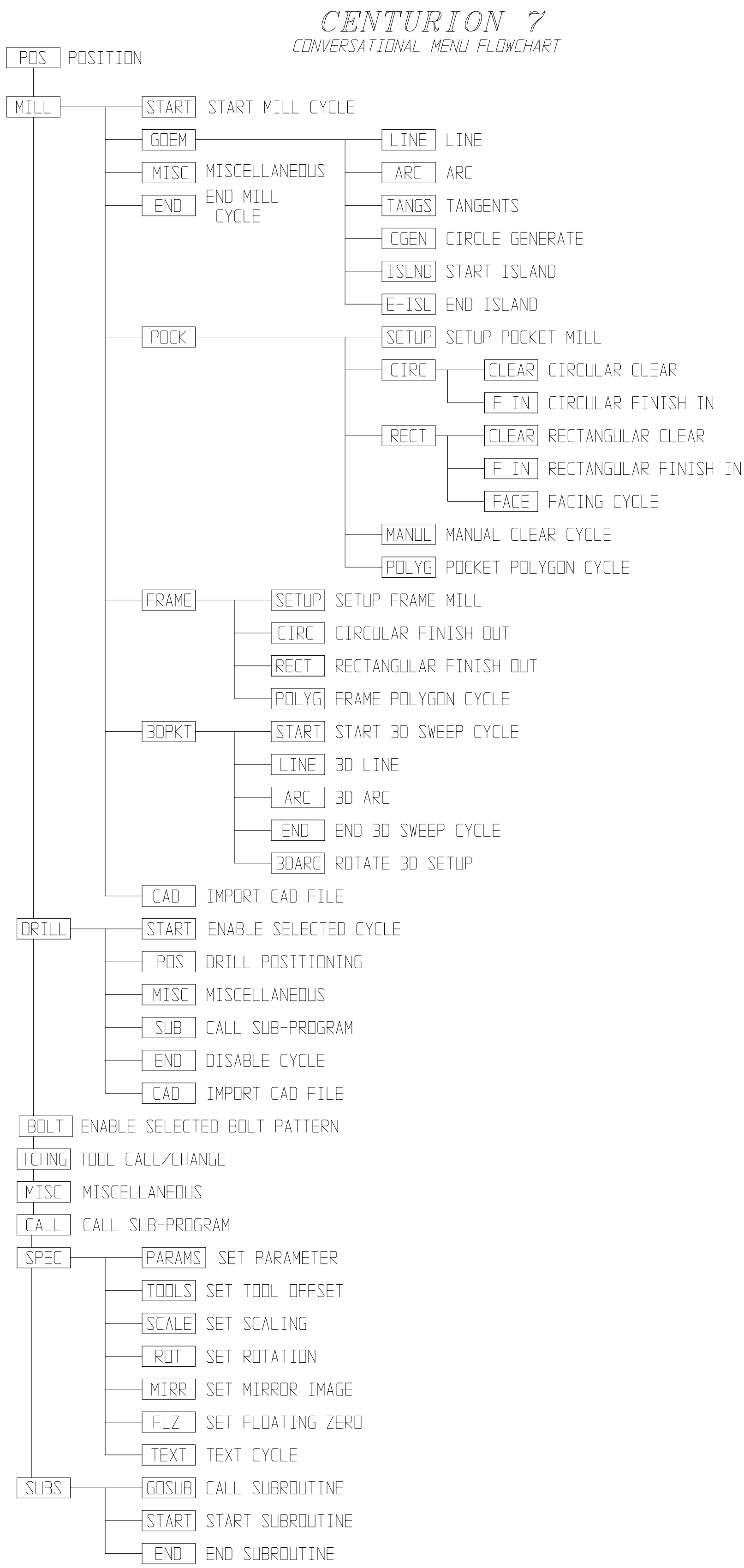
F10 (Command) Main-Util-Command

If the Misc parameter "Command Name" is not blank, the 1st 5 characters will be displayed in the F10 key. When the F10 key is pressed, the command will be executed. If several commands are needed a batch file can be executed. See page 55 for more information on this feature.

Help (AugRv) Main-Util-AugRV

Pressing and holding the auger reverse key will cause the auger to run in reverse. This is used to help un-jam the chip removal system. The auger will run in reverse until the key is released.

The auger will not run (forward or reverse) if the door is open or if the machine is e-stopped.


CENTURION 7
CONVERSATIONAL MENU FLOWCHART
POS POSITION
MILL
START START MILL CYCLE
GOEM
LINE LINE
ARC ARC
TANGS TANGENTS
CGEN CIRCLE GENERATE
ISLND START ISLAND
E-ISL END ISLAND
MISC MISCELLANEOUS
END END MILL CYCLE
POCK
SETUP SETUP POCKET MILL
CIRC
CLEAR CIRCULAR CLEAR
F IN CIRCULAR FINISH IN
RECT
CLEAR RECTANGULAR CLEAR
F IN RECTANGULAR FINISH IN
FACE FACING CYCLE
MANUL MANUAL CLEAR CYCLE
POLYG POCKET POLYGON CYCLE
FRAME
SETUP SETUP FRAME MILL
CIRC CIRCULAR FINISH OUT
RECT RECTANGULAR FINISH OUT
POLYG FRAME POLYGON CYCLE
3DPKT
START START 3D SWEEP CYCLE
LINE 3D LINE
ARC 3D ARC
END END 3D SWEEP CYCLE
3DARC ROTATE 3D SETUP
CAD IMPORT CAD FILE
DRILL
START ENABLE SELECTED CYCLE
POS DRILL POSITIONING
MISC MISCELLANEOUS
SUB CALL SUB-PROGRAM
END DISABLE CYCLE
CAD IMPORT CAD FILE
BOLT ENABLE SELECTED BOLT PATTERN
TCHNG TOOL CALL/CHANGE
MISC MISCELLANEOUS
CALL CALL SUB-PROGRAM
SPEC
PARAMS SET PARAMETER
TOOLS SET TOOL OFFSET
SCALE SET SCALING
ROT SET ROTATION
MIRR SET MIRROR IMAGE
FLZ SET FLOATING ZERO
TEXT TEXT CYCLE
SUBS
GOSUB CALL SUBROUTINE
START START SUBROUTINE
END END SUBROUTINE

SECTION THREE - CONVERSATIONAL INPUT SCREENS

Each conversational program has a text program associated with it. The conversation program file starts with letter P followed by four digits such as P1234. The text file starts with the letter O followed by four digits such as O1234. The text program is created, or posted, from the conversational program. Changes in the conversational program create a new text program from the modified conversational program. The operator can view or change the text program, but his modifications will not be transferred to the conversational program. Reposting the conversational program will overwrite his modified text program.

This section contains diagrams of the conversational input screens and an explanation of each screen.

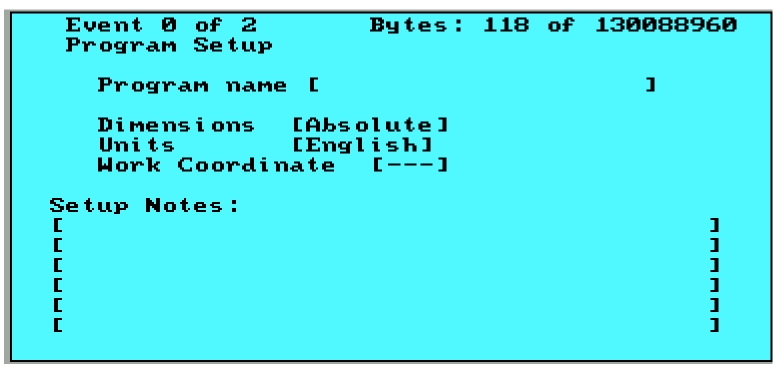
Not all possible combinations of screen inputs are shown; therefore, if additional information concerning any particular screen or field is required, the appropriate section of the manual should be referenced.

Fields that are red are required entries. These fields will appear black after the data is entered.

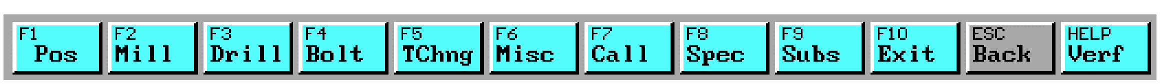
The program setup screen shown below will always appear at the beginning of every program. It initializes certain important functions so no settings remain from a previously run program.

Program setup

The conversational screen for program set-up appears as follows. It is the first event of every conversational program.



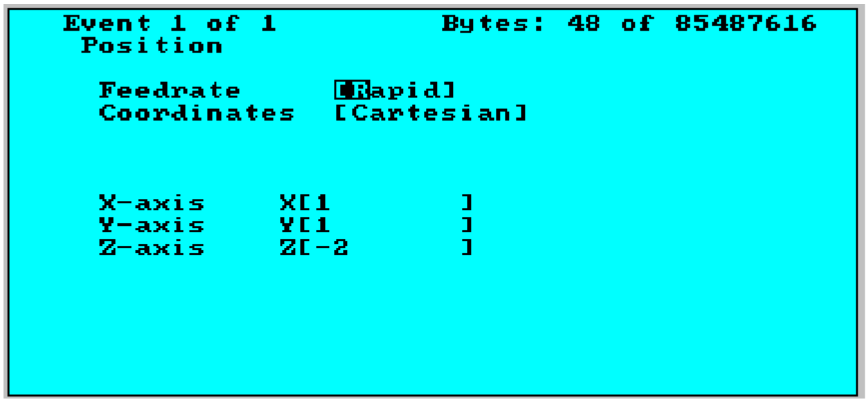
The main setup keys for conversational programming appear as follows.



F1 (Pos) Main-Prog-Conv-Pos

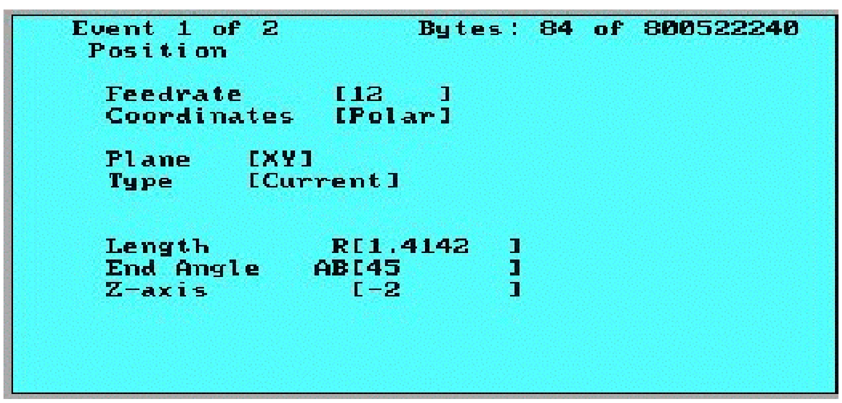
The position screen will normally be used to do rapid positioning; however, feed moves may be made by toggling the feedrate field and entering a feedrate.

The conversational screen for Cartesian rapid positioning appears as follows.



Note: See page 194, Section 4 for further information on positioning.

The conversational screen for polar feed position appears as follows.



Note: See page 196, Section 4 for further information on polar definition of a line.

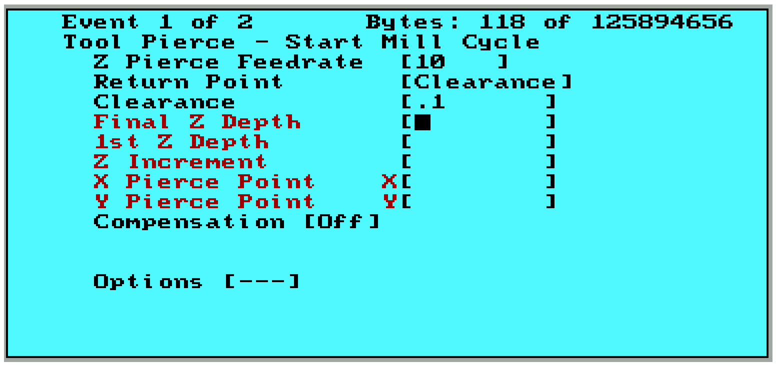
F2 (Mill) Main-Prog-Conv-Mill

The F2 (Mill) selection brings up the following soft keys.

F1 (Start) Mill-Start

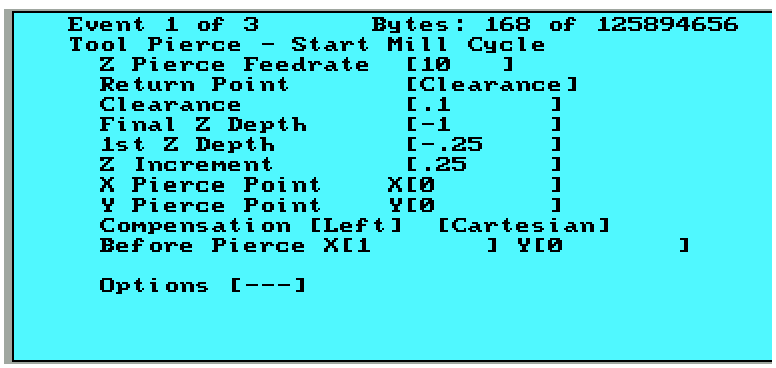
The F1 (Start) screen is used to begin a continuous single or multi-depth milling cycle. Milling will start at the first Z depth specified and continue stepping down by the Z increment until the final Z depth has been reached.

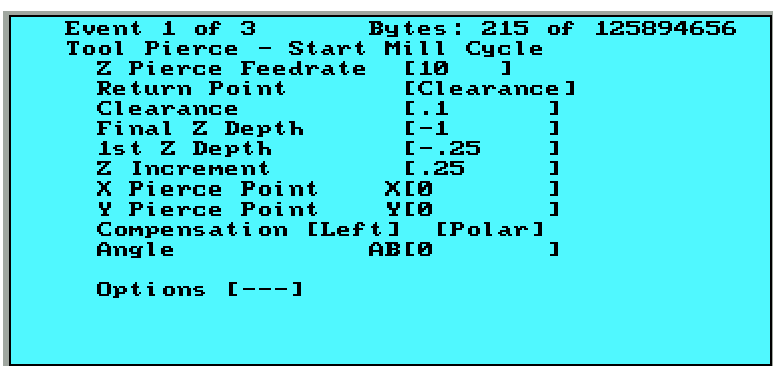
The conversational screen for tool pierce start mill cycle appears as follows.

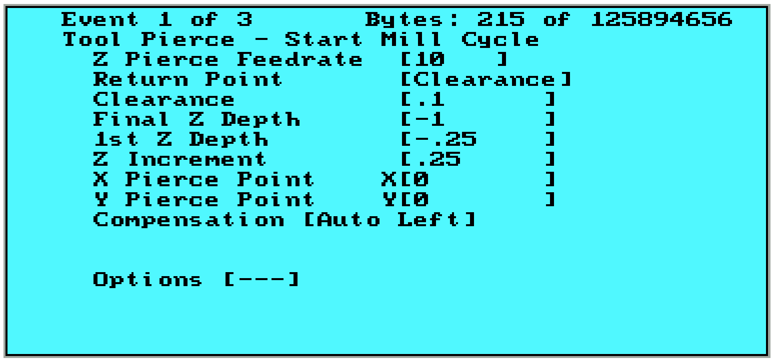


The start mill cycle is normally followed by geometry and ended with an “end mill cycle” event.

Cutter compensation is turned on in the screens below.

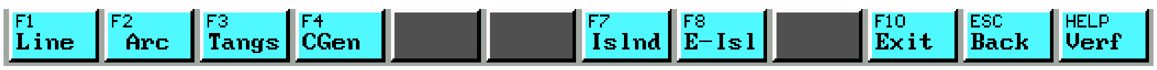






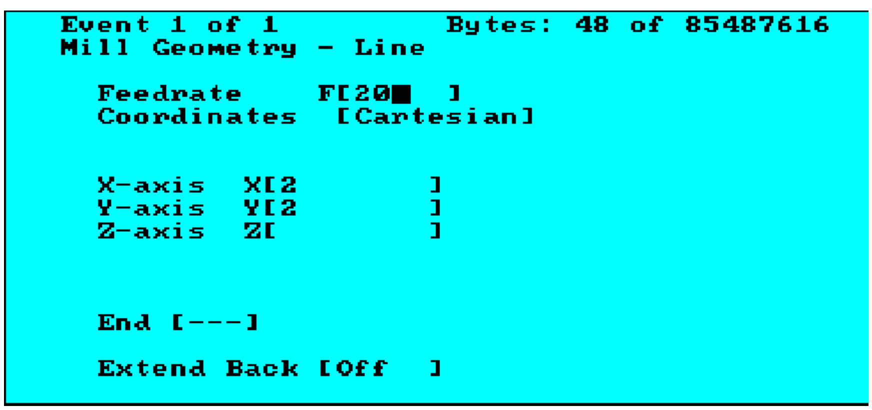
F2 (Geom) Mill-Geom

The F2 (Geom) selection brings up the following soft keys.



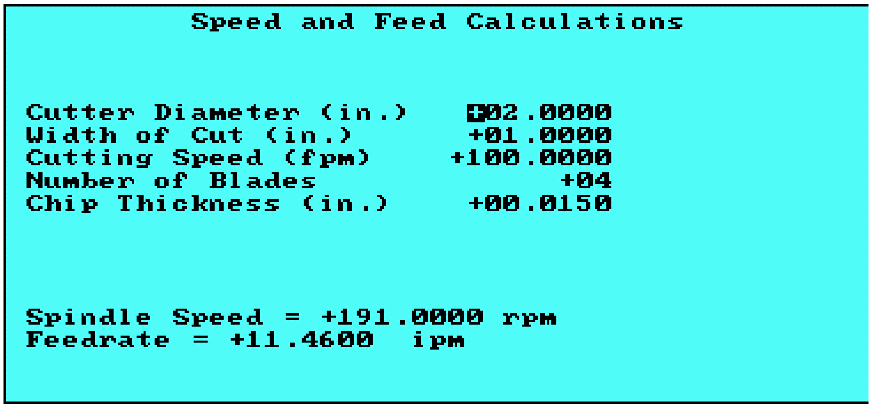
F1 (Line) Mill-Geom-Line

The F1 (Line) key displays the following conversational screen for Cartesian linear interpolation, which is used to execute linear interpolation in feed mode.



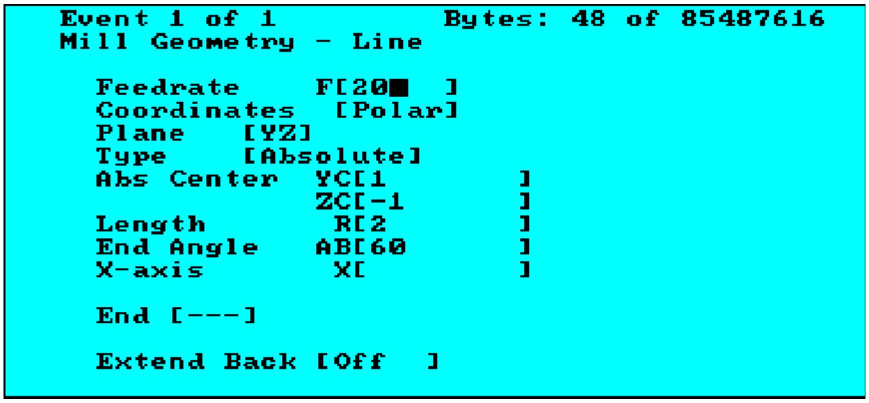
Note: See page 194, Section 4 for further information on positioning.

In conversational programming, for any feedrate or spindle speed input fields, the F12 key will be active. Pressing the F12 key will offer the operator help in calculating the appropriate spindle speed and feedrate for the appropriate inputs. The only exception is the feedrate on the position screen because this feedrate is intended to be a rapid move. After pressing the F12 key, the following screen will appear.



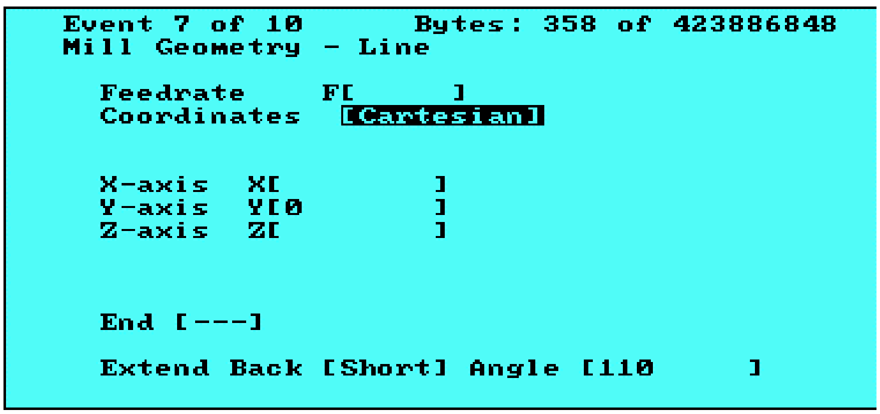
Entering data into the first five fields will give recommended feedrates and spindle speeds.

The conversational screens for polar linear interpolation appear as below.



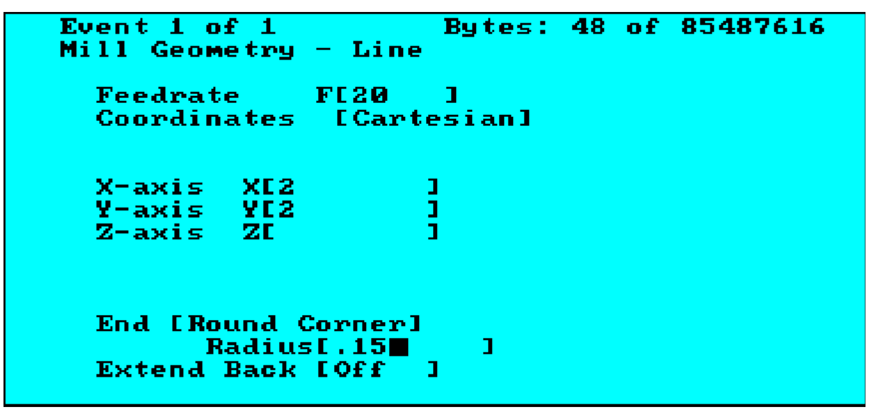
Note: See page 196, Section 4 for further information on polar definition of a line.

When Extend Back [ON] is selected, the following screen appears.



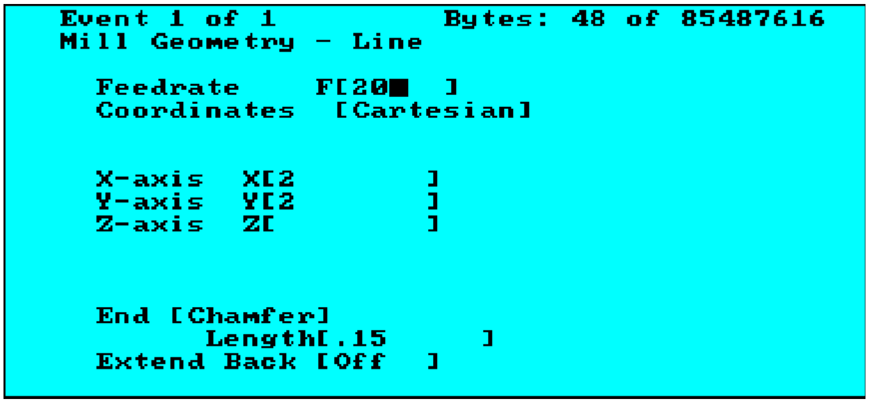
Note: See page 212, Section 4 for further information on back line.

The conversational screen for line with round corner appears as follows.



Note: See page 210, Section 4 for further information on corner rounding.

The conversational screen for line with chamfer appears as follows.



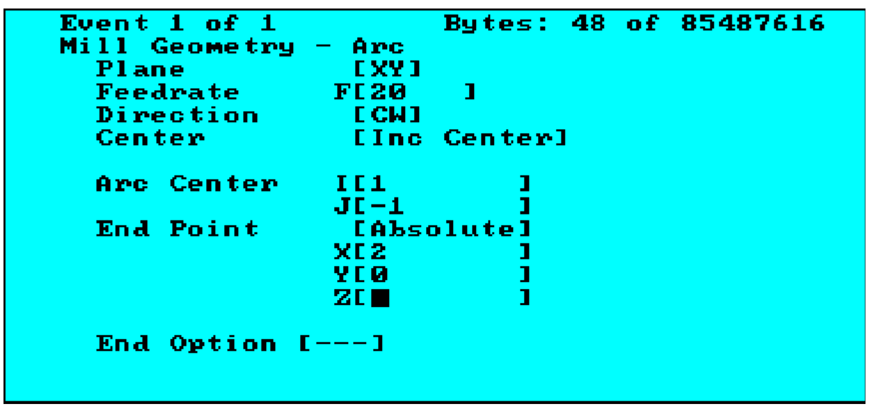
See page 211, Section 4 for further information on angle chamfering.

F2 (Arc) Mill-Geom-Arc

The F2 (Arc) screen is used to execute circular interpolation in feed mode.

Arc Sample 1

The XY plane, incremental center, CW circular interpolation conversational screen appears as follows.



Note: See page 197, Section 4 for further information on circular interpolation.

Arc Sample 2

The ZX plane, absolute center, CCW circular interpolation conversational screen appears as follows.

```
Event 1 of 1            Bytes: 48 of 85487616
Mill Geometry - Arc
  Plane           [ZX]
  Feedrate       F[10    ]
  Direction       [CCW]
  Center          [Abs Center]
  Arc Radius     R[2        ]
  Arc Center    ZC[0        ]
                XC[2        ]
  End Point       [Absolute]
                 Z[0        ]
                 X[4        ]
                 Y[■        ]

  End Option [---]
```

Note: See page 197, Section 4 for further information on circular interpolation.

Arc Sample 3

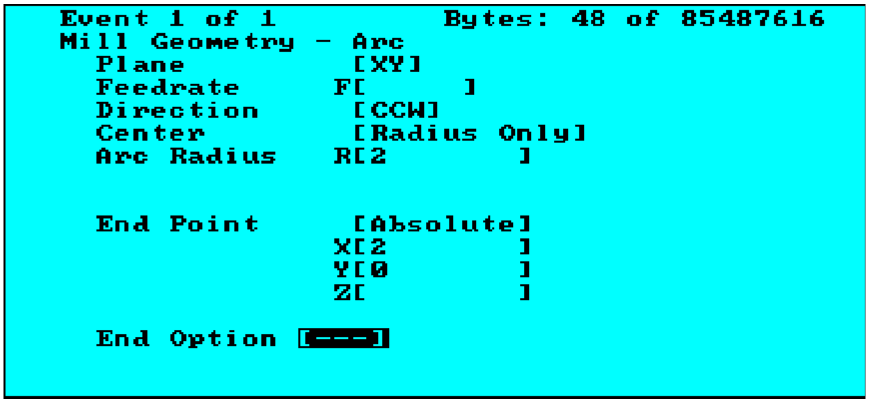
The XY plane, polar, CCW helical interpolation, with round corner appears as follows.

```
Event 1 of 1            Bytes: 48 of 85487616
Mill Geometry - Arc
  Plane           [XY]
  Feedrate       F[15    ]
  Direction       [CCW]
  Center          [Polar]
  Arc Radius     R[3        ]
  Start Angle  AA[45        ]

  End Point       [Polar]
  End Angle    AB[135       ]

                 Z[.5       ]

  End Option [Round Corner]
             Radius [.25■     ]
```

Note: See page 196, Section 4 for further information on describing an arc using polar definitions.

Arc Sample 4

The XY plane, radius only, CCW circular interpolation conversational screen appears as follows.

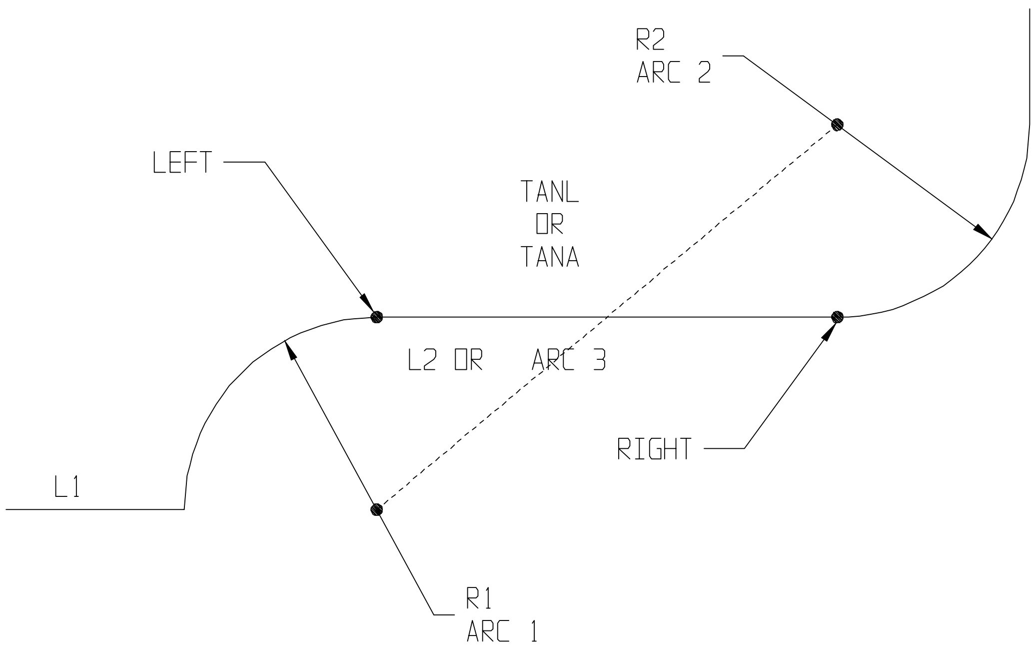


Note: See page 199, Section 4 for further information on describing an arc using a radius.

F3 (Tangs) Mill-Geom-Tangs

The F3 (Tangs) screen is used to compute the intersection points necessary for a tangent arc or tangent line between two arcs. When this function is used the first arc and the tangent line or arc will be entered into the program. The second arc information will only be used for calculation purposes. This feature was developed to enable a series of tangent lines or arcs to be programmed consecutively. Therefore, a tangent line or tangent arc command is normally followed with an arc command describing the second arc.

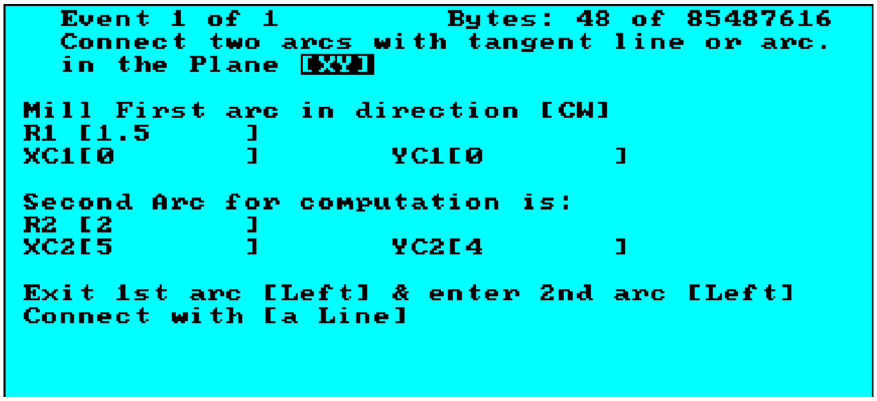
To determine the value of the right or left entries on these screens, draw a line connecting the centers of the two arcs in the direction of tool movement. Then determine if the desired points are to the right or left of this line and enter those values.



The general sequence for the above shape is as follows.

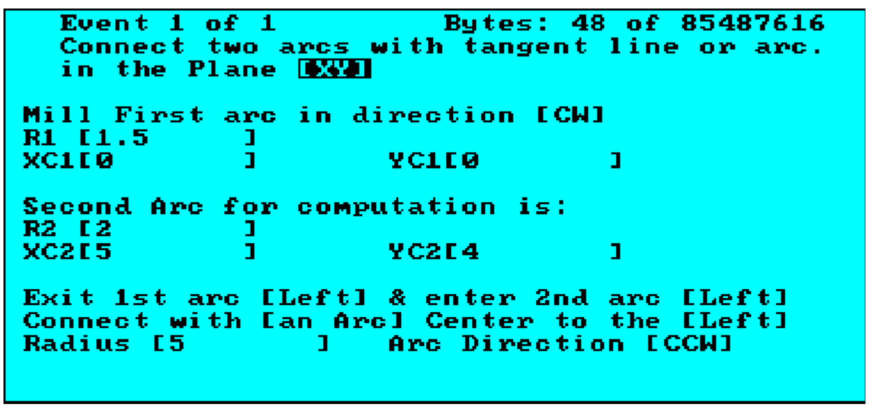
Event 1	Line L1
Event 2	Tangent line or arc function describing arc R1 and line L2 or arc 3
Event 3	Arc R2

Tangent Line
The conversational screen for tangent line appears as below.



Note: See page 330, Section 6 for further information on tangent line.

Tangent Arc
The conversational screen for tangent arc appears as follows.

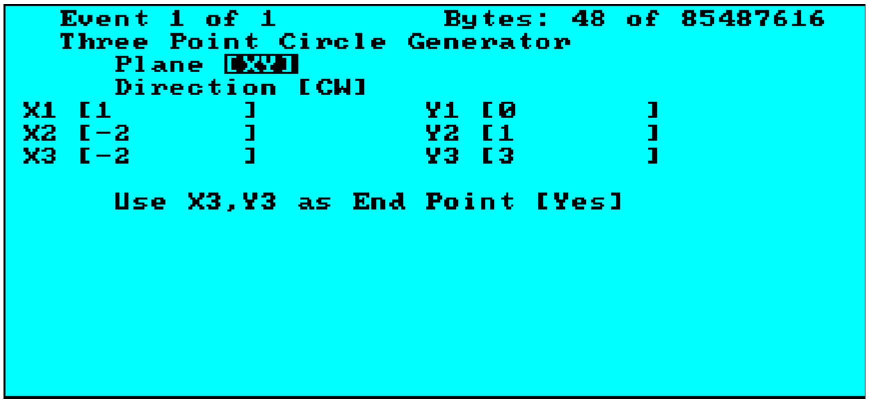


Note: See page 330, Section 6 for further information on tangent arc.

F4 (CGen) Mill-Geom-CGen

To use F4 (CGen), which is the circle generator function, fill in any three points on an arc. These three points will be used to compute the center and radius of the specified arc.

The conversational screen for circle generator appears as follows.

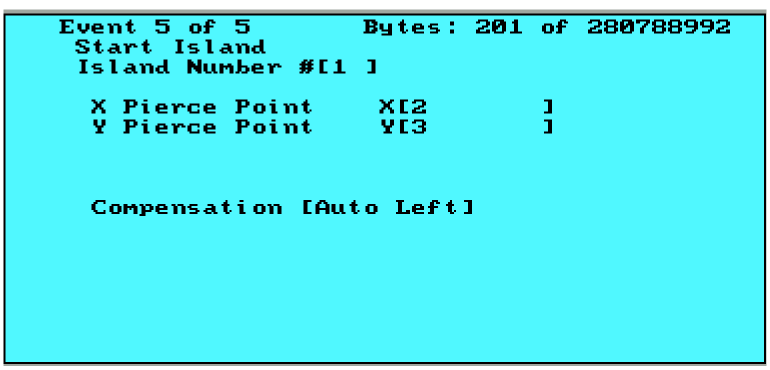


Note: The circle generator screen does not move to the first point on the arc before cutting the arc. In most cases, the circle generator screen is preceded by a line move to this position. Depending on the arc direction, the arc that is generated may not go through the second point.

See Section 6 for further information on circle generator screen.

F7 (Islnd) Mill-Geom-Islnd

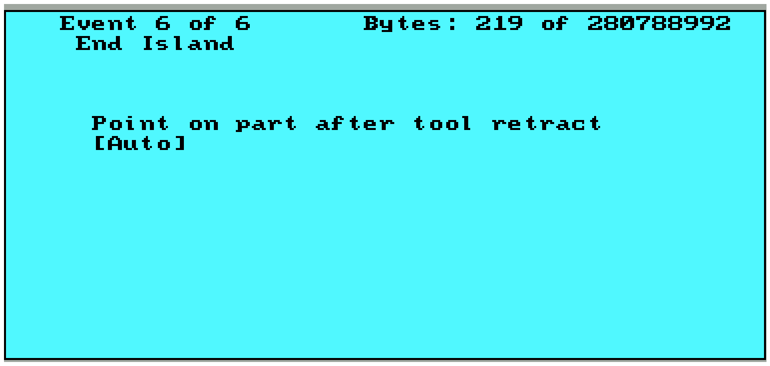
The island screen is used to specify detached geometry in a mill cycle. It can be used in a standard mill cycle or with any option (pocket clear, tapered walls or round walls).



There is no Z information on the start island screen. All of the Z dimension is specified in the start mill cycle. The island is called from the end mill cycle screen.

F8 (E-Isl) Mill-Geom-E-Isl

The end island screen is used to end the geometry in an island.



Following is a sample program using a mill cycle with pocket clear and islands.

Conversational Program C:\CNC\PARTS\P7878
Event 0 of 16
Program Setup

Program name [ ]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

```
  Event 1 of 16
  Tool Pierce - Start Mill Cycle
   Z Pierce Feedrate  [25   ]
   Return Point       [Clearance]
   Clearance          [.1      ]
   Final Z Depth      [-1      ]
   1st Z Depth        [-.3     ]
   Z Increment        [.3      ]
   X Pierce Point    X[0       ]
   Y Pierce Point    Y[0       ]
   Compensation [Left]  [Polar]
   Angle            AB[90      ]

   Options [Pocket Clear 1]
 XYFeedrate[50   ]
 Cut Width [.2      ] Finish Stock[.005    ]
--------------------------------------------------
  Event 2 of 16
  Mill Geometry - Line

   Feedrate    F[50   ]
   Coordinates  [Cartesian]

   X-axis  X[3       ]
   Y-axis  Y[0       ]
   Z-axis  Z[        ]

   End [---]

   Extend Back [Off  ]
--------------------------------------------------
  Event 3 of 16
  Mill Geometry - Line

   Feedrate    F[     ]
   Coordinates  [Cartesian]

   X-axis  X[3       ]
   Y-axis  Y[3       ]
   Z-axis  Z[        ]

   End [---]

   Extend Back [Off  ]
--------------------------------------------------
```

Event 4 of 16
Mill Geometry - Arc
Plane [XY]
Feedrate F[ ]
Direction [CCW]
Center [Abs Center]
Arc Radius R[1.5]
Arc Center XC[1.5]
YC[3]
End Point [Absolute]
X[0]
Y[3]
Z[ ]

End Option [---]

Event 5 of 16
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[0]
Y-axis Y[0]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 6 of 16
Tool Retract
End Mill Cycle

Point on part after tool retract
[Polar]
Angle AB[0]

Call Island #[1] Call Island #[2]
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]

Event 7 of 16
Start Island
Island Number #[1]

X Pierce Point X[1]
Y Pierce Point Y[1]

Compensation [Left] [Polar]

Angle AB[0]

Event 8 of 16
Mill Geometry - Line

Feedrate F[50]
Coordinates [Cartesian]

X-axis X[2]
Y-axis Y[2]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 9 of 16
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[2]
Y-axis Y[1]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 10 of 16
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[1]
Y-axis Y[1]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 11 of 16
End Island

Point on part after tool retract
[Polar]
Angle AB[45]

Event 12 of 16
Start Island
Island Number #[2]

X Pierce Point X[.75]
Y Pierce Point Y[2.75]

Compensation [Left] [Polar]

Angle AB[-90]

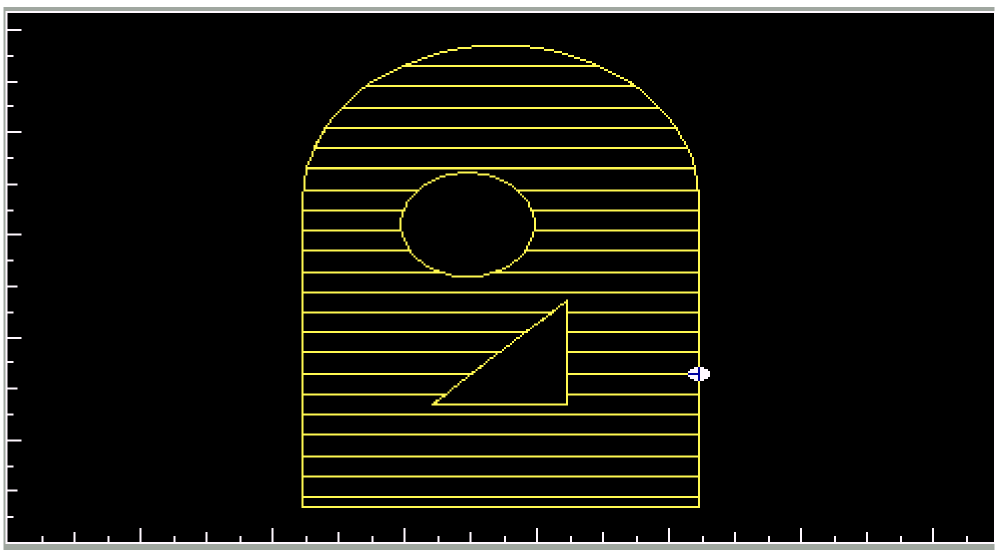
```
  Event 13 of 16
  Mill Geometry - Arc
   Plane        [XY]
   Feedrate     F[50   ]
   Direction    [CW]
   Center       [Abs Center]
   Arc Radius   R[.5      ]
   Arc Center  XC[1.25    ]
               YC[2.75    ]
   End Point    [Absolute]
                X[1.75    ]
                Y[2.75    ]
                Z[        ]

   End Option [---]
--------------------------------------------------
  Event 14 of 16
  Mill Geometry - Arc
   Plane        [XY]
   Feedrate     F[     ]
   Direction    [CW]
   Center       [Abs Center]
   Arc Radius   R[.5      ]
   Arc Center  XC[1.25    ]
               YC[2.75    ]
   End Point    [Absolute]
                X[.75     ]
                Y[2.75    ]
                Z[        ]

   End Option [---]
--------------------------------------------------
  Event 15 of 16
   End Island

   Point on part after tool retract
   [Polar]
         Angle AB[90      ]
--------------------------------------------------
```

Event 16 of 16
End of Program

Spindle off [Yes]
Coolant off [Yes]
Z to Toolchange [Yes]

X Position (home relative)[ ]
Y Position (home relative)[ ]

The previous program will generate the following graphic.



Note: See page 316, Section 5 for more information on pocket clear.

F3 (Misc) Mill-Misc

See page 177, Section 3 for explanation of miscellaneous.

F4 (End) Mill-End

The F4 (End) screen is used to end a previously started mill cycle. An end mill cycle without a start mill cycle, or a start mill cycle without an end mill cycle, will generate a syntax error when the program is run or verified.

<u>Common Error Codes for Milling Cycles</u>

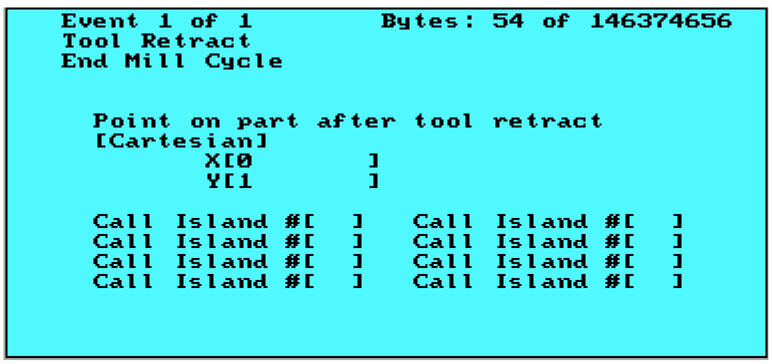
Start Mill Cycle	and **NO**	Tool Retract End Mill Cycle	= **ERROR # 602** Missing WEND Statement

This may be caused by a start mill cycle without an end mill cycle.

Tool Retract End Mill Cycle	and **NO**	Start Mill Cycle	= **ERROR # 601** Missing WHILE Statement

This may be caused by an end mill cycle without a start mill cycle.

The conversational screen for tool retract end mill cycle appears as follows.



<u>Sample Mill Program</u>

Event 0 of 7
Program Setup

Program name [ ]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 7
Tool Pierce - Start Mill Cycle
Z Pierce Feedrate [20]
Return Point [Clearance]
Clearance [.1]
Final Z Depth [-1]
1st Z Depth [-.2]
Z Increment [.3]
X Pierce Point X[0]
Y Pierce Point Y[0]
Compensation [Auto Right]

Options [---]

Event 2 of 7
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[2]
Y-axis Y[ ]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 3 of 7
Mill Geometry - Arc
Plane [XY]
Feedrate F[ ]
Direction [CCW]
Center [Polar]
Arc Radius R[1]
Start Angle AA[-90]

End Point [Polar]
End Angle AB[90]

Z[ ]

End Option [---]

Event 4 of 7
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[0]
Y-axis Y[ ]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 5 of 7
Mill Geometry - Arc
Plane [XY]
Feedrate F[ ]
Direction [CCW]
Center [Polar]
Arc Radius R[1]
Start Angle AA[90]

End Point [Polar]
End Angle AB[-90]

Z[ ]

End Option [---]

Event 6 of 7
Tool Retract
End Mill Cycle

Point on part after tool retract
[Auto]

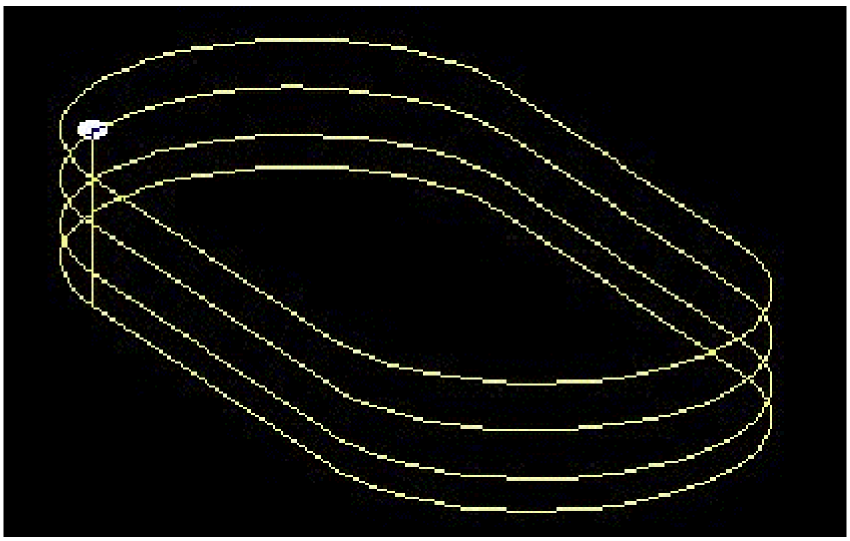
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]

Event 7 of 7
End of Program

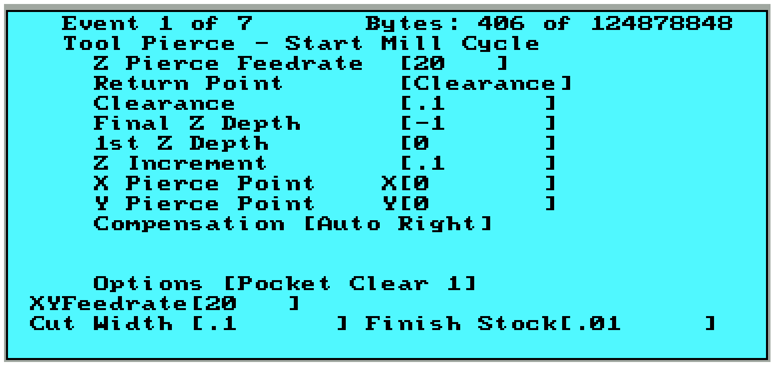
Spindle off [Yes]
Coolant off [Yes]
Z to Toolchange [Yes]

X Position (home relative)[ ]
Y Position (home relative)[ ]

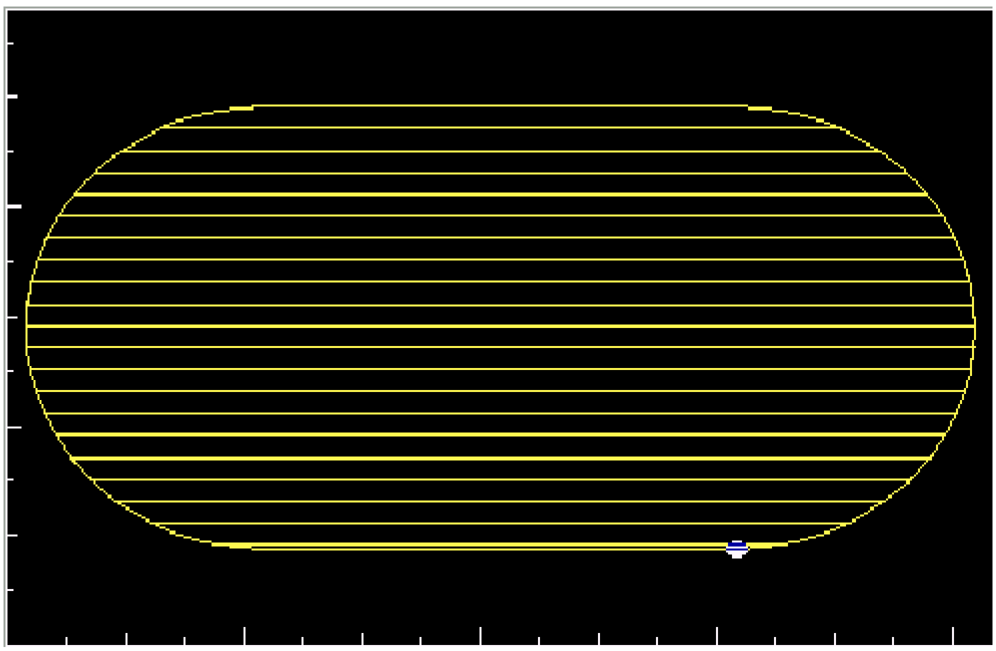
The following graphic is the isometric view of the mill program.



The conversational screen for pocket clear option using tool pierce start mill cycle appears as follows.

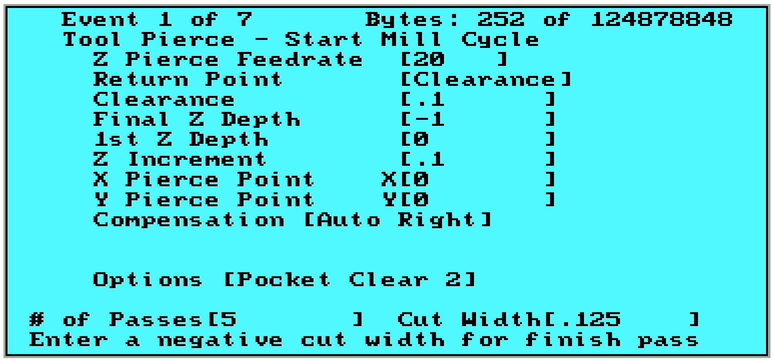


The following graphic is the top view of the sample mill program using the pocket clear 1 option on the tool pierce start mill cycle.

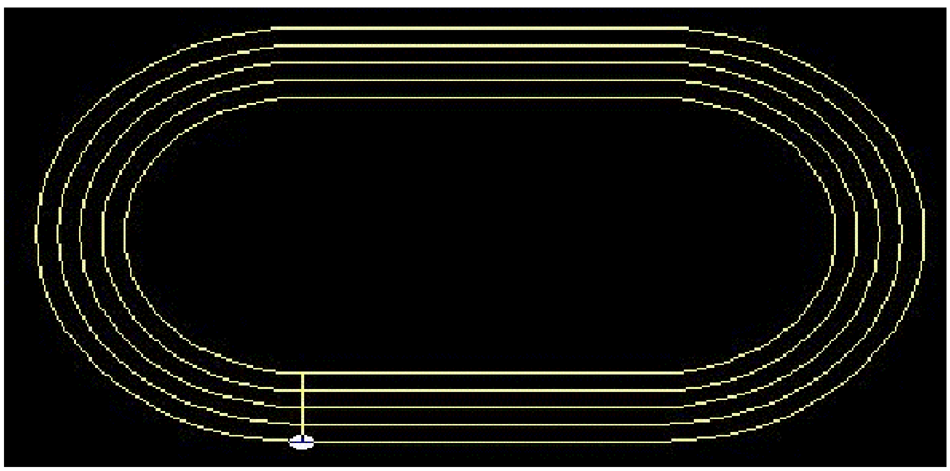


Note: See page 316, section 4 for further information on pocket clear 1.

The conversational screen for pocket clear option using tool pierce start mill cycle appears as follows.



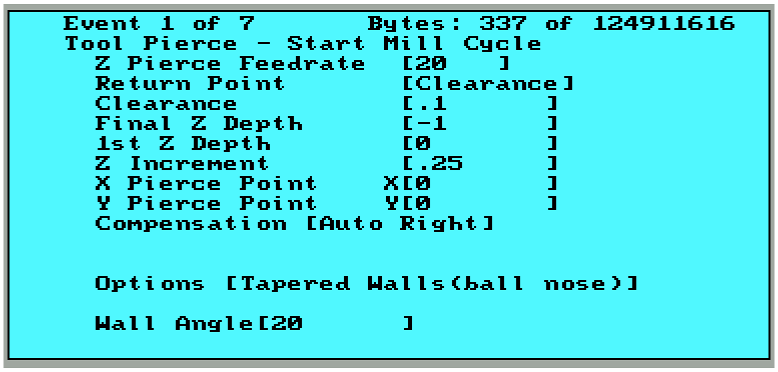
The following graphic is the top view of the sample mill program using the pocket clear 2 option on the tool pierce start mill cycle.



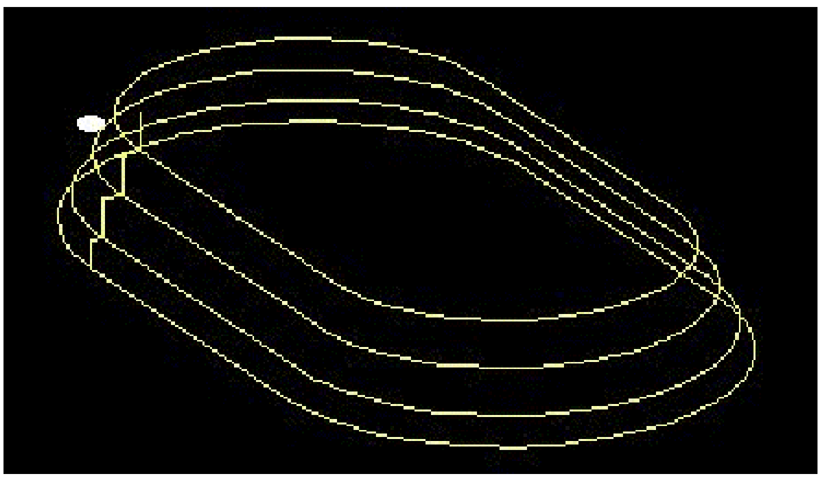
Note: See page 316, section 5 for further information on pocket clear 2.

Note: This option can be used to clear away from a framed mill cycle as well. To use the finish pass option, the original tool radius data for the roughing pass must be bigger than the actual tool radius; if this is the case, use two passes with a negative pass width to attain the true tool radius. This option is meant to clear away from the actual part and not to clear the entire pocket.

The conversational screen for tool pierce start mill cycle with tapered walls option appears as follows.



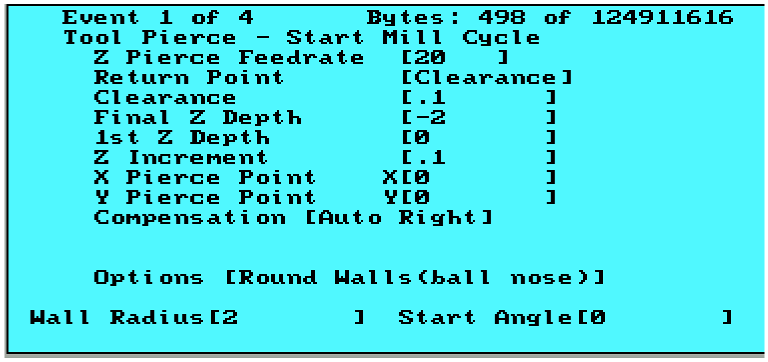
The following graphic illustrates the sample mill program using the tapered walls option on the tool pierce start mill cycle.



0° is a vertical wall; 90° is impossible. The tapered wall also has an option for an end mill. Cutter comp must be on to use the tapered walls option. The first Z depth should be the top of the wall. The first pass is offset by the entire tool radius.

Note: See page 308, Section 5 for further information on tapered walls.

The conversational screen for tool pierce start mill cycle with rounded walls option appears as follows.



Sample Program Using Rounded Walls on End Mill Cycle

Event 0 of 4
Program Setup

Program name [ ]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 4
Tool Pierce - Start Mill Cycle
Z Pierce Feedrate [20]
Return Point [Clearance]
Clearance [.1]
Final Z Depth [-2]
1st Z Depth [0]
Z Increment [.1]
X Pierce Point X[0]
Y Pierce Point Y[0]
Compensation [Auto Right]

Options [Round Walls(ball nose)]

Wall Radius[2] Start Angle[0]

Event 2 of 4
Mill Geometry - Arc
Plane [XY]
Feedrate F[ ]
Direction [CCW]
Center [Polar]
Arc Radius R[1]
Start Angle AA[0]

End Point [Polar]
End Angle AB[0]

Z[ ]

End Option [---]

Event 3 of 4
Tool Retract
End Mill Cycle

Point on part after tool retract
[Auto]

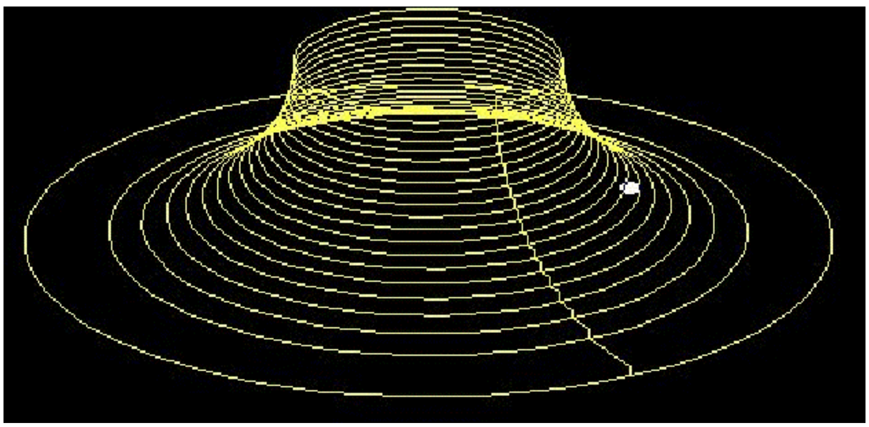
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]
Call Island #[ ] Call Island #[ ]

Event 4 of 4
End of Program

Spindle off [Yes]
Coolant off [Yes]
Z to Toolchange [Yes]

X Position (home relative)[ ]
Y Position (home relative)[ ]

The following graphic illustrates the sample mill program using the round walls option on the start mill cycle.



0° is a vertical wall; 90° is impossible. The round wall also has an option to use an end mill tool. Cutter comp must be on to use the tapered walls option. The first Z depth should be the top of the wall. The first pass is offset by the entire tool radius.

Note: See page 312, Section 5 for further information on rounded walls.

F5 (Pockt) Mill-Pockt

The F5 (Pockt) pocket mill selection brings up the following soft keys.

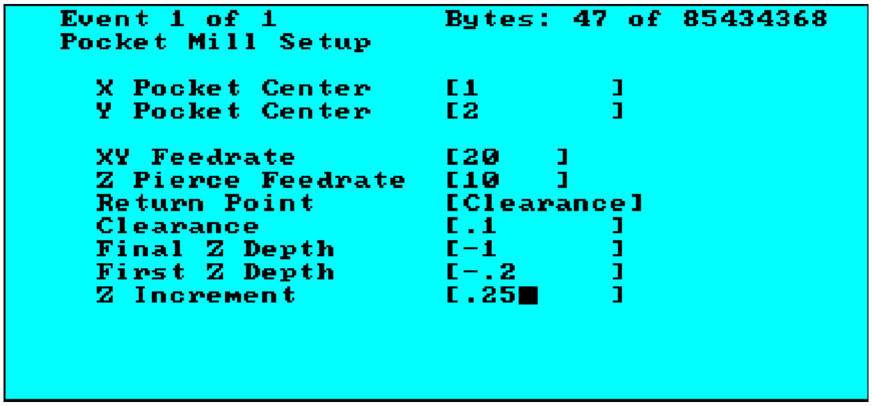
F1 (Setup) Mill-Pockt-Setup

The F1 (Setup) screen is used to set parameters necessary for the circular, rectangular and polygon pocket routines. Setting parameters must be done prior to any pocket routines. The manual pocket clear routine does not need the setup screen.

Mill START and END are **not** to be used with pocket routines.

Note: All milling auto routines must be activated with the tool at the center of the routine.

The conversational screen for pocket mill setup appears as follows.



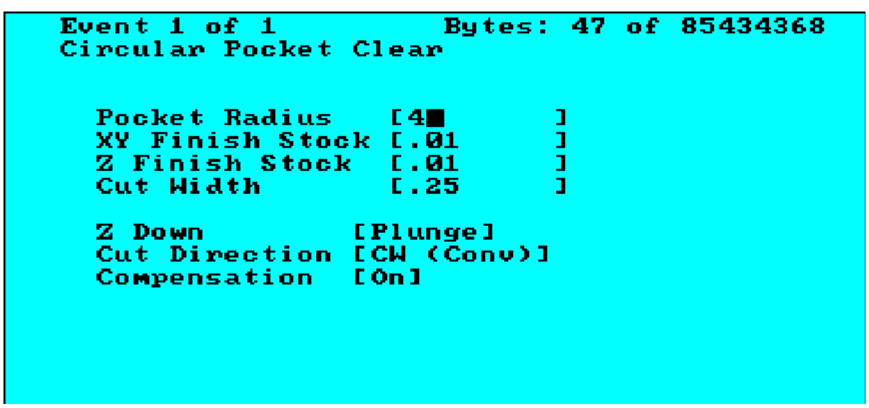
Note: See pages 218, 225 in Section 4 and 316 in Section 5 for further information on pocket routines.

F2 (Circ) Mill-Pockt-Circ

The F2 (Circ) selection brings up the following soft key selections.

F1 (Clear) Mill-Pockt-Circ-Clear

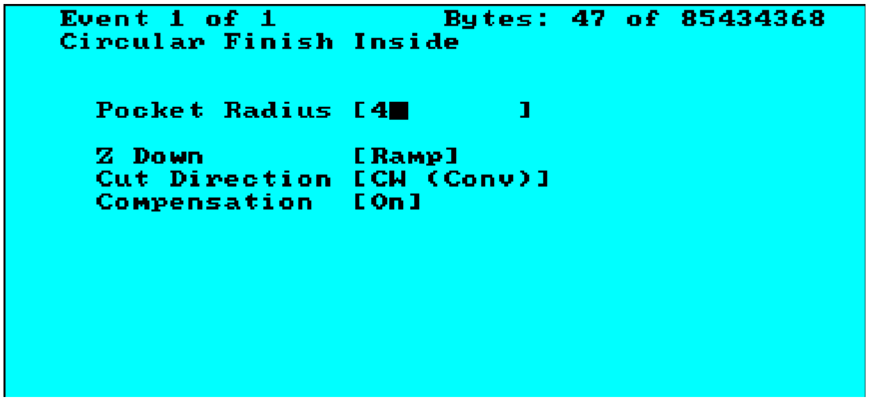
The conversational screen for CW circular pocket clear appears as follows.



Note: See page 218, Section 4 for further information on circular pocket clear.

F2 (Fin) Mill-Pockt-Circ-Fin

The conversational screen for inside CW circular pocket finish appears as follows.



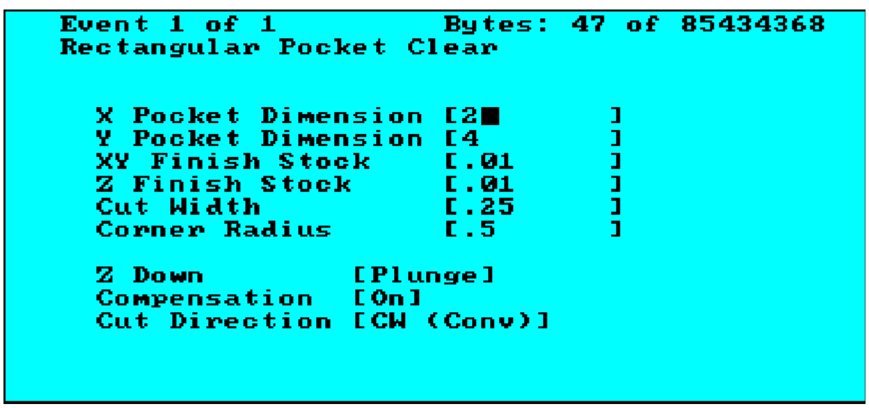
Note: See page 219, Section 4 for more information on circular finish inside.

F3 (Rect) Mill-Pockt-Rect

The F3 (Rect) rectangular pocket selection brings up the following soft keys.

F1 (Clear) Mill-Pockt-Rect-Clear

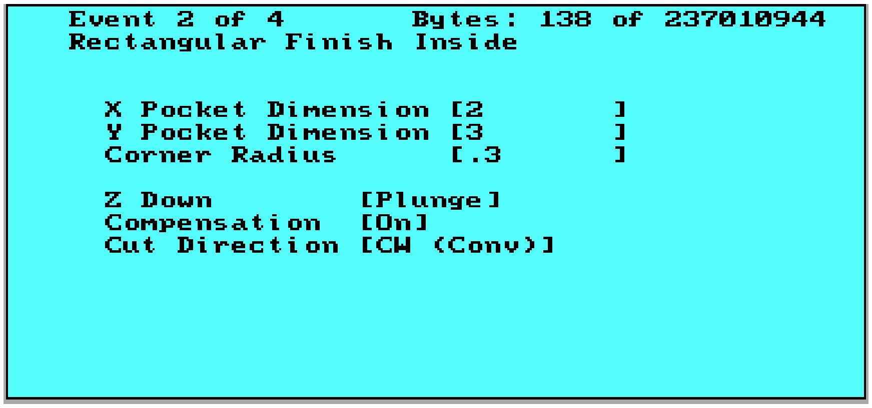
The conversational screen for CW rectangular pocket clear appears as follows.



Note: See page 225, Section 4 for further information on rectangular pocket clear.

F2 (Fin) Mill-Pockt-Rect-Fin

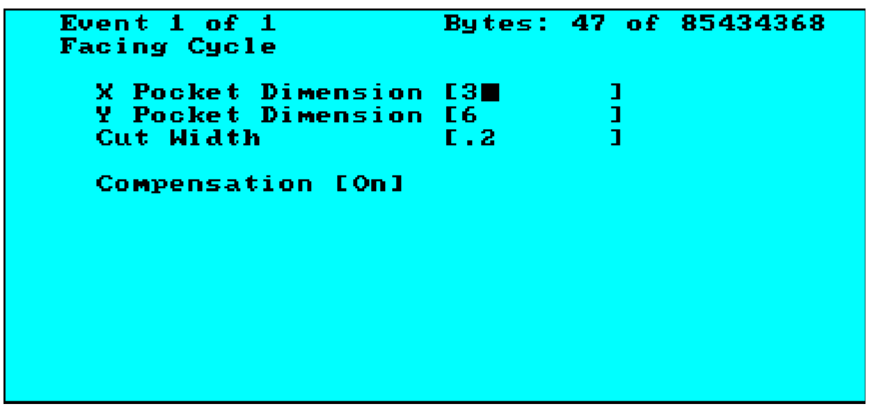
The conversational screen for inside CW rectangular pocket finish appears as follows.



Note: See page 226, Section 4 for further information on rectangular finish inside.

F3 (Face) Mill-Pockt-Rect-Face

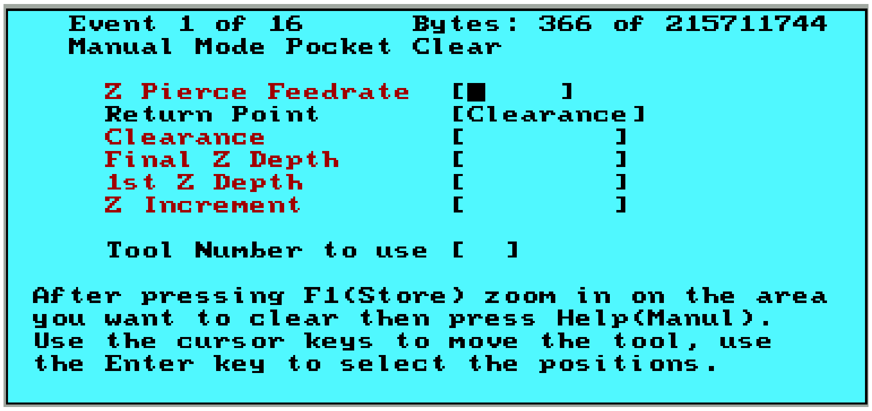
The conversational screen for facing cycle appears as follows.



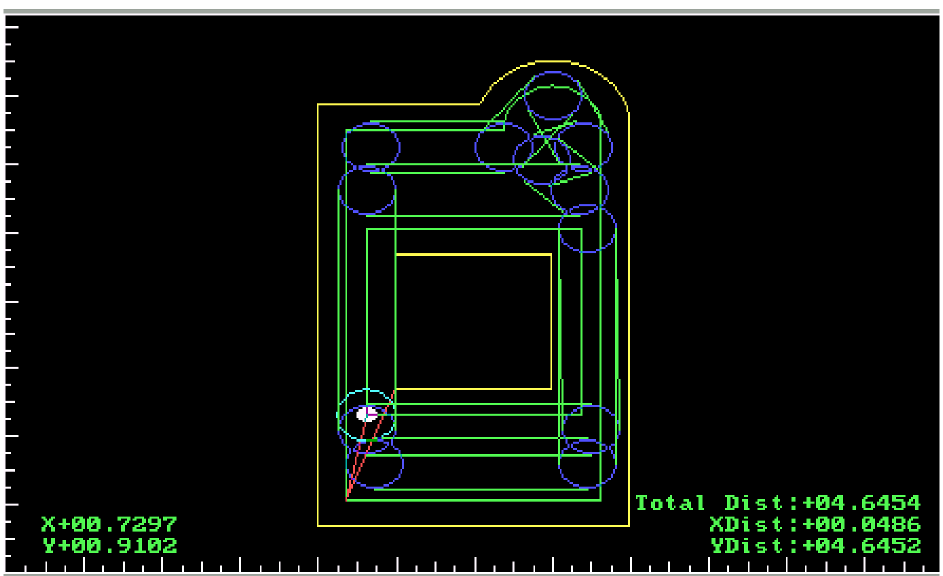
F4 (Manul) Mill-Pockt-Manul

The Manual Pocket Clear is used to allow the operator to move the tool around on the graph screen and generate a tool path using the cursor keys to move the tool and the enter key to select points.

To add manual points in conversational select F2 (Mill) - F5(Pockt) - F4 (Manul). The manual mode pocket clear screen looks like the following:



The fields on the screen are similar to a start mill cycle. The tool number is used to graphically show a tool (from the tool diameter in the tool table). After entering the fields on the screen and pressing F1 (Store). The part being programmed will be verified (part and tool paths). The operator can window, zoom or pan, then select Help (Man) and move the tool around using the cursor keys and enter key to select tool locations.

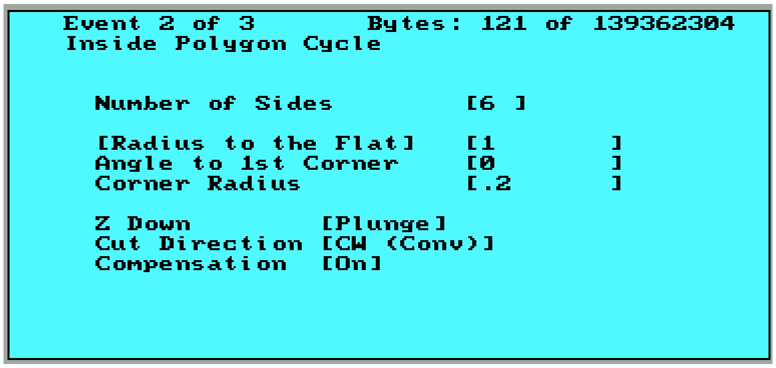


When the operator exits the graph the control stores the points as a mill cycle. Using the 1st point as a start point on the start mill cycle. The remaining points are stored as lines. An end mill cycle event is stored after the last line.

Note: To use this feature it is recommend that the operator program the area that needs the manual data and insert the Manual pocket clear in front of the geometry to clear.

F5 (Polyg) Mill-Pockt-Polyg

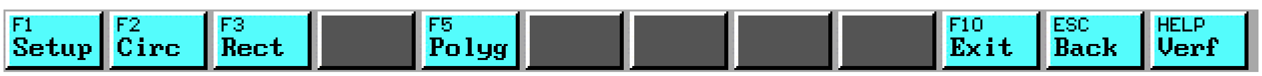
The conversational screen for the polygon cycle appears below.



Note: see page 231 , Section 4 for further information on the polygon cycle.

F6 (Frame) Mill-Frame

The F6 (Frame) frame mill selection brings up the following soft keys.



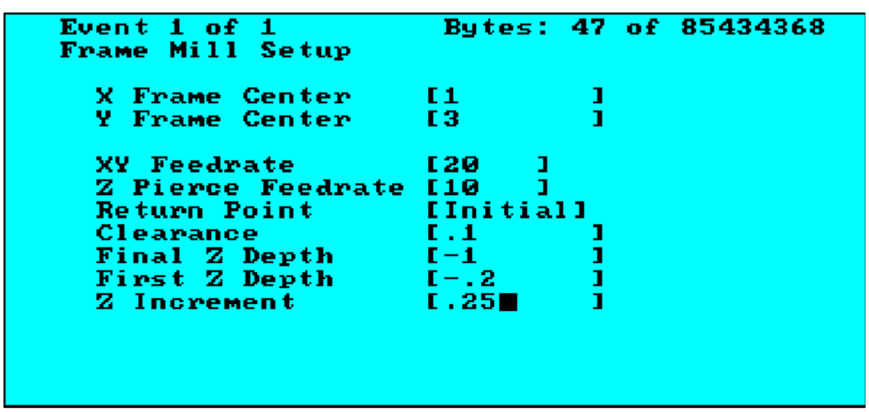
F1 (Setup) Mill-Frame-Setup

The F1 (Setup) screen is used to set parameters necessary for circular and rectangular frame mill routines. Setting parameters must be done prior to any frame milling routines.

Mill START and END are not to be used with frame routines.

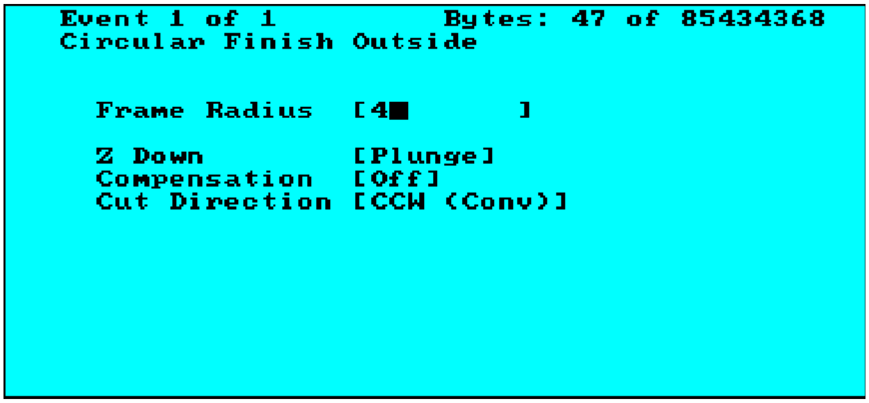
Note: ***All milling autoroutines must be activated with the tool at the center of the routine. The framing autoroutines can use the G65 X___Y___ to specify the center.***

The conversational screen for frame mill setup appears as follows.



F2 (Circ) Mill-Frame-Circ

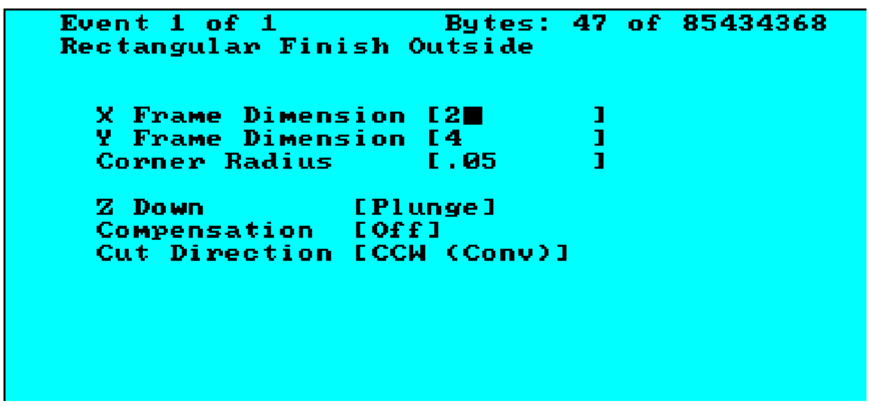
The conversational screen for outside CCW circular frame mill appears as follows.



Note: See page 221, Section 4 for further information on circular finish outside.

F3 (Rect) Mill-Frame-Rect

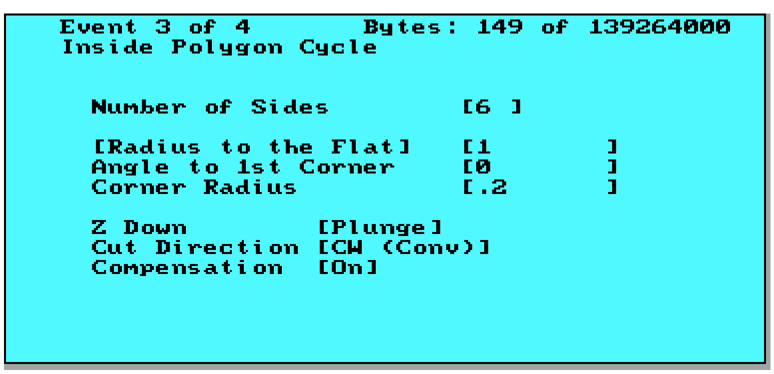
The conversational screen for rectangular finish outside appears as follows:



Note: See page 229, Section 4 for further information on rectangular finish outside.

F5(Poly) Mill-Frame-Poly

The conversational screen for outside polygon appears as follows:



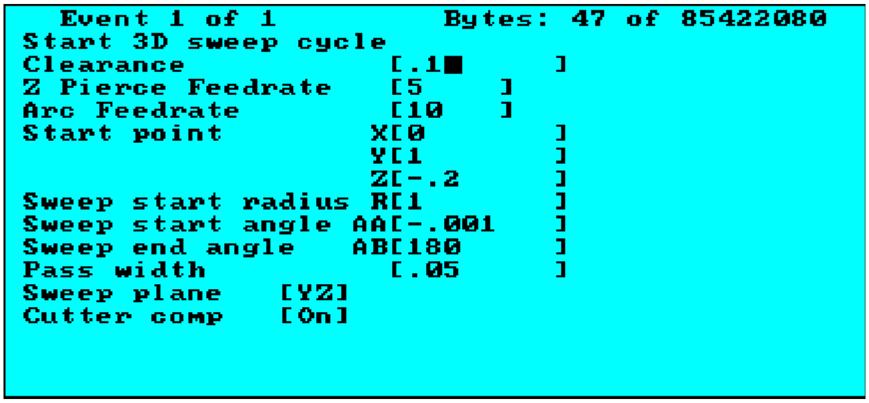
Note: See page 231, Section 4 for more information on the polygon cycle.

F7 (3dPkt) Mill-3dPkt

The F7 (3dPkt) selection brings up the following soft keys.

F1 (Start) Mill-3dPkt-Start

The F1 (Start) key brings up the conversational screen for start 3D sweep cycle, which appears below.



Notes on 3D Sweep Cycle

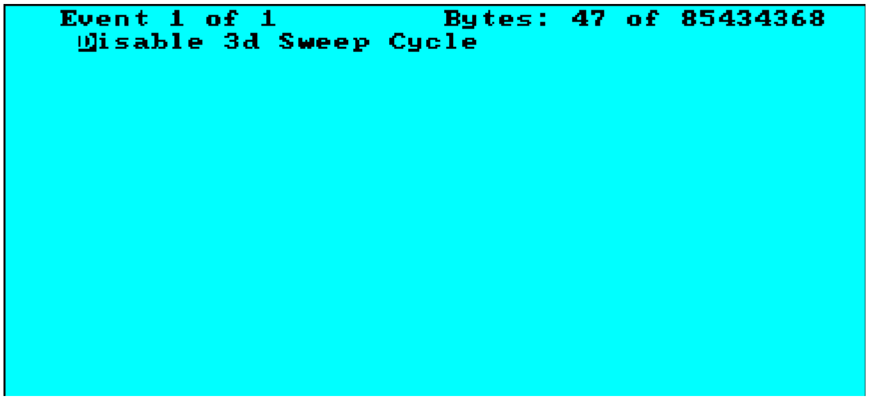
Note 1: If sweep start angle = sweep end angle, then no arc is made.

Note 2: Negative start angles specify a female part, positive start angles specify a male part.

After the start 3D screen is completed, the next step is to program the desired contour using **only lines and arcs**. These should be entered through the Mill-3DPkt-Line or Mill-3DPkt-Arc screen. Once the XY profile of the part has been entered, the above cycle will sweep arcs in the specified plane along the XY programmed lines or arcs until the sweep cycle is ended using the disable 3D sweep cycle event. The net effect of this cycle is to rotate the programmed XY contour into the XZ or YZ plane. In the case of a bottle mold, only the bottle profile would have to be programmed in the XY plane and then rotated into the YZ or XZ plane.

F4 (End) Mill-3dPkt-End

The F4 (End) key must be selected to terminate the 3D pocket cycle or an error will occur. The conversational screen for disable 3D sweep cycle appears as follows.



<u>Sample Program Using</u>
3D Sweep Cycle

```
Event 0 of 6

 Program Setup
  Program name                 [        ]
  Dimensions                   [Absolute]
  Units                 [English]
  Work Coordinate              [---]
 Setup Notes:
 [              ]
 [              ]
 [              ]
 [              ]
 [              ]
 [              ]
--------------------------------------------------
```

Event 1 of 6

Start 3D sweep cycle		
Clearance		[.1]
Z Pierce Feedrate		[15]
Arc Feedrate		[20]
Start point	X	[2]
	Y	[1]
	Z	[0]
Sweep start radius	R	[1]
Sweep start angle	AA	[-.0001]
Sweep end angle	AB	[180]
Pass width		[.05]
Sweep plane		[XZ]
Cutter comp		[On]

Event 2 of 6

3D Geometry - Line		
Feedrate		[ ]
Coordinates	[Cartesian]	
X axis	X	[3]
Y axis	Y	[3]
Z axis	Z	[ ]

Event 3 of 6

3D Geometry - Line		
Feedrate		[ ]
Coordinates	[Cartesian]	
X axis	X	[ ]
Y axis	Y	[4]
Z axis	Z	[ ]

Event 4 of 6

3D Geometry - Arc

Plane	[XY]	
Feedrate	F	[ ]
Direction		[CW]
Center	[Abs Center]	
Arc Radius	R	[1]
Arc Center	XC	[3]
	YC	[5]
End Point		[Absolute]
	X	[2]
	Y	[5]
	Z	[ ]

Event 5 of 6

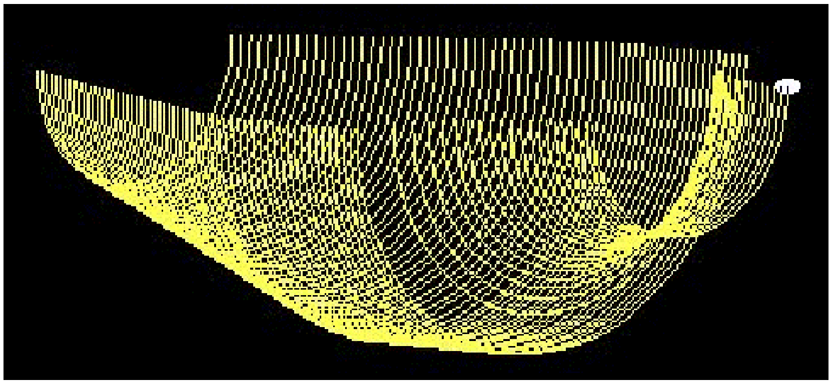
Disable 3d Sweep Cycle

Event 6 of 6

End of Program

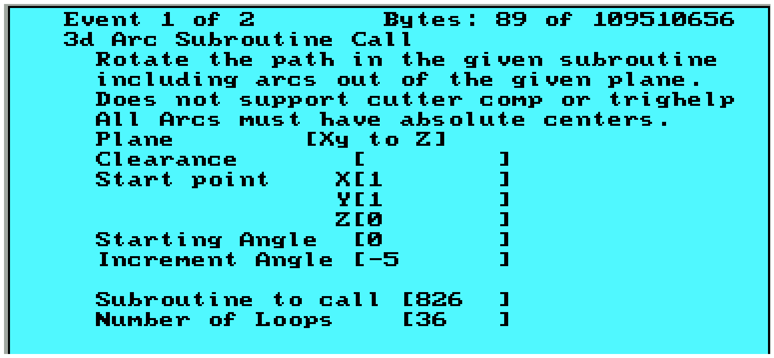
Spindle off	[No]
Coolant off	[No]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

The following graphic illustrates the sample mill program using the 3D sweep cycle.



F5 (3dArc) Mill-3dPkt-3dArc

The F5 (3dArc) brings up the conversational screen for the 3d Arc subroutine call, which appears below.



<u>A sample program using the 3darc is shown below:</u>

Conversational Program C:\CNC\PARTS\P0825
Event 0 of 2
Program Setup

Program name [ ]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

```
  Event 1 of 2
  3d Arc Subroutine Call
    Rotate the path in the given subroutine
    including arcs out of the given plane.
    Does not support cutter comp or trighelp
    All Arcs must have absolute centers.
    Plane        [Xy to Z]
    Clearance      [       ]
    Start point    X[1       ]
                   Y[1       ]
                   Z[0       ]
    Starting Angle  [0       ]
    Increment Angle [-5      ]

    Subroutine to call [826  ]
    Number of Loops    [36   ]
--------------------------------------------------
  Event 2 of 2
  End of Program

    Spindle off     [Yes]
    Coolant off     [Yes]
    Z to Toolchange [No]

    X Position (home relative)[        ]
    Y Position (home relative)[        ]
--------------------------------------------------

Conversational Program C:\CNC\PARTS\P0826
  Event 0 of 5
  Program Setup

    Program name [                    ]

    Dimensions  [Absolute]
    Units       [English]
    Work Coordinate  [---]

   Setup Notes:
   [                              ]
   [                              ]
   [                              ]
   [                              ]
   [                              ]
   [                              ]
--------------------------------------------------
```

Event 1 of 5
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[2]
Y-axis Y[1]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 2 of 5
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[3]
Y-axis Y[3]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 3 of 5
Mill Geometry - Line

Feedrate F[ ]
Coordinates [Cartesian]

X-axis X[ ]
Y-axis Y[4]
Z-axis Z[ ]

End [---]

Extend Back [Off]

Event 4 of 5
Mill Geometry - Arc
Plane [XY]
Feedrate F[ ]
Direction [CW]
Center [Abs Center]
Arc Radius R[1]
Arc Center XC[3]
YC[5]
End Point [Absolute]
X[2]
Y[5]
Z[ ]

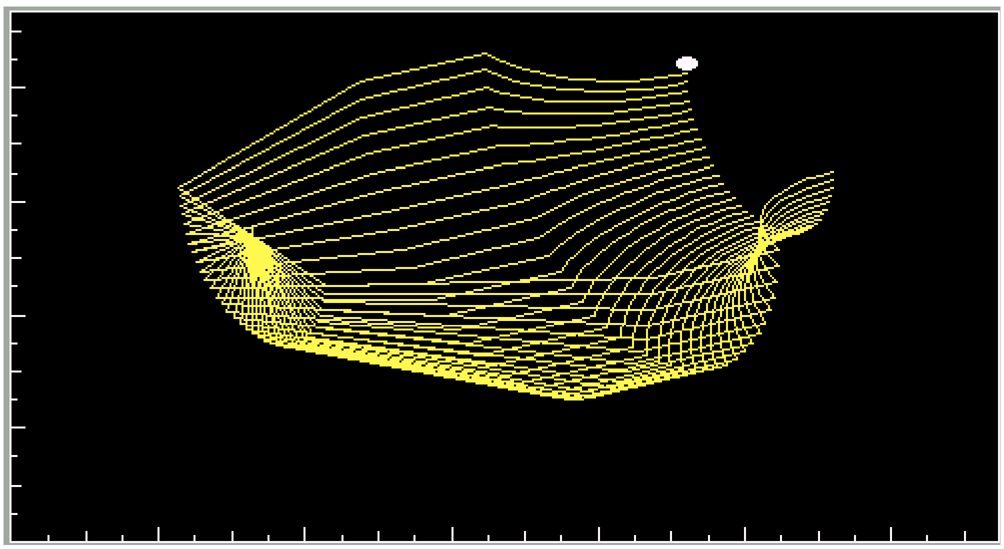
End Option [---]

Event 5 of 5
End of Program

Spindle off [Yes]
Coolant off [Yes]
Z to Toolchange [No]

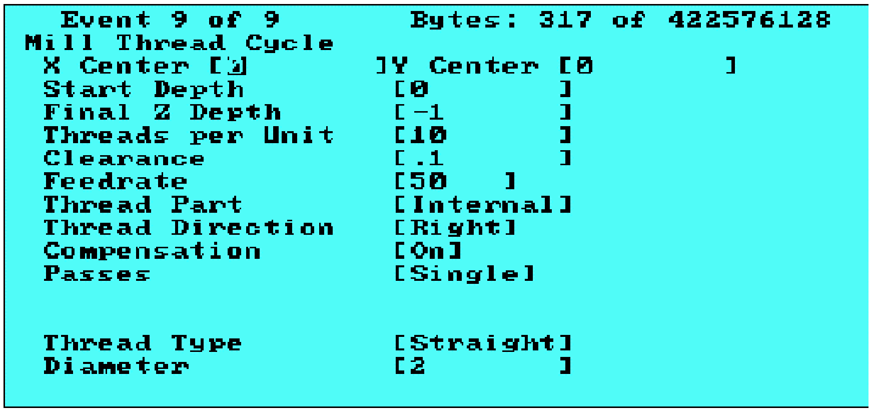
X Position (home relative)[ ]
Y Position (home relative)[ ]

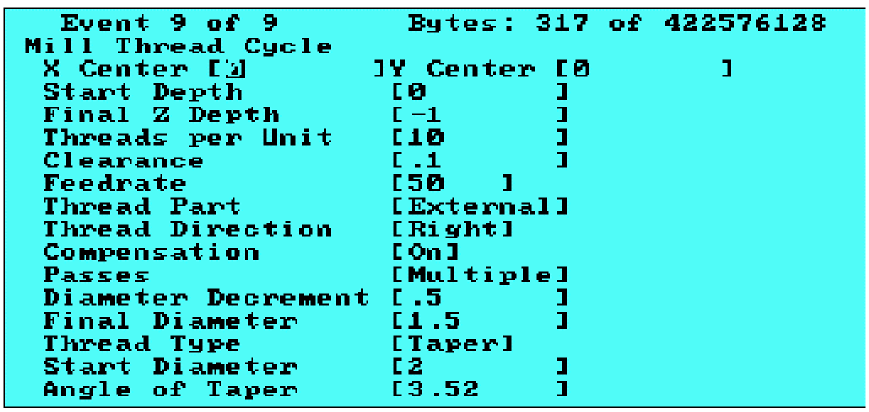
The following graphic illustrates the sample program using the 3d Arc:



F9 (Thred) Mill-Thred

The F9 (Thred) key brings up the thread milling input screen. Two examples are shown below.





Note: See page 234, Section 4 for the correct combination of cutter comp, cut direction.

F3 (Drill) Drill

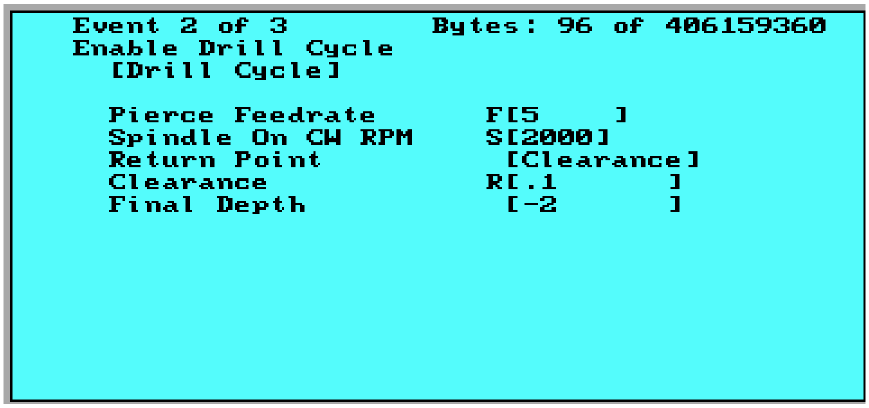
The F3 (Drill) selection brings up the following menu.

All drill cycles must be started prior to execution and ended after the last hole. This is done with the F1 (Start) and F5 (End) selections.

Conversational Screens

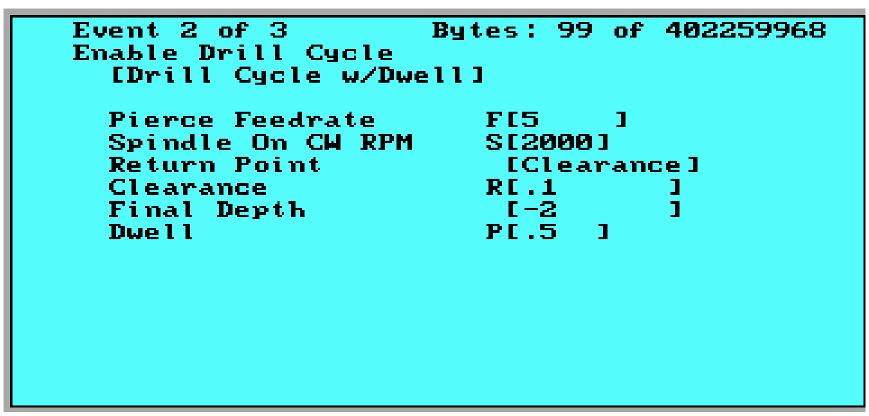
F3 (Drill) start drill cycle screen has a toggle field to select which type of drilling is to be executed. The start drill cycle is normally followed by the positions for the holes and ended with an end drill cycle event. The optional drill cycles are shown below.

Drill
The conversational screen for drill appears as follows.



Note: See page 281, Section 4 for further information on the drill cycle.

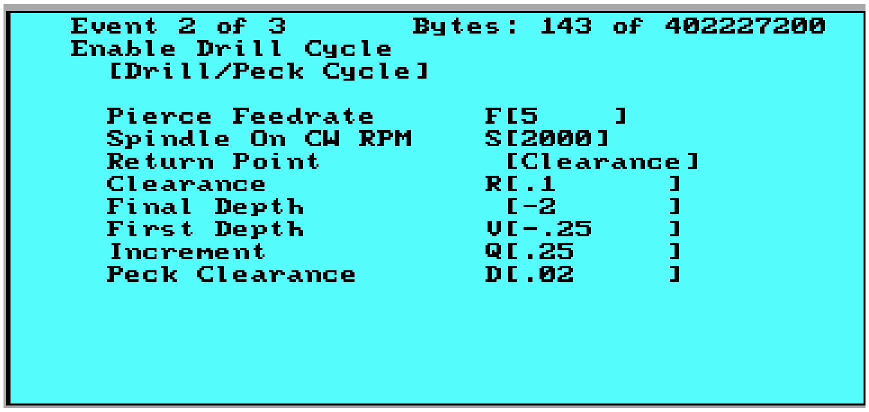
Drill/Dwell
The conversational screen for drill/dwell appears as follows.



Note: See page 281, Section 4 for further information on drill/dwell cycle.

Drill/Peck

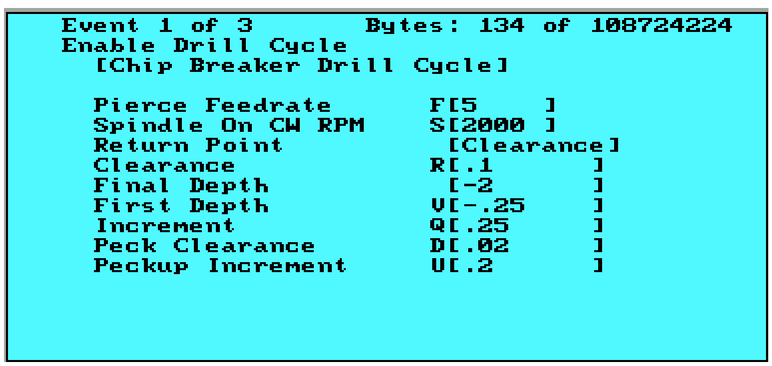
The conversational screen for peck drilling cycle appears as follows.



Note: See page 282, Section 4 for more information on peck drilling cycle.

Chip Breaker Drill

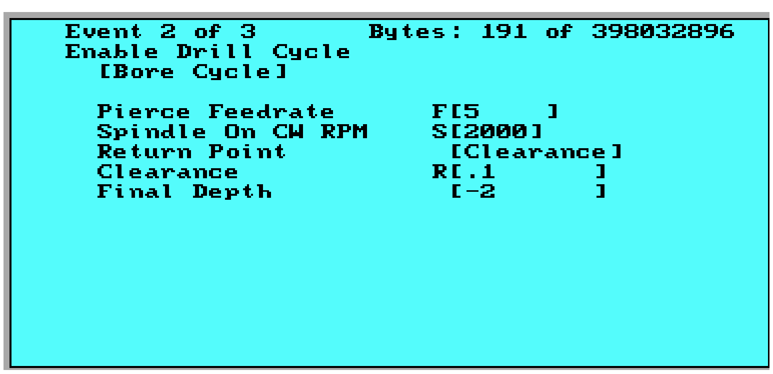
The conversational screen for chip breaker drill cycle appears as follows.



Note: See page 275, Section 4 for further information on chip breaker drilling cycle.

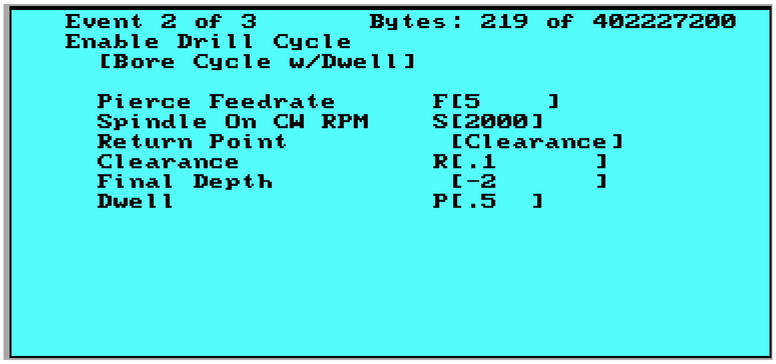
Bore

The conversational screen for bore cycle appears as follows.



Note: See page 284, Section 4 for further information on boring cycle.

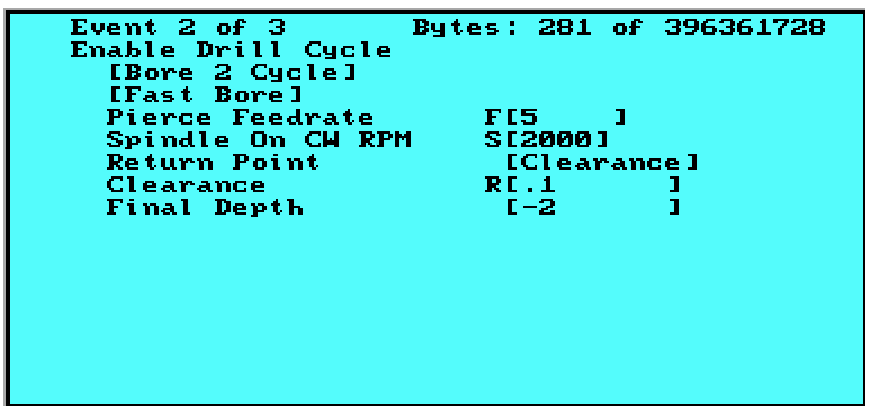
Bore/Dwell
The conversational screen for bore/dwell appears as follows.



Note: See page 289, Section 4 for more information on the bore/dwell cycle.

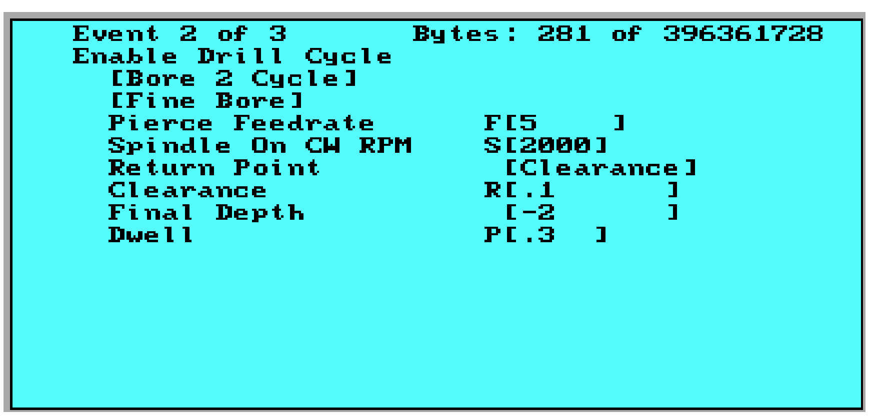
Bore 2
The conversational screens for bore 2 appears as follows.

Fast bore



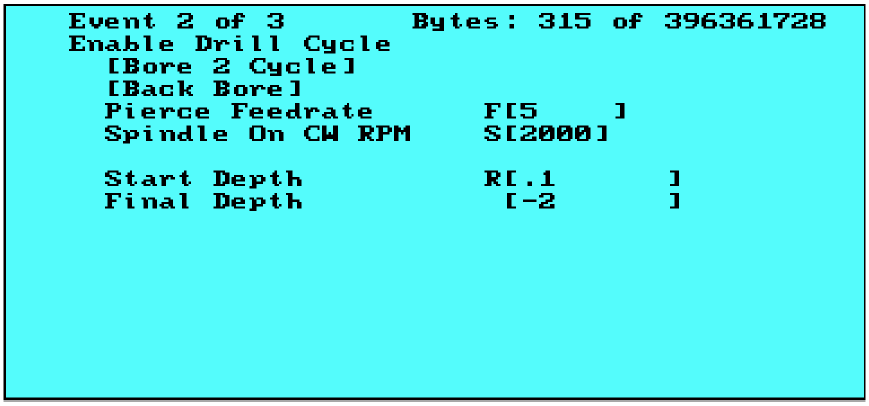
Note: See page 285, Section 4 for further information on fast bore cycle.

Fine bore



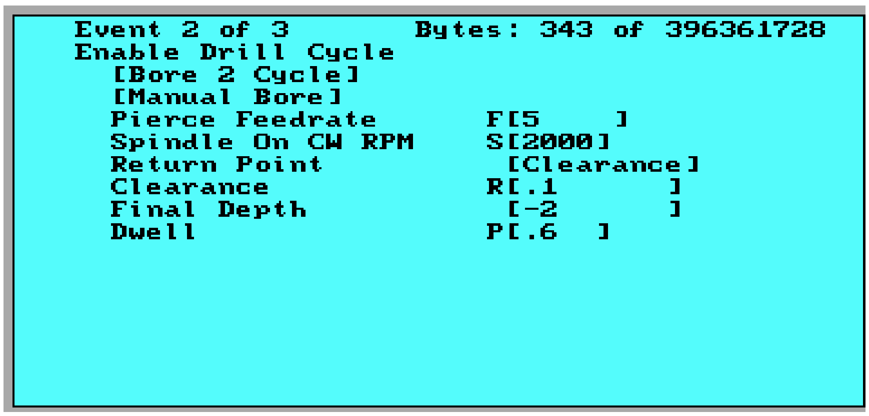
Note: See page 277, Section 4 for further information on fine bore cycle.

Back bore



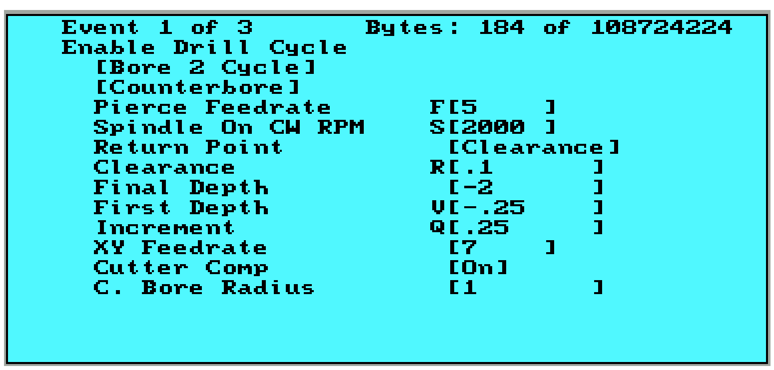
Note: See page 286, Section 4 for more information on the back bore cycle.

Manual bore



Note: See page 280, Section 4 for more information on the manual bore cycle.

Counter bore

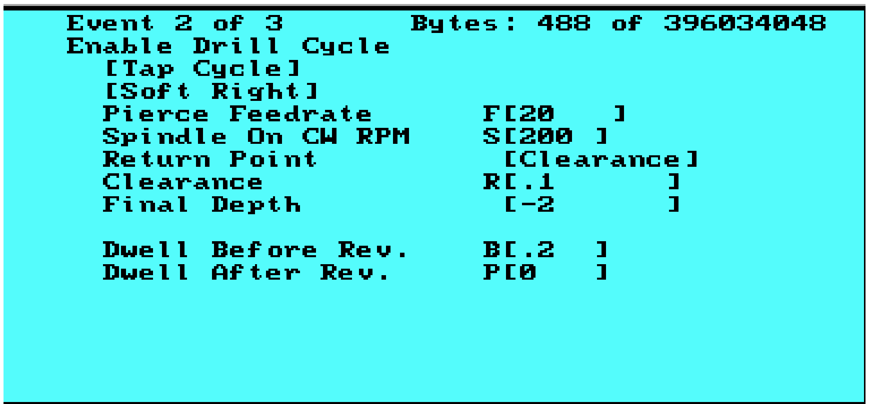


Note: See page 284, Section 4 for more information on the counter bore cycle.

Tap (Drill-Start-Tap)

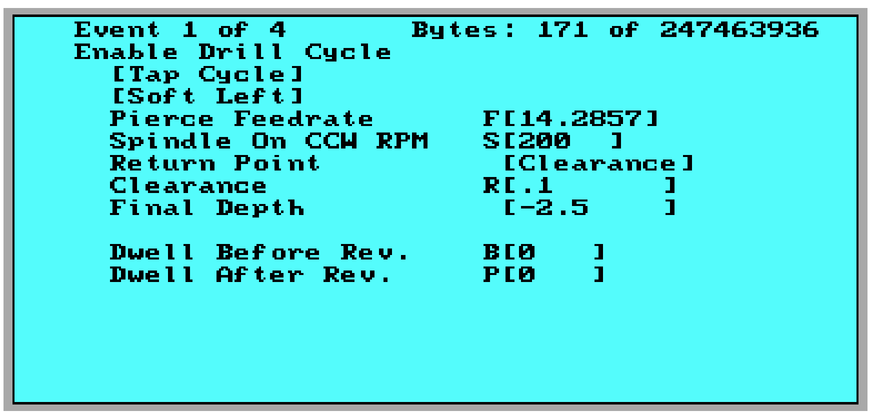
The conversational screens for tap drill cycle appear as follows.

Soft right tap



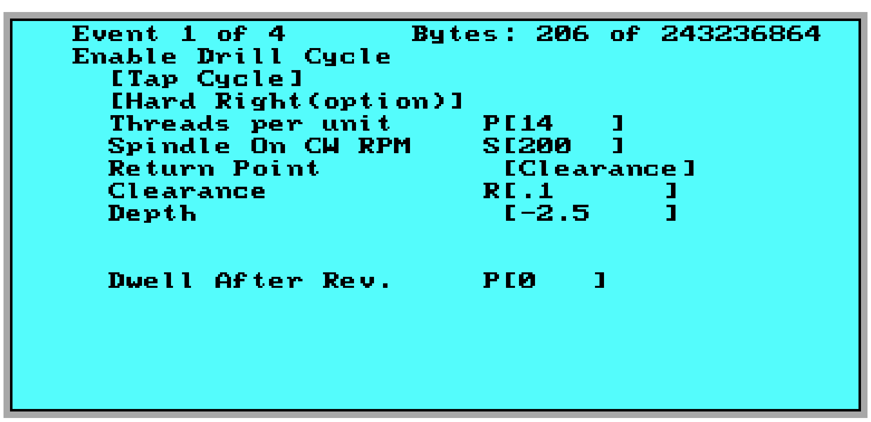
Note: See page 283, Section 4 for more information on the soft right tap cycle.

Soft left tap



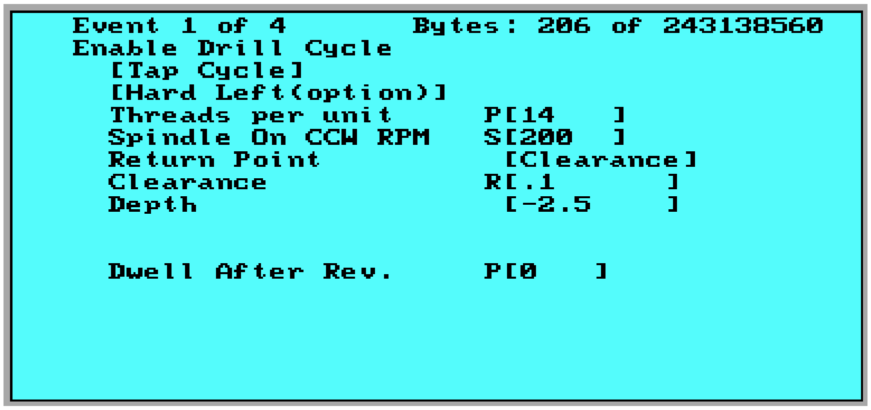
Note: See page 276, Section 4 for more information on the soft left tap cycle.

Hard right tap



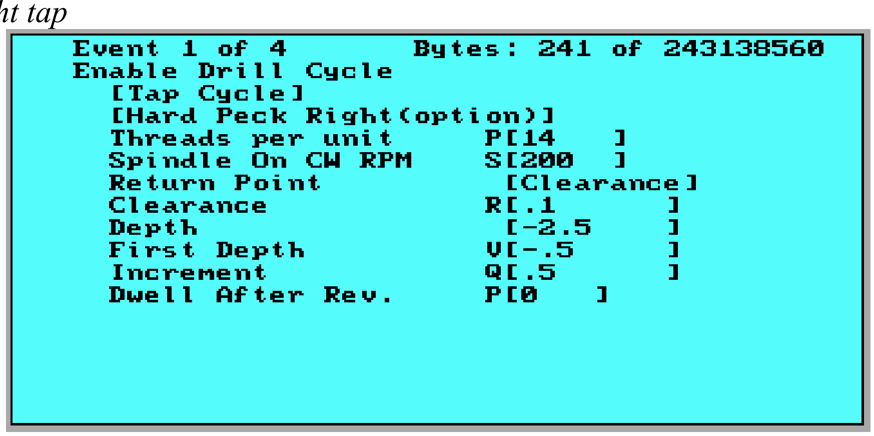
Note: See page 288, Section 4 for more information on the hard tap cycle.

Hard left tap



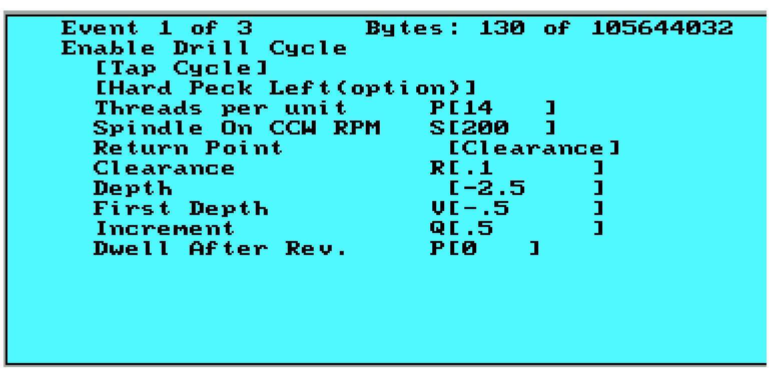
Note: See page 288, Section 4 for more information on the hard tap cycle.

Hard peck right tap



Note: See page 288, Section 4 for more information on the hard tap cycle.

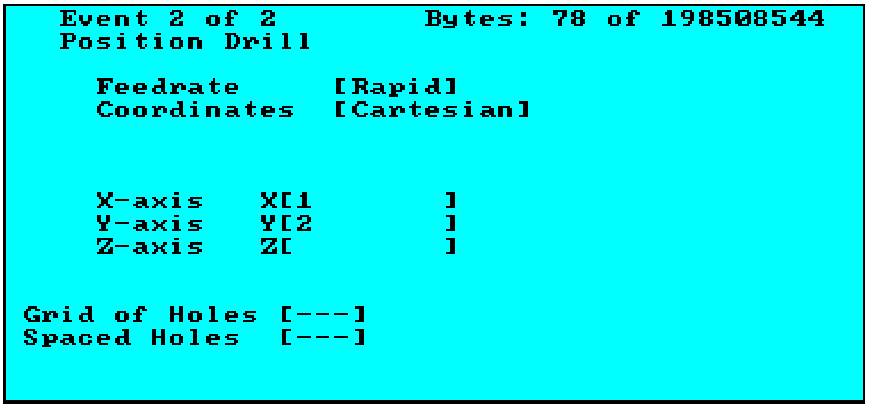
Hard peck leftt tap



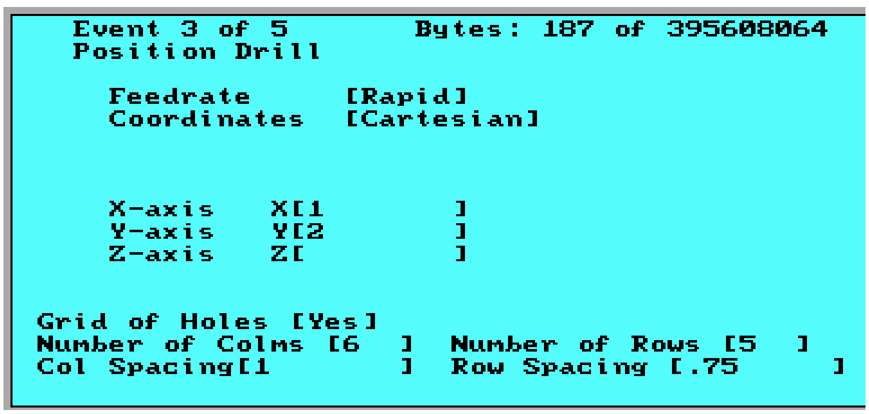
Note: See page 288, Section 4 for more information on the hard tap cycle.

F2 (Pos) Drill-Pos, F3 (Misc) Drill-Misc, F4 (Call) Drill-Call

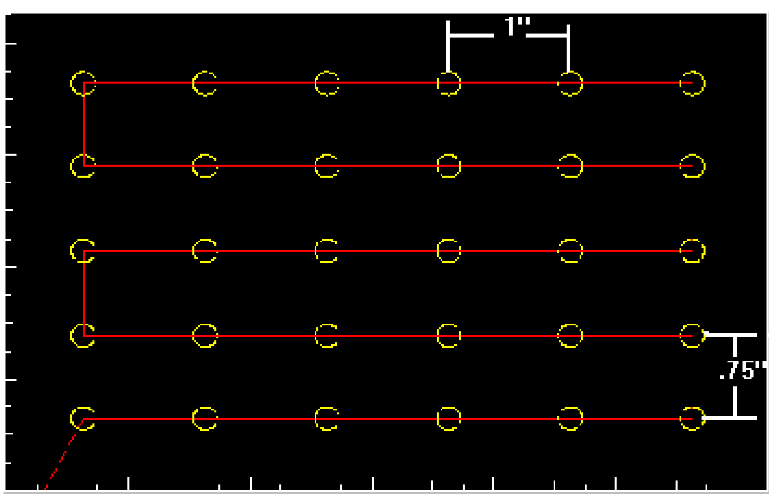
The F2 (Pos) key brings up the position screens. They are used to enter the drill positions.



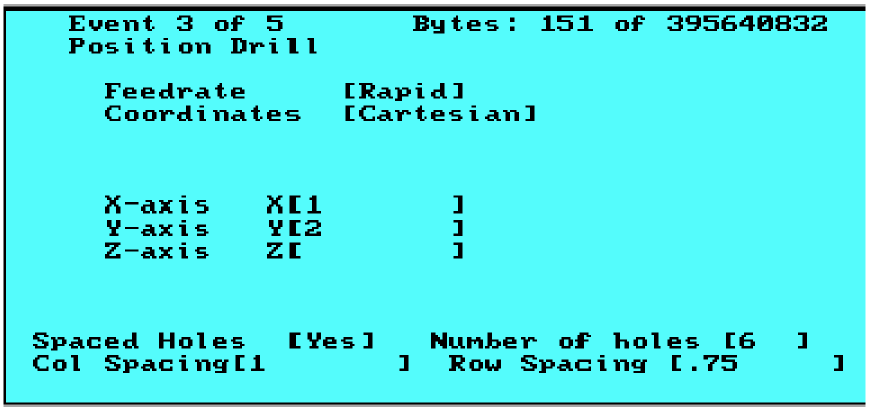
The screen pictured above is the single position hole drill screen. One hole will be drilled or tapped at (1,2). If a dimension is entered in the Z field on this screen it will drill the new depth at (1,2) and subsequent holes. If the only dimension entered appears in the Z field, the Z axis will move to that position; no hole will be drilled, and the Z depth will remain unchanged. This can be useful for clearing clamps or fixtures.



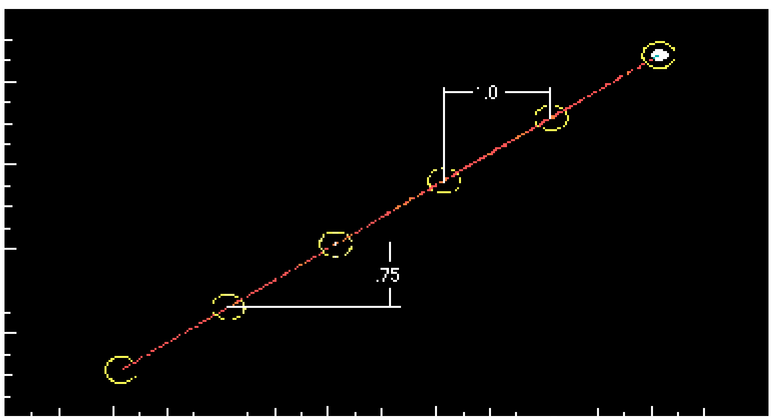
The screen pictured above is the position drill screen using the grid holes option. It will drill a 6 x 5 grid of holes with 1" spacing in the X axis and .75" spacing in the Y axis. The first hole is at (1,2). Below is a picture of the hole pattern created by this event.



The screen pictured below is the position drill screen using the spaced holes option. It will drill or tap six holes, including the hole at (1,2). If the X spacing field is 0, the line of holes would be drilled in a vertical line. If the Y spacing is 0, the holes would be drilled in a horizontal line.



The above screen would create the hole pattern shown below.



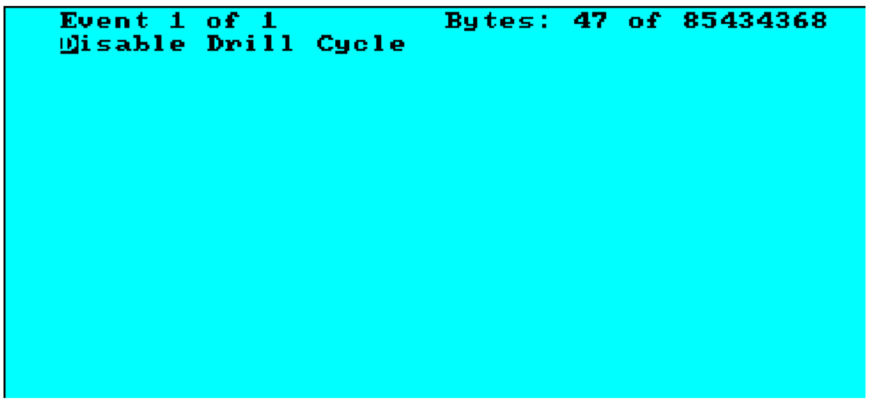
F3 (Misc) brings up the miscellaneous function screen and allows those functions to be programmed during drill cycles. The F4 (Call) screen allows subprograms to be called during a drill cycle. These subprograms would normally contain the drilling positions for different drilling operations.

Note: See page 177, Section 3 for more information on miscellaneous.

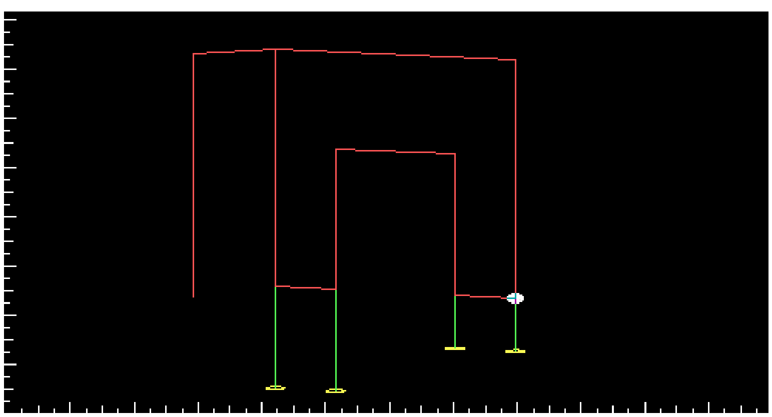
See page 178, Section 3 for more information on call.

F5 (End)

Below is the conversational screen for disable canned cycle.



This screen does not require any entries but must be stored in the program to terminate the active drill cycle. If this screen is not stored, every move will cause a Z axis drill cycle to be performed.



<u>Sample Drilling Program</u>

Event 0 of 17
Program Setup

Program name [Sample drilling program]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 17
Tool Change

Tool [Change]
Tool Change Position X[ ]
Y[ ]

Tool Number T[1]
Tool Description [DRILL]
Next Tool Number [ ]

Spindle Speed S[2000]
Spindle Restart [CW]
Stop For Speed Change [No]
Coolant [Flood]

Event 2 of 17
Enable Drill Cycle
[Drill Cycle]

Pierce Feedrate F[20]
Spindle On CW RPM S[ ]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-2]

Event 3 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[1]
Y axis Y[1] (Z to –2")

Z axis Z[ ]

Grid of Holes [---]
Spaced Holes [---]

Event 4 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[2]
Y axis Y[ ] (Z to –2")
Z axis Z[ ]

Grid of Holes [---]
Spaced Holes [---]

Event 5 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[ ]
Y axis Y[ ]
Z axis Z[3] (Z to 3" to get over a clamp)

Grid of Holes [---]
Spaced Holes [---]

Event 6 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[4]
Y axis Y[ ] (Z to –1")
Z axis Z[-1]

Grid of Holes [---]
Spaced Holes [---]

Event 7 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[5]
Y axis Y[ ] (Z to –1")
Z axis Z[ ]
Grid of Holes [---]
Spaced Holes [---]

Event 8 of 17
Disable Drill Cycle

Event 9 of 17
Tool Change

Tool [Change]
Tool Change Position X[ ]
Y[ ]

Tool Number T[2]
Tool Description [BORE]
Next Tool Number [ ]
Spindle Speed S[2000]
Spindle Restart [CW]
Stop For Speed Change [No]
Coolant [Flood]

Event 10 of 17
Enable Drill Cycle
[Bore Cycle]

Pierce Feedrate F[20]
Spindle On CW RPM S[ ]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-2]

Event 11 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[1]
Y axis Y[1] (Z to –2")
Z axis Z[ ]

Grid of Holes [---]
Spaced Holes [---]

Event 12 of 17
Position Drill

Feedrate [Rapid]

Coordinates [Cartesian]

X axis X[2]
Y axis Y[ ] (Z to –2")
Z axis Z[ ]

Grid of Holes [---]
Spaced Holes [---]

Event 13 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[ ]
Y axis Y[ ]
Z axis Z[3] (Z to 3" to get over a clamp)

Grid of Holes [---]
Spaced Holes [---]

Event 14 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[4]
Y axis Y[ ] (Z to –1")
Z axis Z[-1]

Grid of Holes [---]

Spaced Holes [---]

Event 15 of 17
Position Drill

Feedrate [Rapid]
Coordinates [Cartesian]

X axis X[5]
Y axis Y[ ] (Z to –1")
Z axis Z[ ]

Grid of Holes [---]
Spaced Holes [---]

Event 16 of 17
Disable Drill Cycle

Event 17 of 17
End of Program

Spindle off [Yes]
Coolant off [Yes]
Z to Tool change [Yes]

X Position (home relative) [ ]
Y Position (home relative) [ ]

Note: See page 266, Section 4 for further information on canned cycles.

F4 (Bolt)

The following conversational bolt hole drill screens are displayed upon selecting the bolt hole drill cycles. The first part of the screen contains information used to set up the appropriate drill cycle, whereas the last part contains information used to set up the bolt hole cycle.

```
  Event 2 of 3         Bytes: 115 of 385056768
Bolthole Cycle
[Drill Bolthole Cycle]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]



 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

```
  Event 2 of 3         Bytes: 115 of 385056768
Bolthole Cycle
[Drill Bolthole Cycle w/Dwell]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]
 Dwell                  P[1    ]


 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

```
  Event 2 of 3         Bytes: 229 of 385056768
Bolthole Cycle
[Drill/Peck Bolthole Cycle]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]
 First Depth            V[-.25      ]
 Increment              Q[.25       ]
 Peck Clearance         D[.1        ]

 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

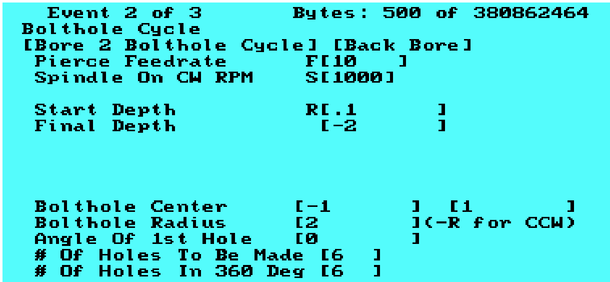
```
  Event 2 of 3         Bytes: 296 of 385056768
Bolthole Cycle
[Woodpecker Drill Bolthole Cycle]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]
 First Depth            V[-.25      ]
 Increment              Q[.25       ]
 Peck Clearance         D[.1        ]
 Peckup Increment       U[.2        ]
 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

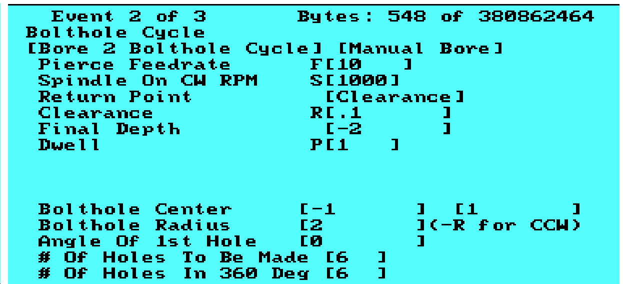
```
  Event 2 of 3         Bytes: 344 of 385056768
Bolthole Cycle
[Bore Bolthole Cycle w/Dwell]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]
 Dwell                  P[.5   ]


 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

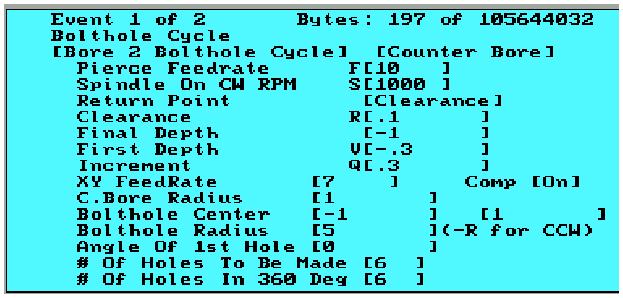
```
  Event 2 of 3         Bytes: 344 of 385056768
Bolthole Cycle
[Bore Bolthole Cycle]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]



 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

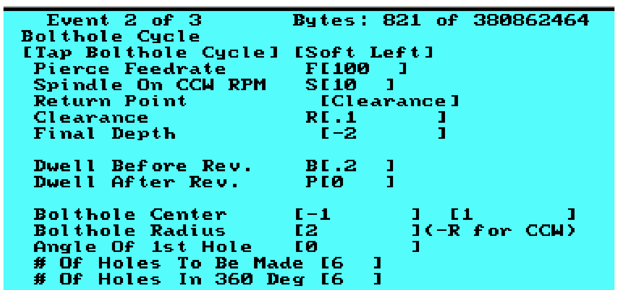
```
  Event 2 of 3         Bytes: 396 of 385056768
Bolthole Cycle
[Bore 2 Bolthole Cycle] [Fast Bore]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]



 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

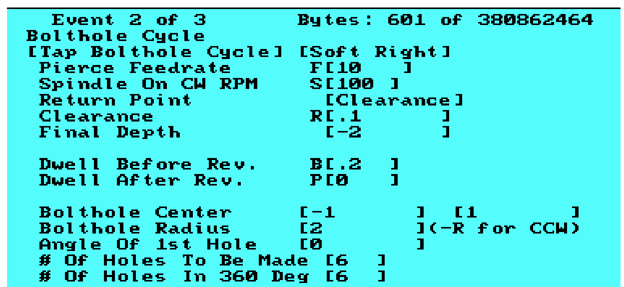
```
  Event 2 of 3         Bytes: 446 of 380862464
Bolthole Cycle
[Bore 2 Bolthole Cycle] [Fine Bore]
 Pierce Feedrate        F[10    ]
 Spindle On CW RPM      S[1000]
 Return Point             [Clearance]
 Clearance              R[.1        ]
 Final Depth              [-2       ]
 Dwell                  P[.7   ]


 Bolthole Center       [-1         ]  [1         ]
 Bolthole Radius       [2          ](-R for CCW)
 Angle Of 1st Hole     [0          ]
 # Of Holes To Be Made [6   ]
 # Of Holes In 360 Deg [6   ]
```

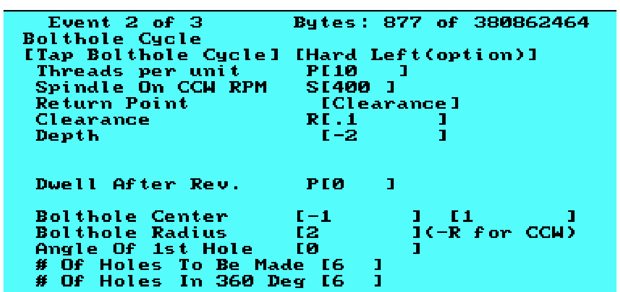

Event 2 of 3 Bytes: 500 of 380862464
Bolthole Cycle
[Bore 2 Bolthole Cycle] [Back Bore]
Pierce Feedrate F[10]
Spindle On CW RPM S[1000]
Start Depth R[.1]
Final Depth [-2]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]

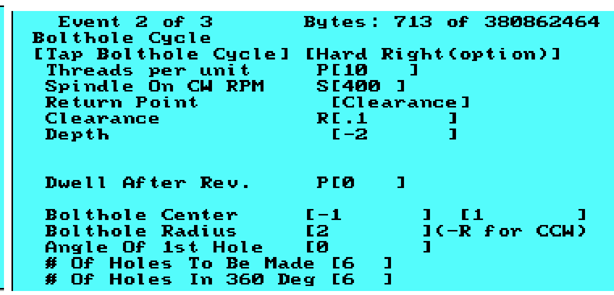

Event 2 of 3 Bytes: 548 of 380862464
Bolthole Cycle
[Bore 2 Bolthole Cycle] [Manual Bore]
Pierce Feedrate F[10]
Spindle On CW RPM S[1000]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-2]
Dwell P[1]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]

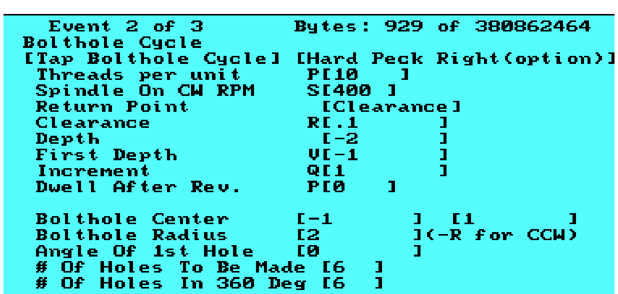

Event 1 of 2 Bytes: 197 of 105644032
Bolthole Cycle
[Bore 2 Bolthole Cycle] [Counter Bore]
Pierce Feedrate F[10]
Spindle On CW RPM S[1000]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-1]
First Depth V[-.3]
Increment Q[.3]
XY FeedRate [7] Comp [On]
C.Bore Radius [1]
Bolthole Center [-1] [1]
Bolthole Radius [5](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]

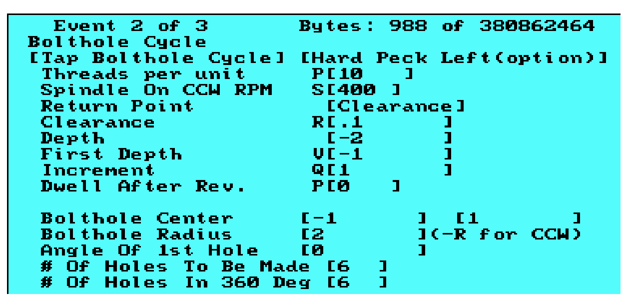

Event 2 of 3 Bytes: 821 of 380862464
Bolthole Cycle
[Tap Bolthole Cycle] [Soft Left]
Pierce Feedrate F[100]
Spindle On CCW RPM S[10]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-2]
Dwell Before Rev. B[.2]
Dwell After Rev. P[0]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]


Event 2 of 3 Bytes: 601 of 380862464
Bolthole Cycle
[Tap Bolthole Cycle] [Soft Right]
Pierce Feedrate F[10]
Spindle On CW RPM S[100]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-2]
Dwell Before Rev. B[.2]
Dwell After Rev. P[0]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]


Event 2 of 3 Bytes: 877 of 380862464
Bolthole Cycle
[Tap Bolthole Cycle] [Hard Left(option)]
Threads per unit P[10]
Spindle On CCW RPM S[400]
Return Point [Clearance]
Clearance R[.1]
Depth [-2]
Dwell After Rev. P[0]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]


Event 2 of 3 Bytes: 713 of 380862464
Bolthole Cycle
[Tap Bolthole Cycle] [Hard Right(option)]
Threads per unit P[10]
Spindle On CW RPM S[400]
Return Point [Clearance]
Clearance R[.1]
Depth [-2]
Dwell After Rev. P[0]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]


Event 2 of 3 Bytes: 929 of 380862464
Bolthole Cycle
[Tap Bolthole Cycle] [Hard Peck Right(option)]
Threads per unit P[10]
Spindle On CW RPM S[400]
Return Point [Clearance]
Clearance R[.1]
Depth [-2]
First Depth V[-1]
Increment Q[1]
Dwell After Rev. P[0]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]


Event 2 of 3 Bytes: 988 of 380862464
Bolthole Cycle
[Tap Bolthole Cycle] [Hard Peck Left(option)]
Threads per unit P[10]
Spindle On CCW RPM S[400]
Return Point [Clearance]
Clearance R[.1]
Depth [-2]
First Depth V[-1]
Increment Q[1]
Dwell After Rev. P[0]
Bolthole Center [-1] [1]
Bolthole Radius [2](-R for CCW)
Angle Of 1st Hole [0]
Of Holes To Be Made [6]
Of Holes In 360 Deg [6]

Sample Bolt Hole Program

Event 0 of 3
Program Setup

Program name [Sample Bolt hole Program]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 3
Tool Change

Tool [Change]
Tool Change Position X[ ]
Y[ ]

Tool Number T[1]
Tool Description [DRILL]
Next Tool Number [ ]
Spindle Speed S[2000]
Spindle Restart [CW]
Stop For Speed Change [No]
Coolant [Flood]

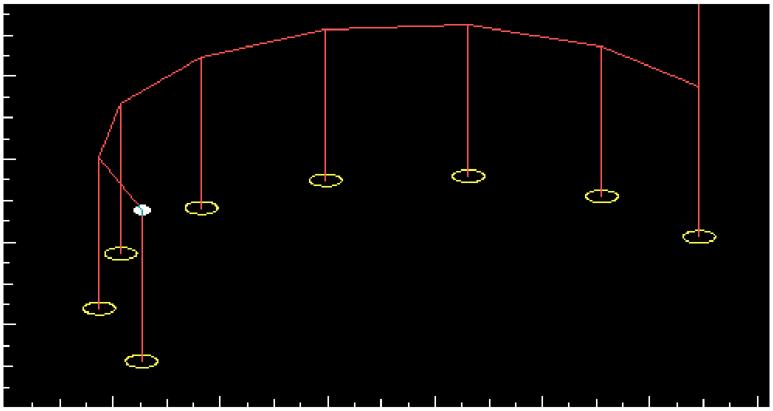
Event 2 of 3
Bolt hole Cycle
[Drill Bolt hole Cycle]
Pierce Feedrate F[10]
Spindle On CW RPM S[ ]
Return Point [Clearance]
Clearance R[.1]
Final Depth [-2]

Bolt hole Center [-4] [2]

Bolt hole Radius	[-3](-R for CCW)
Angle Of 1st Hole	[90]
# Of Holes To Be Made	[8]
# of Holes in 360 Deg	[14]

Event 3 of 3
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[Yes]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]



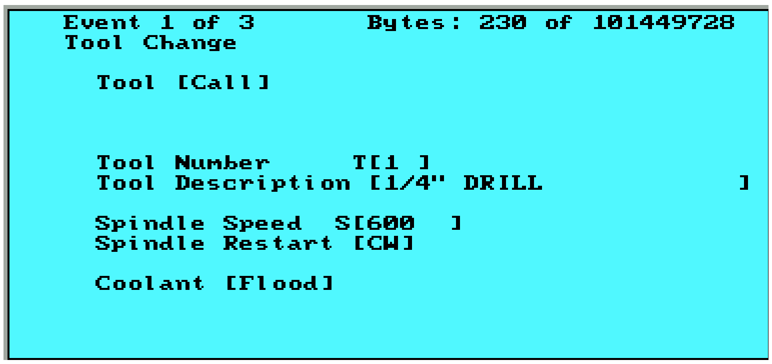
Notes: See page 272, Section 4 for further information on bolt hole routine.

For both the drill and the bolt hole cycles, the conversational input screens are different for versions of software older than #.116. They are also different for foreign translations and #.110 for the English translation. The input fields are the same for all versions of the software, but the appearance and steps for arrival to the input screen are slightly different. For each cycle in the old version a row of keys appears for the programmer to choose the type of cycle that he wants to perform instead of having the toggle field to choose the desired cycle, as in the newer versions. If a conversational program is created with an old style of drill or bolt hole cycle, the conversational program still can be edited with the newer versions of software. However, if a conversational program with the new style drill or bolt hole is created, it cannot be edited on the older versions of software. A fatal error 100, "disk read error," *will appear when an older version of software tries to read the new style of drill or bolt hole cycle in the conversational program. If a conversational program with the newer style of drill or bolt hole cycle is edited on the older software and the fatal error 100 is not given, the control will incorrectly post out the text program. It will eliminate the event containing the newer style of drill or bolt hole cycle. The control creates a text program corresponding to the inputs given in the conversational program, and it is this text program that is run by the control. The text program that the control generates is identical in both styles, so it is possible to edit and run text programs in each of the newer and older versions of software no matter where the text program was created.*

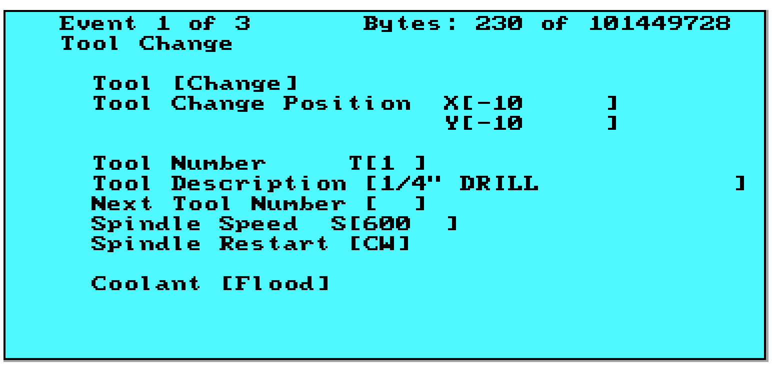
F5 (TChng) Tool Change

When a new tool needs to be put in the machine tool, the tool change screen should be used. The two tool change screens are tool call and tool change. The tool call is used to initiate a new set of tool offsets without physically changing the tool. The tool change puts the machine in a tool change mode and calls for a new tool. When a tool change or tool call is executed, the H and D offsets which are activated will be the ones which are the same as the tool number. For safety reasons, when doing manual tool changes, the machine should **always** be in a tool change mode. The tool screens are shown below.

The conversational screen for tool call appears as follows. No tool change is performed. The tool offsets are called. The spindle and coolant commands are executed.



Conversational screen for tool change.



When a tool change position is entered, if the Z axis is at or above the tool change position, the XY move will be executed prior to any Z axis movement. If Z is below the tool change position, Z will move to the tool change position, and then X and Y will move. If the X and Y number are not filled in, only Z will move during a tool change.

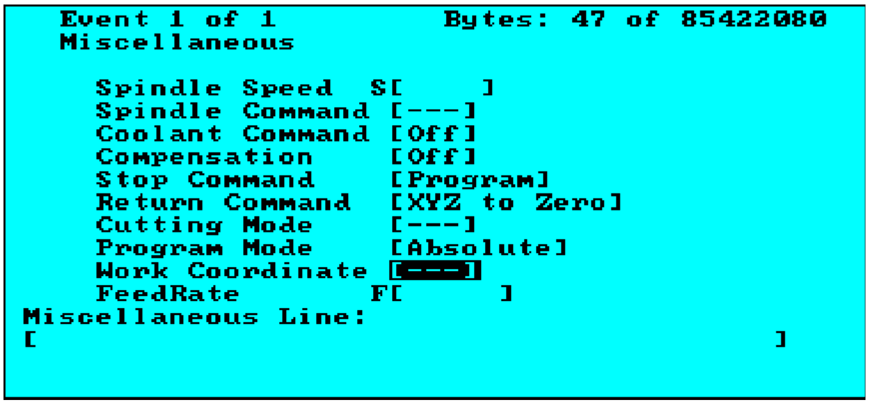
The next tool number field is used for migrating tool changers. The tool holder will index to the next tool while it cuts with the active tool.

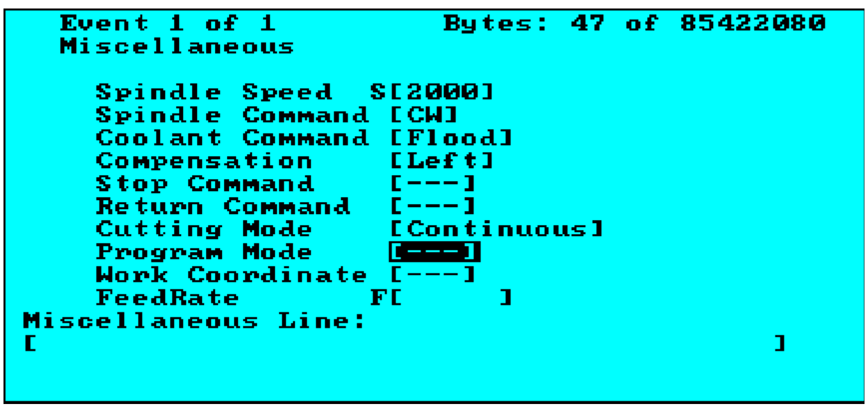
Note: See page 305, Section 5 for further information on tool change.

F6 (Misc)

As a program is being created it may be necessary to add certain miscellaneous functions such as coolant and stop commands. This is done through the F6 (Misc) screen.

Conversational screens for miscellaneous appear below. The miscellaneous line is used to type in any M code that is not part of the standard list.

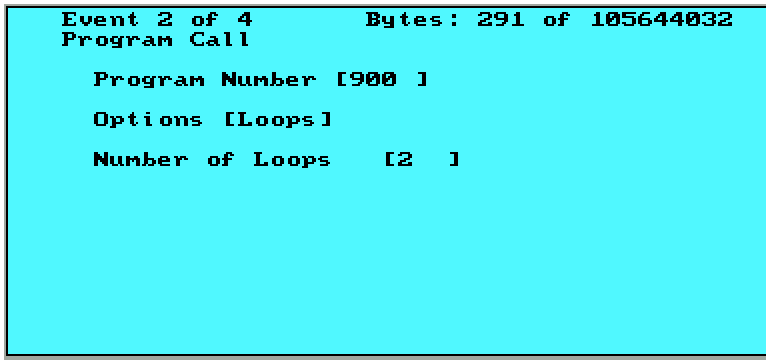




F7 (Call)

The program call screen is used to transfer program execution to another program for a specified number of loops.

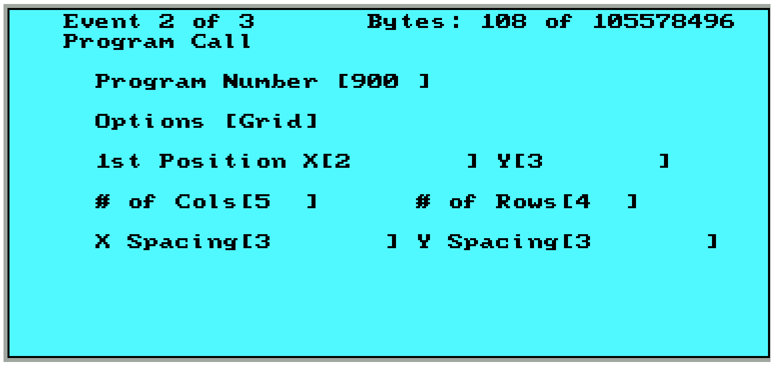
The conversational screen for program call appears below.

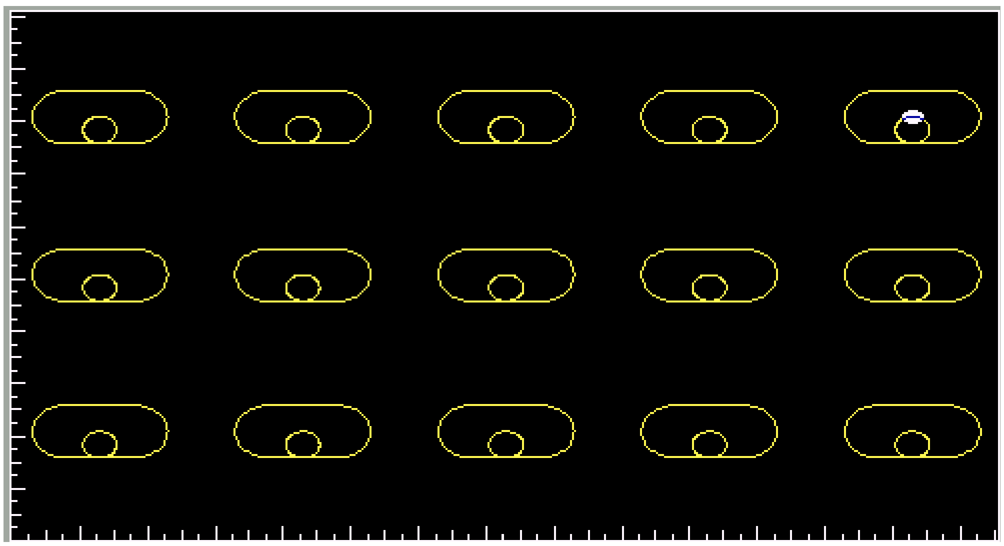


If the number of loops is left blank, the subprogram is called once.

Note: See page 317, Section 5 for further information on subprogram call.

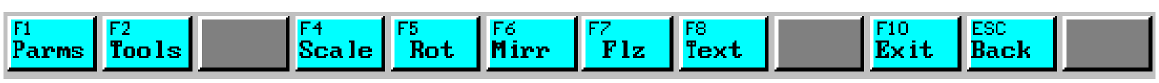
Subprogram calls can be used to create a number of identical parts in a row or a grid. The screen below can be used to call a subprogram that cuts a slot.





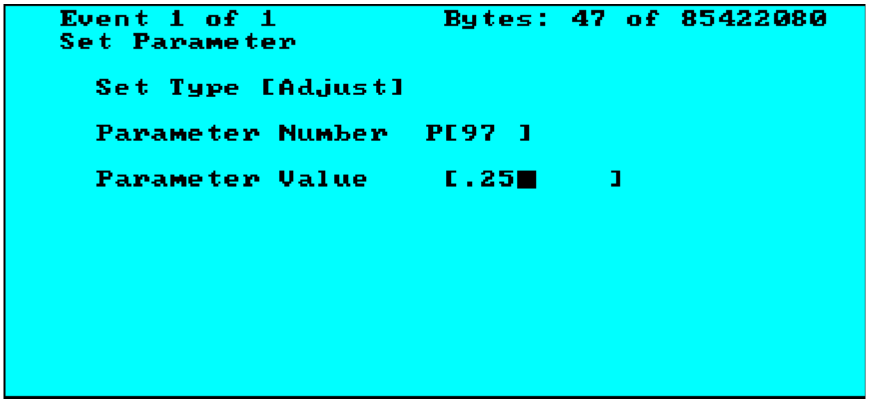
F8 (Spec)

These are screens for setting or adjusting various parameters in the Centurion 6 control. The parameters control various functions, such as tool offsets, scale factors, rotation angles, mirror image, floating zeroes, and the parameters listed in Appendix A. The keys are as follows.



F1 (Parms) Spec-Parms

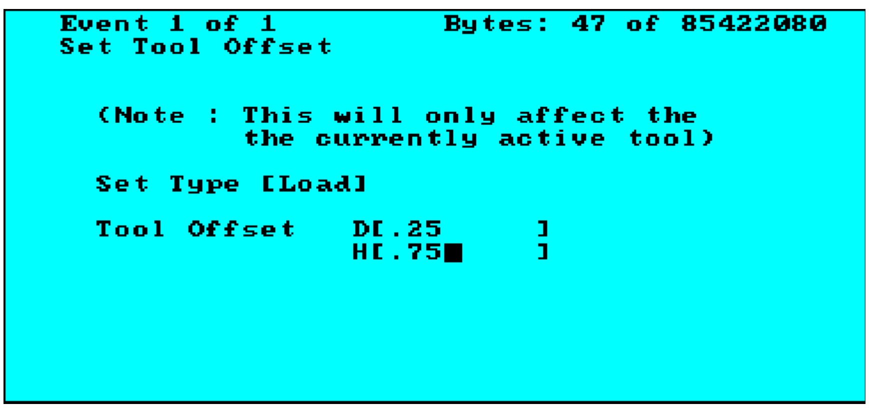
The conversational screen for adjust parameter appears below.



Loading a parameter will set the parameter to the specified value.
Adjusting a parameter will add the specified value to the current setting.

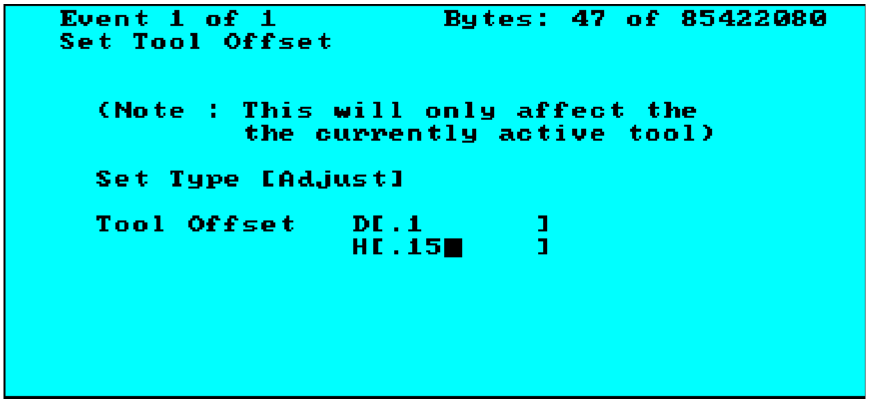
F2 (Tools) Spec-Tools

The conversational screen for set tool offset appears below.



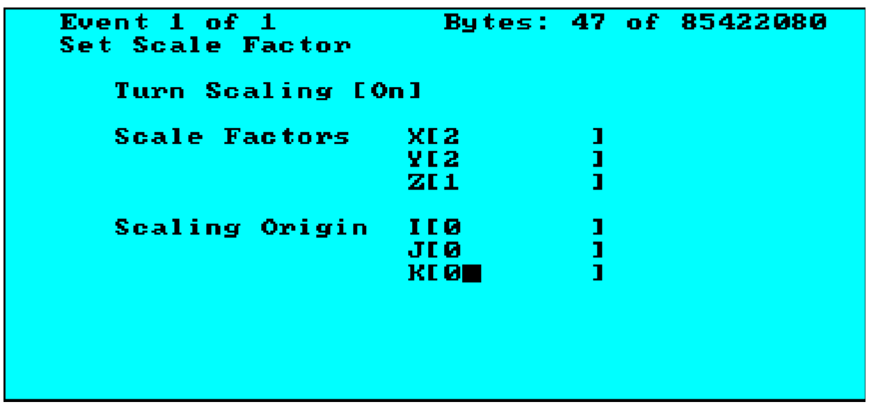
Note: See page 253, Section 4 for further information on tool length offset.

The conversational screen for adjust tool offset appears below.

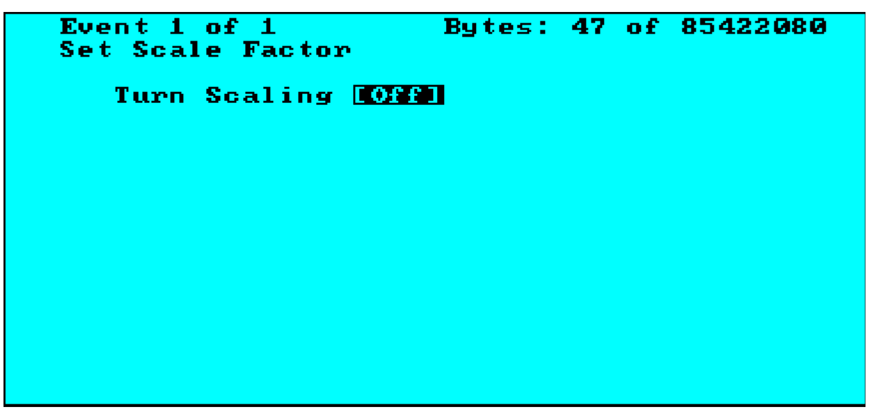


F4 (Scale) Spec-Scale

The conversational screen for turn scale factor on appears below.



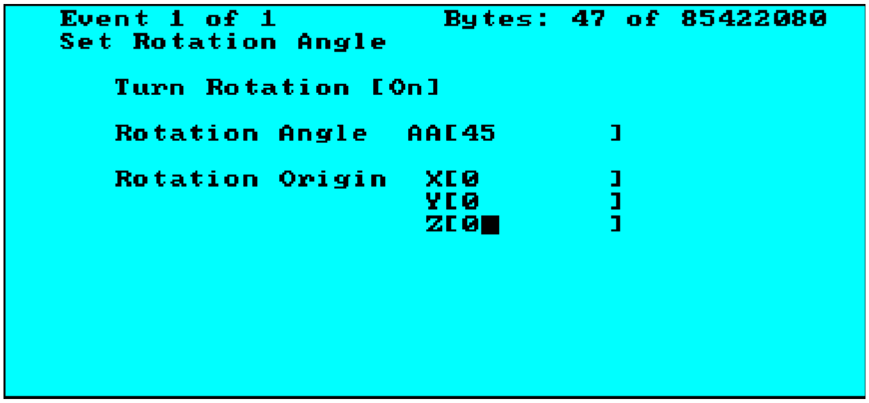
The conversational screen for turn scale factor off appears below.



Note: See page 254, Section 4 for further information on set and cancel scaling.

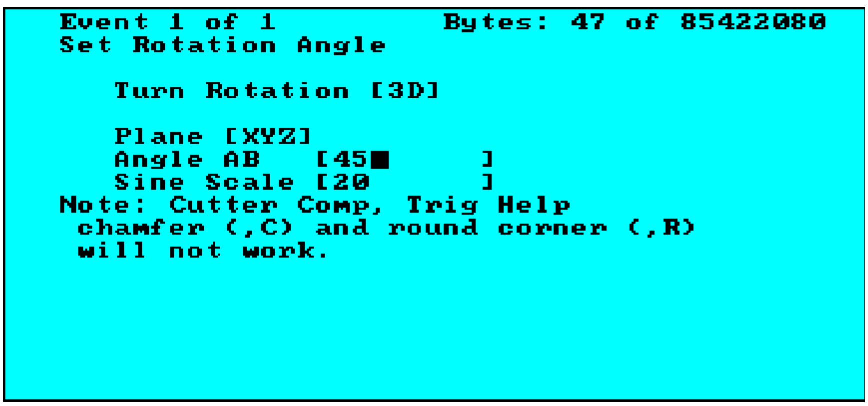
F5 (Rot) Spec-Rot

The conversational screen for turn rotation on appears as follows.



Note: See page 260, Section 4 for further information on coordinate system rotation.

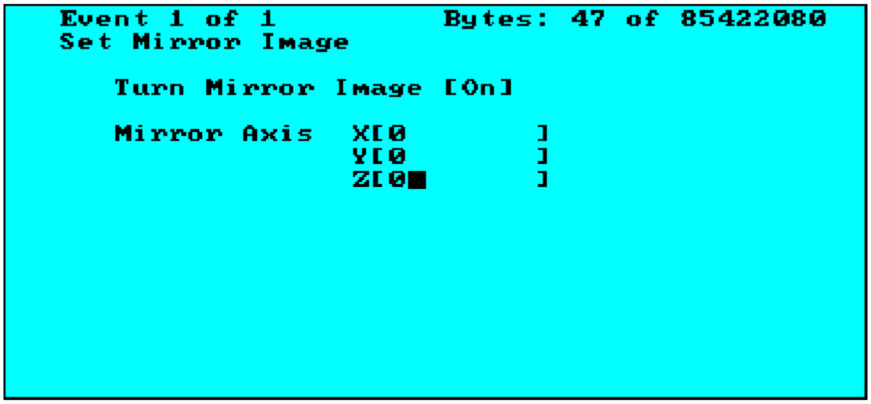
The conversational screen for set 3D rotation angle appears below.



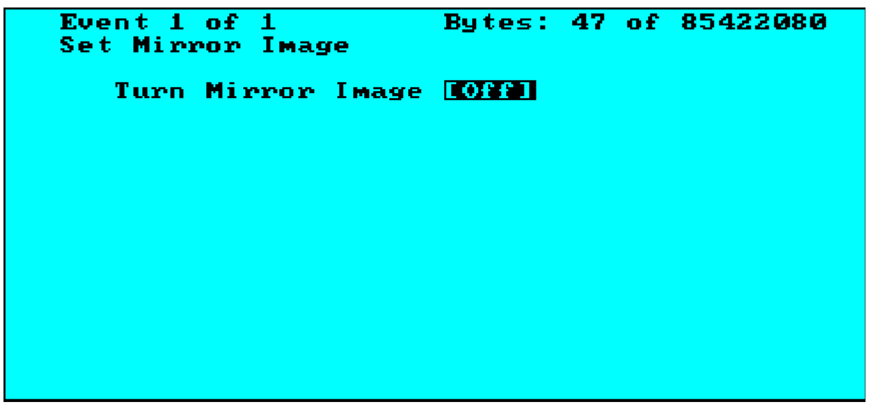
3D rotation is used to tilt parts out of the plane.

F6 (Mirr) Spec-Mirr

The conversational screen for set mirror image on appears below.



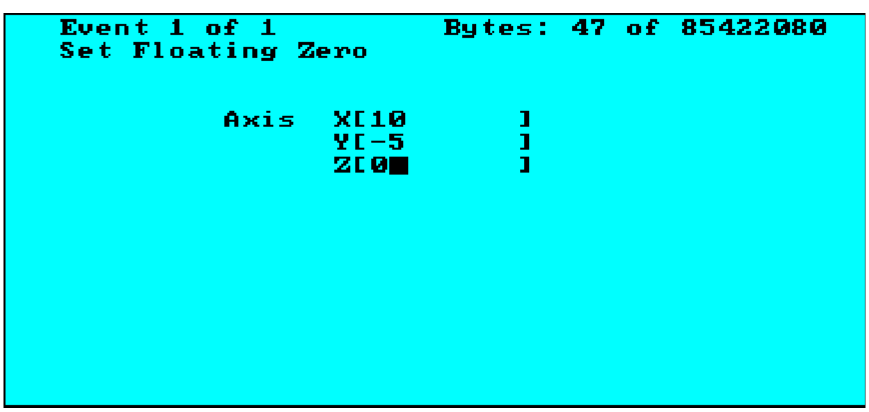
The conversational screen for set mirror image off appears below.



Note: See page 265, Section 4 for further information on mirror image set and cancel.

F7 (Flz) Spec-Flz

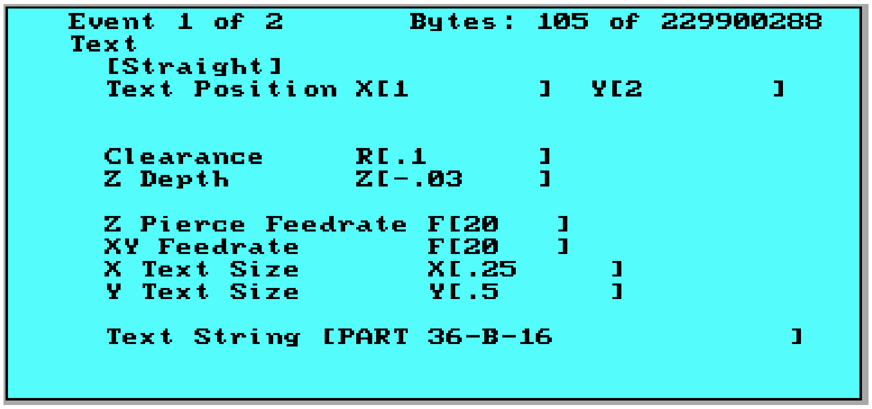
The conversational screen for set floating zero appears below.



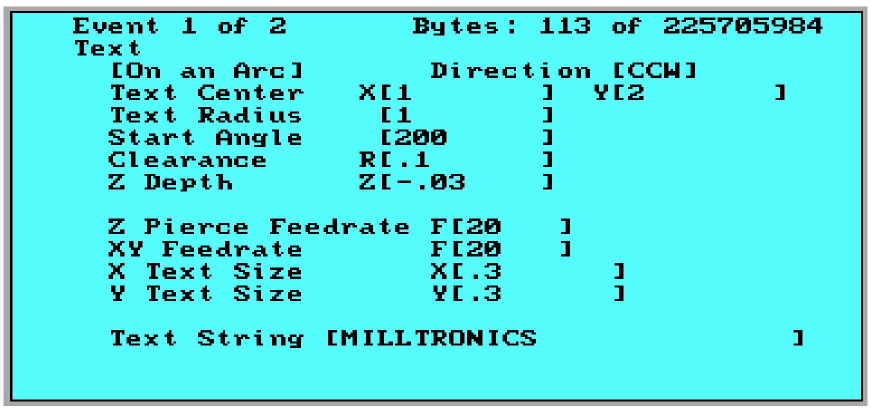
Note: See page 294, Section 4 for further information on floating zeros.

F8 (Text) Spec-Text

The conversational screen for text appears below.

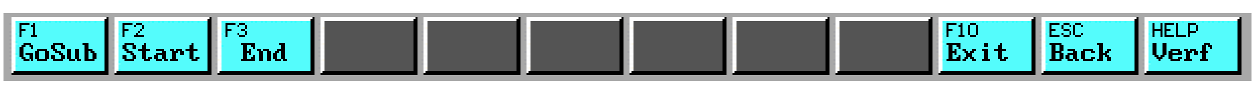


The conversational screen for text on an arc:



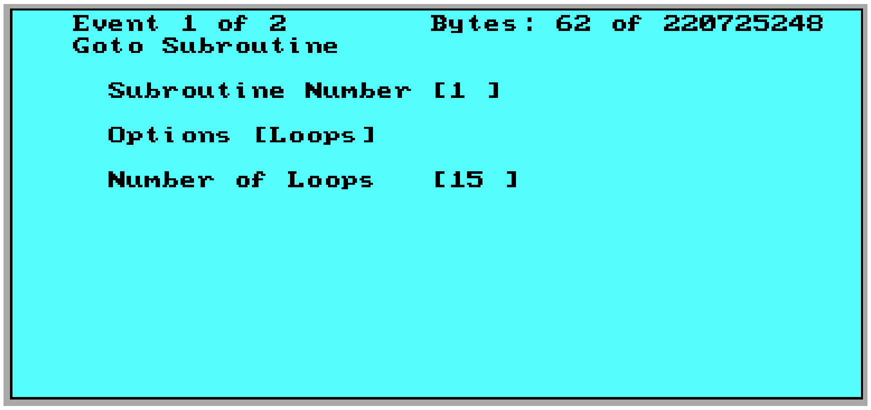
F9 (Subs)

These screens are used to define and call subroutines.
The keys show are as follows:

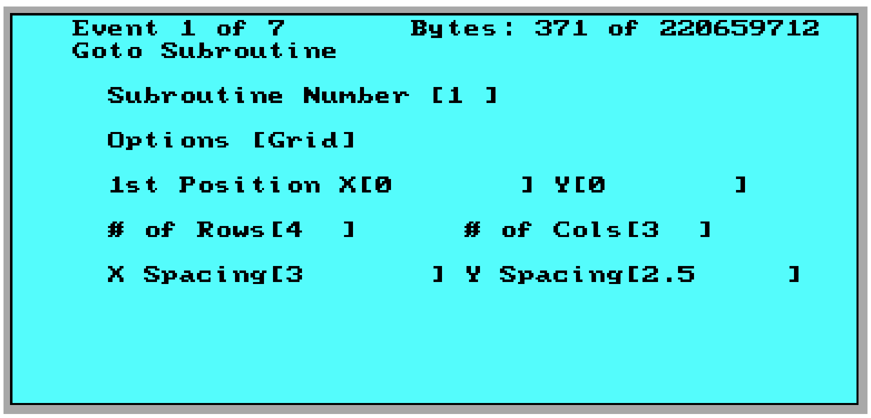


F1 (Gosub) Subs-Gosub

Gosub is used to call a subroutine. The screen below calls subroutine 1 fifteen times. If the number of loops is left blank, the subroutine is called 1 time.



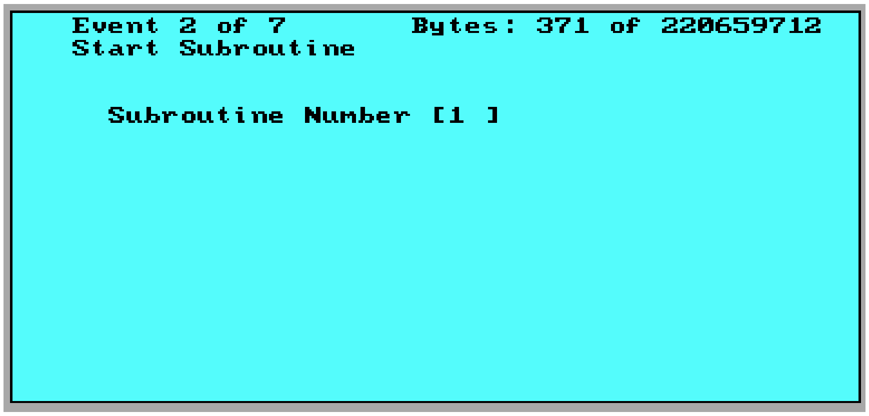
Another option on the gosub screen is used to call the subroutine and repeat it in a rectangular pattern.



Note: The other option is to call the subroutine using XY spacing between each. The grid and the spaced are similar to holes in the drill cycles on page 162, Section 3.

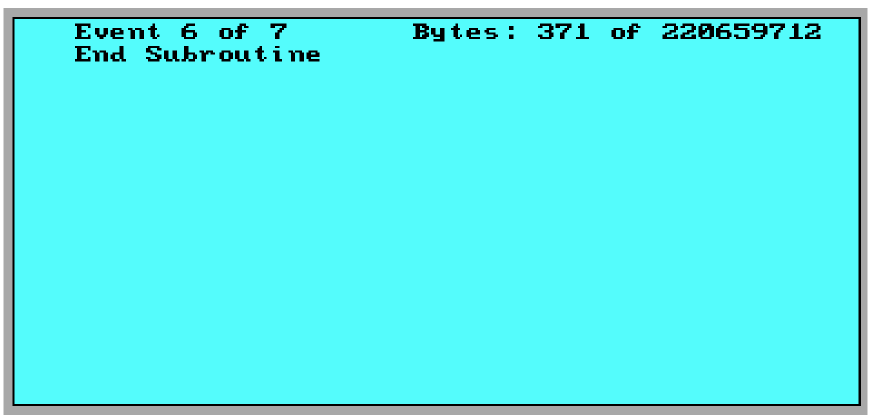
F2 (Start) Subs-Start

The start subroutine screen defines the start of a subroutine.



F3 (End) Subs-End

The end subroutine screen defines the end of the subroutine.



Sample Program Using Subroutines

Conversational Program C:\CNC\PARTS\P0523
Event 0 of 12
Program Setup

Program name [ ]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 12
Tool Change

Tool [Change]
Tool Change Position X[ ]
Y[ ]

Tool Number T[1]
Tool Description [ ]
Next Tool Number [ ]
Spindle Speed S[600]
Spindle Restart [CW]

Coolant [Flood]

Event 2 of 12
Goto Subroutine

Subroutine Number [1]

Options [Grid]

1st Position X[0] Y[0]

# of Rows[4] # of Cols[3]

X Spacing[3] Y Spacing[2.5]

```
---------------------------------------------------
Event 3 of 12
Tool Change

 Tool                       [Change]
 Tool Change Position       X[    ]
                            Y[    ]

 Tool Number                T[2 ]
 Tool Description           [          ]
 Next Tool Number           [ ]
 Spindle Speed              S[1200 ]
 Spindle Restart            [CW]

 Coolant                    [Mist]
---------------------------------------------------
Event 4 of 12
Goto Subroutine

 Subroutine Number          [2 ]

 Options                    [Grid]

 1st Position               X[0    ] Y[0    ]

 # of Rows[4 ]   # of Cols[3 ]

 X Spacing[3    ] Y Spacing[2.5    ]
---------------------------------------------------
Event 5 of 12
Start Subroutine

 Subroutine Number          [1 ]
---------------------------------------------------
```

Event 6 of 12
Pocket Mill Setup

X Pocket Center [0]
Y Pocket Center [0]

XY Feedrate [10]
Z Pierce Feedrate [5]
Return Point [Clearance]
Clearance [.1]
Final Z Depth [-.4]
First Z Depth [-.1]
Z Increment [.1]

Event 7 of 12
Circular Finish Inside

Pocket Radius [1]

Z Down [Plunge]
Cut Direction [CCW (Climb)]
Compensation [On]

Event 8 of 12
End Subroutine

Event 9 of 12
Start Subroutine

Subroutine Number [2]

Event 10 of 12
Text
[On an Arc] Direction [CW]
Text Center X[0] Y[0]
Text Radius [1]
Start Angle [180]
Clearance R[.1]
Z Depth Z[-.03]

Z Pierce Feedrate F[20]
XY Feedrate F[30]
X Text Size X[.3]
Y Text Size Y[.4]

Text String [MILLTRONICS]

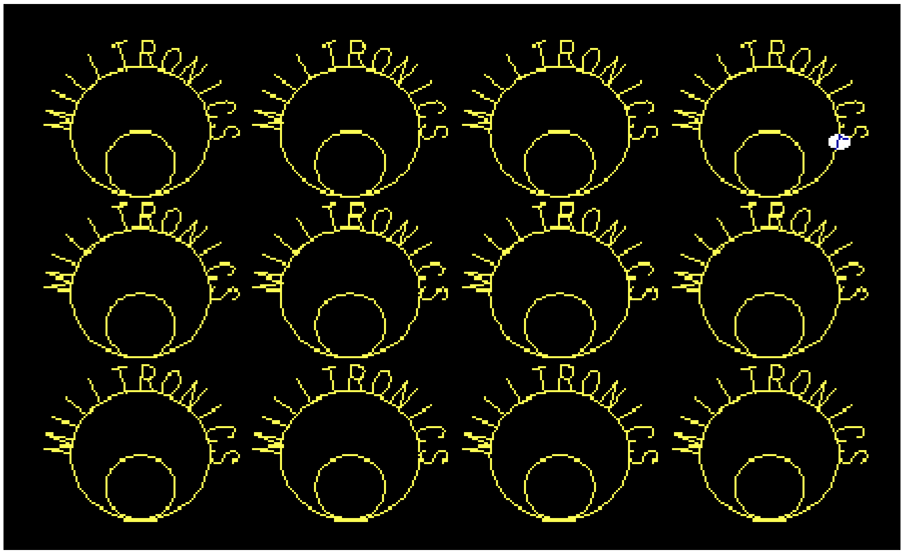
Event 11 of 12
End Subroutine

Event 12 of 12
End of Program

Spindle off [No]
Coolant off [No]
Z to Toolchange [No]

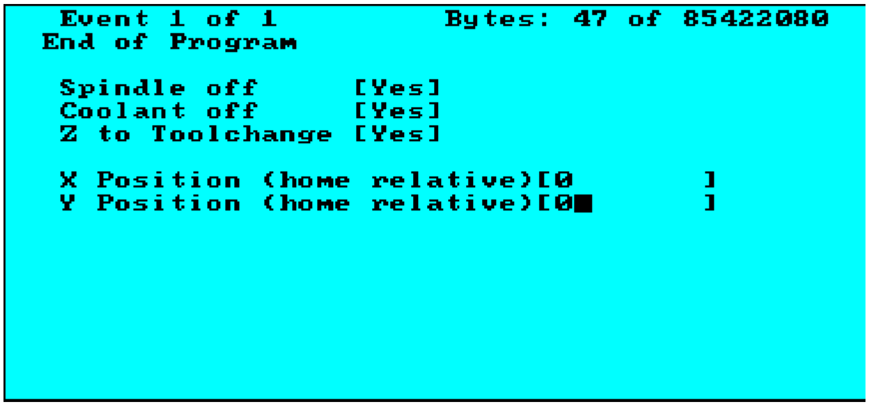
X Position (home relative)[ ]
Y Position (home relative)[ ]

The above program makes the following:



End of Program

The conversational screen for end of program appears below.



These codes are used if the operator is programming the Centurion 6 in the text mode or MDI mode. They are also generated from conversational programs. It should be noted that most programmers, particularly new programmers, use the conversational programming mode. If you are planning to use text mode of programming, pay close attention to this section for it explains these codes. If you are planning to use conversational mode of programming, you can skim this section and concentrate on the conversational section.

The preparatory function code is a two-digit number preceded by the letter G. Preparatory functions are used to determine the program operating mode and are divided into two types: one-shot and modal. **One-shot** G codes are only in effect during execution of the block in which they are present. **Modal** G codes establish operating modes, which remain in effect until replaced by another mode in the same category.

The following table lists G codes accepted by the Centurion 6 control system. Each code will have a detailed explanation later in the manual.

G Codes

		Active On Power-up	Modal	One Shot
00	Rapid Positioning		X	
01	Linear interpolation	X	X	
02	Circular/helical interpolation CW		X	
03	Circular/helical interpolation CCW		X	
04	Dwell			X
09	Exact stop			X
10	Set data on		X	
11	Set data off	X	X	
12	Clear floating zero			X
17	XY plane	X	X	
18	XZ or ZX plane		X	
19	YZ plane		X	
*20	Inch input	X	X	
*21	Metric input		X	
*22	Safe zone check on		X	
*23	Safe zone check off	X	X	
24	Circular pocket clear			X

* G20, G21, G22 and G23 are selectable by parameters on power up.

		Active On Power-up	Modal	One Shot
25	Circular finish inside			X
26	Circular finish outside			X
28-30	Reference point return			X
31	Z to clearance			X
32	Z to tool change			X
33	Facing cycle			X
34	Rectangular pocket clear			X
35	Rectangular finish inside			X
36	Rectangular finish outside			X
39	Threading mill cycle			X
40	Cutter compensation cancel	X	X	
41	Cutter compensation left		X	
42	Cutter compensation right		X	
43	H offset added		X	
44	H offset subtracted		X	
45	Auto cutter compensation left		X	
46	Auto cutter compensation right		X	
47	Auto cutter compensation off			
49	Cancel H offset	X	X	
50	Scaling cancel	X	X	
51	Scaling set		X	
52	Local coordinate system set		X	
53	Machine coordinate system			X
54	Work coordinate 1 system (G540...G549)	X	X	
55-59	Work coordinate 2-6 system (G5#0...G5#9)		X	
60	Single direction positioning			X
61	Exact stop mode		X	
63	Tapping mode		X	
64	Cutting	X	X	
65	Non-movement / Program Call			X
68	Set rotation		X	
69	Cancel rotation	X	X	
70	Cancel mirror	X	X	
71	Set mirror		X	

		Active On Power-up	Modal	One Shot
72	Bolt hole routine		X	
73	Woodpecker		X	
74	Left hand tapping		X	
75	Counter bore		X	
76	Fine bore		X	
77	Custom drill cycle		X	
78	Manual bore		X	
79	Custom drill cycle		X	
80	Cancel canned cycle	X	X	
81	Drill		X	
82	Drill/dwell		X	
83	Peck/drill		X	
84	Right-hand tapping		X	
85	Bore		X	
86	Fast bore		X	
87	Back bore		X	
88	Hard tap		X	
89	Bore/dwell		X	
90	Absolute dimension	X	X	
91	Incremental dimension		X	
92	Work coordinate chg. (Set fl. zero)		X	
93	Inverse time feed mode		X	
94	Feed per minute	X	X	
95	Feed per revolution		X	
98	Canned cycle initial level return		X	
99	Canned cycle R point level return		X	
271	Pocket Clear			X
666	Polygon Circle			X
990	Store parameters			X
991	Read parameters			X
995	Read byte parameters			X
996	Set byte parameters			X
997	Force error			
998	Beep			X

Note: Unrecognized G codes will cause an error 549 to occur.

Interpolation functions

There are four modes of interpolation:

G0 Rapid linear
G1 Feed linear
G2 Clockwise arcs
G3 Counterclockwise arcs

Positioning (G00) rapid traverse (modal)

Example: G0 X3 Y2

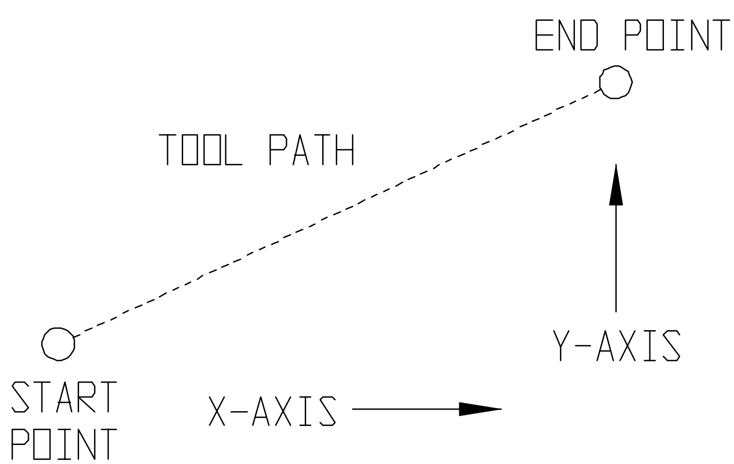
G00 specifies positioning in rapid traverse mode. There is no need to program rapid traverse rates because the rates are preset by parameters. Rapid traverse rates can be overridden by the feedrate override switch on the machine operator's panel.

G00 moves the tool at a rapid traverse rate to a position in the work coordinate system for both incremental and absolute commands.

Format: G00 ~-;

where ~- is: a combination of optional axis address (of X, Y, Z, A, B, C) as X-Y-Z-A-...

The programmed feed remains in the feedrate register and can be activated by canceling the G00 command with a G01 command. The motions of all axes in G00 mode will be interpolated with all axes reaching the end point simultaneously.



Note: The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder. Accordingly, the rapid traverse rate cannot be specified in the address F. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position. In-position *means that the axis position is within a specified range. (This range is determined by the machine tool builder.)*

G00 mode automatically accelerates and decelerates in a linear fashion allowing the controlled axis to start and stop smoothly. The rate of accel/decel can be changed by the machine tool builder.

Positioning (G01) Feed travels (modal)

Example: G1 X5 Y2 (The control defaults to G1 mode at the start of each program.)

This command actuates the linear interpolation mode. The feedrate is set to a cutting feed by the F address and is modal. An example follows.

(G91) G01 X20 Y10 F20

The feedrate specified by the F address is the vector rate along the path; it is not the rate of each axis.

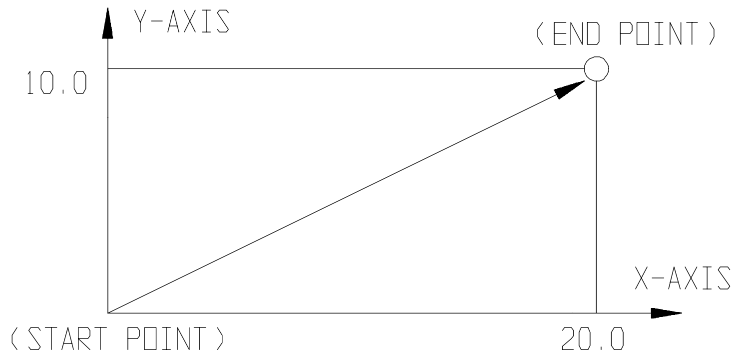
The feedrate of linear interpolation (G01) and circular interpolation (G02, G03) are commanded with numbers after the F address (FXXX.X).

The F address can appear anywhere in a block and specifies the rate of motion in inches or millimeters per minute.

Tangential Feedrate Control:

The cutting feed is controlled so that speed along the tangential path is always the commanded feedrate.

Feedrate Override:

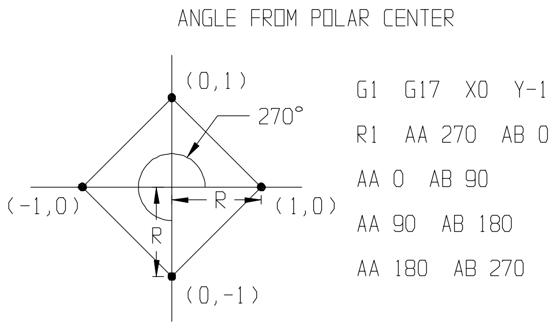


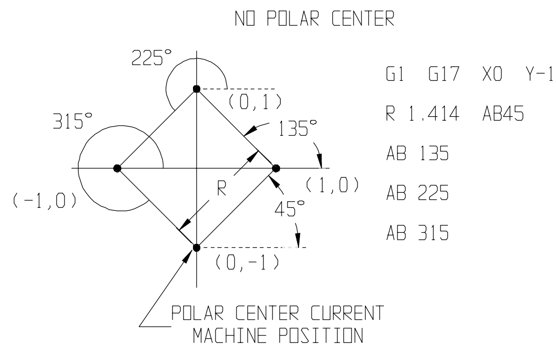
The per minute feed can be overridden using the feedrate override button on the machine operator's panel by 0 to 140% (per every 10%). Feedrate override cannot be applied to functions in which override is inhibited (e.g. tapping cycle).

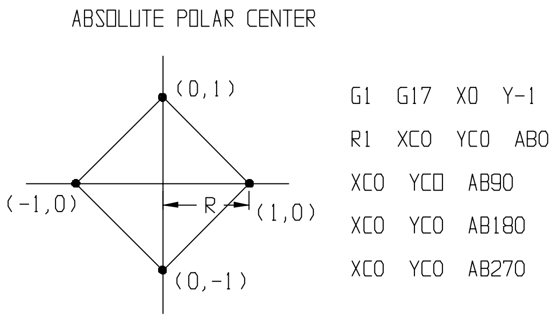
Polar definition of a line

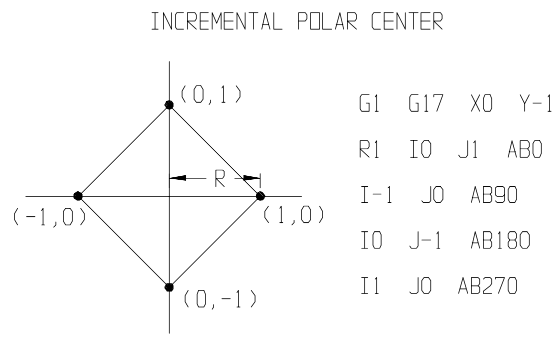
A polar line is specified by a polar radius/length (R), an angle (AB), and a polar center (AA or I, J, K, or XC, YC, ZC).

Polar definitions are valid in any plane. The 3 o'clock position is always 0°. Positive angles result in CCW rotation (from 0°) of the polar radius, while negative angles result in CW rotation (from 0°) of the polar radius. Polar lines can be used when estimating lengths during trig help. If the polar radius/length (R) or angle (AB) is not specified, the previous values will be used. If the polar center is not specified, it is taken to be the current machine position. Some examples follow.









Circular interpolation (G02, G03)

The general command format to move along a circular arc is as follows.

G17	G02	X Y I J or	XC YC R	or	R	or	AA R F		
G18	or	X Z I K	or	XC ZC R	or	R	or	AA R F	
G19	G03	Y Z J K	or	YC ZC R	or	R	or	AA R F	
	or								
	AB R								

*(1) *(2) *(3 or 6) *(4 or 5) *(7)

*These numbers are referenced in the chart that follows.

Plane Select	Direction	Radius and/or Center	End point	Feedrate
G17	G2	R XC YC (or ZC)	XY (or Z)	F
G18	G3	R AA	AB	
G19		I J (or K)	XY (or Z) WC	
		R or -R		

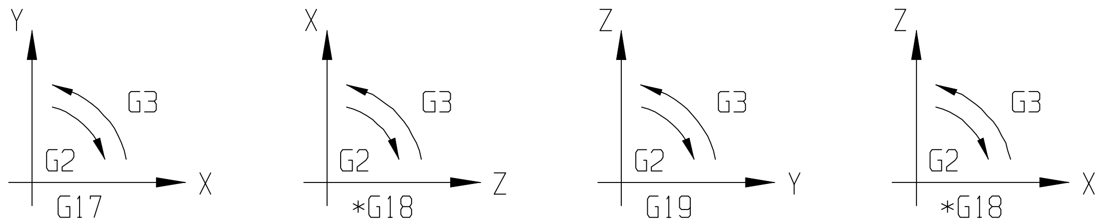
Circular Interpolation

<table>
<tr><th></th><th colspan="2">DATA TO BE GIVEN</th><th>COMMAND</th><th>MEANING</th></tr>
<tr><td rowspan="3">1</td><td colspan="2" rowspan="3">Plane selection</td><td>G17</td><td>Specify arc on XY plane</td></tr>
<tr><td>G18</td><td>Specify arc on ZX plane</td></tr>
<tr><td>G19</td><td>Specify arc on YZ plane</td></tr>
<tr><td rowspan="2">2</td><td colspan="2" rowspan="2">Direction of rotation</td><td>G02</td><td>Clockwise (CW)</td></tr>
<tr><td>G03</td><td>Counterclockwise (CCW)</td></tr>
<tr><td rowspan="2">3</td><td rowspan="2">End point position</td><td>G90 mode</td><td>One, Two, or Three of X, Y, and Z</td><td>End point position in work coordinate system</td></tr>
<tr><td>G91 mode</td><td>One, Two, or Three of X, Y, and Z</td><td>Distance from start point to end point</td></tr>
<tr><td rowspan="3">4</td><td colspan="2">Distance from start point to center</td><td>Two of I, J, and K</td><td>The signed distance from start point to center $R=\sqrt{(I^2+J^2)}$</td></tr>
<tr><td colspan="2" rowspan="2">Arc radius</td><td>R</td><td>Arc radius (if no center is specified a center is calculated from the start point and end point)
(-R is the longer arc)</td></tr>
<tr><td>Two of XC, YC, ZC</td><td>The signed absolute coordinates of the arc center</td></tr>
</table>

Polar Definition

<table>
<tr><th></th><th>DATA TO BE GIVEN</th><th>COMMAND</th><th>MEANING</th></tr>
<tr><td rowspan="2">5</td><td>Arc radius</td><td>R</td><td>Arc radius</td></tr>
<tr><td>Start angle</td><td>AA</td><td>Angle from center to start</td></tr>
<tr><td>6</td><td>End angle</td><td>AB</td><td>Angle from center to end point CW direction</td></tr>
<tr><td>7</td><td>Feedrate</td><td>F</td><td>Velocity along arc</td></tr>
<tr><td>8</td><td>To line</td><td>W, C</td><td>Angle</td></tr>
</table>

Clockwise and Counterclockwise Directions



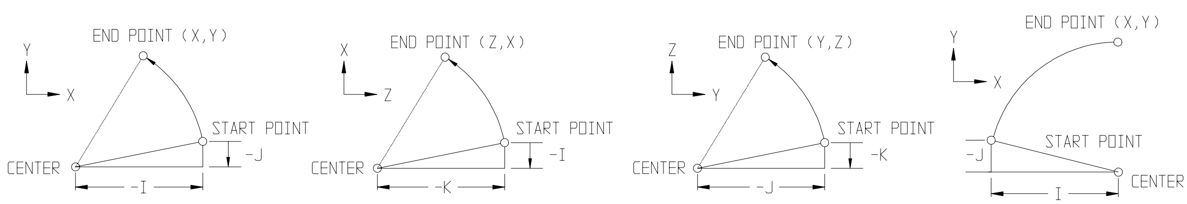
The view above is from the positive direction of the Z, Y, or X axis to the negative direction on XY, XZ, YZ, or ZX plane in a right-hand Cartesian coordinate system.

Method I

Describing an Arc Using Incremental Center

The end point of an arc is specified by address X, Y, or Z and is expressed as an absolute or incremental value depending on G90 or G91. In incremental, the coordinate of the end point is related to the start point of the arc. The arc center is defined by I, J, and K for the X, Y, and Z axes. The numerical value foll by the tool
owing I, J, or K is the distance from the start point to the arc center in X, Y, or Z axes. I, J, and K are always incremental values independent of G90 and G91.

The sign of I, J, and K depends on the relationship of the center to the start point as shown below.



Method II

Describing an Arc Using a Radius

When describing an arc using a radius value, there are a number of valid formats. The various command formats are as follows.

G17	G02	X____	Y____	R____
G18	or	X____	Z____	R____
G19	G03	Y____	Z____	R____

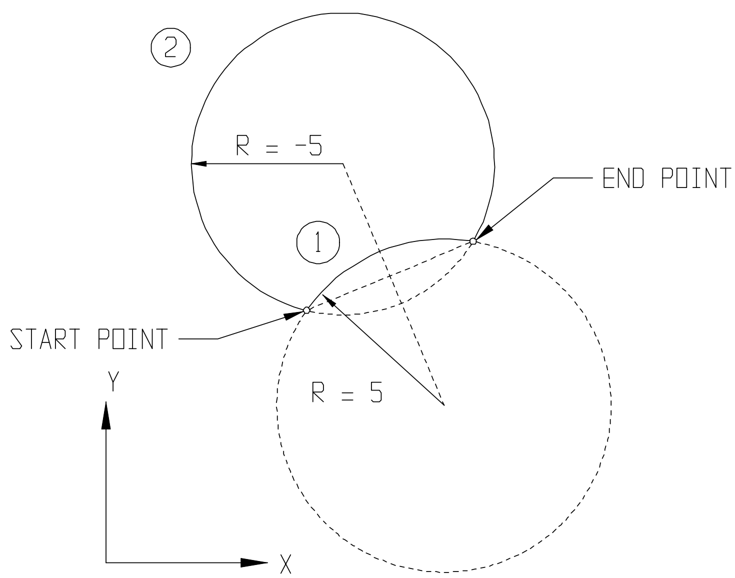
Arc End Points

The radius is always specified as its true value. The end points are incremental or absolute depending on G90 and G91. If a radius is used without a center point, there are two types of arcs that can be generated. One is less than 180°, and the other is greater than 180°, as shown in the figure that follows. When the arc exceeds 180° the radius must be specified as a negative value.

Examples:

For arc 1 (less than 180°)
G2 X6 Y2 R5 F30

For arc 2 (greater than 180°)
G2 X6 Y2 R-5 F30



Note: If the arc cannot span the start and end points, an error 505 "radius too small to span given points" *will occur.*

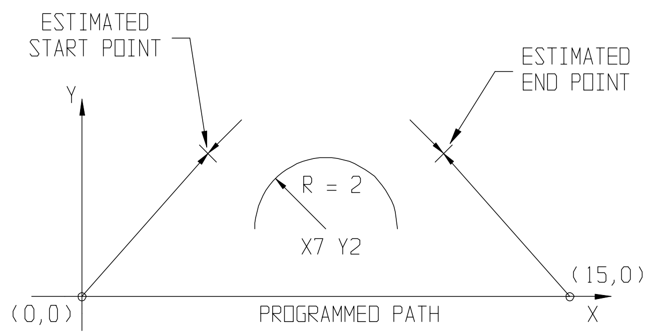
Method III
Describing an Arc Using Absolute Center and Trig Help

G17	G02	X___	Y___	XC___	YC___	R___
G18	or	X___	Z___	XC___	ZC___	R___
G19	G03	Y___	Z___	YC___	ZC___	R___
		End Point		Center Point		Radius

Trig Help will allow the programmer to estimate both the start and end points of any arc. The control will calculate the true start and end points based on the moves preceding and trailing the arc. Where there are two possible correct answers, the control will choose the point closest to the estimated point. If the slope of the line entering or leaving the arc is such that no intersection occurs, the line will be made tangent to the arc.

Examples of Trig Help follow.

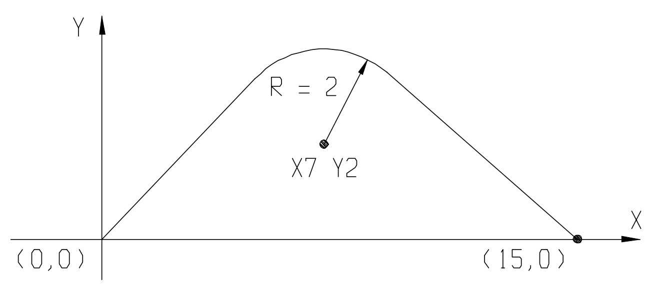
Program 1



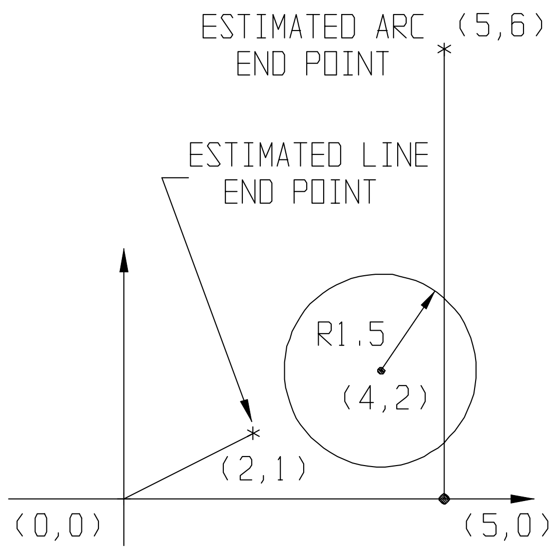
Programmed path

G1 X0 Y0
X4 Y4 (estimated start point)
G2 R2 XC7 YC2 X10 Y4 (estimated end point)
G1 X15 Y0

Path generated by Program 1



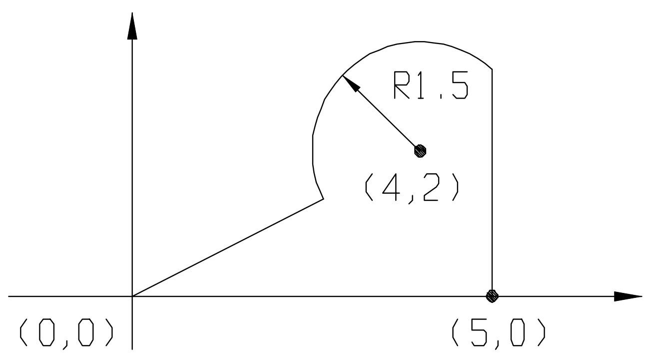
Program 2



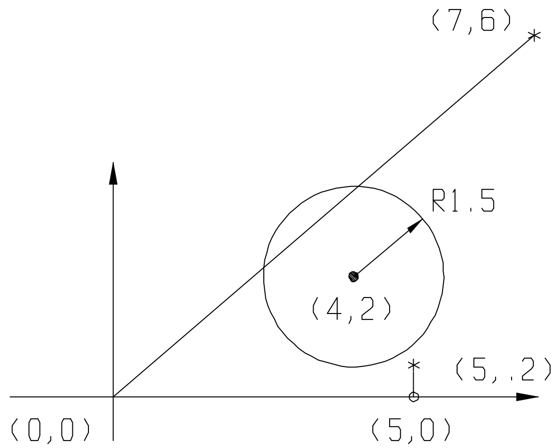
Programmed path

G1 X0 Y0
X2 Y1 (estimated start point)
G2 R1.5 XC2 X5 Y6 (estimated end point)
G1 X5 Y0

Path generated by Program 2



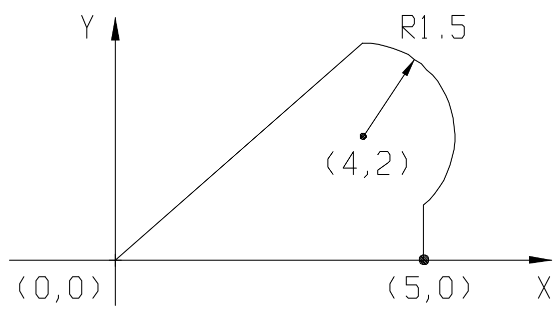
Program 3



Programmed path

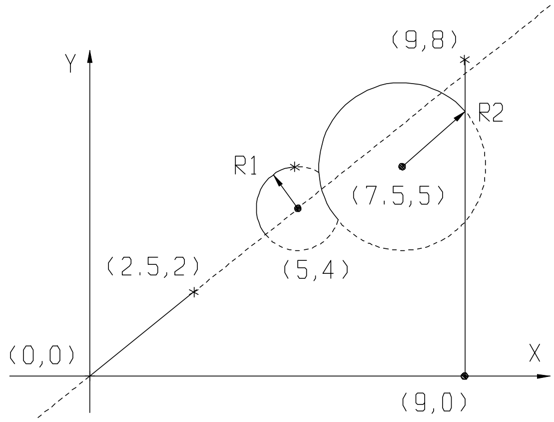
G1 X0 Y0
X7 Y6 (estimated start point)
G2 R1.5 XC4 YC2 X5 Y.2 (estimated end point)
G1 X5 Y0

Path generated by Program 3



In general, when dealing with lines and arcs, if the line is programmed short of the arc it will be extended to the arc. If the line is programmed past the arc it will be shortened to the arc, and if the line does not intersect the arc it will be made tangent.

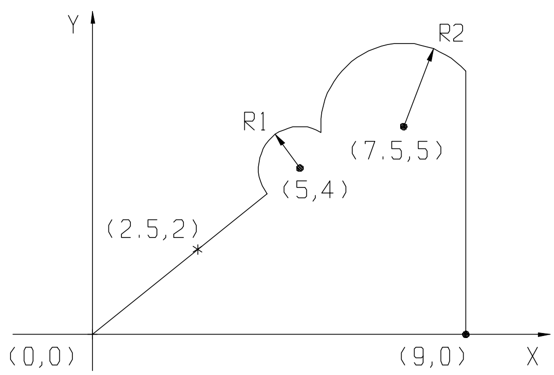
Program 4



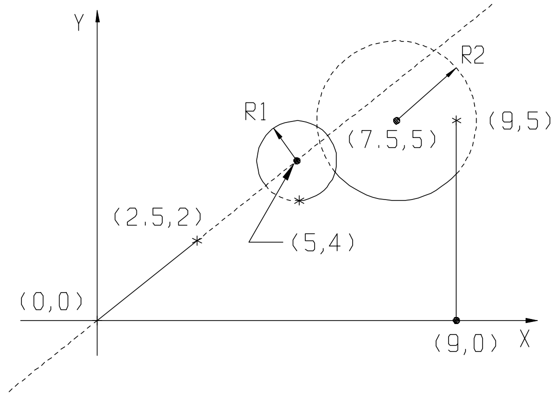
Programmed path

```
G1 X0  Y0
X2.5 Y2 (estimated start point)
G2 R1 XC5 YC4 X5 Y5 (estimated end point)
R2 XC7.5 YC5 X9 Y8 (estimated end point)
G1 X9 Y0
```

Path generated by Program 4



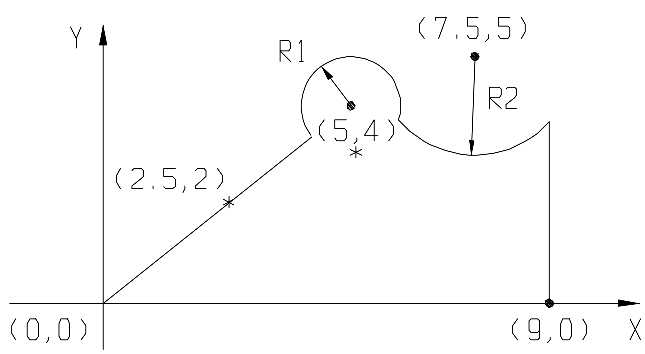
Program 5



Programmed path

G1 X0 Y0
X2.5 Y2 (estimated point)
G2 R1 XC5 YC4 X5 Y3 (estimated end point)
G3 R2 XC7.5 YC5 X9 Y5 (estimated end point)
G1 Y0

Path generated by Program 5



In general, when estimating arc-to-arc intersections, the easiest end points to pick are one of the quadrant points (0°, 90°, 180°, or 270°).

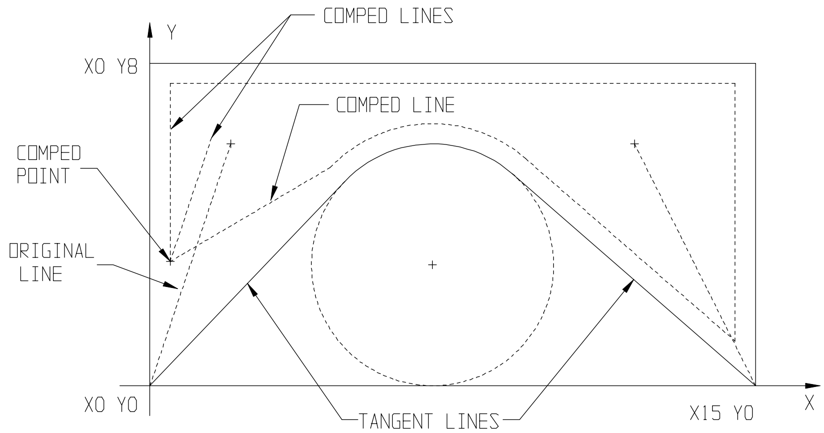
Things To Remember When Estimating Points

- Estimating can be used with line to circle, circle to circle, and circle to line paths.
- The center and radius of arcs cannot be estimated.
- For line-to-circle and circle-to-line, the start and end point estimates **should** lie on the line; i.e. the **slopes** of the lines entering or leaving the arc **must** be correct.
- If a line intersects an arc at two points, the estimated point should be **closer** to the desired point of intersection.
- If the above conditions are met, there is no limit on how far the estimated point is away from the correct point.
- When estimating the intersection of one arc to another arc, the easiest point to pick on an arc is at one of the quadrant points (0°, 90°, 180°, or 270°).

The following is a brief discussion on cutter compensation and Trig Help when Trig Help is used to bring a line tangent to an arc.

Sample Program:

```
G41 D1
G65 X1 Y8
X0
Z-1
X0 Y0
X2 Y6 (This line is pulled in tangent.)
G2 R3 XC7 YC3 X12 Y6
G1 X15 Y0
Y8
X0
G65 X0 Y0
Z0
```



The block X2 Y6 is pulled in tangent to the arc. The cutter compensation has already taken into consideration the previous two lines, and it has calculated the compensated point based on the original line rather than the tangent line. The compensated path for this program will not cut the correct part. **To avoid this problem, you must verify the program with cutter compensation off (or 0 tool radius).** Note the actual tangent point (X4.8276, Y5.0690). Substitute this point for the estimated point (X2 Y6).

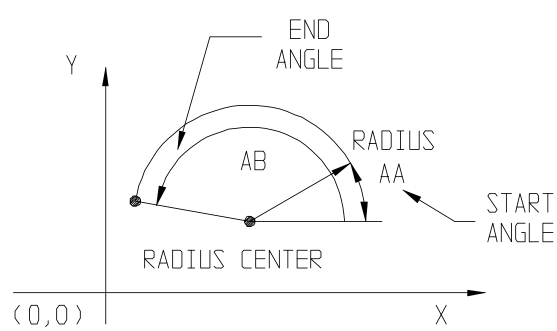
This is not a problem for the other tangent line in the program.

Method IV

Describing an Arc Using Polar Definitions

The polar definitions do not change from absolute to incremental. The center of the arc is always considered the pole and all angles are related to it. The basic polar definition is as follows.

G17	G2			
G18	or	AA_____	AB_____	R_____
G19	G3	start angle	end angle	radius



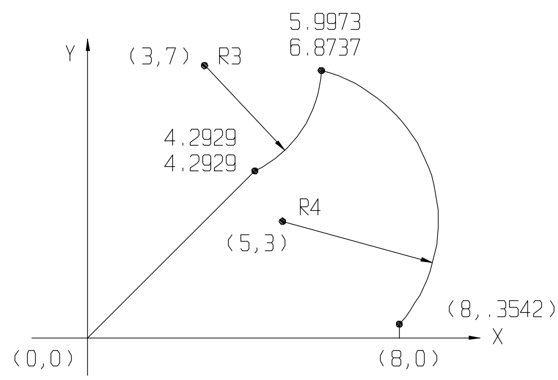
The polar format for arcs can be mixed with the Cartesian formats. The following are legal formats.

G17 G2 X_____ Y_____ AA_____ R_____
end point start angle
G17 G2 AB_____ XC_____ YC_____ R_____
end angle center point
G17 G2 I_____ J_____ AB_____
center point

The above formats are written for the XY plane but are valid in any plane or direction. Trig Help is only valid in polar when using an arc with valid center point and radius.

Program 1

The following programs will all produce the same part. The programming method used is totally optional.



1. Absolute coordinates (Polar No Trig Help)

```
G90
G1 X0 Y0
X4.2929 Y4.2929
G3 R3 AA295.53 AB357.59
G2 R4 AA77.56 AB318.59
G1 Y0
X0
```

2. Absolute coordinates (Polar Trig Help)

```
G90
G1 X0  Y0
R1 AB45
G3 R3 XC3 YC7 AB0
G2 R4 XC5 YC3 X8 Y.5
G1 Y0
X0
```

Note: When using Trig Help, you must have a valid arc center and radius. That is why the G2 and G3 lines have a fixed format.

3. Absolute coordinates (Cartesian No Trig Help)

```
G90
G1 X0 Y0
X4.2929 Y4.2929
G3 R3 XC3 YC7 X5.9973 Y6.8737
                or
G3 I-1.2929 J2.7071 X5.9973 Y6.8737
                or
G3 R3 X5.9973 Y6.8737
G2 R4 XC5 YC3 X8 Y.3542
                or
G2 R4 X8 Y.3542
G1 Y0
X0
```

4. Absolute coordinates (Cartesian Trig Help)

```
G1 X0 Y0
X1 Y1
G3 R3 XC3 YC7 X6 Y7
G2 R4 XC5 YC3 X8 Y.5
G1 X8 Y0
X0
```

Note: Most of the dimensions are approximations; the control calculates the exact dimensions.

5. Incremental coordinates

```
G91
G1 X0 Y0
X4.2929 Y4.2929
G3 I-1.2929 J2.7071 X-1.7044 Y2.5058
          or
G3 R3 XC3 Yc7 X-1.7044 Y2.5808
          or
G2 I-.9973 J-3.8737 X2.0027 Y-6.5195
G2 X2.0027 Y-6.5195 I-.9973 J-3.8737
          or
G2 R4 X2.0021 Y-6.5195
          or
G2 R4 XC5 YC3 X2.0027 Y-6.5195
G1 Y-.3542
X-8
```

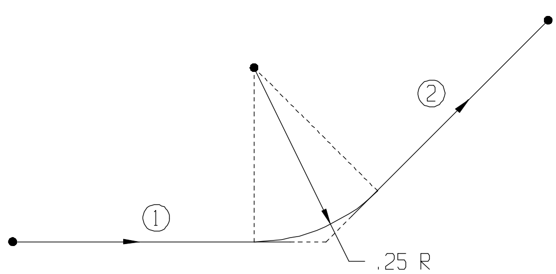
Note: In Incremental, Trig Help cannot be used as each point is related to the current position.

Trig Help can be shut off by setting bit 2 of the miscellaneous special flags parameter. This may be desirable for programs generated from some Cad/Cam systems.

Corner rounding (,R)

By adding ,R___ to the end of blocks commanding linear or circular interpolation, corner rounding can be automatically inserted.

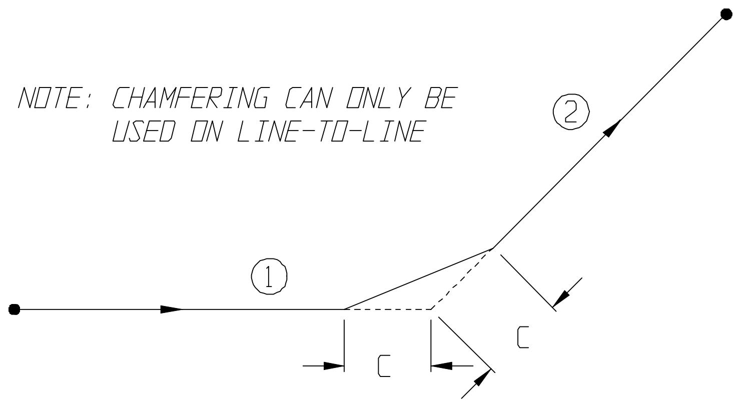
```
G91 G01 X0 Y0
X1,R.25
X1 Y1
```



Angle chamfering (,C)

By adding ,C___ to the end of blocks commanding linear interpolation, angle chamfering is automatically inserted.

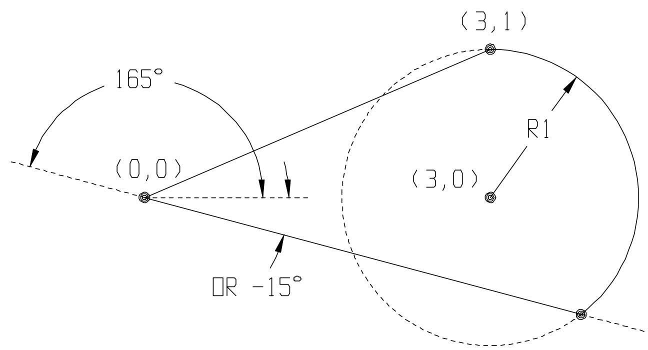
G91 G01 X0 Y0
X1,C.25
X1 Y1



Back line

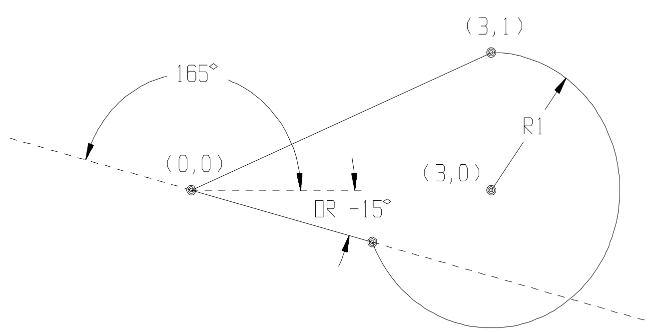
The back line function can be used on any line command. This function reverses the direction of a programmed line. It would normally be used when you know the end point of the line and not its start point. The end point would be programmed and the line would be extended backwards to the start point. All Trig Help functions are still valid when using this function.

```
X0 Y0
X3 Y1
G2 R1 XC3 YC0 AB270
G01 X0 Y0 BACK C2 W165
```



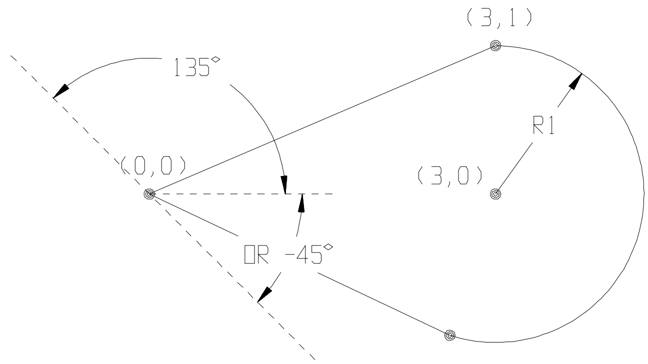
Back	Extends line backwards from (0,0)
C2	Uses the arc intersection farthest from (0,0)
W165	Extends the line from (0,0) at an angle of 165°

```
X0 Y0
X3 Y1
G17 G2 R1 XC3 YC0 AB270
G01 X0 Y0 BACK C0 W165
```

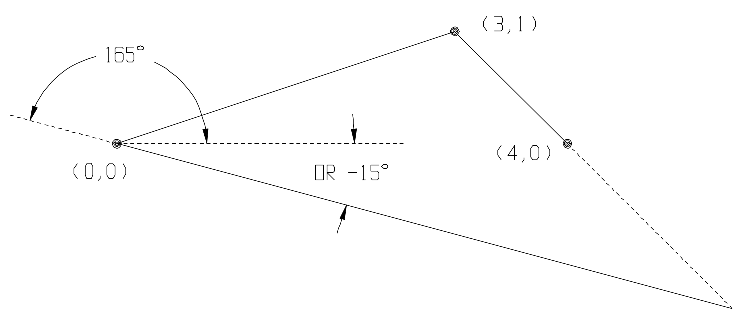


C0	Uses closest intersection

```
X0 Y0
X3 Y1
G2 R1 XC3 YC0 X3 Y-1
G01 X0 Y0 BACK C0 W135
```



W135 This line does not intersect with the arc; therefore, the line will be rotated until it is tangent.
X0 Y0
X3 Y1
X4 Y0
X0 Y0 BACK C0 or C2 W165



This example used a back line between two lines to program an unknown point.

Notes on Circular and Linear Milling

The feedrate in circular and linear is equal to the feedrate specified by the F address. This feedrate is the tangential feedrate along the arc and the vector feed on the linear moves.

Note 1: I0, J0, and K0 can be omitted.

Note 2: If X, Y, and Z are all omitted – or if the end point is located at the same position as the start point and the center is commanded by I, J and K – an arc of 360° (a complete circle) is assumed.

G02I _____; (a complete circle)

When only R is used, an arc of 0° is programmed.

G02R _____; (The cutter does not move.)

Note 3: The feedrate is measured along the arc after the cutter compensation is applied.

Note 4: If I, J, K, and R addresses are specified simultaneously, the arc radius specified by address I, J, K takes precedence and the R is ignored.

Note 5: The X Y Z I J K R AA AB commands are retained by the control. If an interpolation block is left incomplete, the missing axis information will be defaulted to the value previously entered.

Helical cutting (G02, G03)

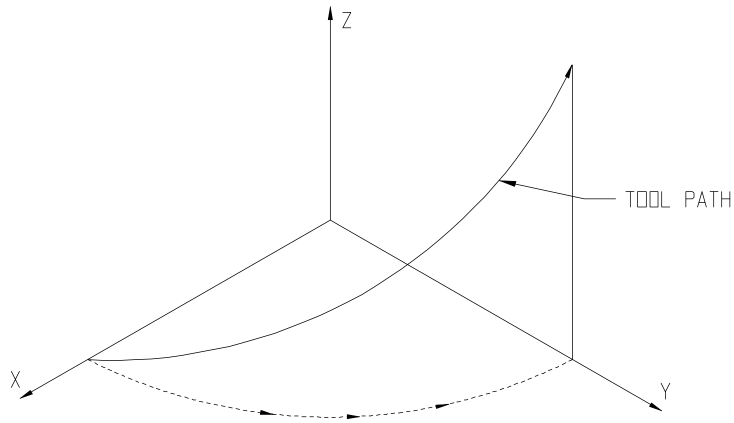
Helical interpolation is enabled by specifying another axis which moves synchronously with the circular interpolation. That is, the tool can be moved helically.

```
      G02                  R____
G17   or    X____ Y____    or   Z____ F____;
      G03                 I____ J____
```

```
      G02                  R____
G18   or    X____ Z____    or   Y____ F____;
      G03                 I____ K____
```

```
      G02                  R____
G19   or    Y____ Z____  or   X____ F____;
      G03                 J____ K____
```

The above formats for helical milling illustrate the general concept. Any of the previous arc formats can be used to do helical cutting by simply adding the third axis end point to the arc command.



An F address specifies a feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows.

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feedrate so the linear axis feedrate does not exceed any of the various limit values.

Dwell command (G04)

The G4 code must be immediately followed by an FXXX.X instruction. This instruction will then cause the program to stop or dwell for XXX.X seconds.

General Format:

G4F2.5 will cause the program to dwell for 2.5 seconds.

G4F25 will cause the program to dwell for 25 seconds.

Note: A, P, or X can be used in place of F following the G4 command.

Exact stop (G09)

Moves commanded in blocks with G09 decelerate at the end point, and in-position check is performed. This function is used when sharp edges are required for workpiece corners in cutting feed.

Set data on/off (G10, G11)

This function allows all the CNC's configuration, setup, axis, and offset table parameters to be loaded via a program rather than through the front panel. (This function is the only way to change parameters 700 and higher from a program.) The format for loading the parameters is as follows.

G10	Set data On
P*** = value	
P*** = value	P*** = parameter number
P*** = value	to be loaded
P*** = value	
G11	Set data Off

Commands other than P***= value, while in G10 mode, will give an error 543 *"Illegal G10 statement."*

When a G11 is performed, the CNC will start using the new parameter settings. Refer to Appendix A on parameter assignments.

Clear floating zero (G12)

This function will clear the floating zeros. (G12 in a program will cancel the floating zeros, but they are restored at the end of a program. G12 in MDI will clear the floating zeros.)

XY plane (modal) (G17)

Selects the XY plane for all polar and arc moves. This command remains in effect until switched by another plane command. The G17 command can appear anywhere on the line. The control always powers up in G17 Mode. The control defaults to G17 Mode at the start of each program.

XZ plane (modal) (G18)

Selects the XZ or ZX plane for all polar and arc moves. This command remains in effect until switched by another plane command. The G18 command can appear anywhere on the line.

YZ plane (modal) (G19)

Selects the YZ plane for all polar and arc moves. This command remains in effect until switched by another plane command. The G19 command can appear anywhere on the line.

Inch dimensioning mode (modal) (G20)

This function will cause the system to go into the inch mode. In this mode the system will accept dimensions in inches.

Active power-up is selectable.

G20 cancels G21.

Metric dimensioning mode (modal) (G21)

This function will cause the system to go into the metric mode. In this mode the system will accept dimensions in millimeters (mm). In metric the actual machine position may not exactly agree with the program position because of the conversion. Feedrate in the metric mode is in millimeters per minute (mmpm).

Note: The CNC does a conversion - -from metric to inch and inch to metric - on all tool offsets. This means that a 1.0 inch offset entered in the inch mode will change to 25.4 mm when the system is switched to metric. The reverse happens for metric entries.

Safe zone on/off (G22, G23)

This control is equipped with a programmable safe zone. Any area of the machine's travels can be designated as a safe zone. This is an area the machine cannot enter. If the machine is programmed into this area when the safe zone check is enabled, an *"attempted to move into safe zone"* error will be displayed. The safe zone is defined in the coordinate parameters. The check is turned on with a G22 and off with a G23. Active power-up is selectable.

Autoroutines

These G codes select basic patterns which are used repeatedly in most milling applications. These patterns are circular and rectangular finish cuts, circular and rectangular pocket clearing and polygons. These routines use the CNC's parametrics to input the various cutting and size differences encountered in different applications. The following will explain each G code and give an example of its use in a program.

All inside autoroutines must be activated with the tool at the center of the routine. Outside autoroutines can use a G65 (no-move block) to define the center.

Circular pocket clear (G24)

The G24 autoroutine is used to clear a circular pocket by starting in the center and spiraling out to the programmed diameter.

Circular Pocket Clear Program

```
N1    G20 G90 (Inch/Absolute)
N2    G00 X0 Y0 (rapids to center of pocket)
N3    S1000 M3 D1 G43 H1 (spindle CW-1000 RPM, calls tool #1's offsets)
N4    F25 (X-Y feedrate)
N5    P150=1 (pocket radius)
N6    P153=.015 (X-Y finish stock)
N7    P154=.005 (Z finish stock)
N8    P155=.25 (cut width)
N9    G24 G99 G42 G2 R.1 P199=0 Z-.5 V-.3 Q.2 F10
      *1  *2  *3  *3  *4  *5     *6  *7  *8  *9
```

*1 Executes circle pocket clear autoroutine
*2 Returns Z to clearance plane
*3 CW right or "G3 G41" CCW left **
*4 .1 clearance plane
*5 P199=0 plunge for all autoroutines except facing cycle
*6 -.5 final Z depth
*7 -.3 first Z depth ***
*8 .2 Z increment, signed Z increment will use the absolute valve
*9 10 ipm Z feedrate

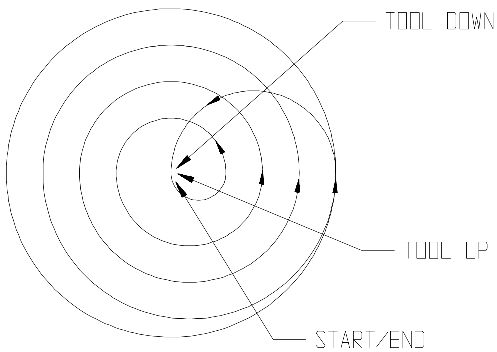
** For all autoroutines, G41 and G42 do the same thing: they turn cutter compensation on. The examples shown are the correct cutter compensation direction. The control will compensate correctly on all autoroutines if the cutter compensation is on (G41 or G42).The climb/conventional cut is specified by G2 or G3.

*** If the first Z depth is less than the final Z depth, the first Z depth is set to the final Z depth.

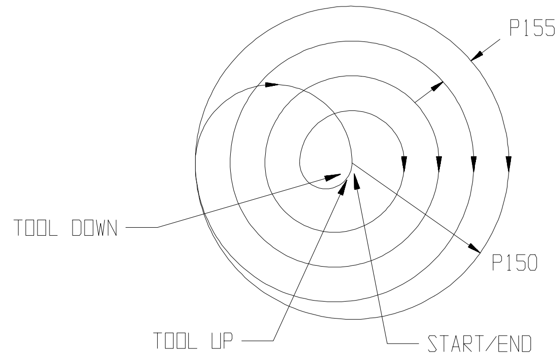
Note: It is also possible to use the following.

P140=for Clearance plane
P141=for Final Z depth
P142=for Z initial level
P143=for Z increment
P144=for 1st Z depth
P145=for Z feedrate

CW Circular Pocket Clearing



CCW Circular Pocket Clearing



Block #	Block Entry Info
N9	G2 G42
Block 9	Selects CW circle, and turns ON right cutter compensation

Block #	Block Entry Info
N9	G3 G41
Block 9	Selects CCW circle and turns ON left cutter compensation

Circular finish inside (G25)

If a tool radius is specified, cutter compensation can be used in all autoroutines. The control will automatically decrease or increase the tool path by the radius of the tool.

The figure below “Inside CW Finish Circle” shows the tool path of the following program. The figure below “Inside CCW Finish Circle” shows the same program with the change indicated in line N9.

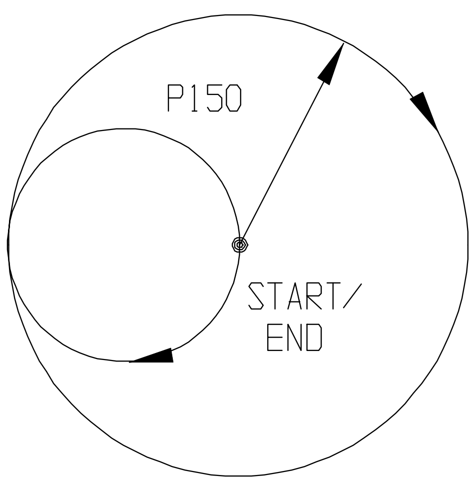
Circular Finish Inside Program

N1	G20 G90 (Inch/Absolute)
N2	G00 X0 Y0 (rapid to center of pocket)
N3	S1000 M3 D1 G43 H1 (spindle CW-1000 RPM; calls tool #1's offsets)
N4	G99 (return Z to clearance plane)
N5	F20 (X-Y feedrate)
N6	P150=1 (pocket radius)
N7	P153=0 (X-Y finish stock) **
N8	P154=0 (Z finish stock)
N9	G25 G42 G2 V-.3 R.1 P199=1 F5 Z-.5 Q.2
	*1 *2 *2 *3 *4 *5 *6 *7 *8

*1	Executes circle finish inside
*2	CW right or "G41 G3" CCW left
*3	First Z depth
*4	Clearance plane
*5	Ramp Z down
*6	Z feedrate
*7	Final Z depth
*8	Z increment

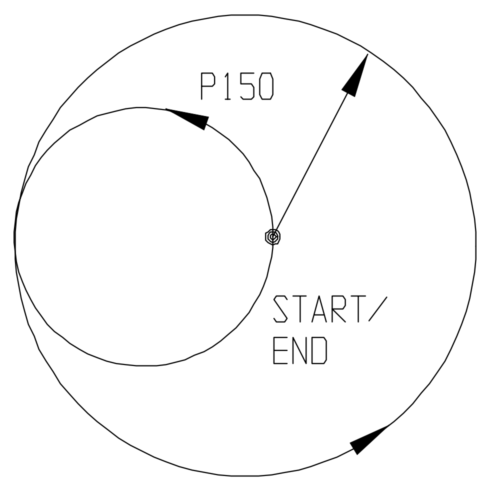
** XY finish stock and Z finish stock are zeroed at the beginning of each program.

Inside CW Finish Circle



N9 G42 G2 selects CW circle and **right** cutter compensation

Inside CCW Finish Circle



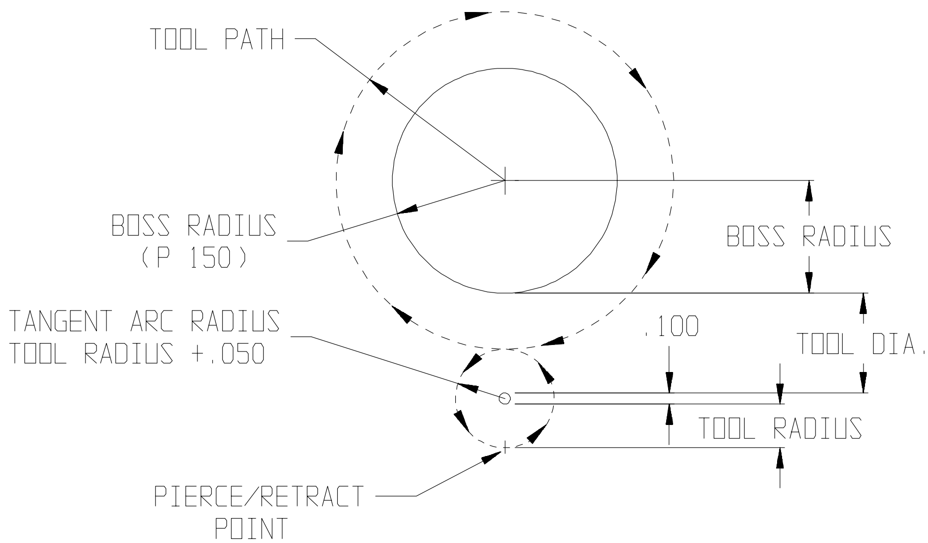
N9 G41 G3 selects CCW direction and **left** cutter compensation

Note: Parameter P150 is the pocket radius. If no finish stock is desired, parameters P153 and P154 should be set to zero. The F20 programmed in N5 is the XY feedrate and the F5 in N9 affects only the Z axis feed. Once parameters are set to a value they do not change and can be utilized further in the program. When an autoroutine is called, any parameters that are not re-initialized will default to the previous value of the parameter.

Circular finish outside (G26)

The G26 autoroutine is identical in operation to the G25 autoroutine except it cuts the outside of a circular boss rather than the inside. Because the G26 needs to position to the outside of the boss, it will use the following formula to calculate the distance from the center of the boss to the pierce/retract point.

Distance = Circle + .1 + [3 X Tool]
radius radius



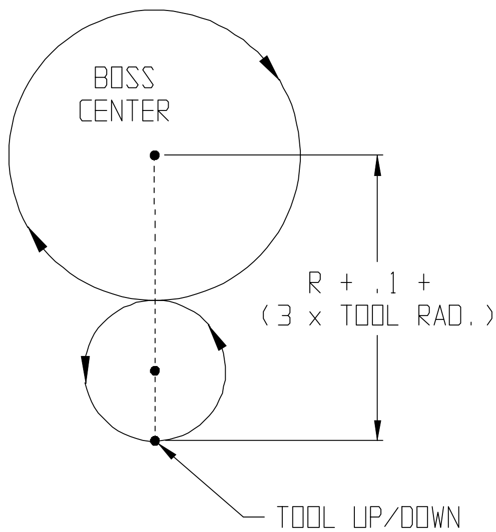
Circular Finish Outside Program

N1 G20 G90 (Inch/Absolute)
N2 S1000 M3 D1 G43 H1 (spindle CW-1000 RPM, calls tool #1's offsets)
N3 F20 (X-Y feedrate)
N4 P150=1 (Boss radius)
N5 P153=0 (X-Y finish stock)
N6 P154=0 (Z finish stock)
N7 G26 G98 G41 G2 R.1 P199=0 Z-.5 V-.3 Q.2 F5
*1 *2 *3 *3 *4 *5 *6 *7 *8 *9

*1 Executes circle finish outside
*2 Return to initial point
*3 CW left or "G42 G3" CCW right
*4 Clearance plane

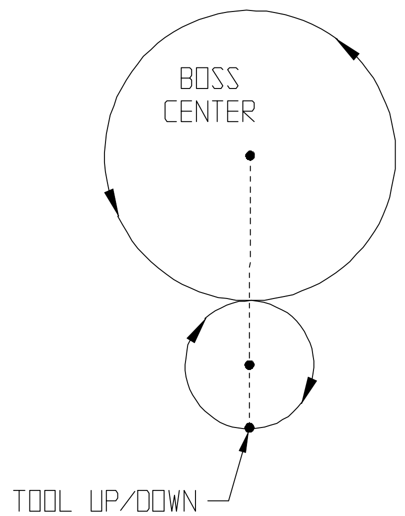
*5 Plunge
*6 Final Z depth
*7 First Z depth
*8 Z increment
*9 Z feedrate

Outside CW Finish Circle



N7 G2, G42 selects CW direction and **left** cutter compensation

Outside CCW Finish Circle



N7 G3, G42 selects CCW direction and **right** cutter compensation

Reference point return (G28, G29, G30)

These commands allow the machine to be commanded to a fixed point (reference point) by first passing through an intermediate point on the way to the reference point. First a fixed reference point in XYZ is entered via the front panel into the reference point parameters. The reference point is relative to machine zero. Once this point is established it will remain unchanged until replaced by another front panel command. Each time a G28 or G30 is commanded it will return the machine to the designated reference point. Positioning to the intermediate and reference points are done in rapid traverse. If a G28 or G30 is executed with no axis definitions it has no effect. If one or two axes are commanded as intermediate points, only the commanded axis will move to the intermediate point and reference point. Once an intermediate point is programmed, it will be remembered until the next G28 is executed (i.e. for use in a G29).

The command format is as follows.

Reference point set at	X-10
	Y 0
	Z-0.1

Example 1:	G28	(No axis movement)	
Example 2:	G91 G28	Z 0	(Z to -0.1 Relative to machine zero)
Example 3:	X1 Y0 Z-2		
	G28 X3	X3 then X-10	(Relative to machine zero.)
Example 4:	X-3 Y2 Z-8	Z	
	G28 Z-7	-7 then Z-0.1	(Relative to machine zero.)
	G29	Z to -7	

The G29 command is the converse of a G28. G29 will return the machine to the programmed point via the last intermediate point stored by a G28 command. The command format is as follows.

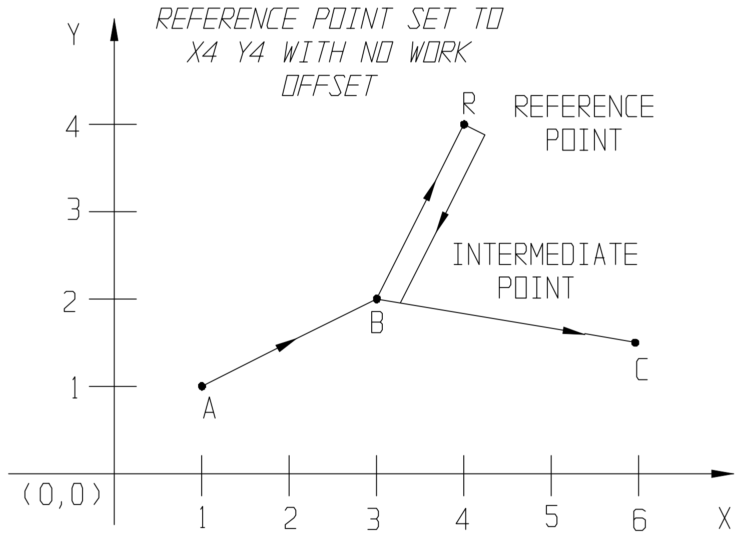
G29 X____ Y____ Z____

programmed point

In general, G28 is used for a tool change or part loading/unloading position. It is normally used to move to the reference point only. For example, G91, G28, Z0, Z moves to reference point.

When G29 is executed by itself, only the axis that was commanded with the previous G28/G30 goes to the intermediate point.

Example of G28 and G29:



X1 Y1	Point A
G28 X3 Y2	Point B then Point R
G29 X6 Y1.5	Point B then Point C

G30 2nd, 3rd, 4th Reference Point Return

This function works in an identical manner to the G28 reference point return except that a 2nd, 3rd, and 4th reference point can be called. The command format is as follows.

G30 P2	X____ Y____ Z____
G30 P3	X____ Y____ Z____
G30 P4	X____ Y____ Z____
reference point	intermediate point

If no P is specified, P2 is assumed. $P \leq 1$ and $P \geq 5$ are illegal and will result in an error 542 *"G30 illegal return to reference parameter on G30 block"*.

Z to clearance (G31)

The G31 function will retract Z to the clearance position. This position defaults to the last clearance position but may be changed by editing parameter 140 or set in canned cycles with the "R" parameter.

Z to tool change (G32)

The G32 function will retract Z to the tool change position. This position is set by the machine tool builder but may be changed by editing the tool change coordinate parameter.

Facing cycle (G33)

The G33 autoroutine is used to face a rectangular surface. It always starts in the upper right-hand corner of the rectangle and moves back and forth along the long side of the rectangle and steps over on the short side of the pocket. Successive depths will raise the tool up .1" above the current depth and rapid to the top right corner for the next depth.

Facing Cycle Program

```
N1    G20 G90 (inch/absolute)
N2    S1000 M3 D1 G43 H1 (spindle CW1000 RPM, calls tool #1's offsets)
N3    G0 X1 Y2 (rapid to the center of the pocket)
N4    F20 (X-Y feedrate)
N5    P151=1 (X dimension)
N6    P152=3 (Y dimension)
N7    P155=.1 (cut width)
N8    G33 G99 G41 R.1 Z-.5 V-.3 Q.1 FS
       *1  *2  *3  *4  *5  *6  *7  *8
```

*1 Execute facing cycle
*2 Returns to Z clearance
*3 Cutter compensation on (G41 or G42 will do the same thing)
*4 Clearance plane
*5 Final Z depth
*6 First Z depth (If no V is specified the 1st Z depth is the clearance plane minus the Z increment)
*7 Z increment
*8 Z feedrate

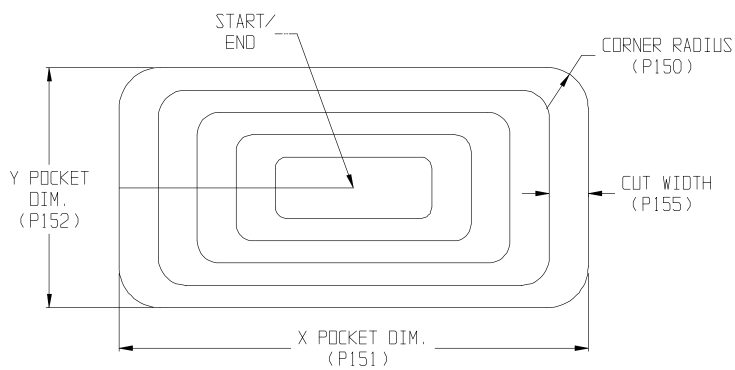
Rectangular pocket clear (G34)

The G34 autoroutine is used to clear a rectangular pocket by starting in the center and working its way out to the finish dimensions. The operation of the autoroutine is identical to the circular routines except that a rectangle with radiused corners is cut. The rectangular routines have the addition of parameters P151 (X pocket dimension) and P152 (Y pocket dimension). Parameters P150, P153, P154, and P155 retain the same meaning as in the circular routines. The X and Y dimensions are the overall pocket dimensions, and if the corner radius is set to 0 the corners will be cut at the tool radius.

Rectangular Pocket Clear Program

N1 G20 G90 (Inch/Absolute)
N2 S1000 M3 D1 G43 H1 (spindle CW-1000 RPM, calls tool #1's offsets)
N3 G00 X0 Y0 (rapids to pocket center)
N4 F20 (X-Y feedrate)
N5 P150=.75 (corner radius)
N6 P151=4 (X pocket dimension)
N7 P152=2 (Y pocket dimension)
N8 P153=.015 (X-Y finish stock)
N9 P154=.005 (Z finish stock)
N10 P155=.5 (cut width)
N11 G34 G99 G42 G2 R.1 P199=1 Z-.5 V-.3 Q.2 F5
*1 *2 *3 *3 *4 *5 *6 *7 *8 *9

*1 Executes rectangular pocket clear
*2 Returns Z to clearance plane
*3 CW right or "G41 G3" CCW left
*4 Clearance plane
*5 Ramp Z down
*6 Final Z depth
*7 First Z depth (If no V is specified the 1st Z depth is the clearance plane minus the Z increment)
*8 Z increment
*9 Z feedrate



If N11 is G42 G2, the cut direction is CW.
If N11 is G41 G3, the cut direction is CCW.

Rectangular finish inside (G35)

The G35 autoroutine is used to remove the finish stock left by the rectangular clear routine, or it is used to remove some amount of stock around the inside of a rectangle. The G35 autoroutine works in an identical manner to the G34 autoroutine. It starts at the center. The circle from the middle of the autoroutine to the edge will always be along the longest side of the pocket.

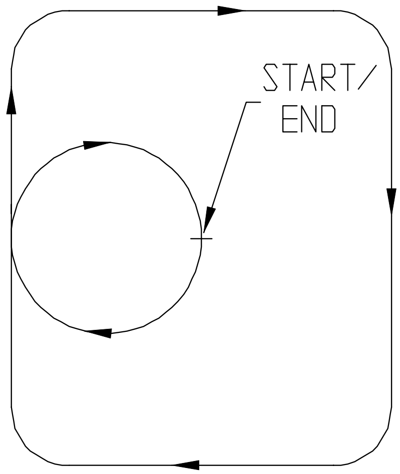
Rectangular Finish Inside Program

N1 G20 G90 (Inch/Absolute)
N2 S1000 M3 D1 G43 H1 (spindle CW-1000 RPM, calls tool #1's offsets)
N3 G00 X0 Y0 (rapids to center of rectangle)
N4 F20 (X-Y feedrate)
N5 P150=.25 (corner radius)
N6 P153=0 (X-Y finish stock)
N7 P154=0 (Z finish stock)
N8 P151=4 (X pocket dimension)
N9 P152=2 (Y pocket dimension)
N10 G35 G99 G42 G2 R.1 P199=1 Z-.5 V-.3 Q.2 F5
*1 *2 *3 *3 *4 *5 *6 *7 *8 *9

*1 Executes rectangular finish inside
*2 Returns Z to clearance plane
*3 CW right or "G41 G3" CCW left
*4 Clearance plane
*5 Ramp Z down
*6 Final Z depth
*7 First Z depth (If no V is specified the 1st Z depth is the clearance plane minus the Z increment)
*8 Z increment
*9 Z feedrate

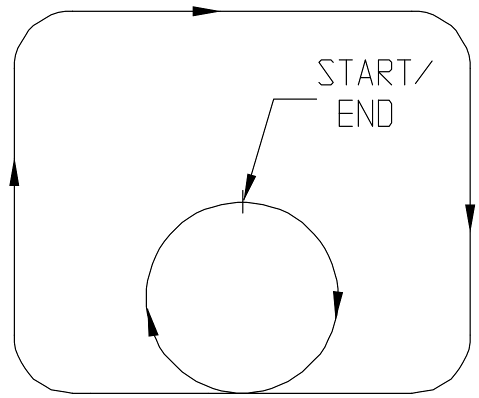
Block #	Line Entry Info
N10	G2 G42 selects CW direction and **right** cutter comp

Inside CW Finish
Rectangular $X \leq Y$
$P151 \leq P152$



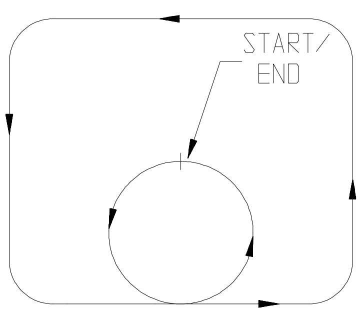
Block #	Line Entry Info
N10	G2 G42 selects CW direction and **right** cutter comp

Inside CW Finish
Rectangular X>Y
P151>P152



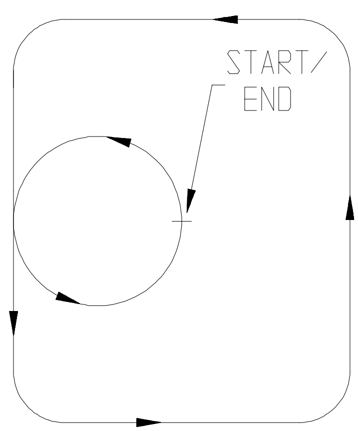
Block #	Line Entry Info
N10	G3 G41 selects CCW direction and **left** cutter comp

Inside CCW Finish
Rectangular
Corners and $X \leq Y$
$P151 \leq P152$



Block #	Line Entry Info
N10	G3 G41 selects CCW direction and **left** cutter comp

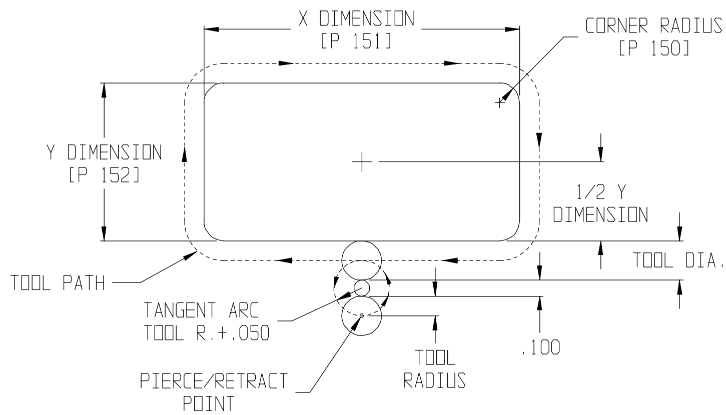
Inside CCW Finish
Rectangular w/Circular
Corners and X>Y
P151>P152



Rectangular finish outside (G36)

The G36 autoroutine is used to remove finish stock around the outside of a rectangular boss. The G36 autoroutine works in an identical manner to the G34 autoroutine. It starts in the center, makes a rapid move to the outside of the part, then feeds the tool down. The circle to the edge is always on the bottom side of the boss. The formula the CNC uses to calculate the distance from the center to the feed down point is as follows.

Y = (3 X tool radius) + .1 + ½ Y pocket width

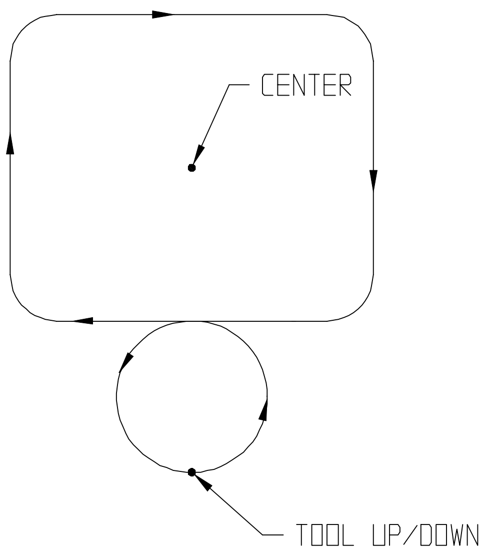


<u>Rectangular Finish Outside Program</u>

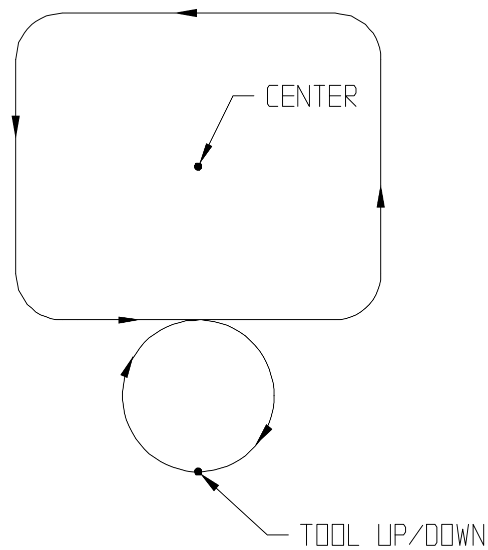
N1 G20 G90 (Inch/Absolute)
N2 S1000 M3 D1 G43 H1 (spindle CW 1000 RPM, calls tool #1's offsets)
N3 G00 X0 Y0 (rapid to X0 Y0)
N4 G98 (returns Z to initial level)
N5 F20 (X-Y feedrate)
N6 P150=.25 (corner radius)
N7 P153=0 (X-Y finish stock)
N8 P154=0 (Z finish stock)
N9 P151=4 (X frame dimension)
N10 P152=2 (Y frame dimension)
N11 G36 G41 G2 V-.25 R.1 Z-.5 Q.25 F10 P199=0
*1 *2 *3 *4 *5 *6 *7 *8

*1 Executes rectangular finish outside
*2 CW left or "G42 G3" CCW right
*3 First Z depth (If no V is specified the 1st Z depth is the clearance plane minus the Z increment)
*4 Clearance plane
*5 Final Z depth
*6 Z increment
*7 Z feedrate
*8 Z plunge

Block #	Line Entry Info	Block #	Line Entry Info
N11	G3 G42 selects CCW direction and **right** cutter comp	N11	G2 G41 selects CW direction and **left** cutter comp



Outside CCW Finish

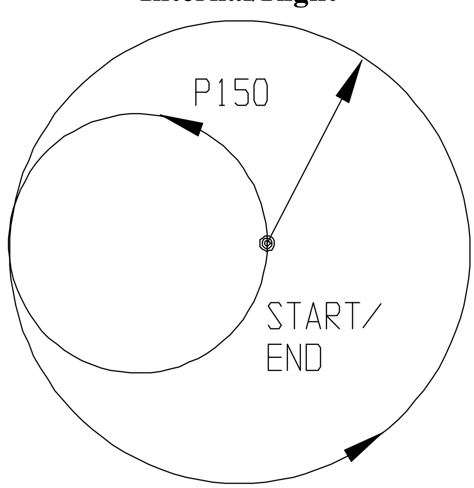


Outside CW Finish

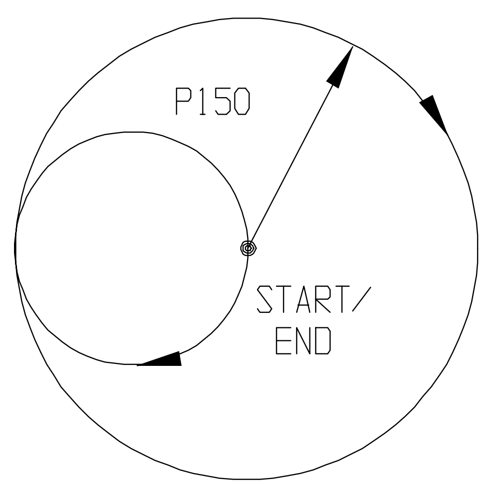
Threading (G39)

The G39 autoroutine is used for cutting threads. It is possible to program either internal or external threads. G39 works on the same principles as the G25 circular finish inside and the G26 circular finish outside. If internal threads are programmed, G39 autoroutine starts at the center and moves as in the following diagrams.

Internal/Right



Internal/Left



For internal threads, the start point is the center of the thread for cutter compensation both on and off. If external threads are programmed, it starts at the center and rapids to the feed down point using the following formula for cutter compensation off.

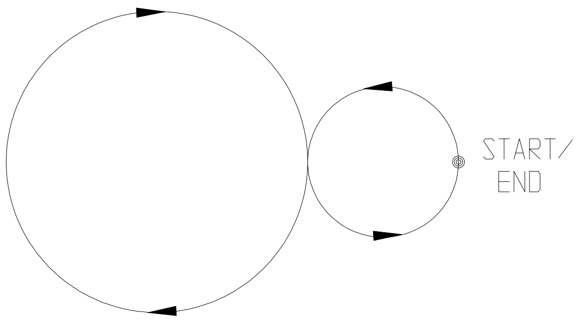
X = thread radius + [2 X tool radius] + 0.1

Cutter compensation on the control uses the following formula to calculate the feed down point.

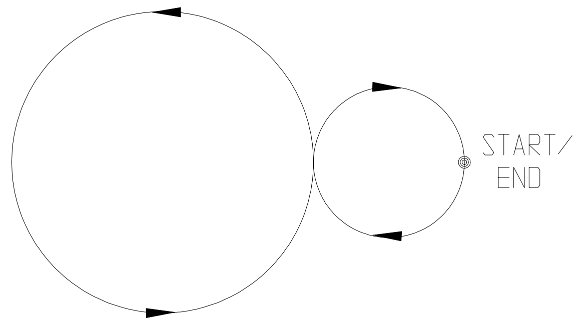
X = thread radius + [3 X tool radius] + 0.1

For external threads the machine will move as in the following figures.

External/Right



External/Left



Polygon Cycle (G666)

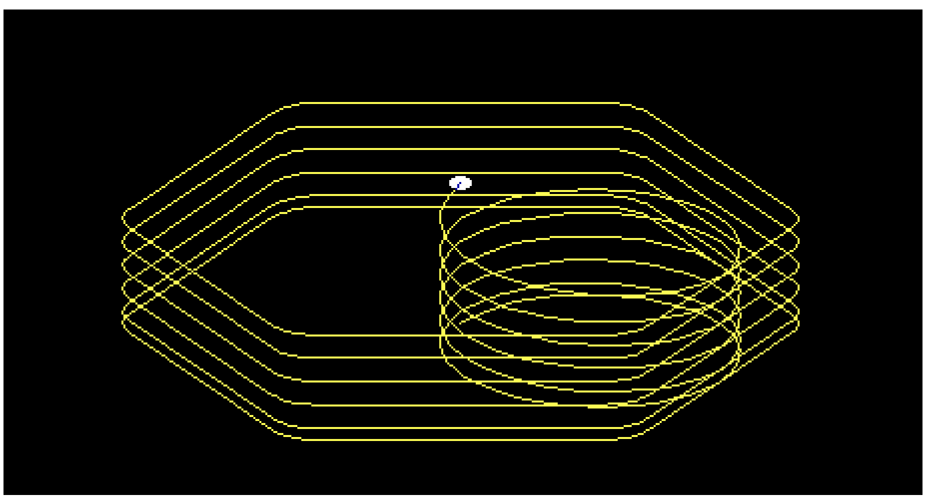
The G666 autoroutine is used for cutting polygons (equally sided shapes), such as triangles, squares, pentagons, hexagons, etc.

The polygon cycle can be used to cut inside or outside. The inside polygon starts in the middle and spirals to the shape, then cuts the polygon and spirals back to the center. Outside polygons must be positioned to the center (or use the G65 X___ Y___ to specify the center). The move from the center to the outside uses the same formula as the circular finish outside or the rectangular finish outside, for the framing polygon cycle.

Inside polygon program
G0 X2 Y3 (Center)
F20 (XY Feedrate)
P126=2 (Radius to the corner)
P127=0 (Angle to the 1st corner)
P125=6 (Number of sides)
P128=.3 (Corner radius)
P132=0 (Inside/outside)
G666 G99 G3 G41 R.2 P199=1 Z-1 V-.1 Q.2 F10
*1 *2 *3 *3 *4 *5 *6 *7 *8 *9

*1 Executes the polygon cycle
*2 Returns to the clearance place
*3 Climb cut CCW/left
*4 Clearance plane
*5 Ramp Z down
*6 Final Z depth
*7 1st Z depth
*8 Z increment
*9 Z feedrate

The above program creates the part below.

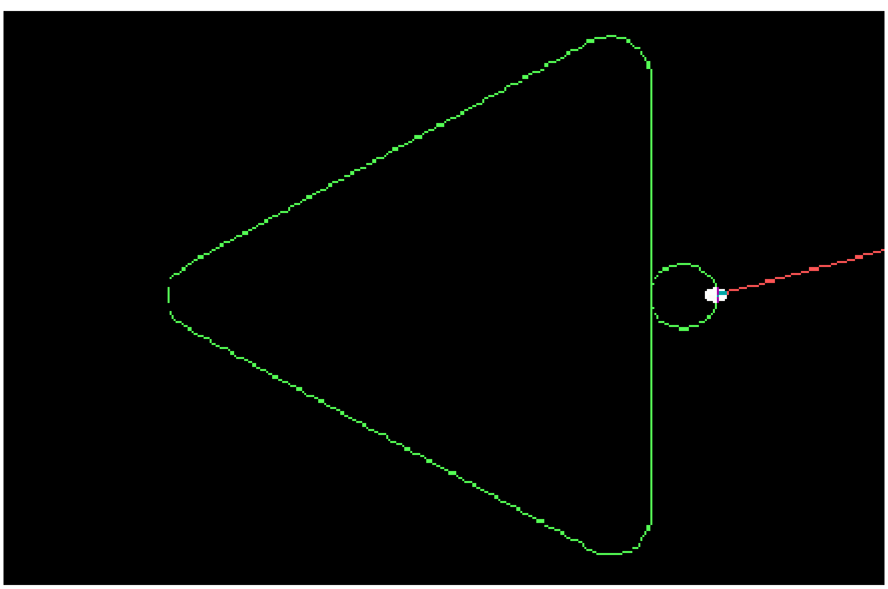


Outside Polygon Program

```
G0 Z2
G65 X2 Y3 (Center)
F20 (XY Feedrate)
P126=3 (Radius to the corner)
P127=60 (Angle to the 1st corner)
P125=3 (Number of sides)
P128=.1 (Corner radius)
P132=1 (Inside/outside)
G666 G98 G2 G41 R.1 P199=0 Z-1 V-.2 Q.2 F5
*1    *2  *3  *3  *4   *5   *6  *7  *8 *9
```

*1 Executes the polygon cycle
*2 Returns Z to the initial position
*3 Climb cut CW/left
*4 Clearance plane
*5 Plunge Z down
*6 Final Z depth
*7 1st Z depth
*8 Z increment
*9 Z feedrate

The above program creates the part below.



N1	G20 G90	(Inch, Absolute)
		N2 G0 X0 Y0 (Rapid position to X center, Y center for internal)(use a G65 for external)
N3	F100	(Feedrate)
N4	P121 = 0	(Angle of taper specified by the half angle or angle with the centerline, 0 for a straight thread)
N5	P122 = 10	(Threads per unit)
N6	P123 = 1	(1 for internal threads, 2 for external threads)
N7	P125 = .03	(Diameter increment/decrement, zero for a single pass)
N8	P126 = 2	(Final Diameter for multiple passes)**
N9	P127 = 0	(X center)
N10	P128 = 0	(Y center)
N11	P150 = 1.91	(Diameter for a straight thread, start diameter for tapered thread and multiple passes)
N12	G39 G42 G2 Z-1 V0 R.1	
	*1 *2 *3 *4 *5 *6	

*1 Executes thread milling cycle
*2 Cutter compensation [Off(G40), Left(G41), Right(G42)]***
*3 Thread direction (G2 right, G3 left)***
*4 Final Z depth (P141)
*5 1st Z depth [(P144, If no V is specified, the control calculates the 1st depth to be P140 (the clearance plane) minus P143 (the Z increment)]
*6 Clearance plane (P140)

**For multiple passes on internal threads, the final diameter must be larger than the start diameter. Conversely, for multiple passes on external threads the final diameter must be smaller than the start diameter.

***The correct combinations of cut direction and cutter compensation are:
G42 G2 for internal right threads.
G41 G3 for internal left threads.
G41 G2 for external right threads.
G42 G3 for external left threads.

Cutter compensation (G40, G41, G42)

G40 Compensation Off
G41 Left Compensation
G42 Right Compensation

This section will explain how the cutter compensation works and give suggestions on how to use it optimally.

Cutter compensation is the displacement of the tool path – perpendicular to the programmed path – by the amount equal to the cutter radius. The programmed path can be figured by the programmer for a zero tool radius. If the parts program is written for a zero tool radius, i.e.

directly off the print, then by entering the actual tool radius into the system and activating cutter compensation, the operator can make the control calculate the displaced path.

Throughout the program the control keeps a record of the previous programmed point, the current programmed point, and the next programmed point along the tool path.

With two geometries such as a line and an arc, the cutter radius, and whether it is a left or a right compensation, the control can calculate the current compensated point. The control will also employ its Trig Help function discussed earlier to connect lines and arcs during cutter compensation.

After each successful calculation of a compensated point, the current programmed point becomes the previous programmed point, the next programmed point becomes the current programmed point, and a new programmed point is read up to become the next programmed point. This mechanism is repeated until the end of the program is reached. This sequence should be understood clearly, for it will clarify many points that occur later concerning how the compensation works.

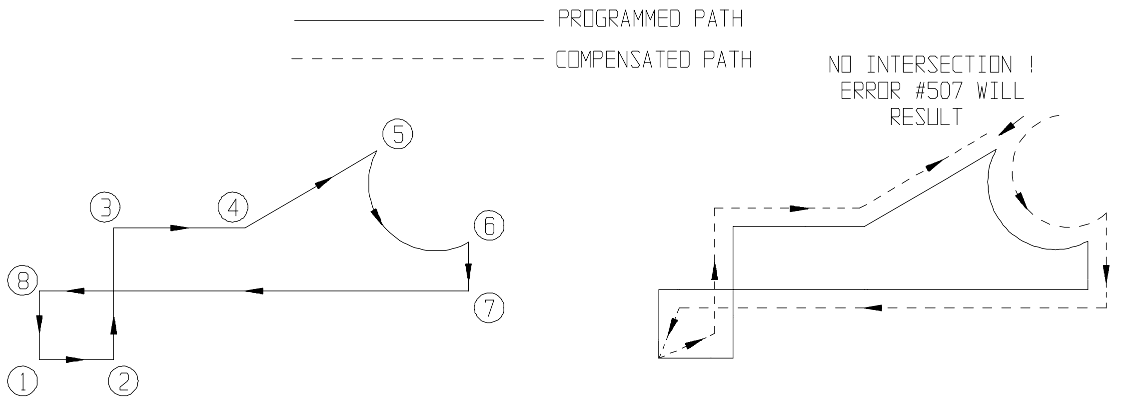
The compensation in this control is intersectional. Given the three points mentioned above, the control calculates the intersection of the compensated path between the previous and the current programmed points **and** the compensated path between the current and the next programmed points. These paths can be a mixture of straight lines and arcs.

Because of the intersectional nature of the compensation package, there must be an intersection of all the displaced paths for the system to work. **If there is no intersection between two paths, the control will give an error 507 or 509.**

Note: All cutter compensation examples are shown without Z axis moves.

To see how cutter compensation is done, it is useful to select the F6 (Both) key when verifying a part.

Explanation of How Displaced Tool Paths Cannot Have an Intersection

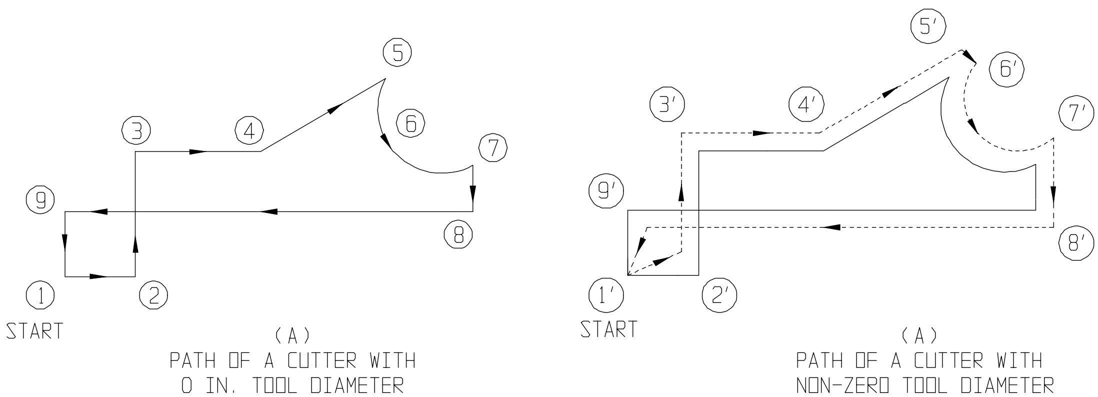


(a) Path of a cutter with 0" tool diameter (b) Path of a cutter with non-zero tool diameter

The solution of the above part is to introduce a 00.0001" chamfer or round corner at Point 5 between the non-intersecting surfaces.

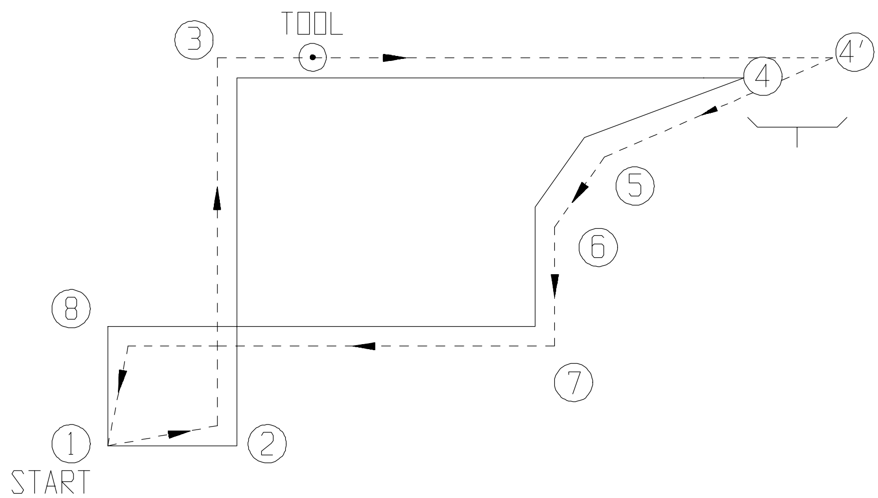
Explanation of How a 00.0001" Chamfer Should Be Introduced to Solve a Non-intersection Problem

Note: The ",C" used to chamfer can only be used between two lines.



In some cases the system will find an intersection, but it will be unreasonably far away from the part. Again, in such cases, a 00.0001" chamfer or round corner should be inserted to solve this problem.

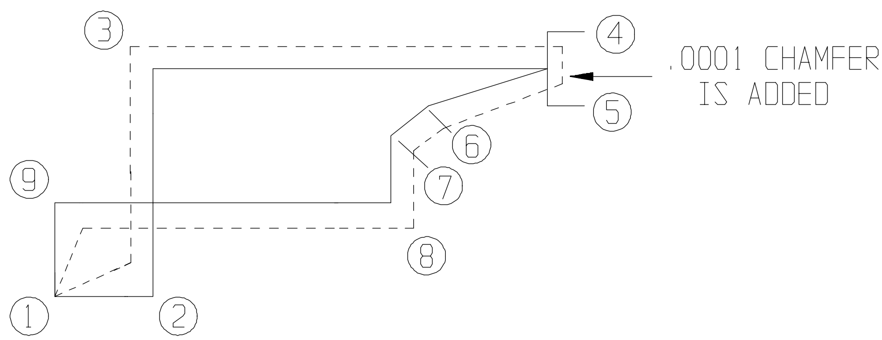
Outside "V" Cutter Compensation



Note: Compensation point (4') is displaced more than the tool radius away from (4).

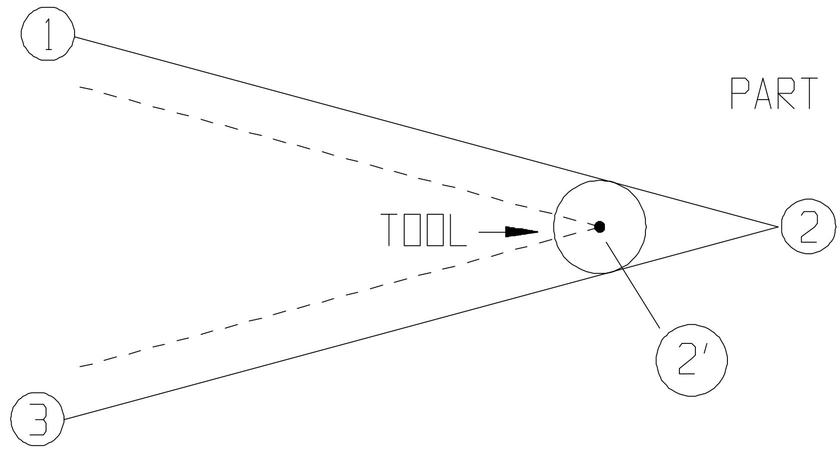
The figure below shows how a 00.0001" chamfer or round corner added at point (4) has saved an unnecessary departure.

Outside "V" Cutter Compensation Solution



The figure below shows how the compensated point for an inside "V" will stay away from the programmed point by more than the tool radius.

Inside "V" Cutter Compensation



Note: The tool stays away from the programmed point (2) by a distance more than the tool radius. If the compensated point (2') was any closer to (2), the tool would gouge the sides of the part.

Sample part exercise

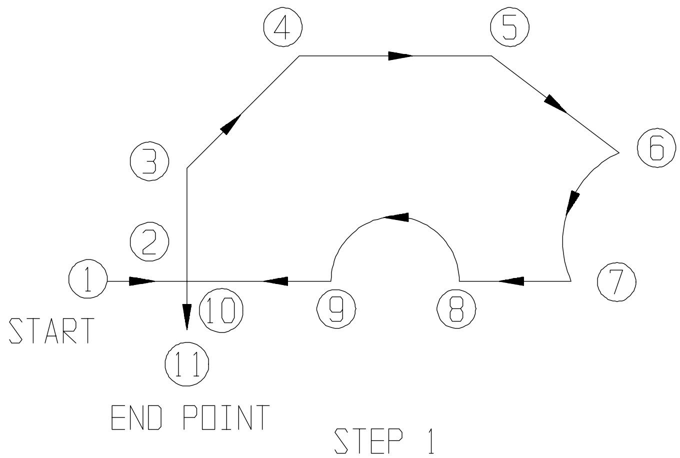
As the system requires three points to generate a compensated point, care should be taken when the cutter compensation is turned on or off.

The magnitude for the axis movement for leaving or entering the part **must** be greater than the amount of cutter compensation expected. The following figure under Step 1 shows that the machine position at point (1) is identical for both the compensated as well as the uncompensated tool path. This is because the first point cannot be compensated as it has no previous point to make up a three point set for the compensation routine. Therefore, the compensation should be turned on **before** the tool enters the work.

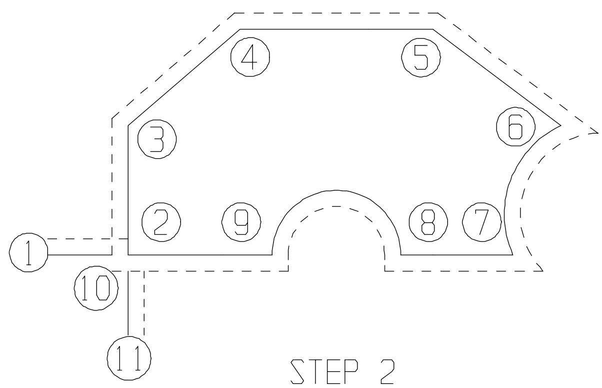
For ease of programming, the tool should enter and leave the part **perpendicular** to the part surface. This is not a strict requirement, but it simplifies understanding how the cutter compensation will behave entering and exiting the workpiece. If in doubt about how the displaced tool path will look, it is advisable to rough sketch, by hand, lines parallel to the part surface from start to end or use the F6 (Both) key when verifying. Often problems will become apparent immediately.

Determining How the Compensated Path Will Look

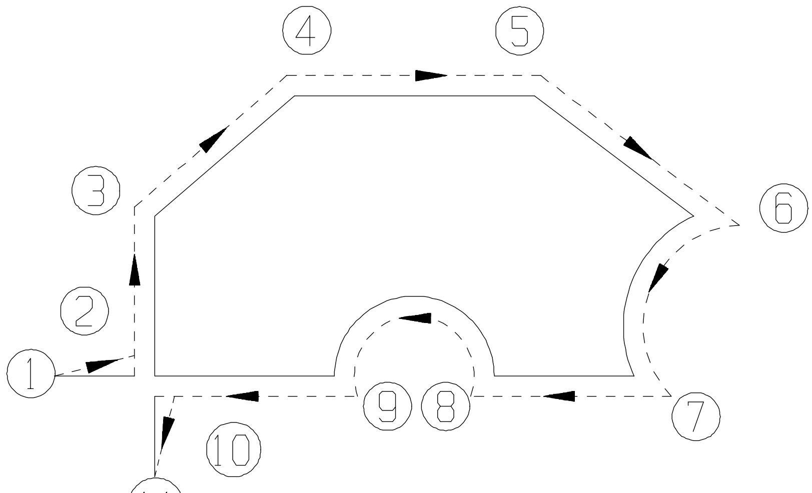
Step 1. Sketch actual part and label points in sequence.



Step 2. Sketch lines displaced by tool radius away from part surface from point 1 to point 11.

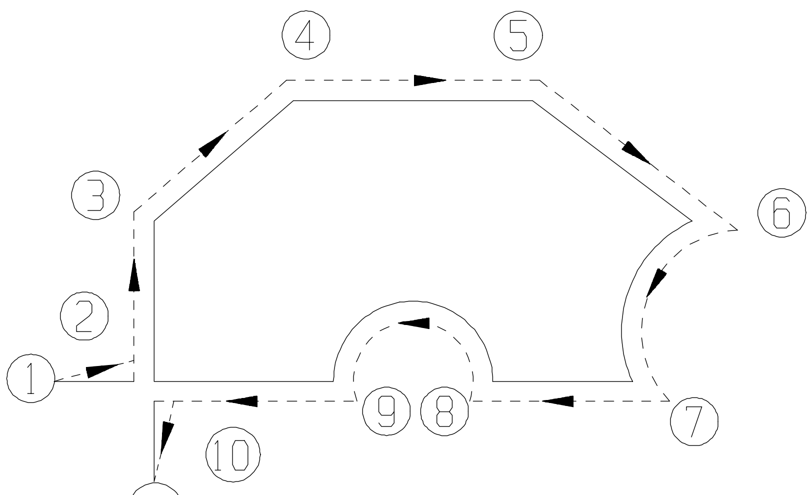


Step 3. Check to verify all paths in the sequence intersect. If yes, then (except for the start and end points) connect the displaced path and label points of intersection. If even one intersection cannot be found, the part will not run unless the error is corrected.



STEP 3

Step 4. Since points 1 and 11 do not have two points on either side, they will be the uncompensated points. Therefore, connect them to their neighboring compensated points to arrive at the actual displaced cutter path.

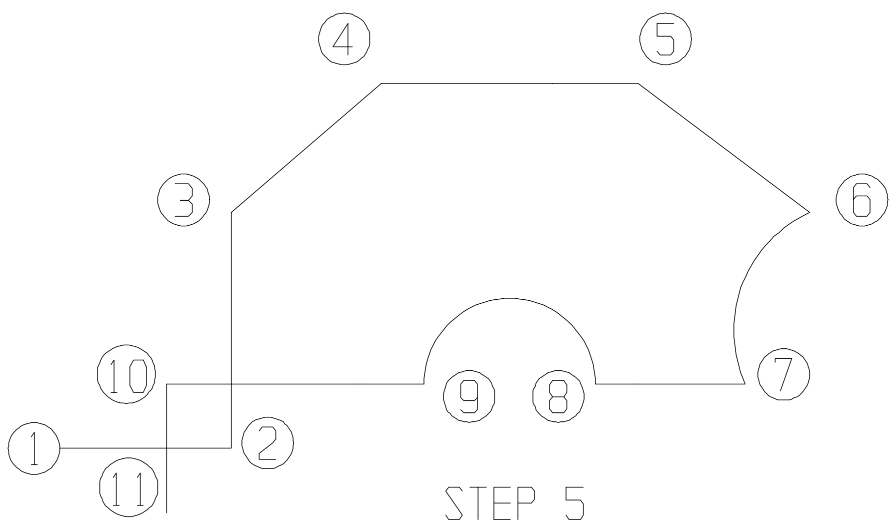


STEP 4

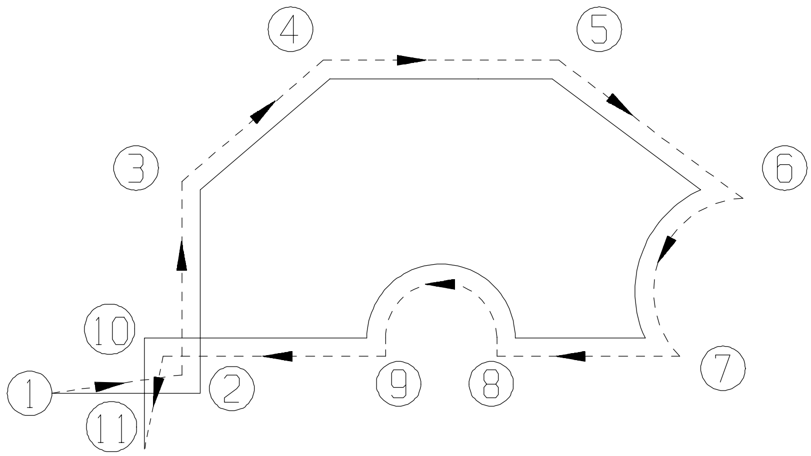
Step 5. The above displaced path is what the system will trace if the part is run. However, a problem has become apparent from the rough sketch.

Note that the lower left-hand corner will be left uncut because the tool going from (1) to (2) will leave a small notch of uncut material. A similar case is obvious in the tool path from (10) to (11). There again the corner will be left uncut.

The solution is to rearrange the start and end points so that the corner is properly cut.



Step 6. Note how points 1, 2, 10, and 11 have been moved slightly. The result will be as follows.



STEP 6

Note: It is now seen that when the tool moved from (2) to (3), and (9) to (10), the corner will be properly cut.

In Step 6, the cutter compensation was turned on by using the G41 command for turning on left cutter compensation at point (2) and turned off by using the G40 command at point (11).

<u>Notes on Steps 1-6</u>

Note 1: Points (1) and (11), being the start and end points, were chosen so that they lie sufficiently away from the part surface. Sufficient implies a distance ***more*** *than the total expected compensation.*

Note 2: Circle at (9) and (8) points to complications that will arise if the tool radius is too large. As the radius is increased, (9) and (8) will keep moving closer together until, at one point of the radius, they will become identical. If the radius is increased further, the tool radius will have become too large to make the circle and the system will give an error telling the operator that an intersection cannot be found at that line.

Note 3: If a compensated path can be successfully sketched by hand, then it will run on the system. However, if the sketch yields a missing intersection, the control will give an error 507 or 509.

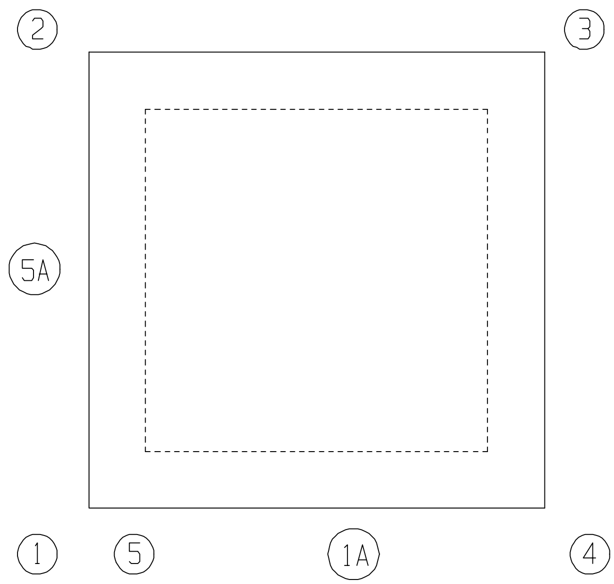
Note 4: Until the operator becomes familiar with cutter compensation, it is advisable that rough sketches be made for the compensated path before the part is run as a program.

Note 5: Cutter compensation is shut off at the start of each program.

How To Compensate for a Cavity

If the part is a cavity, then the start and end points will have to change. Simply changing the G41 to G42 (right cut) will not help. This is because the tool will still come down on the part. The reason for that is as explained earlier: the system uses the previous, current, and the next **programmed** points to calculate its compensation. This procedure does not occur at point 1 and point 5.

Relocation of Points 1, 2, 10, and 11 for a Cavity



In the above figure, the tool should be brought down at point 1 and taken back up at 5.

The purpose of adding point 1a is to give advance information to the system about line 4 to 5. The length of 1a to 1 can be as small as 00.0001". It is important to note that for the correct advance information the slope of 1a to 1 **has to be the same** as the slope of 4 to 5. In this case the slope is zero.

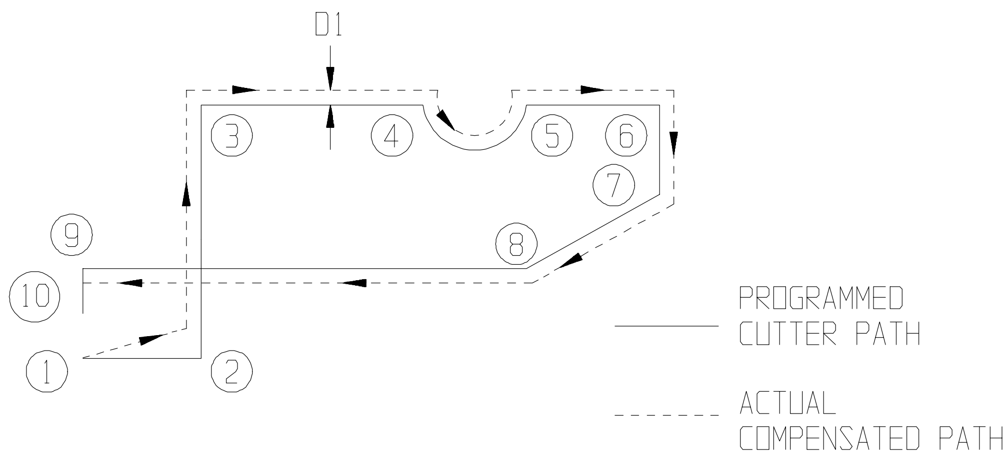
Similarly, the purpose of adding point 5a is to give **past** information to the system about line 1 to 2. Again, the slope of 5 to 5a has to be the **same** as the slope of 1 to 2.

Regardless of the cutter radius, the tool can now be brought down on the corner at (2) and brought back up at 5.

Programming with Cutter Compensation

When programming with cutter radius compensation, the first and last move the cutter makes should be done off the part per the figure below. The movement made prior to cutting should be at least the distance of the cutter diameter being used.

Block #	Block Entry Info
N1001	G0 X-1 Y-1
N1002	G41 X0 D1 (D offset = tool radius)
N1003	G1 Y3 F10
N1004	X3.5
N1005	G3 R.5 XC4 YC3 X4.5 Y3
N1006	G1 X6.8
N1007	Y2
N1008	X4.5 Y0
N1009	X-1
N1010	G40 Y-.5



No-movement (G65)

The G65 is used for starting and ending cutter compensation. The G65 code placed on a line with coordinates will cause these coordinates to be used for cutter compensation points but skipped during machine movement.

G65 X___ Y___ Z___ The machine will not move to the XYZ coordinates.

Note: G65 can use polar move as well as Cartesian coordinates.

G65 R____ AB____

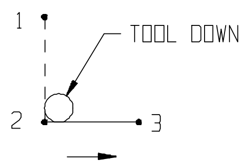
G65 will allow the programmer to turn cutter compensation on and get the tool to drop or retract at a specific point without doing any extra moves. Generally the no-move point would be chosen to be a point on the part that directly precedes the tool down point. On a tool retract, the no-move point would be a point on the part directly after the tool up point. The no-move point does not have to lie on the part, but points on the part generally work the best. The following sets of figures under "Starting and Ending Cutter Compensation" show various tool start and retract positions given different no-move points, which are indicated by the dotted line (---). Point 2 is the desired tool start or retract point. Point 1 is the no-move point in the cutter compensation on (pierce) case, and Point 3 is the no-move point in the cutter compensation off (retract) case. If cutter compensation is turned on or off using the following format, the tool up and down position can be easily predicted.

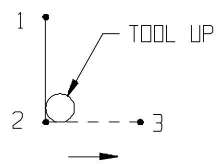
Starting and Ending Cutter Compensation

G41 Tool Left
D1 = Tool Radius (Previously Set in D1)

PIERCE	RETRACT
1=point on part before pierce point	1=last position before retract
2=pierce point	2=tool retract position
3=first cut move	3=point after retract

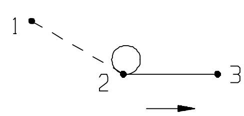
G41 D1
G65 X0 Y1
X0 Y0
X1 Y0

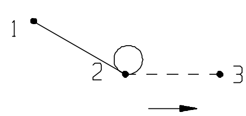




G41 PREVIOUSLY ON
X0 Y1
X0 Y0
G65 X1 Y0
G40

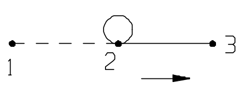
G41 D1
G65 X-1 Y1
X0 Y0
X1 Y0

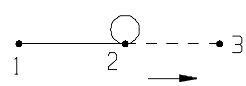




G41 PREVIOUSLY ON
X-1 Y1
X0 Y0
G65 X1 Y0
G40

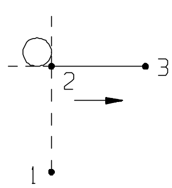
G41 D1
G65 X-1 Y0
X0 Y0
X1 Y0

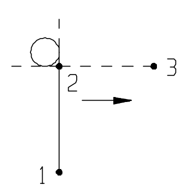




G41 PREVIOUSLY ON
X-1 Y0
X0 Y0
G65 X1 Y0
G40

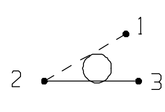
G41 D1
G65 X0 Y-1
X0 Y0
X1 Y0

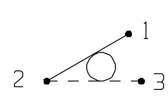




G41 PREVIOUSLY ON
X0 Y-1
X0 Y0
G65 X1 Y0
G40

G41 D1
G65 X1 Y1
X0 Y0
X1 Y0



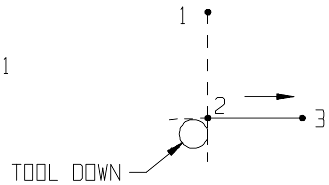


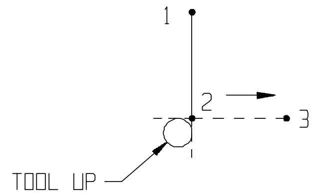
G41 PREVIOUSLY ON
X1 Y1
X0 Y0
G65 X1 Y0
G40

G42 Tool Right
D1 = Tool Radius (Previously Set in D1)

PIERCE	RETRACT
1=point on part before pierce point	1=last position before retract
2=pierce point	2=tool retract position
3=first cut move	3=point after retract

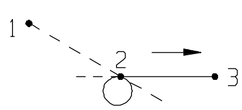
G42 D1
G65 X0 Y1
X0 Y0
X1 Y0

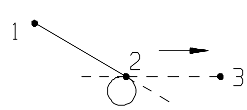




G42 PREVIOUSLY ON
X0 Y1
X0 Y0
G65 X1 Y0
G40

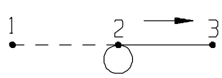
G42 D1
G65 X-1 Y1
X0 Y0
X1 Y0

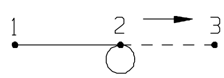




G42 PREVIOUSLY ON
X-1 Y1
X0 Y0
G65 X1 Y0
G40

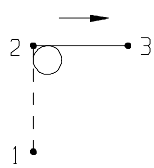
G42 D1
G65 X-1 Y0
X0 Y0
X1 Y0

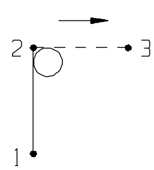




G42 PREVIOUSLY ON
X-1 Y0
X0 Y0
G65 X1 Y0
G40

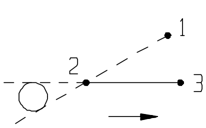
G42 D1
G65 X0 Y-1
X0 Y0
X1 Y0

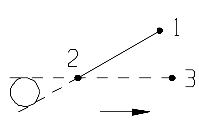




G42 PREVIOUSLY ON
X0 Y-1
X0 Y0
G65 X1 Y0
G40

G42 D1
G65 X1 Y1
X0 Y0
X1 Y0

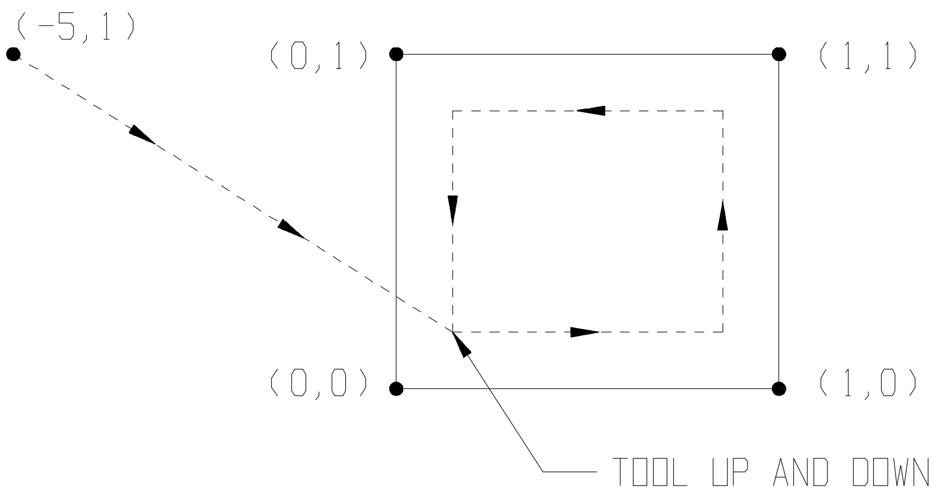




G42 PREVIOUSLY ON
X1 Y1
X0 Y0
G65 X1 Y0
G40

Sample Program for Enter-Exit Cutter Compensation

G0 X-5 Y1	part load/unload point
G41 D1 F10	cutter comp. on offset #1
G65 X0 Y1	no move compensation point
X0 Y0	
G1 Z-1	tool down
X1	
Y1	
X0	
Y0	
G65 X1 Y0	cutter comp. off no move exit point
G40 G0 Z0	tool up



Note that the tool enters and exits the part tangent to both walls because of the G65 lines.

Notes on Cutter Compensation

Note 1: Turning compensation on can be done both in a block without axis movement or in a block containing axis movement.

Note 2: There is no restriction on how many successive blocks can have no axis information.

Note 3: There is no restriction on how lines enter and leave arcs. They can have any angle of intersection as long as an intersection exists.

Note 4: Given the correct centers and radii for two intersecting circles, the system automatically checks and corrects the programmed point of intersection, i.e. Trig Help.

*Note 5: All autoroutines use the present axis position as their center. For this reason it should be made sure that the cutter compensation is turned **off** in a program using these routines so that the axis can position to the programmed center. If a compensation center is used, the entire pocket will be shifted.*

Note 6: If the programmed point rather than the compensated point is desired, a G40 command should be added to the block containing that point. The G40 could have been a part of the previous block, providing that block had no axis information.

Note 7: Similarly, if the point after the desired programmed point is to be compensated, a G41 or G42 should be added to the block containing that next point or to any non axis command block between the two.

Note 8: Roughing and finishing passes can be easily made by first entering a tool radius value larger than the actual measured tool radius by the amount of stock to be left on the part for the finish pass.

When the program is run, the resultant part will be oversize. However, by entering the actual cutter radius into the system and running the program once more, the finished part size will be obtained.

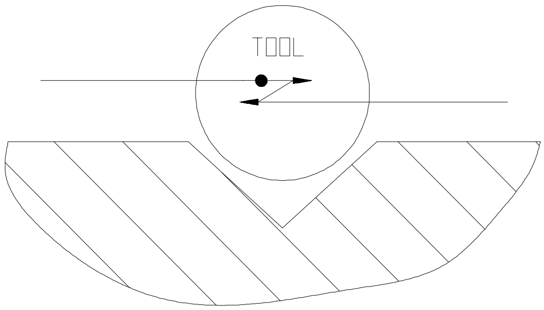
*Thus by entering a **larger** or **smaller** tool radius, the part can be made **under** or oversize.*

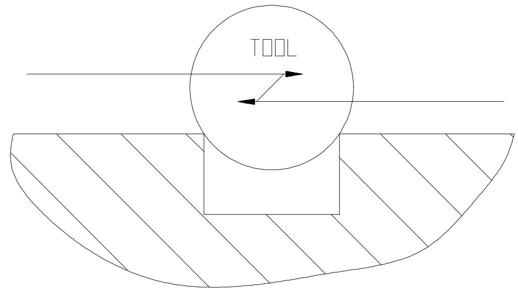
Note 9: In general, when using cutter compensation, no inside feature on the part can be smaller than the tool radius. This includes such things as slots, arcs, and vees. If a part contains such features, they should be replaced by straight lines, cutter compensation should be turned off, or a smaller tool should be used.

Note 10: Cutter compensation can be shut off by setting the special flag parameter to 4.

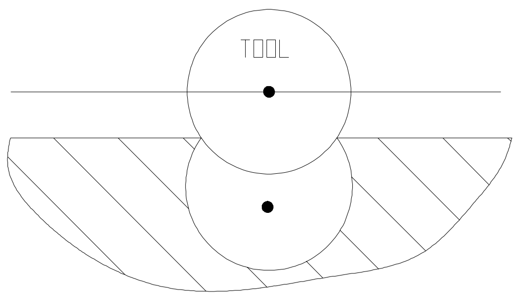
Note 11: A G41 with a negative tool radius is the same as a G42 with a positive tool radius.

Note 12: A G42 with a negative tool radius is the same as a G41 with a positive tool radius.





In the above cases, the tool will back up as it tries to place itself tangent to the walls of the slots or v.



This case will give a *"compensated line/arc do not intersect"* error.

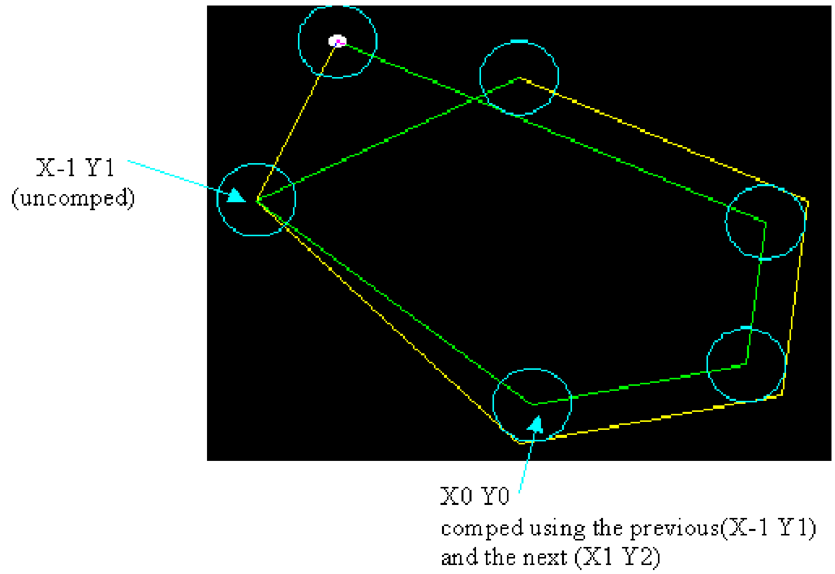
Auto cutter compensation (G45, G46, G47)

The automatic cutter compensation commands, G45 (auto left), G46 (auto right), and G47 (auto off), work primarily like the normal cutter compensation commands using the previous, current, and next programmed points to calculate the current compensated point. They only differ in the way the first and final programmed points are compensated. The control automatically calculates the previous programmed point for the first compensated point and the next programmed point for the final compensated point. These points are calculated so the compensated path will begin and end at points the cutter radius away from the first and last programmed point on either the left (G45) or the right (G46) side of the programmed path.

Sample Programs

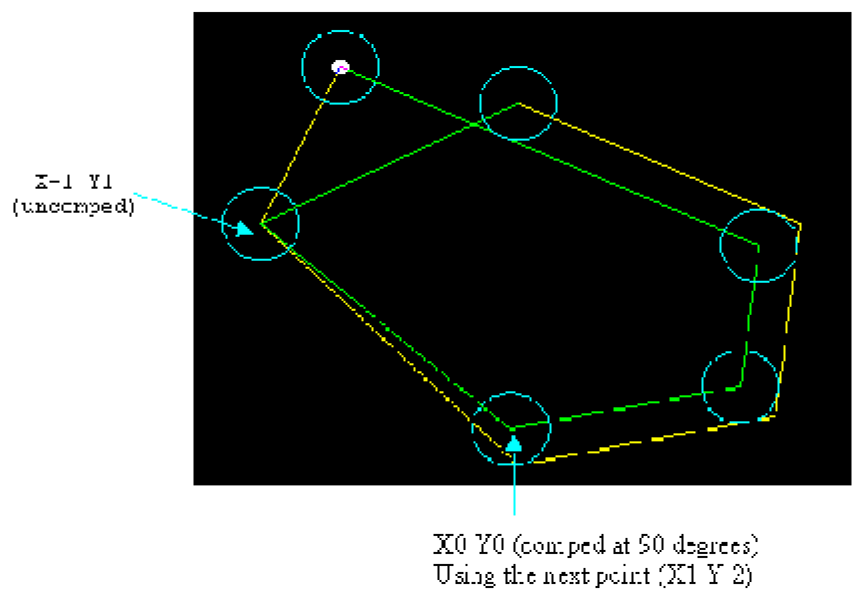
Cutter comp on using a G41

X-1 Y1
G41 X0 Y0
X1 Y.2
X0 Y1.5



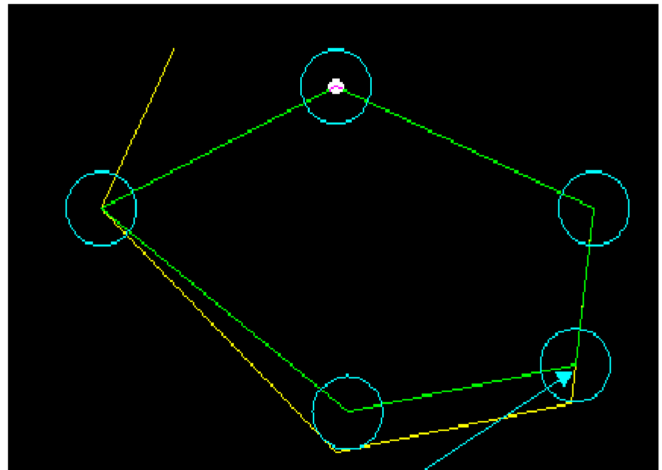
Cutter comp on using a G45

X-1 Y1
G45 X0 Y0
X1 Y.2
X1.1 Y1
X0 Y1.5



Cutter comp off using a G40

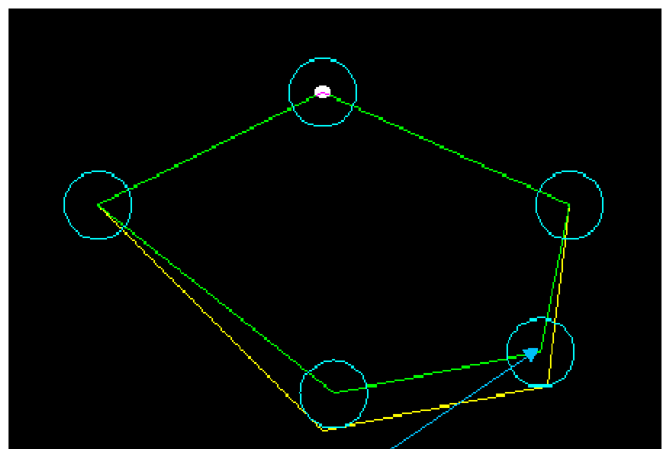
X-1 Y1
G41 X0 Y0
X1 Y.2
G40 X1.1 Y1
X0 Y1.5



X1 Y.2 (using the intersection of the comped line (X0 Y0 to X1 Y.2) and the uncomped line (X1 Y.2 to X1.1 Y1)

Cutter comp off using a G47

X-1 Y1
G41 X0 Y0
X1 Y.2
G47 X1.1 Y1
X0 Y1.5



X1 Y.2 (using the 90 degree comped line (X0 Y0 to X1 Y.2)

Tool length offset (G43, G44, G49)

A tool length offset is activated using a G43 or G44 command.

Command format:

G43 G44 or	Z ____ H____;	(Z moves to dimension selected referenced to tool length offset selected by H)
G43 G44 or	H____;	
G43	H10; 1st offset H14; 2nd offset H13; 3rd offset H15; 4th offset	

The direction of the offset is controlled by G43 and G44; the magnitude of the offset is set by the offset value in the H table.

G43 is a + offset (value in H table is added to axis)
G44 is a - offset (value in H table is subtracted from axis)

Once a G43 or G44 offset is activated it will remain in effect until canceled by a G49 or changed with another H offset. The H offsets can be changed throughout the program without canceling the previous offset with a G49 or H00. The new H offset will automatically take effect in either the G43 or G44 mode, whichever is active at the time the new H offset is executed. The offsets for the current tool are active at the start of all programs and any H offsets before G43 or G44 will zero out the tool offset. After a G49 or H6 is executed, H offsets are ignored until a G43 or G44 is executed. The H offsets will always be added to the axis perpendicular to the current plane at the time the H offset is activated.

G17 XY plane H add/sub Z axis
G18 XZ plane H add/sub Y axis
G19 YZ plane H add/sub X axis

If an H offset is activated in the G17 XY plane and then the plane is switched to G18 XZ plane, the offset will remain in effect and still be added to the Z axis. However, if another H offset is activated while still in the G18 XZ plane, it will be added to the Y axis. Both offsets will be in effect, one on the Z axis and one on the Y axis, until they are canceled by a G49 or changed by another H offset.

H offsets from the tool table
H01 = 1.5
H02 = -.5
H03 = -1.25
H04 = 5

Various program lines and results

G17 G43 H1	
G90 Z0	Z moves to 1.5 (from home)
Z1 H3	Z moves to -.25 (from home)
G44 H3	
Z0	Z moves to 1.25 (from home)
H4	
Z0	Z moves to -5 (from home)
G19 G43 H2	
X0	X moves to -.5 (from home)
H0	
X0	X moves to 0 (from home)
G49 X0 Z0	Z moves to 0 and X moves to 0 (from home)

G90 and G91 modes have no bearing on how the H offsets are added/subtracted to the final axis position.

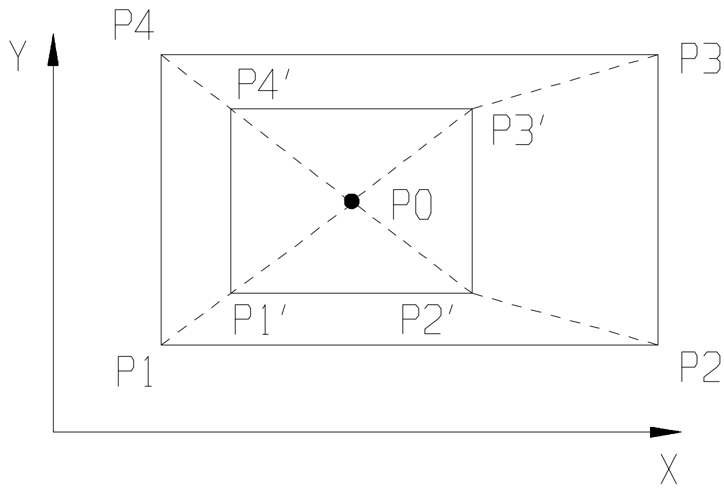
Cancel scaling (G50) Set scaling (G51)
Scaling can be commanded at any time during a program by using the G51 command.

Command format:

G51 I____ J____ K____ X____ Y____ Z____

I, J, K are the scaling center. If I, J, K are not specified in the G51 line, the scaling center will default to the last center used. The scaling center is set to 0 at the start of each program and after a G50.

X, Y, Z are the scale factors for each axis. The range of each scale factor is ±999.9999 to ±000.0001. The scale factors, once set, remain in effect until changed or canceled by a G50. At the start of each program all scale factors are set to 1.



P1 - P4	original program no scaling
P1'- P4'	scaled program
P0	scaling center

Notes on Scaling

Note 1: Once set, scaling remains in effect until canceled by a G50.

Note 2: If arcs are being scaled, the primary axis scale factor is used.

Note 3: G27, G28, G29, G30, and G92 are not affected by the scale factors.

Note 4: To scale all axes to the same scale factor use G51 P.

Note 5: G50 sets scale factors to 1 and scaling centers to zero.

Coordinate systems

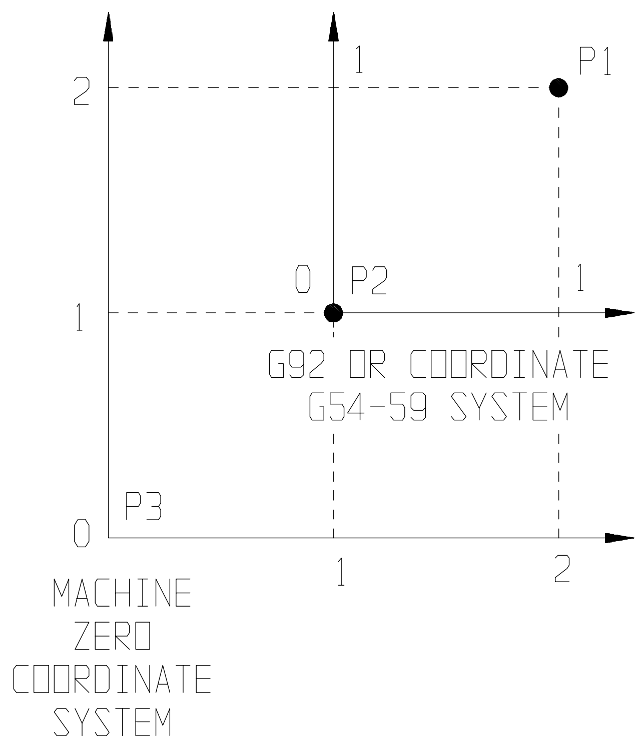
Machine zero is a fixed point on the machine. The machine tool builder normally decides the machine zero point. A limit switch and encoder marker pulse on each axis sets it.

The machine zero point is established when the F1(Home) command is first executed. Once the machine zero point is established, it is not changed by reset, coordinate system call (G54-G59), coordinate system shift (G92), or local coordinate system setting (G52).

Software limits are set from the machine zero point.

Machine coordinate system (G53)

A G53 code preceding any XYZ move will cause those dimensions to be relative to the machine zero point.



X1 Y1	move to P2
G92 X0 Y0	
X1 Y1	move to P1
X-1 Y-1	move to P3
or	
G53 X0 Y0	move to P3

A coordinate system used to align the work part dimensions to the machine's programs is called a **work coordinate system**. The work coordinate system is set by either of the following methods.

1. using a G92 command
2. using a G53 command
3. using G54 -G59 commands
4. using A G52 command

Work coordinate systems (G54 - G59)(G5#0…G5#9)

The dimensions of the work coordinates are always relative to the G92 Floating Zero Point. To set a work coordinate system, press F7 (Parms) - F2 (Coord). The work coordinate menu will appear allowing you to enter the offset coordinates for each work coordinate system. The Home Position offsets are parameters which shift all coordinate systems relative to the Machine Zero Point. Normally the Machine Zero Point and the Home Position are the same.

The following extended work coordinate systems are available.

G540 - G549
.
. (60 total work offsets)
.
G590 - G599

Note: G540 - G590 are the same as G54 - G59

G55 X1 Y1	moves to X1 Y1 in work offset 2
G59 X1 Y1	moves to X1 Y1 in work offset 6
G54	is always the power on coordinate system

Note: G54 - G59 offsets are not zeroed on power-up or after homing. The control will remain in the selected coordinate system until another G54-G59 is executed.

Local coordinate system (G52)

The G52 command is similar to the G92 command in that it uses the current coordinate system zero as its reference point instead of the current machine position (G92). When using the G52 command, think of it as "shift the work coordinate by"
X____ Y ____ Z____.

Notes on Local Coordinate System

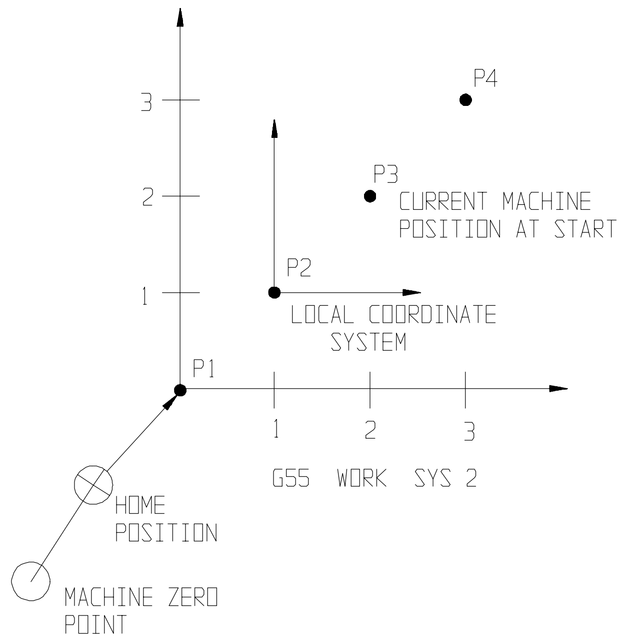
Note 1: A G52 is modal; therefore it will affect all coordinate systems once set.

Note 2: To cancel a G52, enter G52 X0 Y0.

Note 3: *G52 offsets are not affected by the position of the machine. G92 offsets are affected by the position of the machine.*

Note 4: *G52 offsets are zeroed on power-up, after homing, after setting work offset in handwheel or jog, and after any G92 command.*

Note 5: *G52 offsets are restored to there initial values after the program ends.*



G55	
X2 Y2	moves to P3
G52 X1 Y1	sets zero at P2 dim. rel. P1
X1 Y1	stays at P3
X2 Y2	moves to P4

Using G92

G55	
X2 Y2	moves to P3
G92 X1 Y1	sets zero at P2 dim. rel. P3
X1 Y1	stays at P3
X2 Y2	moves to P4

Single direction or one shot rapid positioning (G60)

For accurate positioning without backlash, positioning from one direction is available.

G60 X___ Y___

G60 is a one-shot G code and is used in place of G00.

Notes on Single Direction Positioning

Note 1: *The amount of overrun is pre-set by the machine tool builder.*

Note 2: *Overrun direction is not affected by mirror imaging.*

Note 3: *If "G00 unidirectional approach" was set by the machine tool builder, the same positioning sequence would happen with each G00 move.*

Exact stop mode (modal) (G61)

When G61 is commanded, deceleration is applied to the end point of the cutting block and in-position is performed per block thereafter. G61 is valid until G63 (tapping mode) or G64 (cutting mode) is commanded.

Tapping mode (modal) (G63)

When G63 is commanded, feedrate override and spindle speed override are ignored (always regarded as 100%), and block mode and feedhold becomes invalid. G63 is valid until G61 (exact stop mode) or G64 (cutting mode) is commanded.

Cutting mode (modal) (G64)

G64 is the default at the beginning of each program. When G64 is commanded, deceleration based on the angle between blocks at the end point of each block thereafter is performed, and cutting goes on to the next block. This command is valid until G61 (exact stop mode) or G63 (tapping mode) is commanded.

Calling a program (G65 P)

G65 can also be used for calling a program. The program to be called is specified by the value of P.

Example: *G65 P4371 (calls program #4371 and sets parameter #16 to 4371 and parameter #7 to 65)

To pass arguments to the program, other addresses can be added to the block.

Example: *G65 P1402 A500 (calls program #1402 and sets parameter #1 to 500, parameter #16 to 1402, and parameter #7 to 65)

** Parameters not specified are set to -999.*
The addresses refer to the parameters as follows.

Address	Parameter #
A	1
B	2
C	3
.	.
.	.
X	24
Y	25
Z	26

Notes on Calling a Program with G65

Note 1: If the program specified by address P does not exist, an error will occur.

Note 2: The program called is the rounded value of address P. Example: G65 P12.75 (calls program #13)

Note 3: If no P is in the G65 block, the block is treated as a non-movement block.

Note 4: Addresses not specified are set to -999.

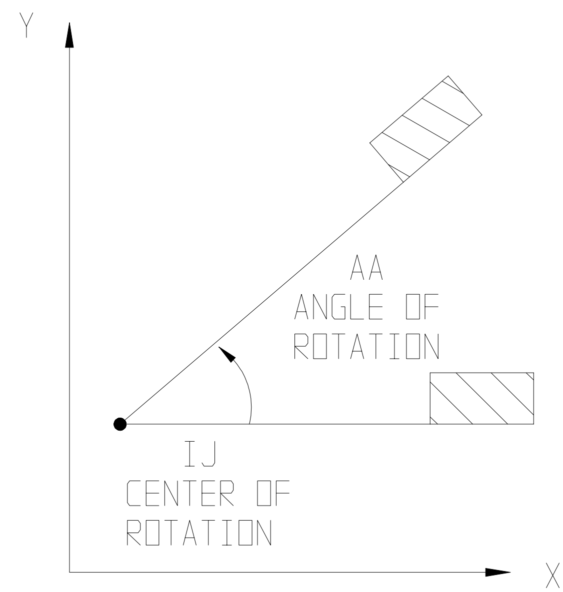
Coordinate system rotation (G68) Cancel Rotation (G69)

G68 can be used to rotate a programmed shape about a predefined center point. The plane of rotation is defined by G17, G18, G19; the center of rotation is defined by IJK and the angle of rotation by AA. The command format is as follows.

G68 AA+_____ I____ J____ K____

AA+ is CCW
AA- is CW

IJK specify the center of rotation in the plane selected by G17 G18 G19. The center of rotation defaults to the current coordinate system zero point at the beginning of each program. If the IJK's are not present in the G68 block, the center of rotation will be the last center specified. The rotation angle and rotation center are zeroed at the start of each program.



G69 zeros the rotation angle and rotation center.

Care needs to be taken when using rotation in conjunction with other functions. Functions such as mirror image, scaling, and cutter compensation need to be carefully considered when used together with rotations. Some of the basic rules are as follows.

1. Cutter compensation should be off (G40) when rotation is called. (Cutter compensation can be turned on after rotation is called.)

2. If scaling is on before rotation, the rotation center will be scaled; if rotation is called before scaling, the rotation center will not be scaled.

3. The order of on and off is **first on, last off.**

   G51 . . . . scaling on
   G68 . . . . rotation on
   G41 . . . . cutter compensation on
   .
   .
   .
   G40 . . . . cutter compensation off
   G69 . . . . rotation off
   G50 . . . . scaling off

4. If the rotational center is scaled, it will remain scaled until replaced by a new center, or canceled with G69.

5. Rotation is always done in the active plane.

6. X, Y, and Z can be used instead of I, J, and K for rotation center.

7. R can be used instead of AA for rotation angle.

3D Rotation (G0, G1, G2, G3, G68 AND G69)

G0, G1, G2, and G3 respond to 3D rotation when a G68 ABm has been entered.

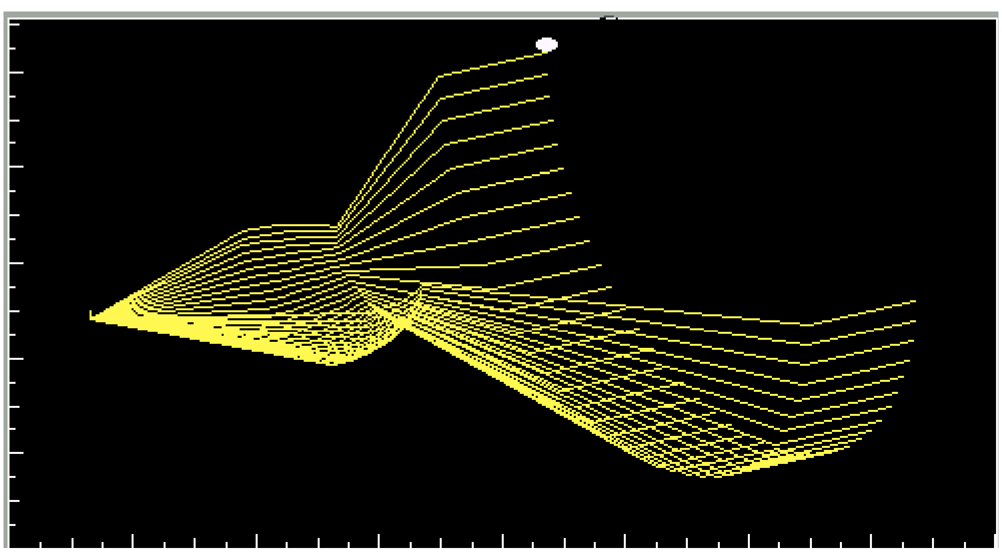
G68 ABm

The ABm signifies 3D rotation. The angle m in degrees is the rotation of the primary axis into the tertiary. For example, G17 G68 AB30 causes a 30 degree rotation of X coordinates into Z coordinates.

Sample program:

```
G31
P1=0 G69
N1 G0 X0 Y0
G1 Z0
G17
G68 AB[P1]
(part to rotate)
X1 Y2
G3 R1 AA0 AB45
G1 X3 Y5
Y6
G69
G31
P1=P1-5
IF P1 >= -180 GOTO 1
```

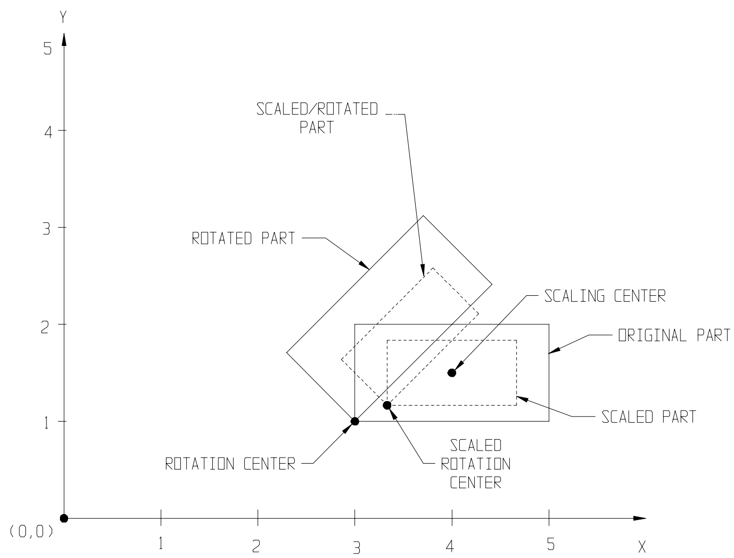
The program above will make the following part:



Note: Any AA in plane rotation is ignored. Cutter compensation and trig help are not fully supported in 3D rotation.

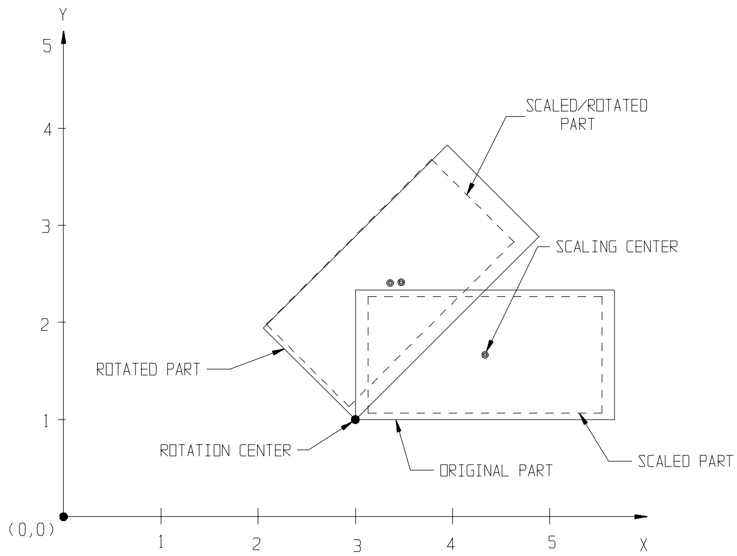
G69 Cancels all rotation, including 3D.

Part Scaled then Rotated



G51 I4 J1.5 X.7 Y.7
G68 I3 J1 AA45
X3 Y1
X5
Y2
X3
Y1
G69
G50

Part Rotated then Scaled



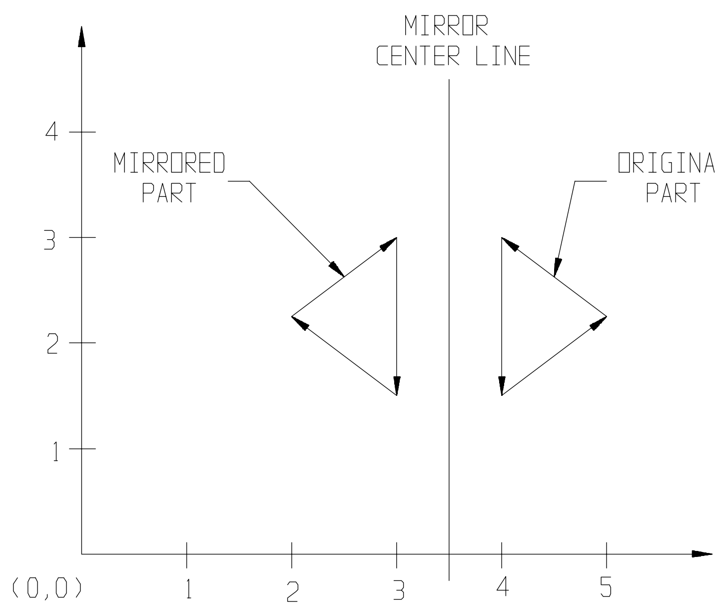
G68 I3 J1 AA45.00
G51 I4 J1.5 X.9 Y.9
X3 Y1
X5
Y2
X3
Y1
G50
G69

Cancel mirror image (G70)

Set mirror image (G71)

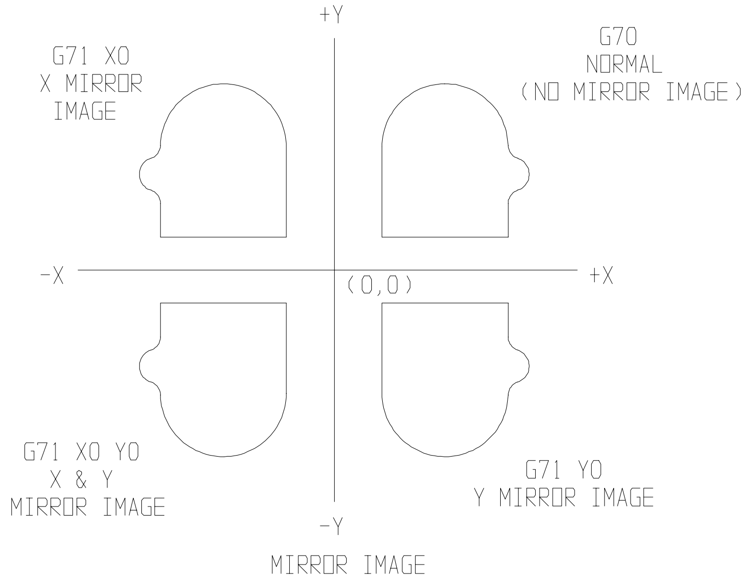
The mirror image commands allow mirroring about any centerline. The mirror image centerline is not affected by either scaling or rotation being on or off. Mirror image is shut off at the start of each program.

The command is as follows. G71 X____ Y____ Z____



X,Y, and Z specify the axes to mirror. Their values specify the distance from the current coordinate zero to create the mirror centerline. There must be at least one X, Y, or Z after the G71 command.

```
G71 X3.5
X4 Y1.5
X5 Y2.25
X4 Y3
Y1.5
G70
```



G70 cancels mirror image.

Mirroring in one axis will reverse climb cutting and conventional cutting. Mirroring an axis is similar to scaling by –1.

Canned cycles

A canned cycle simplifies a program by using a single block with a G code to specify the machining operations usually specified by several blocks.

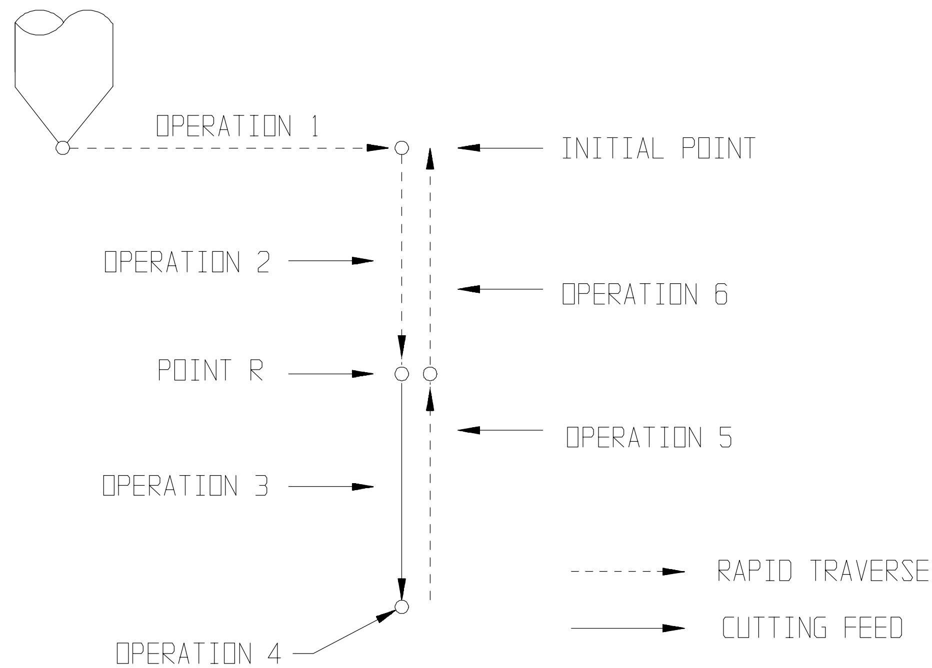
Canned Cycles

G code	Drilling -Z	Operation at Hole Bottom	Retraction +Z	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G74	Feed	Dwell → spindle CW	Feed	Left hand Tapping cycle
G75	Feed	-	Rapid	Counter bore
G76	Feed	Dwell →orient spindle → move in X Y	Rapid → moves in X Y	Fine Boring cycle
G77				Custom Drill Cycle
G78	Feed	Dwell → stop spindle → handwheel →	Rapid	Manual Boring cycle
G80	-	-	-	Cancel
G81	Feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid	Peck drilling cycle
G84	Feed	Dwell → spindle CCW	Feed	Tapping cycle
G85	Feed	-	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid	Fast Boring cycle
G87	Feed	Move in XY	Feed Rapid	Back Boring cycle
G88	Feed	Dwell	Feed	Hard tap
G89	Feed	Dwell	Feed	Boring cycle

Generally, a canned cycle consists of a sequence of six operations as shown below.

Operation 1: Positioning of axes X and Y (or 4th and 5th if enabled)
Operation 2: Rapid traverse to point R
Operation 3: Hole machining
Operation 4: Operation at the bottom of a hole
Operation 5: Retraction to point R
Operation 6: Rapid traverse up to the initial point

Canned Cycle Operation



Positioning is normally performed on the XY plane, and hole machining is performed with the Z axis. Positioning and hole machining must use this plane and axes combination. Canned cycles can be used in any plane.

Canned cycle operations consist of three basic modes which are specified by particular modal G codes as shown below.

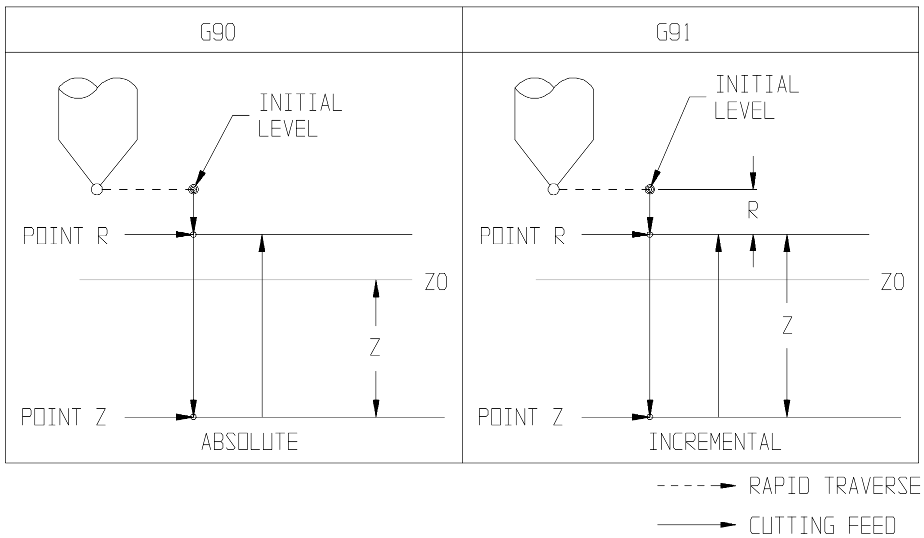
(1) Data format	G90 Absolute G91 Incremental
(2) Return point level	G98 Initial point level G99 R point level
(3) Drilling mode	G73 G80 G81 . .

.
G89

Note: The initial level means the value of the Z axis when the canned cycle is first turned on.

The figure below shows how to specify data in G90 or G91 mode.

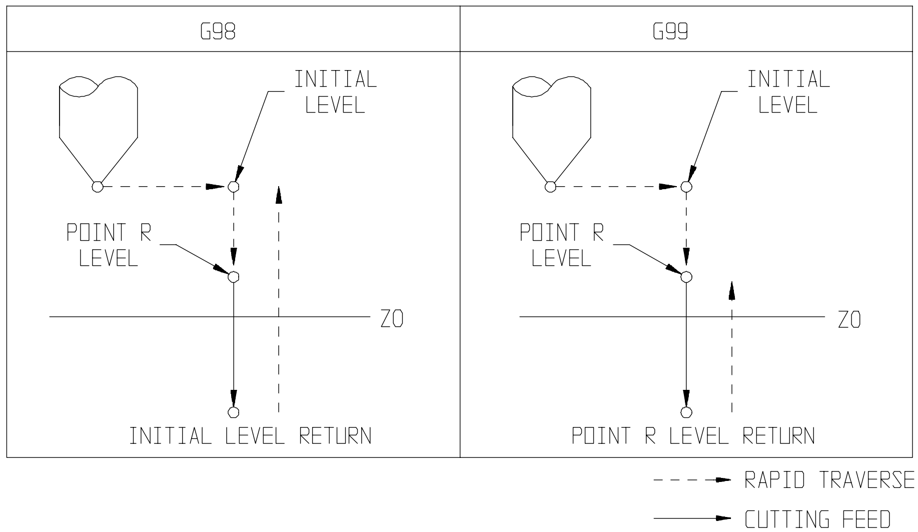
Absolute and Incremental Programming



If the tool is to be returned to point R or to the initial level, it is specified by G98 or G99. (See figure below.) Use G99 for the first hole, and use G98 for the last hole. When the canned cycle is repeated in G98 mode, the tool is returned to the initial level after each hole.

In the G99 mode the initial level does not change, and the tool is returned to the R point after each hole.

Initial Level and Point R Level



The drilling data is specified following G73,G74,G76,G77,G78, G81 to G89. Data is stored in the control as modal values and is retained for future use in other cycles.

The machining data in a canned cycle is specified as shown below.

G __ __	X__ Y__	Z__ R__ V__ Q__ P__ F__
Drilling Mode	Hole Position Data	Drilling Data

Drilling mode . . .	G__ __See canned cycle table.
Hole position data	X Specifies hole position by an incremental or absolute Y value. The path and feedrate are the same as G00 positioning.
Drilling data	Z . . . Specifies the distance from point R to the bottom of the hole with an incremental value or the position of the hole bottom with an absolute value. R . . . Specifies the distance from the initial level to the point R with an incremental value or the position of point R with an absolute value. V . . . Specifies the first Z depth in G73 and G83. Q . . . Specifies the increment value for G73 and G83. P . . . Specifies the dwell time at the bottom of the hole. The relationship between the time and the specified value is the same as for G04. F . . . Specifies the feedrate. B . . . Specifies the dwell time before spindle reverse in soft left or right tapping.

For drilling cycles you may use:

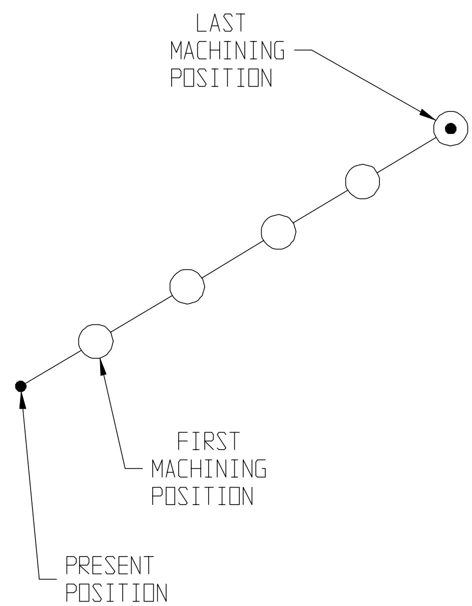
P140	for	Clearance plane
P141	for	Final Z depth
P142	for	Z initial level
P143	for	Z increment
P144	for	1st Z depth
P145	for	Z feedrate
P146	for	Peck up increment
P147	for	Peck clearance
P148	for	Dwell before spindle reverses in tap cycles
P149	for	Dwell

The drilling mode (G__ __) remains unchanged until another drilling mode is specified or the canned cycle is canceled with a G80. Once the drilling data has been specified in a canned cycle, it is retained until it is changed. All required drilling data needs to be specified when the canned cycle is started; only data to be changed needs to be specified during the cycle.

Canned cycles are canceled at the beginning of each program.

G73 through G78, G81 to G89 without an axis move will not drill a hole. The G codes turns on the drilling cycle only.

Equally spaced holes can be programmed by use of the L address.



G81 R___ Z___ F___
G91 X___ Y___ L___

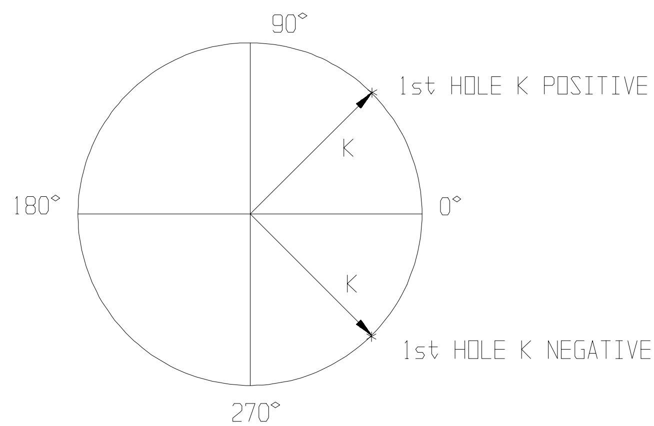
X___ Y____ specifies the first and subsequent hole positions in the incremental mode (G91). In the absolute mode (G90), a hole would be repeatedly drilled at the same position. K can be used instead of L to specify the number of holes.

Bolt hole routine (G72)

The bolt circle autoroutine can be used with any of the drilling cycles. Drilling cycles, when used with this autoroutine, differ in that hole positions are not specified. The G72 line indirectly specifies all the hole positions based on specific input: number of holes in 360°, number of holes to be drilled, the radius of the bolt circle, the starting angle of the first hole, and the center of the bolt circle. The control will then calculate the position of each hole and rapid to each hole in straight line moves. The angle of the first hole is the angle from the 3 o'clock position. A positive starting angle is counterclockwise from 3 o'clock. A negative starting angle is clockwise from 3 o'clock.

Note: Holes will be drilled clockwise from the first position if the bolt hole radius is positive. Holes will be drilled counterclockwise if the bolt hole radius is negative.

Definition of Starting Angle



The following is a format of the G72 command.

G72 X___ Y___	R___	Q___	P___	K___
position of bolt hole center	radius of bolt circle	# of holes in 360°	# of holes to be drilled	angle of first hole

Program to Drill a 5 Hole 1" Radius Bolt Circle

N1	G20 G90 (Inch/Absolute)
N2	S1000 M3 G43 H1 (spindle CW 1000 RPM, activates tool #1's length offsets)
N3	G81 G99 Z-1 R.1 F10

G81	Drill
G99	Return to R point
Z-1	Drill depth
R.1	R plane
F10	Z feedrate

N4 G72 X0 Y0 R1 Q5 P5 K0

G72	Bolt hole routine
X0 Y0	Center at X0, Y0
R1	Radius of 1 inch
Q5	5 holes in 360°
P5	Drill 5 holes
K0	Start angle 0°

Notes on Bolt Hole Routines

Note 1: If P is less than Q a partial bolt circle will be drilled.

Note 2 To go counterclockwise use a negative radius.

Note 3: If a move to the center of the bolt hole pattern is not desired the following scheme may be used.

```
G81 G99 Z-1 R.1 F10
G65 X0 Y0
G72 R1 Q5 P5 K0
```

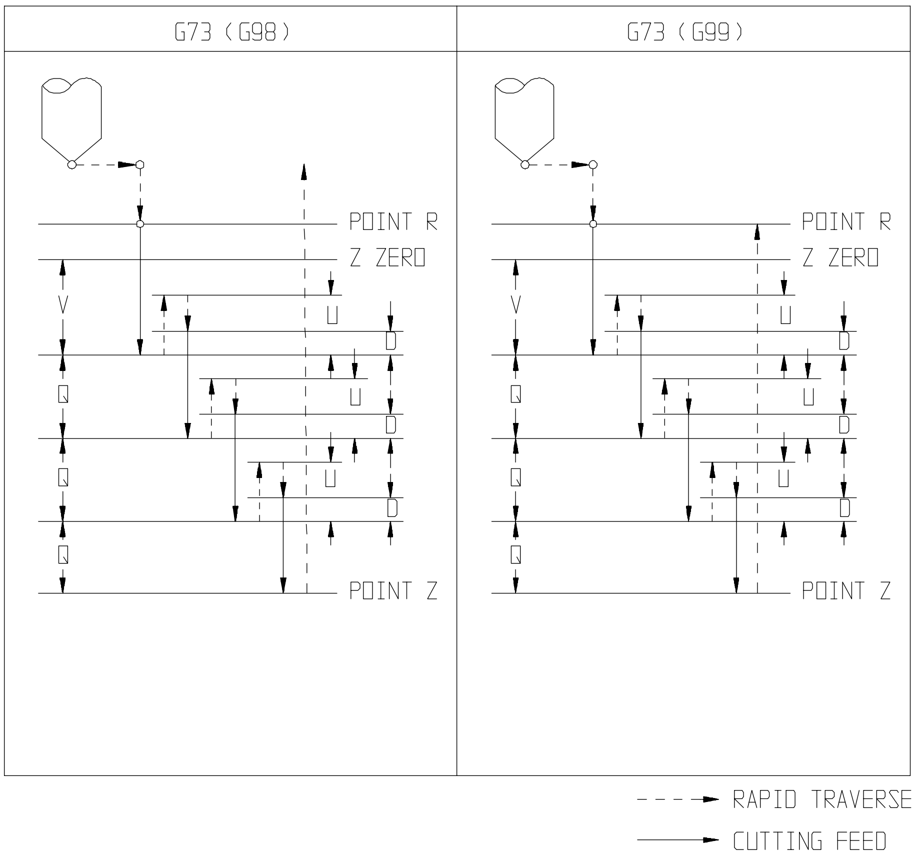
Note 4: The G65 cannot be on the G72 block because there is also a P on the block that will cause a program call to program #5.

You may also use:

G81 G99 Z-1 2.1 F10
P156=1 (Bolt hole radius)
P157=45 (Bolt hole start angle)
P158=5 (# of holes in 360°)
P159=5 (# of holes to be made)
G72 G65 X0 Y0 (Bolt hole center)

Note 5: The initial Z level corresponds to where Z is when the drilling code (G73 through G78,G81 to G89) is executed.

High speed peck drilling cycle (G73)



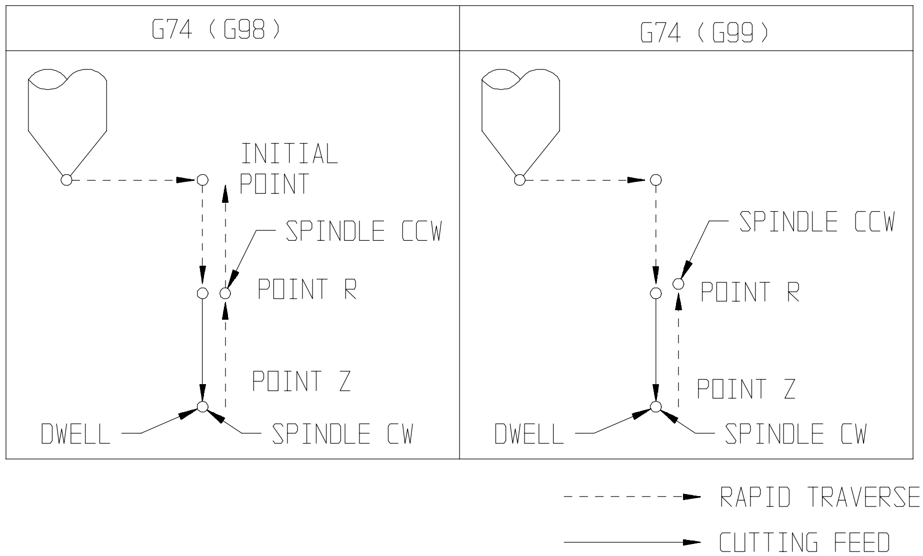
G73 G98/G99 Z___ R___ V___ Q___ U___ D___ F___

The G73 command specifies the high speed peck cycle. This cycle will do the following.

1. Rapids to point R
2. Feeds down to point V
3. Rapids up U value
4. Rapids down to D value
5. Feeds down by Q value or Z point (whichever is less)
6. Repeat steps 3-5 until point Z is reached
7. Rapids to initial point/point R as determined by G98/G99

Note: The V command is optional. If left out, the first depth would equal R___ - Q___.

Left hand soft tapping cycle (G74)



G74 G98/G99 Z___ R___ B___ P___ F___

The G74 command specifies the left hand soft tapping cycle. At each axis position, this cycle will do the following.

1. Rapid to point R
2. Feeds to point Z
3. Dwells before reversing (specified by the B code)
4. Reverses spindle (CW)
5. Dwells after reversing (specified by the P code)
6. Feeds to point R
7. Reverses spindle (CCW)
8. Rapids to initial point, if specified by the G98 code

Note: During tapping, the feedrate override and spindle override switches are ignored. When in the block mode, the cycle does not stop until the tap is completed. When feedhold or halt is applied, the tap will stop when it is out of the hole.

Calculating Feeds and Speeds for Tapping

$$\frac{1}{\text{PITCH}} = \text{LEAD}$$

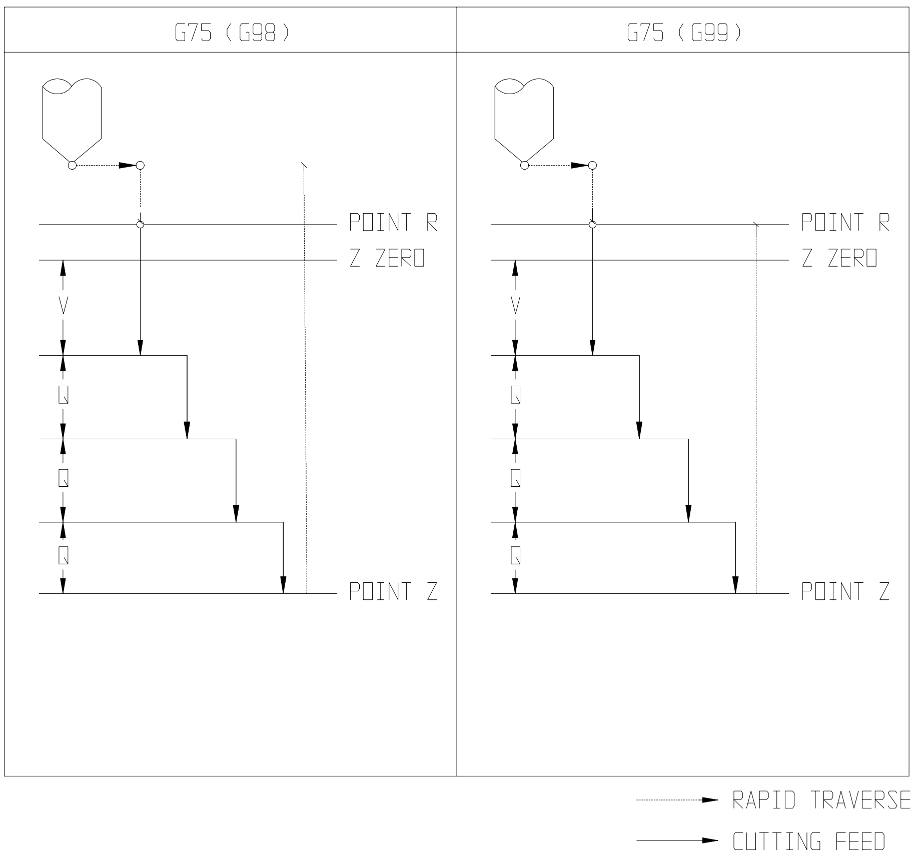
RPM X LEAD = FEEDRATE

Example: 1/4-20 tap, spindle rpm 400
1/20 = .05 (lead)
400 x .05 = 20 (feedrate)

Feedrate may need adjustment for proper operation of tap holder. If tap is pulled too far in the holder, feedrate should be increased. If tap is pushed into the holder, feedrate should be decreased.

Counter Bore (G75)

The counter bore is identical to the circular finish inside cycle (G25). It always does a climb cut (counter clockwise). It always does a plunge to the center (not a ramp). It can be used in bolt hole patterns or any other drilling patterns.



F__ (XY Feed Rate)
P150=__ Counter Bore Radius
P503=__ O is cutter comp off, 1 is cutter comp on
G75 G98/G99 Z__ R__ V__ Q__ F__

The G75 command specifies the counter bore cycle. The cycle will do the following.

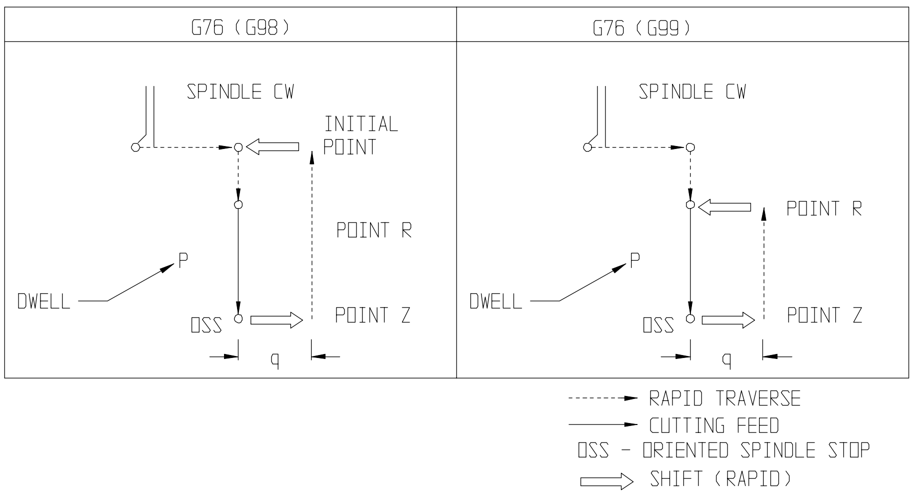
1. Rapids to point R.

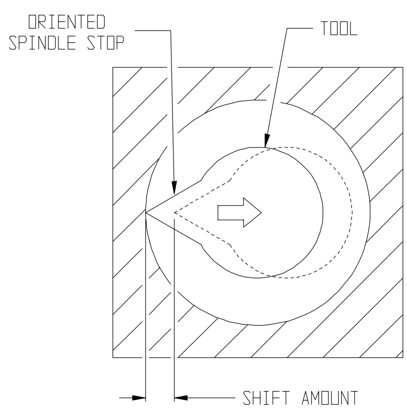
2. Feeds down to point V.
3. Counter bores the hole (to radius P150).
4. Feeds down by Q value or Z point (whichever is less).
5. Repeats steps 3-4 until point Z is reached.
6. Rapids to initial point / point R as determined by G98/G99.

Note 1: The V command is optional. If left out, the 1st depth would equal R__ - Q__

Note 2: It is possible to do a conventional cut (clockwise) by using mirror image.

Fine bore cycle (G76)





The distance and angle is specified by control parameters bore relief angle and bore relief distance.

G76 F____ P____ R____ Z____ G98/G99

The G76 command specifies the fine bore drilling cycle. At each axis position, this cycle will execute the following.

1. Rapids to point R
2. Feeds down to point Z
3. Dwells by P seconds at the bottom
4. Orients the spindle
5. Moves in XY (distance and direction specified by CTRL parameters 'Bore Relief Angle' and 'Bore Relief Distance')
6. Rapids out of hole to point R/initial point, specified by G98/G99
7. Moves back to original XY position

Custom drill cycle (G77)

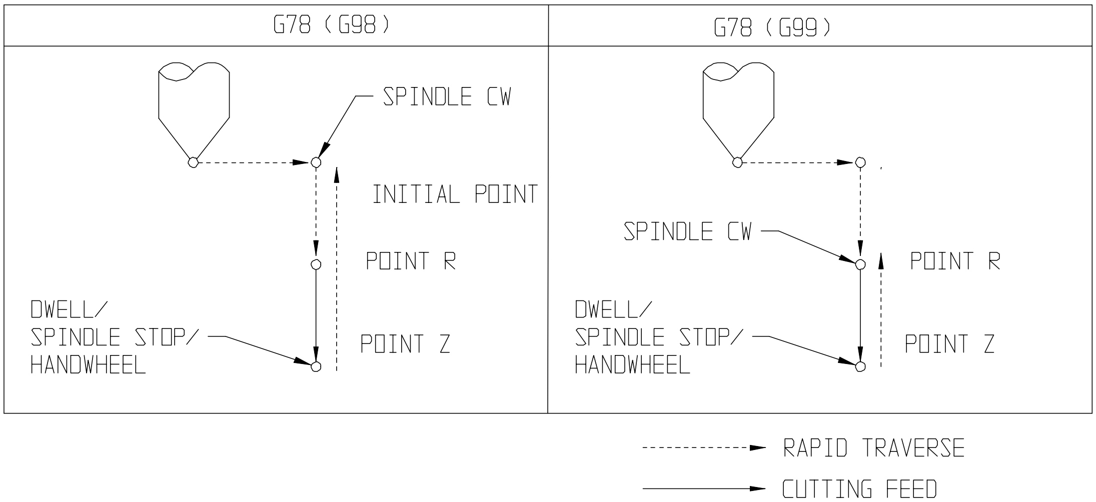
This is a custom drill cycle. It can be used to perform a user-defined cycle. The cycle must be defined by a custom G code, G79.

Example:

```
G77 R.1 Z-2
X1 Y2 (does what is in G79)
Y-2   (does what is in G79)
 .
 .
 .
X3   (does what is in G79)
G80
```

Custom drill cycle can be used in bolt hole cycles, grid of holes, or spaced holes. If G79 is not a custom code the error 549, unrecognized G code, will occur.

Drilling cycle, manual bore (G78)



G78 F____ P____ R____ Z____ G98/G99

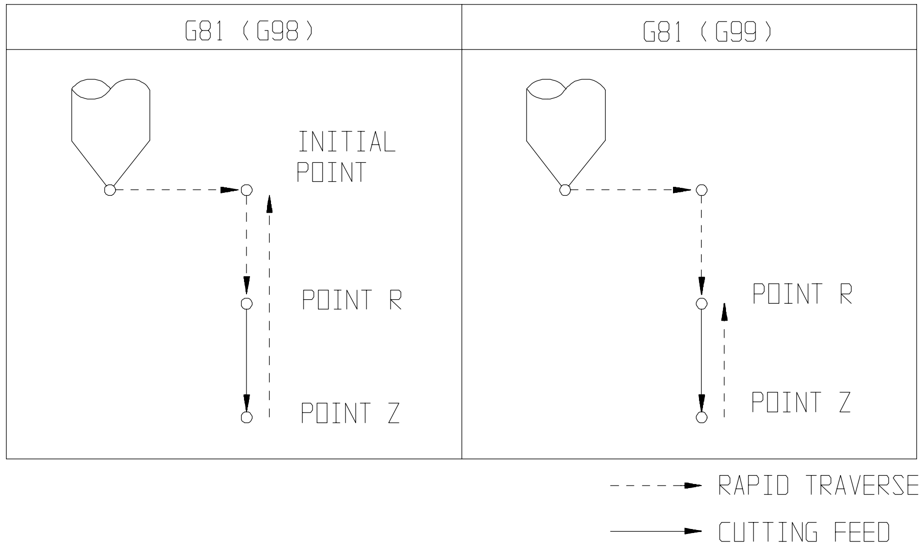
The G78 command specifies the manual bore drilling cycle. At each following axis position, this cycle will do the following.

1. Rapids to point R
2. Feeds down to point Z
3. Dwells P seconds
4. Stops the spindle
5. Enters the handwheel mode (user can handwheel the axes, turn spindle on/off, remove tool, etc.)
6. Exits handwheel mode by pressing Enter or ESC
7. Rapids out of hole to point R/initial point

Canned cycle cancel (G80)

The canned cycle (G73 through G78, G81 to G89) is canceled and normal operation is subsequently performed.

Drilling cycle (G81)

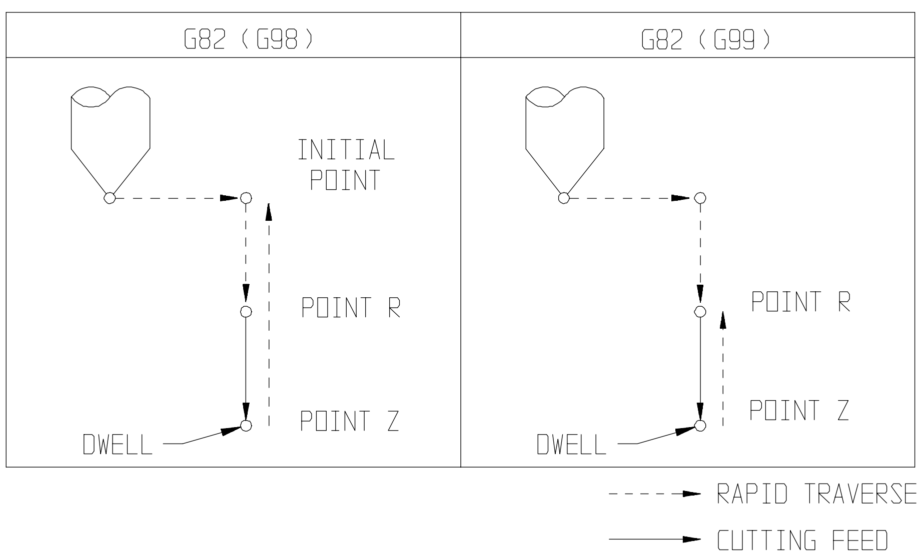


G81 G98/G99 Z___ R___ F___

The G81 command specifies the drilling cycle. This cycle will do the following.

1. Rapids to point R
2. Feeds down to point Z
3. Rapids to initial point/point R as determined by G98/G99

Drill/Dwell cycle (G82)

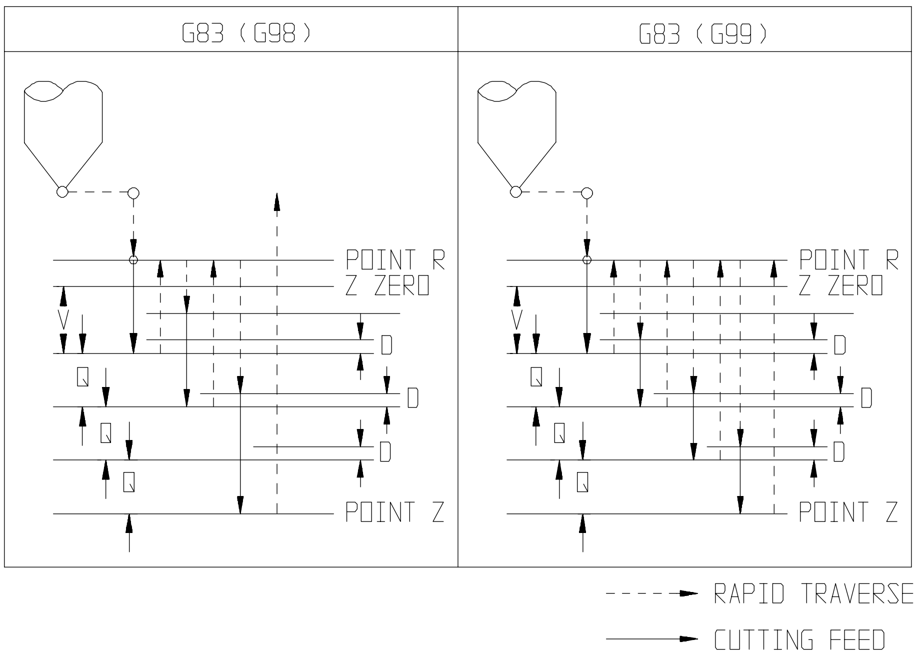


G82 G98/G99 Z___ R___ P___ F___

The G82 command is similar to the G81 command; however, a dwell (specified by the P command) is performed at the bottom of the hole. This cycle will do the following.

1. Rapids to point R
2. Feeds down to point Z
3. Dwells by P___ seconds
4. Rapids to initial point/point R as determined by G98/G99

Peck drilling cycle (G83)



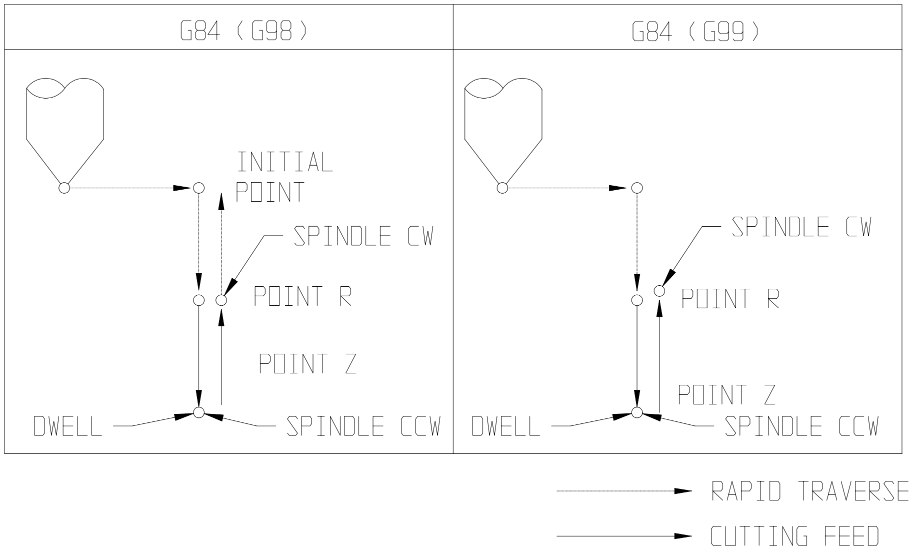
G83 G98/G99 Z___ R___ V___ Q___ D___ F___

The G83 command specifies the peck drill cycle. This cycle will do the following.

1. Rapids to point R
2. Feeds to Point V
3. Rapids up to point R
4. Rapids down to D value
5. Feeds down by Q value or Z point (whichever is less)
6. Repeats steps 3-5 until point Z is reached
7. Rapids to initial point/point R as determined by G98/G99

Note: The V command is optional; if left out, the first depth would equal R___ - Q___.

Right-hand soft tapping cycle (G84)



G84 G98/G99 Z___ R___ B___ P___ F___

The G84 command specifies the right-hand tapping cycle. At each axis position this cycle will do the following.

1. Rapids to point R
2. Feeds to point Z
3. Dwells before reversing (specified by the B command)
4. Reverses spindle (CCW)
5. Dwells after reversing (specified by the P command)
6. Feeds to point R
7. Reverses spindle (CW)
8. Rapids to initial point, if specified by the G98 code

Note: During tapping the feedrate override and spindle override switches are ignored. Once the tap cycle is started, it does not stop until the tap is completed if block mode is applied. If feedhold or halt is applied, the tap will stop when it is out of the hole.

Tapping Feeds and Speeds

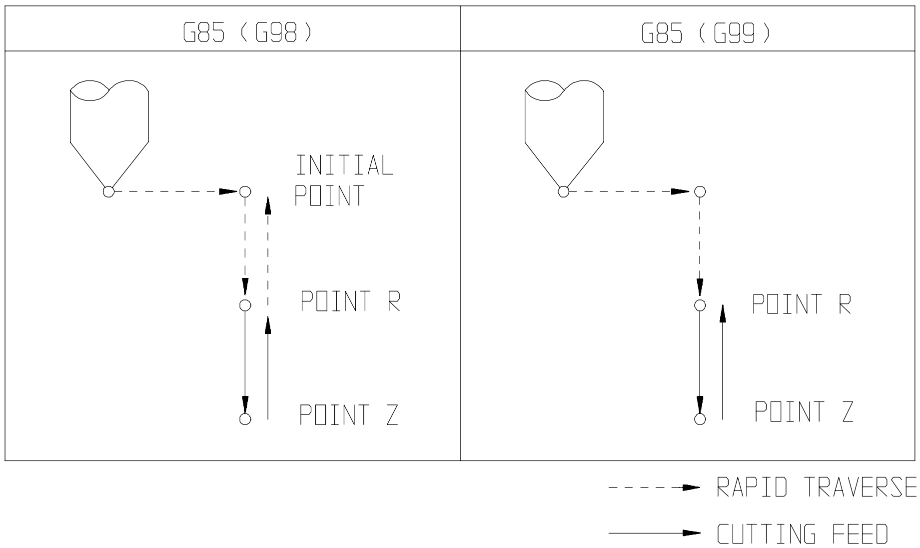
$$\frac{1}{\text{PITCH}} = \text{LEAD}$$

RPM X LEAD = FEEDRATE

Example: 1/4-20 tap, spindle rpm 400
1/20 = .05 (lead)
400 x .05 = 20 (feedrate)

Feedrate may need adjustment for proper operation of the tap holder. If the tap is pulled too far in the holder, feedrate should be increased. If the tap is pushed into the holder, feedrate should be decreased.

Boring cycle (G85)

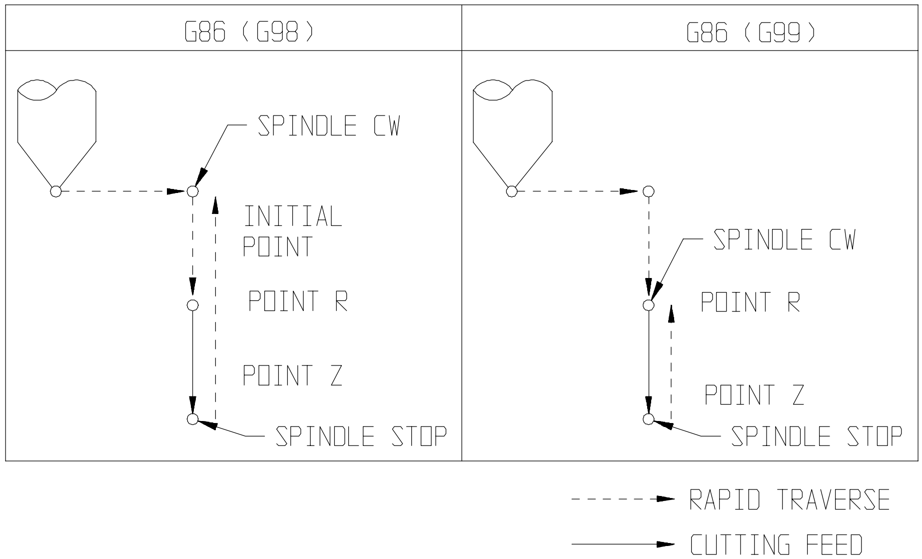


G85 G98/G99 Z___ R____ F___

The G85 command specifies the boring cycle. At each axis position this cycle will do the following.

1. Rapids to point R
2. Feeds to point Z
3. Feeds to point R
4. Rapids to initial point if specified by the G98 code

Fast bore cycle (G86)

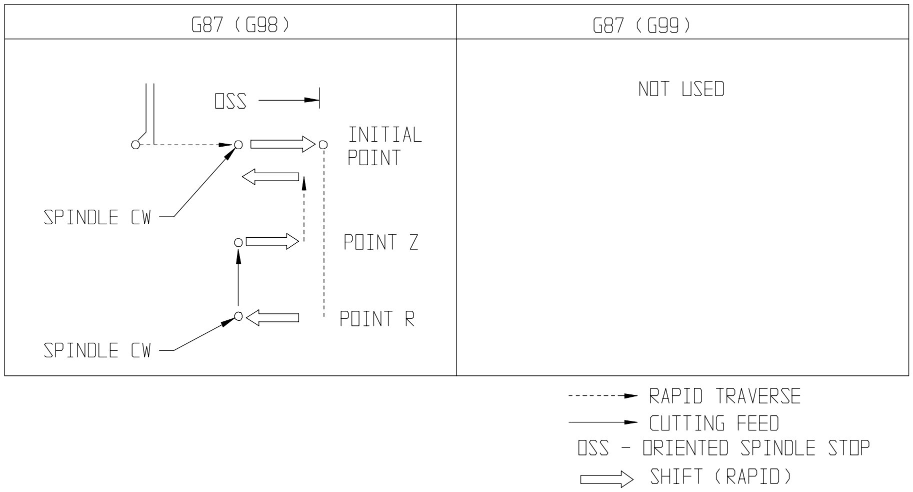


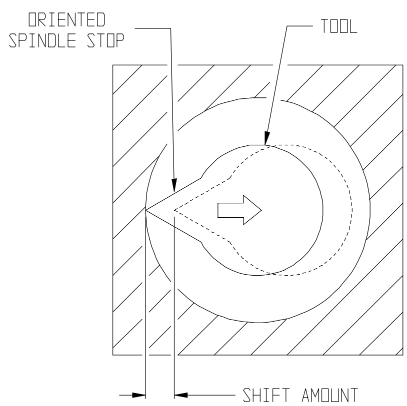
G86 G98/G99 Z___ R___ F___

The G86 command specifies the fast bore cycle. At each axis position this cycle will do the following.

1. Spindle starts (CW)
2. Rapids to point R
3. Feeds down to point Z
4. Spindle stops
5. Rapids to point R
6. Rapids to initial point/point R as determined by G98/G99
7. Spindle Starts (CW)

Back Boring cycle (G87)





G87, G98 F____ R____ Z____

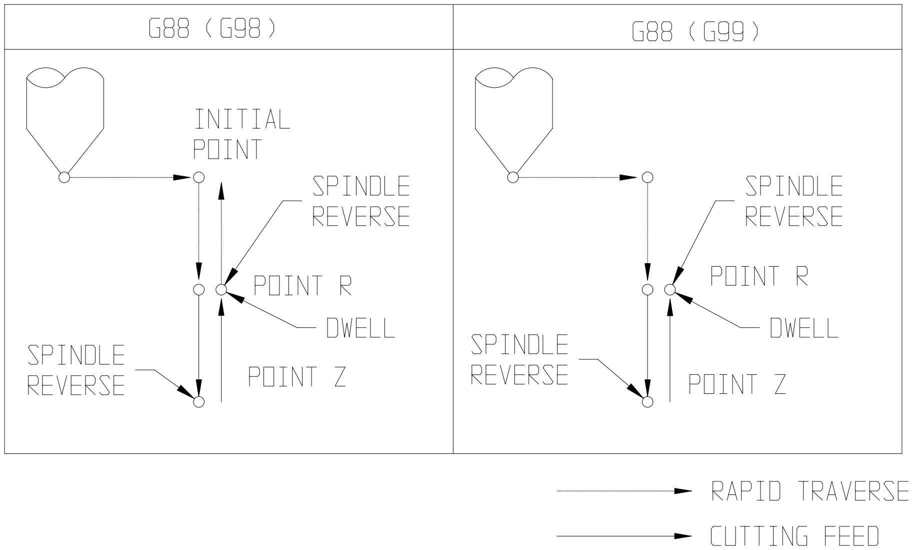
The distance and angle is specified by control parameters "Bore Relief Angle" and "Bore Relief Distance".

The G87 command specifies the back bore drilling cycle. At each axis position, this cycle will execute the following.

1. Orients the spindle
2. Rapids in XY (using the CTRL parameters used in the G76)
3. Rapids to point R (normally at the bottom of the hole)
4. Moves to original XY position
5. Starts the spindle
6. Feeds up to Z depth
7. Orients the spindle
8. Moves in XY (specified by Bore Relief Angle and Bore Relief Distance)
9. Rapids up to initial point

10. Moves back to original XY position
11. Restarts the spindle

Hard tap cycle (G88)



G88 G98/G99 Z____ R____ F____ P____ (Q____ V____)

The G88 command specifies the hard tapping cycle. P(dwell) can be used if the distance between holes is small to give the spindle time to reverse to its proper direction.
This cycle will do the following.

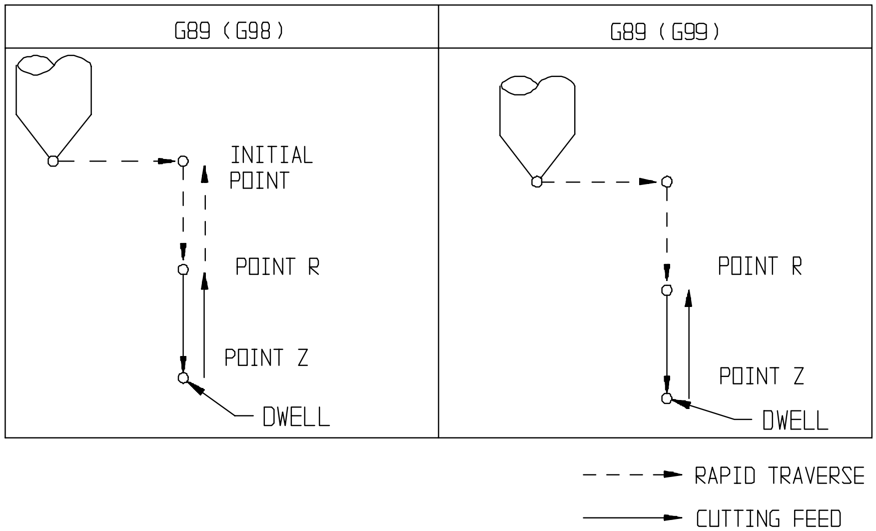
1. Rapids to point R
2. Feeds point Z
3. Reverses the spindle
4. Feeds to point R
5. Reverse the spindle
6. Dwells for P___ seconds
7. Rapids to initial point, if specified by the G98 code

Note 1: If the spindle is running clockwise when the G88 is initiated, right tapping is done. If the spindle is running counterclockwise when the G88 is initiated, left tapping is done.
Note 2: The tap cycle will not stop until the end of the tap cycle even if feedhold or block mode are applied. If halt is applied the tap reverses and stops when it is out of the hole.
Note 3: If Q__ is specified on the G88 block, peck tapping is preformed. The Q__ specifies the peck increment. The V__ specifies the first depth to tap.

F specifies the lead
Example: ¼ - 20 tap
1/20 = .05 lead
Use F.05

Hard tapping is an option supplied by the machine tool builder and should only be used if the option has been installed.

Bore/Dwell cycle (G89)



G89 G98/G99 Z___ P___ F___

The G89 command specifies the bore with dwell cycle. At each following axis position this cycle will do the following.

1. Rapid to point R
2. Feed to point Z
3. Dwell at bottom (specified by P code)
4. Feed to point R
5. Rapid to initial point, if specified by the G98 code

Sample program to drill holes in the YZ plane.

```
G19
G81 R.1 X-1 F20 (X depth is -1)
Y3 Z2 (drills a hole at Y3 Z2)
Z5 (drills a hole at Y3 Z5)
Y-1 (drills a hole at Y-1 Z5)
G80
```

Notes on Canned Cycle Specifications

Note 1: The spindle must be turned on by M code, M3 or M4, before the canned cycle is specified.

M3 Spindle CW
.
.
.
G __ __ ......Correct

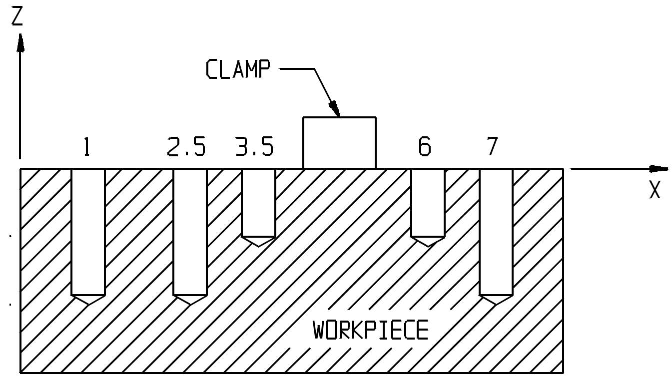
M5 Spindle Stop
.
.
.
G __ __ ......Incorrect (M3 or M4 must be specified before this block.)

Note 2: If the block contains an X and/or Y move, drilling is performed in canned cycle mode. If the block does not contain an X and/or Y move, drilling is not performed. However, when "G4 X___" is specified, drilling is not performed even if X is specified.

Note 3: If a block contains a Z position by itself, drilling will not be performed. However, the Z axis will rapid to this point. This can be used to manipulate a tool up and over obstructions without disabling the canned cycle.

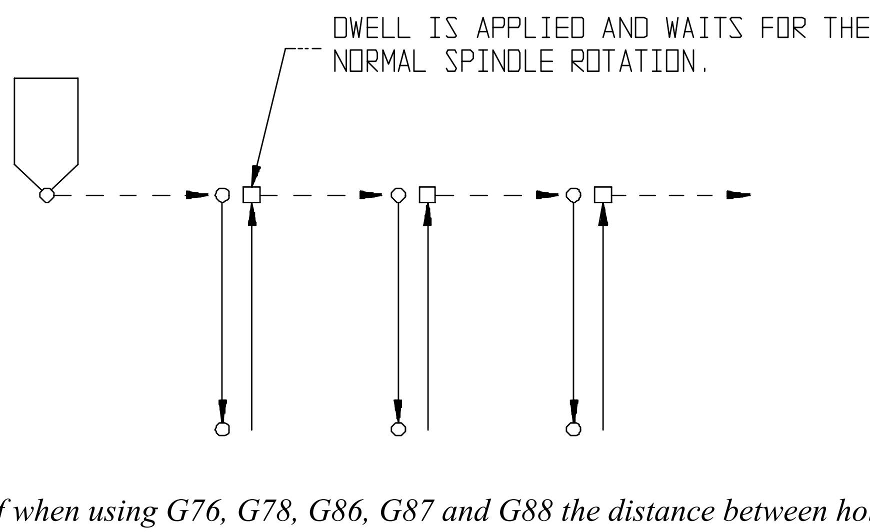
G00 X____
G81X____ Y____ Z____ R____ F____
F (Drilling is not performed. X Y feedrate is updated.)
M (Drilling is not performed. Only the miscellaneous function is executed.)
G4 P (Drilling is not performed. Drilling data P is not changed by G04 P__.)

Note 4: Specify drilling data in the block where drilling is performed. Entries (V, Q, B, Z, R, F, or P) are stored as modal data.



Drill Example:

G90
G81 G0 R.1 Z-2 F10 (drill clearance .1, depth -2, Z feed 10)
X1 (drill hole -2 deep at X1)
X2.5 F5 (drill hole -2 deep at X2.5, Z feed 5)
X3.5 Z-1 (drill hole at X3.5 -1 deep)
Z2 (rapid Z to +2 to clear clamp)
X6 (drill hole at X6)
X7 Z-2 (Drill hole at X7 -2 deep)
G80 (cancels drill cycle)



Note 5: If when using G76, G78, G86, G87 and G88 the distance between holes is tool small, a dwell (G04) needs to be inserted between moves to give the spindle time to reverse directions.

G00 M___
G86 X___ Y___ Z___ R___ F___
G04 P___ (Dwell is performed, but drilling is not.)
X__ Y___
G04 P___ (Dwell is performed, but drilling is not.)
X__ Y___
G04 P___ (Dwell is performed, but drilling is not.)
. .
. .
. .
This may not have to be considered if spindle up-to-speed is available on the machine tool.

Note 6: Operator Precautions

a) Single block
When a canned cycle is performed in the single block mode, the control stops at the end of Operations 1, 2, and 6. Therefore it must be started at least three times to drill one hole.

b) Override
The feedrate override and spindle override are assumed to be 100% during the operation of canned cycles G74 and G84.

Absolute/Incremental Mode

There are two modes of specifying moves: G90 (absolute) specifies a fixed location, and G91 (incremental) specifies a distance from the current location of the tool.

Absolute mode (modal) (G90)

This function causes the control to go into its normal absolute operating mode. In this mode, all dimensions are referenced from a single reference point. This reference can either be the home zero point, which is a fixed point on the machine, or an operator defined work coordinate point.

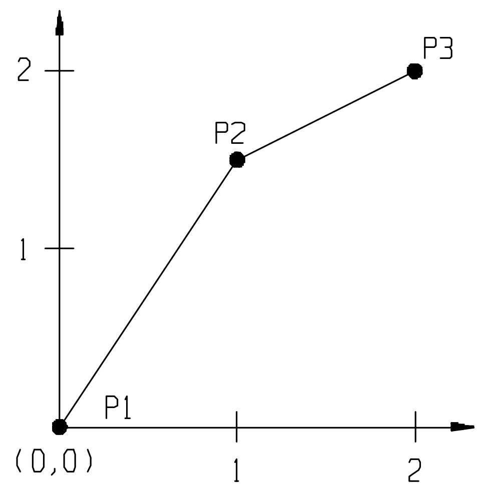
Because of the table/saddle limitation, X Y Z dimensions relative to the home zero can only be negative on standard mills built by Milltronics. Dimensions relative to work coordinates can be either negative or positive, depending on where the operator sets the zero coordinate.

G90 is active at the beginning of each program. G90 cancels G91.

Note: The machine defaults to G90 (absolute) when starting a program.

Absolute Positioning

G90	X0 Y0	P1
	X1 Y1.5	P2
	X2 Y2	P3



Incremental mode (modal) (G91)

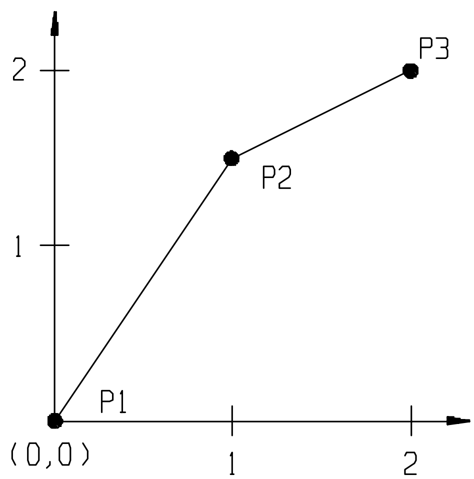
This function causes the control to go into the incremental mode. In this mode all dimensions are entered relative to the machine position in the previous block. In the case of MDI, the dimensions are relative to the current machine position. Dimensions in G91 can be either positive or negative. Care should be taken when using G91. Whenever activating tool offsets, R-plane dimensions, or setting floating zeroes via G92, the control should not be in the G91 mode.

XC, YC, ZC (centers), AA (start angle) and AB (end angle) are always an absolute.

G91 cancels G90.

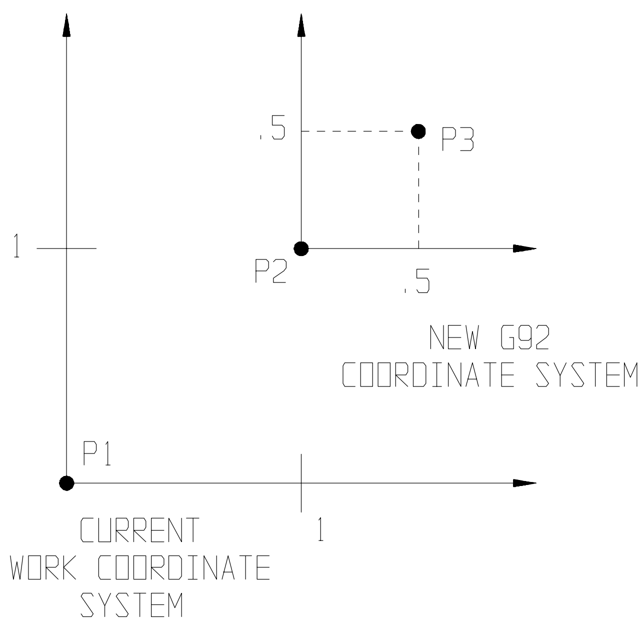
Incremental Positioning

G90 X0 Y0	P1
G91 X1 Y1.5	P2
X1 Y.5	P3



Floating zero (G92)

This command establishes the work coordinate system. The position of the tool becomes the programmed position in the current work coordinate system. When using this G92 command, think of it as "call this position" X_ Y_ Z_.



If the machine is positioned at P2, which is a command of X1 Y1, and then a G92 X0 Y0 is commanded, the next time X.5 Y.5 is commanded the machine will position to P3. If the machine is positioned at P1 and G92 X-1 Y-1 is commanded, the next time X.5 Y.5 is commanded the machine will position to P3.

When using G92's for calling subprograms, the G92 is saved prior to calling the subprogram and restored when returning from the subprogram.

Example:

X0 Y0	O0002
Call 2 (cut at 0,0)	G92 X0 Y0
X5 Y0	X__ Y__
Call 2 (cut at 5,0)	X__ Y__
X5 Y5	X__ Y__
Call 2 (cut at 5,5)	M99 (return)

Using this convention, you can cut the same shape in several locations.

See notation on M98 (page 317) for additional information on calling subprograms with G92's.

<u>*Notes on Floating Zero*</u>

Note 1: When using a G92 Z__, tool length offsets should be canceled or accounted for as they will affect the new floating zero position on the Z axis.

Note 2: G92 should not be used when cutter compensation is active; the control should always be in G40 mode.

Note 3: The distance shifted via a G92 in one work coordinate system will be applied to other work coordinate systems when they are activated via G54 - G59 commands. If this is not desirable, a new G92 must be set when changing coordinate systems. G92 offsets are zeroed on power-up and after homing the machine.

Note 4: G92 X__ Y__ can be thought of as call this position X__ Y__.

Note 5: On power-up the G54 coordinate system is active.

Note 6: G92 are restored to there initial values after a program ends.

Inverse Time Feed Mode (G93)

G93 is a modal G code. If in G93, inverse time mode feedrates are specified in units of either inverse seconds (1/sec) or inverse minutes (1/min). The two units used depend upon the miscellaneous parameter that set the units for inverse time mode.

When in inverse time mode, the amount of time a move block takes is the inverse of the feedrate regardless of the distance of the move. The two exceptions to this are 1) rapid moves and 2) moves that will slow down if any axis in the move goes faster than the maximum feed parameter for that particular axis. While in inverse time mode, the feedrate must be specified in every move block, or an error 611 will be reported.

Example:

G93
X-10Y-2.4A-3F.25 (Assuming feedrate units of 1/sec, the move will take 1/.25 = 4 seconds regardless of where the machine is moving from.)
X-5 (This line generates an error 611 because no feedrate is specified.)

Note: Rapid moves still move at the rapid feedrate.

Feed Per Minute (G94)

This G code is a modal G code that instructs the control to interpret feed commands as inches/minute or mm/minute for linear moves, degrees/minute for rotary moves, and inches/minute or mm/minute for a combination of linear and rotary moves. When a combination of linear and rotary moves is programmed, the rotary moves match the time it takes to make the linear moves. The only exception is if the time to make the linear move is especially short, the rotary axis would try to move faster than it is capable. In this case, the move is slowed using the length of the rotary move and the maximum feedrate for the rotary axis. The control always powers up in G94 feed per minute.

Examples:

F100 X12	Starts at (0,0,0) and makes a total linear move of 12 inches and the move takes 12 x 60/100 = 7.2 seconds.
F28 X1Y3Z4	Starts at (2,5,1) and makes a total linear move of 3.74 inches and the move takes 3.74 x 60/28 = 8.02 seconds.
F600 A180	Starts at (0,0,0,0,0) and makes a total rotary move of 180° and the move takes 180 x 60/600 = 18 seconds.
F250 A200 B100	Starts at (0,0,0,120,300) and makes a total rotary move of 215.41° and the move takes 215.41x 60/250 = 51.70 seconds.
F100 X12 A180	Starts at (0,0,0,0,0) and makes a linear move of 12 inches and the move takes 12 x 60/100 = 7.2 seconds.
F28 X1 Y3 Z4 A30 B120	Starts at (2,5,1,0, -333) and makes a linear move of 3.74 inches and the move takes 3.74 x 60/28 = 8.02 seconds.
F10 X-1 A1800	Starts at (0,0,0,0,0) and, since the rotary axis would have to try to move faster than it is capable, the control uses the parameters for the rotary axis to make a move of 1800° taking 1800 x 60/5000 =

21.6 seconds, where 5000 is the maximum feedrate for rotary axis A.

Feed Per Revolution (G95)

This G code is a modal G code that instructs the control to interpret feed commands as mm or inches per revolution of the spindle. G1 F.005 would cause the axis to advance .005" for every revolution of the spindle.

*Note: The machine **must have the hard tapping option** to use this G code.*

Return to initial level or to R level (G98/G99)

These two G codes are only used when the control is in one of the Z axis canned cycles (G73 thru G89) or autoroutines (G24-26, G34-36). A G98 will cause a canned cycle to return the Z axis to the same level it was at when the cycle was activated. A G99 will cause a canned cycle to return the Z axis to the current R level.

```
G98
X1 Y1 Z1
G81 X5 Y-4 Z-1.3 R.2 F10    X5 Y-4     then Z.2 then Z-1.3 then Z1
X2 Y3                                  X2 Y3 then Z.2 then Z-1.3 then Z1
G99 X3 Y-1                             X3 Y-1 then Z.2 then Z-1.3 then Z.2
G80                                    cancel cycle
```

A discussion follows on several specialized and non-standard G codes.

G271 (Pocket Clear)

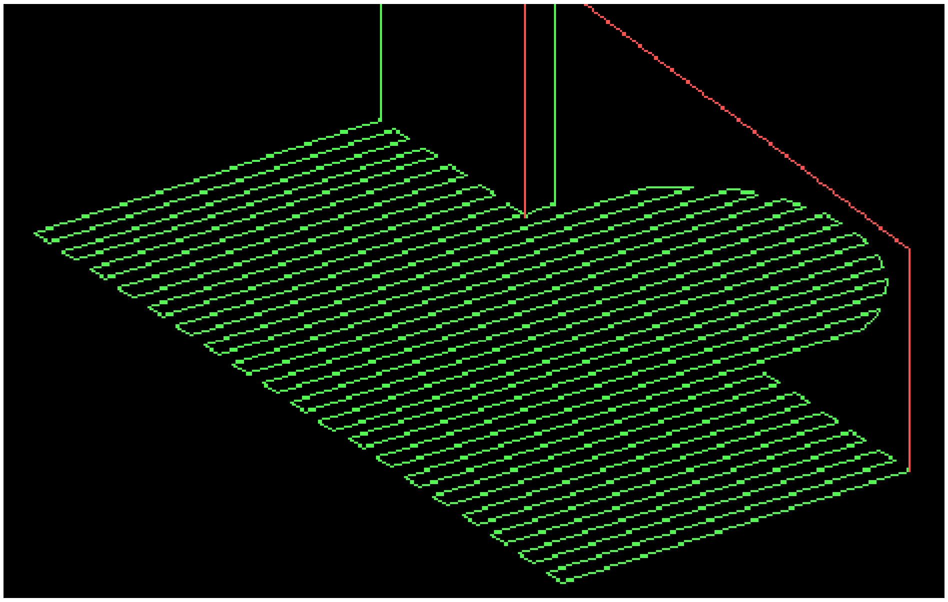
The cycle works by generating the path with cutter comp using the tool radius plus finish stock. Lines are spaced the cut with apart inside the path.

Sample program:

```
P145=10 (Z Feederate)
F25 (XY Feed-rate to clear the pocket)
G271 P1234 Q1235 R.1 Z-1 D.1 I.05
('R' is the R plane, 'Z' is the Z-depth, 'D' is the Cut Width and 'I'
is the Finish Stock)
N1234 (The 'P' specifies the start of the pocket)
G41
G65 X0 Y1
X0 Y0
X2
Y1
G3 R1 AA-90 AB90
G1 Y4
X0
Y0
```

G65 X1 Y0
G40
N1235 (The 'Q' specifies the end of the pocket)

This program will clear a pocket that looks like this.



The Cut Width is the distance between 1 pass and the next.

The Finish stock is not removed with a final pass. To remove the stock another mill cycle needs to be programmed. The Finish Stock is not an option if the cutter comp is off.

Note 1 : The 'I' value requires that cutter comp is on.
If cutter comp is on in the wrong direction (outside instead of inside) a bigger pocket will be cleared.

Note 2 : If the end point is not the same as the start point a line will be added from the end point to the start point.

Note 3 : Feedrates and spindle speeds in the pocket geometry are ignored during the clearing cycle.

Note 4 : If path has geometry that crosses other geometry the cycle does it's best to clear the pocket, but will more then likely not do a satisfactory job.

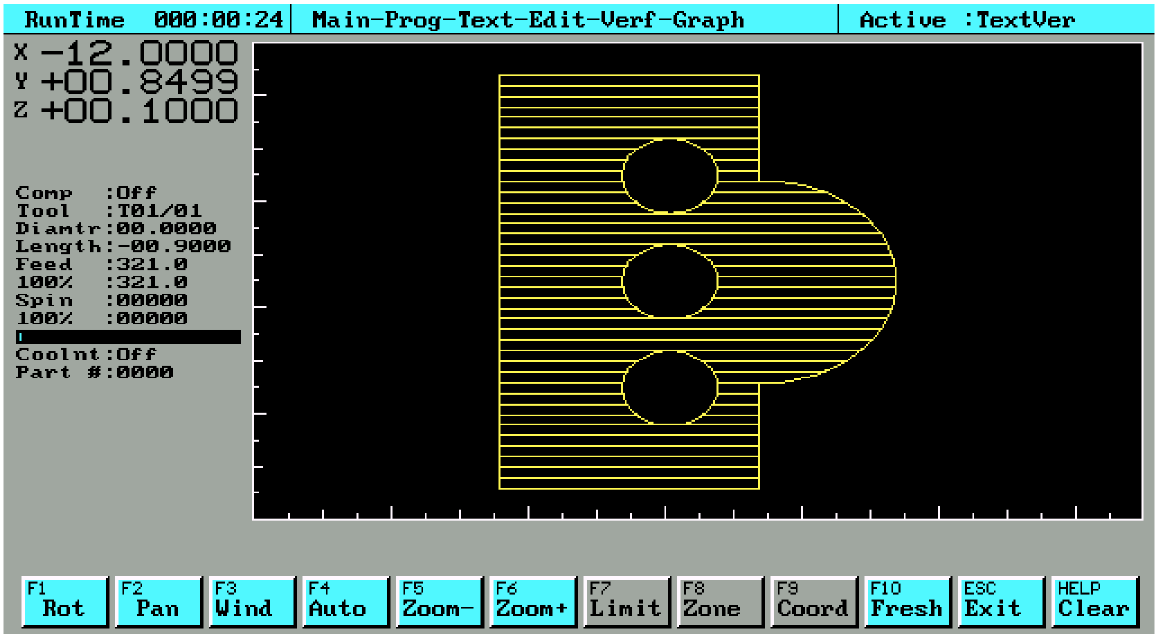
Note 5 : The cycle may require that holes are drilled at the plunge points. This would require that the plunge points be observed and added to a drill cycle.

The cycle can be used to clear around islands.

Sample program with islands:

```
P145=10 (Z Feedrate)
F321 (XY Feed-rate to clear the pocket)
G271 P1234 Q1235 R.1 Z-1 D.1 I.05
('R' is the R plane, 'Z' is the Z-depth, 'D' is the Cut Width and 'I' is
the Finish Stock)
N1234 (The 'P' specifies the start of the pocket)
G41 G65 X0 Y1
G0 X0 Y0
G1 X2
Y1
G3 R1 AA-90 AB90
G1 Y4
X0
Y0
G65 X1 Y0
G40
P516 = 1 (Specifiles and Island)
G45 G0 X1 Y1
G2 R.3 AA180 AB180
G47
P516 = 1 (Specifiles and Island)
G45 G0 X1 Y2
G2 R.3 AA180 AB180
G47
P516 = 1 (Specifiles and Island)
G45 G0 X1 Y3
G2 R.3 AA180 AB180
G47
N1235 (The 'Q' specifies the end of the pocket)
```

This program will clear a pocket that looks like:



A discussion follows on several specialized and non-standard G codes.

Store Restore parameters (G990/G991)

Pp Ll Qq G990 (store parameters)
Pp Ll Qq G991 (restore parameters)
G990/991 allow parameters to be saved and restored using file names C:/RAM/Q0000-Q9999.
Parameters are:

Pp (base parameter number, default 0),
Ll (number of parameters, default 10),
Qq (file identification, default 0).

The G990 and G991 MUST follow the Pp, Ll or Qq.

CAUTION: Have no other commands on the line.

Example:

N0020 (My subroutines)
P0 L10 Q20 G990 (Save parameters P0-P9)
•
•
(Freely use and modify parameters P0-P9)
•
•
P0 L10 Q20 G991 (restore P0-P9)
M98 (return to caller with original P0-P9)

Read byte parameter (G995)

P1=b G995 (sets P0 to value of byte b)

Example:

P1=79
G995 (sets P0 to value of byte 79, G18 plane select, XZ=0, ZX=1)
valid P1 values are 0 to 639

Valid P1 values are 0 to 639
Valid P0 values are 0 to 255

Note: The G995 is identical to P0=PB##.

Write byte parameter (G996)

P1=b P0=V sets byte parameter b to value v.

Example:

P1=82
P0=1
G996 (set byte parameter #82, offset round taper walls to yes, example 0=no, 1=yes)

Note: The G996 is identical to PB##=#

For a full list of the byte parameters, see APPENDIX..

Force Error (G997)

Forces an error code to be displayed.
Error code generated is round (parameter #1).

Example:

P1=408
G997 (Forces a 408 Y axis excess error to be displayed. Y axis does not cause an excess error, it only displays the error).

Note: P1=0 will not produce an error.

All valid error codes on the control are between 1 and 999.

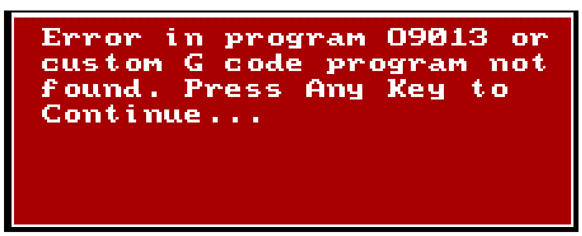
Beep (G998)

G998 will cause the speaker to beep if a speaker is installed.

Custom G codes

Custom G codes can be created to execute a user-defined cycle such as a nonexistent drilling cycle. To setup the new custom G code, first create a text program (numbered from 9010 to 9019) that defines the new G code. Next, enter the number of the new custom code in the F3 (Power) parameter section after the newly created text program.

The text program for the custom G code can reside in the RAM directory or parts directory. The program in the RAM directory holds precedence over the program in the parts directory. If you call any custom M or G code from within a custom code, it will execute its normal function. If a syntax error exists in a custom code, the following window will appear upon power-up.



Example: Set F3 (Power) parameter custom G code O9014 to 005, and enter program into C:/RAM/O9014.

Each time you execute a G5 it is similar to a call to C:/RAM/O9014.

A three-digit number can be used to identify the newly created custom code. See page 41, Section 2 on custom M and G codes.

SECTION FIVE - MISCELLANEOUS FUNCTIONS (M CODES)

These codes are used if the operator is programming the Centurion 6 in the text mode or MDI mode. They are also generated from conversation programs. It should be noted that most programmers – particularly new programmers – use the conversational programming mode. If you are planning to use text mode programming, pay close attention to this section explaining these codes. If you are planning to use conversational mode programming, you can ignore or skim over this section and concentrate on the conversational section.

The Miscellaneous Function codes are one or two digit numbers preceded by the letter M. If the code is less than 10, zero entry is optional (M02 or M2). These codes are used to perform a variety of machine and control functions as listed in the following table.

Note: The majority of the M codes that work with I/O are specific to Milltronics machines; however, they are flexible and can be tailored to specific applications. There may be several other M codes not listed here that deal with optional features on specific machines.

M codes

M codes	Function	Executed Before Move	Executed After Move
M00 M01 M02 M30	Program Stop Optional Stop End of Program End of Program/ Spindle Off		X X X X
M03 M04 M05	Spindle on CW Spindle On CCW Spindle Off	X * X *	 X *
M06	Tool Change		X *
M07 M08 M09	Mist Coolant On Flood Coolant On Coolant Off	X * X *	 X *
M10 M11 M19	Clamps Brake Unclamps Brake Orient Spindle (ATC Option)	X * X * X *	
M31	E-Stop	X *	
M32	Test Wait Channel	X *	
M90	Graphics Off	X *	
M91	Graphics On	X *	
M93	3D Sweep On	X	
M94	3D Sweep Off	X	
M95	Tapered Wall	X	
M96	Rounded Wall	X	
M97	Pocket Clear	X	
M98	Subroutine Call Statement	X *	
M99	End of Subroutine Statement	X *	

** These functions are selectable for either before or after move.*

Caution: The control will accept more than one M code on a line; however, it is recommended that only one M code per line be programmed. When more than one M code per line exists, the order of execution is somewhat undefined and the program may not run as expected. In general the M codes will execute in numerical order "M00 first M99 last" unless they have been defined to execute after the move statements. (See Post M codes Table on page 55, Section 4.)

After coolants are turned on, they go off and on with the spindle and need not be programmed on when the spindle is running.

Program stop (M00)

Execution of the program is halted on the block containing M00 and prompts the operator to press the **Cycle Start** button. Program execution will be resumed when the **Cycle Start** button is pushed. If M00 is on a line with a move command, the move will be executed before the stop. If there is a comment on the M00 block, it will be displayed as a prompt.

Optional stop (M01)

M01 is the same as M00 except it is only executed if the optional stop switch is enabled. If there is a comment on the M01 block, it will be displayed as a prompt to the operator.

Block skip (/)

A line of program can be skipped or ignored by the control. Inserting a / (slash) at the beginning of a line and enabling the F5 (BSkip) will cause the control to skip that line. In the example below with F5 (BSkip) disabled, the machine will move to the first, second, and third points. When F5 (BSkip) is enabled, the machine will move to the first then third points. Block two is skipped.

```
N1    X0 Y0
/N2   X2 Y2
N3    X4 Y0
```

End of program (M02, M30, M99)

Any of these codes can be used to indicate the end of a program. However, M02 and M99 will leave the spindle and coolants on; the M30 will turn them off. All three codes will return to the beginning of the program and start over when the **Cycle Start** button is pressed. The end of a program is the same as an M2 or M99.

Spindle on/off (M03, M04, M05)

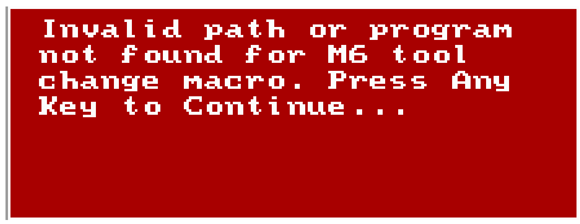
These codes turn the spindle on: CW (M03), CCW (M04) and OFF (M05). The spindle-on commands will be executed before an axis command. M05 will execute after an axis command.

Tool change (M06)

An M06 moves Z to the tool change position.

This command strobes the M function bus and sends out a 150 millisecond (msec) pulse on the M06 I/O output. It then halts program execution and prompts the operator to change the tool. If a comment is on the M6 block, it will be displayed to prompt the operator. The control shuts off the spindle and coolant, and then it waits until it receives a tool-change-complete signal. The spindle cannot be turned on until the tool change is completed. After the tool-change-complete signal, the program will resume running. If the tool change parameters are set to manual tool change, pressing the **Cycle Start** button is required after the tool-change-complete signal is received to resume program operation. For safety reasons a manual tool change should **never** be attempted unless the machine is in an M06 tool change command.

If the machine is equipped with an automatic tool changer, an F3 (Power) parameter, M6 (Tool Change) MACRO exists to automatically change the tools. The tool change macro points to a program that executes the tool change. These programs are specific to the tool changer and its inputs and outputs. If an error exists in the program or the program is not present in the specified directory, an error will appear upon power-up of the machine.



Coolants on/off (M07, M08, M09)

These codes turn the coolants on (M07 mist, M08 flood) before an axis command is executed. The coolant off command (M09) will be executed after an axis command.

Clamp for rotary table (M10)

This energizes the clamp or brake for the optional rotary table.

Release clamp on rotary table (M11)

This releases the brake for rotary table option.

Note: There is an F9 (Ctrl) "auto rotary brake" parameter that turns the brake on and off automatically.

Tool changer codes (M19-M28)

These functions are used with the optional automatic tool changer. An M19 will orient the spindle. The tool drum is controlled by M20 (tool drum home), M21 (tool drum CW) and M22 (tool drum CCW). The tool arm is controlled by M23 (Arm in) and M24 (Arm out). The drawbar is controlled by M25 (drawbar on) and M26 (drawbar off). The orient roller is controlled by M27 (orient roller in) and M28 (orient roller out).

Disable Drives (M31)

It may be desirable to disable the drives (emergency stop) after running a long program when the machine is unattended. The M31 will shut off the drives (emergency stop the machine).

Channel (M32)

This code causes the control to wait for the wait channel, X input 7, then continues the program.

Miscellaneous M codes (M65/75, M67/77, M68/78, M69/79, M50/60)

The standard M code is controlled by M65 (on) and M66 (off). These optional M codes control the four spare functions. Spare function one is controlled by M67 (on) and M77 (off). Spare function two is controlled by M68 (on) and M78 (off). Spare function three is controlled by M69 (on) and M79 (off). Spare function four is controlled by M50 (on) and M60 (off).

Graphics off/on (M90, M91)

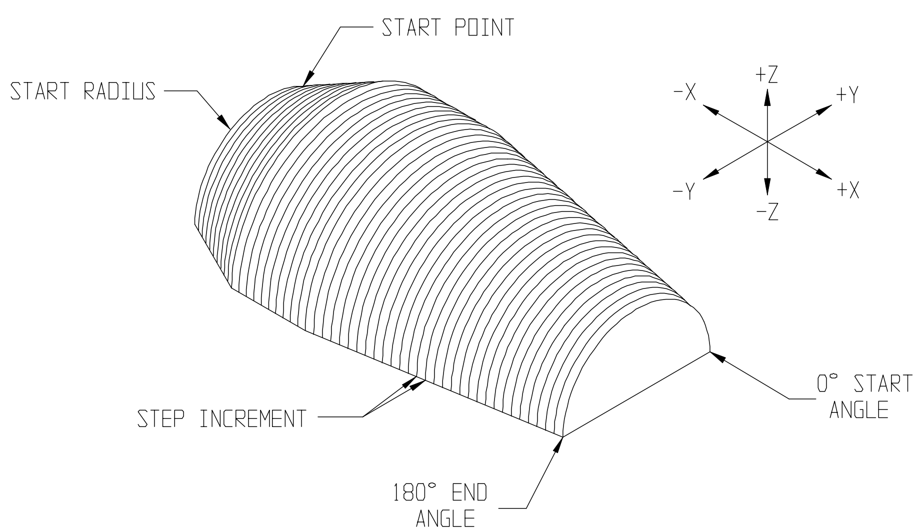
Occasionally, the graphics function on the CNC should be turned off to prevent redundant graphics. We suggest that when writing a program with loops, the operator should execute an M90 (Graphics Off) after the first loop. This prevents redundant lines from building up in the graphic memory. After the loop is finished, M91 (Graphics On) can be executed to display the next section of the program. If the last command executed was an M90 (Graphics Off) and the program is started over, the control will always restore the CNC to a M91 (Graphics On) state. The size of the graphics file is limited to a size set by the machine tool builder. If this size is exceeded, no additional graphics will be added to the screen.

3D Sweep off/on (M93, M94)

The 3D Sweep feature can be used to sweep XZ or YZ arc through XY (and Z) geometry. The following parameters are used to define these arcs.

P120 arc plane (8 = XZ plane, 9 = YZ plane)
P127 starting radius
P128 starting angle
P129 ending angle

The arcs are swept along geometry following the M94 command. Z pierce feedrate is parameter #145, the clearance is parameter #140, and the pass width is parameter #130.



The starting point is stored in the following parameter.

121 for X axis
122 for Y axis
123 for Z axis

To create a female part the starting angle must be between greater than 180° and less than 360°. For female parts that start at 0° use -.0001° or 359.9999°. For female parts that start at 180° use 180.00001° or -179.9999°.

Z can change in the geometry as well.

Example:	P120=9	YZ plane
	P127=1	start radius
	P128=-.0001	start angle, female part
	P129=180	end angle
	P130=.1	pass width
	P145=10	Z feedrate
	F15	sweep feedrate
	P121=0 P122=0 P123=0	start point
	P140=.1	R plane
	M94	3D sweep on
	X3 Z-1	Ramp down to Z-1, X0 to X3
	M93	3D sweep off

Creates a ramp on trough that drops from Z0 to Z-1.

Notes on 3D Sweep off/on

Note 1: 3D sweep assumes a ball-nose tool is being used.

Note 2: 3D sweep will not work with round corner or chamfering.

Note 3: Cutter compensation can be used by setting parameter #167 to 1. It works in the XZ or YZ plane only.

Tapered Walls (M95)

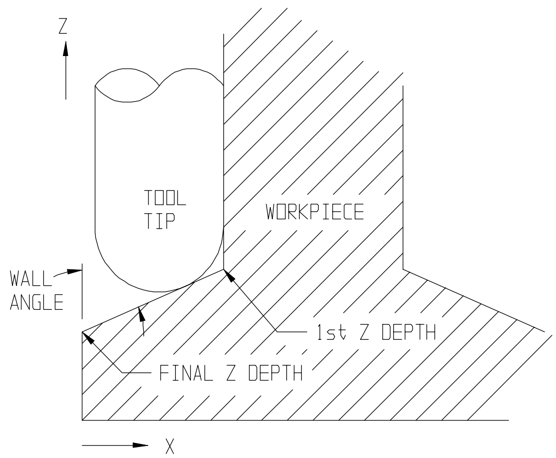
M95 can be used for tapering walls in pockets or on islands. This command takes wall angle, first depth, final depth, and the Z increment as parameters. The M95 must be within a while-wend loop. The tool radius and parameter #160 (the current Z depth) are modified based on the above parameters. The M95 sets parameter 162 to 2 when the cycle is completed. Cutter compensation must be on to use the tapered wall feature.

The M95 assumes that a ball-nosed tool is used. Tapered walls also uses an offset round/tapered walls parameter to determine the location of the first cut.

Example 1: Offset Island Taper

Offset round/tapered walls parameter = *yes*

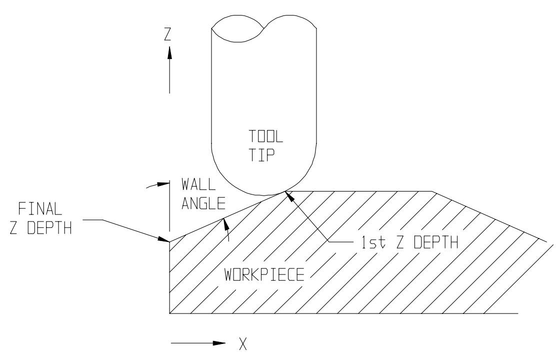
First cut is offset to avoid over cutting vertical wall.



Example 2: No Offset Island Taper

Offset round/tapered walls parameter = *no*

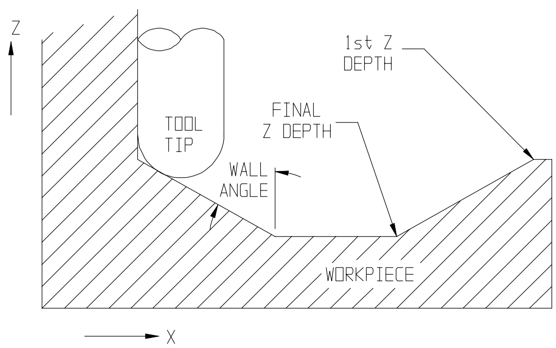
Offset is not needed to cut this part.



Example 3: Offset Cavity Taper

Offset round/tapered walls parameter = *yes*

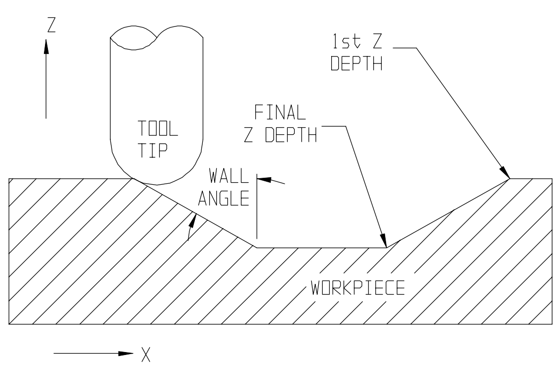
First cut is offset to avoid cutting vertical wall.



Example 4: Offset Cavity Taper

Offset round/tapered walls parameter = *no*

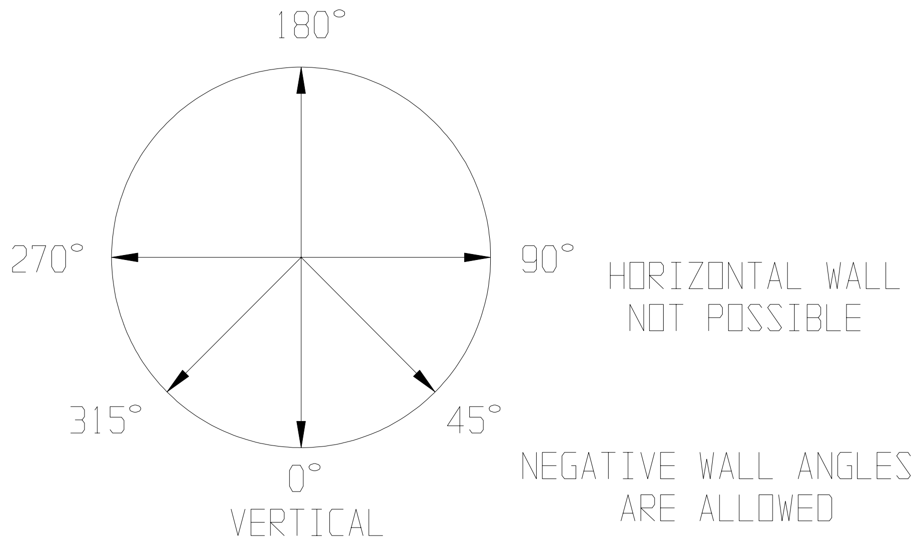
There is no need to offset tool on this part.



Note: The first cut at first Z depth is always offset by the entire tool radius.

First Z depth should be at the top of the surface to cut.

Wall angles are described as follows, regardless of pockets or islands.



If you are using a ball-nosed tool with the tapered walls, use an M95 EO (or M95). If you are using an end mill, use M95 E1.

Example Program:

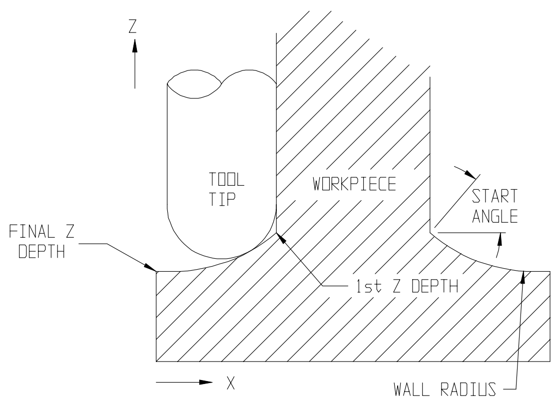
This program makes a 2" x 3" cavity with 30° walls.

P140=.1	Clearance
P141=-1	Final Z depth
P143=.1	Z increment
P144=0	1st Z depth, top of cut
P160=P144	
P162=0	To enter while loop
While P162 =0	Start of loop
G41 G65 X0 Y1	Cutter comp lead in
X0 Y0	
G1 F10 Z [P160]	Z down
X2	Geometry
Y3	Geometry
X0	Geometry
Y0	Geometry
G65 X1 Y0	Lead out
P163=30	Wall angle
M95	Taper walls command
Wend	End while loop

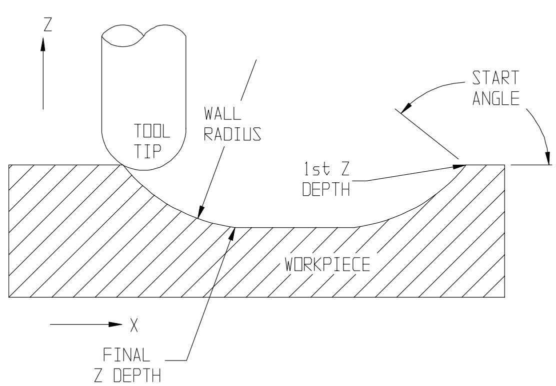
Rounded Walls (M96)

M96 can be used for rounding walls in pockets or on islands. This command takes a start angle, wall radius, first and final Z depths, and the Z increment as parameters. The M96 must be within a while-wend loop. The tool radius, parameter #160 (the current Z depth) and cutter compensation must be on to use the round walls feature. The M96 sets parameter #162 to 2 when the cycle is completed. The M96 assumes that a ball-nosed tool is used. Rounded walls also use an offset round/tapered walls parameter to determine where the first cut is.

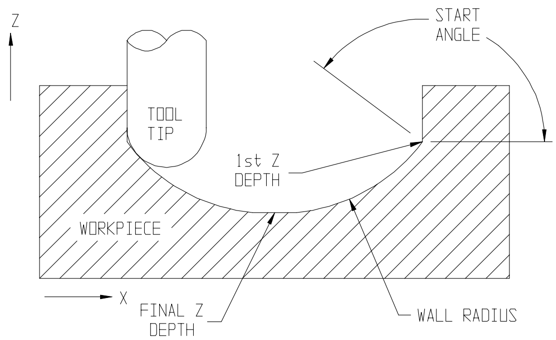
Example 1: Offset Island Rounded Walls
Offset round/tapered walls parameter = *yes*
First cut is offset to avoid overcutting vertical wall.



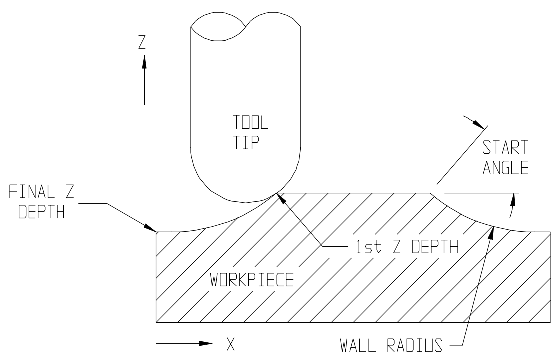
Example 2: No Offset Island Rounded Walls
Offset round/tapered walls parameter = *no*
Offset is not needed to cut this part.



Example 3: Offset Cavity Rounded Wall
Offset round/tapered walls parameter = *yes*
First cut is offset to avoid cutting vertical wall.

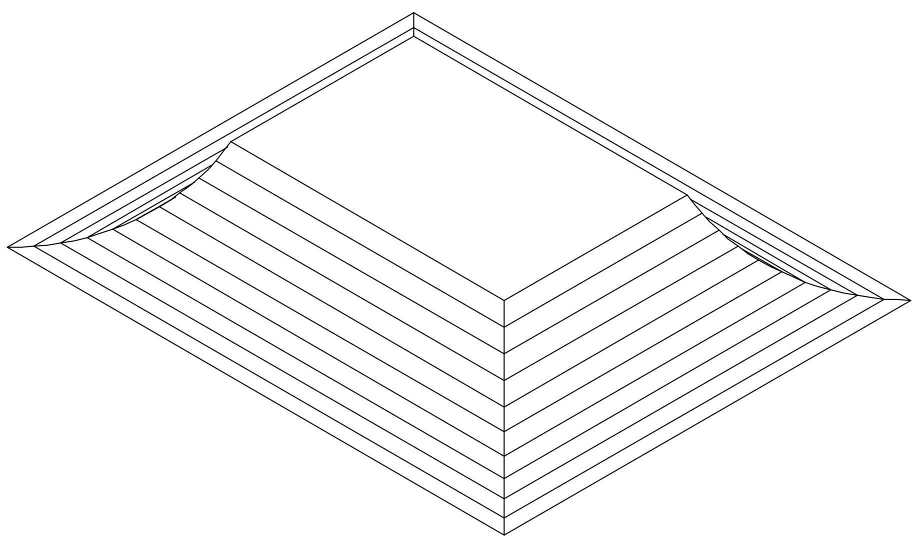


Example 4: No Offset Cavity Rounded Wall
Offset round/tapered walls = *no*
No need to offset tool on this part.



This program makes a 2" x 3" island with rounded walls. The wall has a 2" radius and starts at a slope of 30°.

P140=.1	Clearance
P141=-1	Final Z depth
P143=.1	Z increment
P144=0	1st Z depth
P160=P144	
P162=0	To enter while loop
G0	
While P162=0	Start of while-wend loop
G42 G65 X0 Y1	Cutter comp lead in
X0 Y0	
G1 F10 Z[P160]	Z down
G1 X2	Geometry
Y3	Geometry
X0	
Y0	Geometry
G65 X1 Y0	Lead out
P163=30	Start angle
P164=2	Wall radius
M96	Round wall command
Wend	



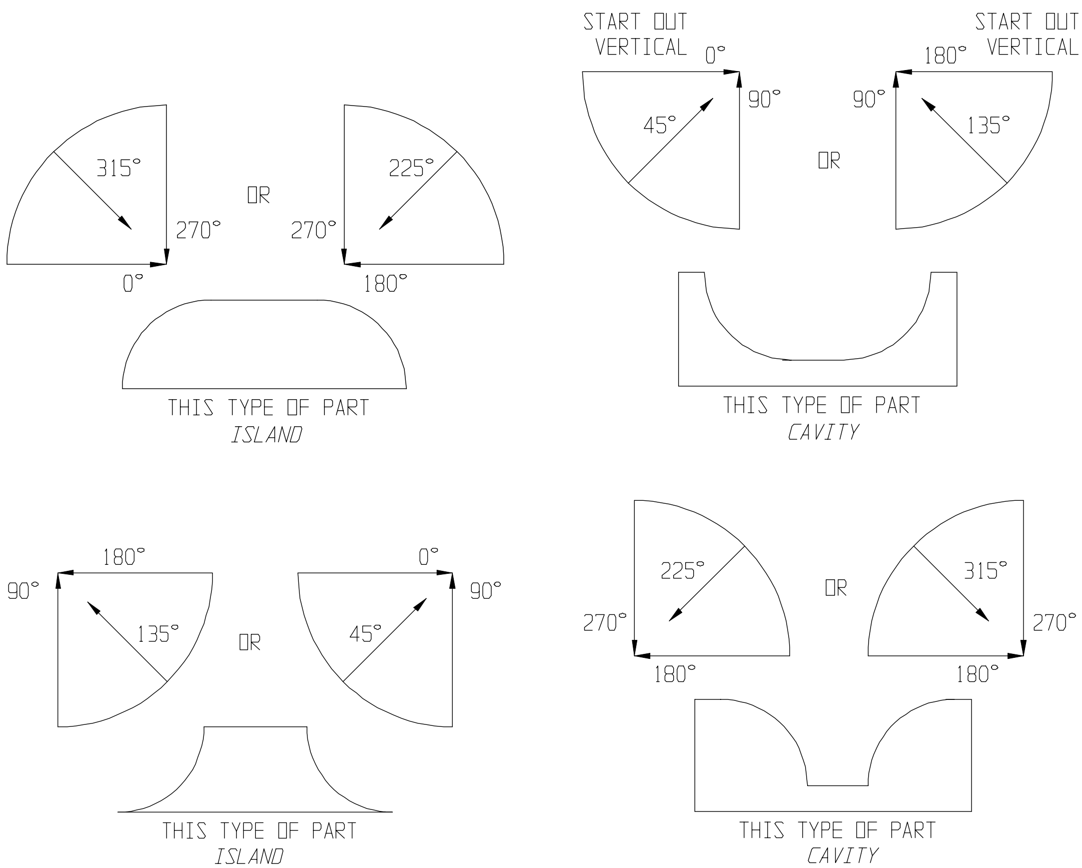
Notes on Rounded Walls

Note 1: *The first cut at the first Z depth is always offset the entire tool radius.*

Note 2: *The first Z depth should be at the top of the surface to cut.*

Note 3: *If the wall radius and start angle will not span the first Z depth and final Z depth, an error will occur.*

Start angles are defined and illustrated in the following drawings. Negative start angles are allowed.



If you use an E0 or no E on the M95 (or M96) block, the control uses a ball nose on the wall. Any other E# on the M95 (or M96) instructs the control you are using an end mill. The offset round/tapered walls has no effect on the end mill option.

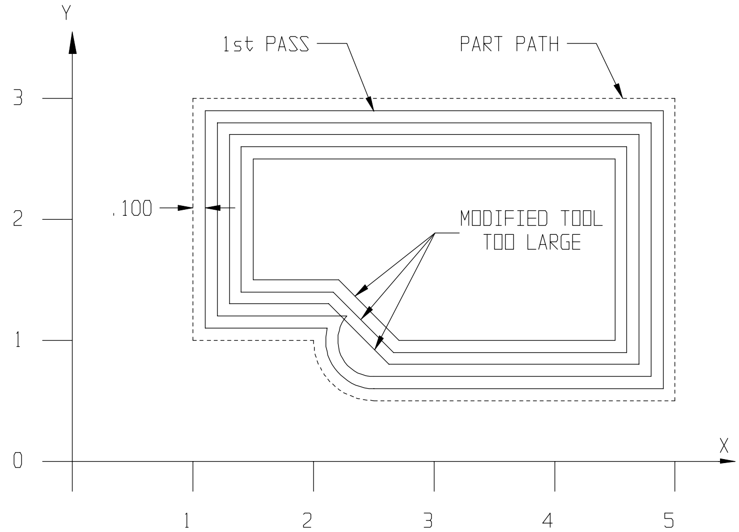
Pocket Clear (M97)

M97 can be used for clearing pockets as well as clearing away material from islands. It can also be used for finish passes on irregular pockets or islands. This command takes two parameters: the number of passes to make and the cut width of each pass. The command must be within a while-wend loop. The cut width is added or subtracted to the cutter radius and moves toward or away from the part. Cutter comp must be on to use the pocket clear feature. If a compensated arc intersection error for the modified tool radius occurs, a compensated line is substituted from the arc start point to the arc end point.

Example:

```
G0 Z.1                          Rapid to .1 above 0
D1                              Load tool radius #1
P1=-.1                          First Z depth
WHILE P1>-.3                    Last Z depth =-.3
G41 G65 X1 Y2                   Non-move cutter comp lead in
X1 Y1                           Start point
G1 F10 Z[P1]                    Z down
X2
G3 R.5 XC2.5 YC1 X2.5 Y .5
G1 X5
Y3
X1
Y1
G65 X2 Y0                       No-move cutter comp off
G40                             Cutter comp off
P163=5                          Make 5 passes
P164=.1                         Pass width =.1
M97                             Clear the Pocket
P1=P1-.1                        Next Z depth
G0 Z.1                          Rapid above part
WEND                            End while loop
```

The previous program makes this for a .1" radius tool:



To clear away from an island with the same shape, change the G41 to G42.

To make a finish pass, load the tool table with a tool radius bigger than the actual tool in use.

P163=2 Make 2 passes
P164=-.01 Correction to actual tool size/finish stock to clean up

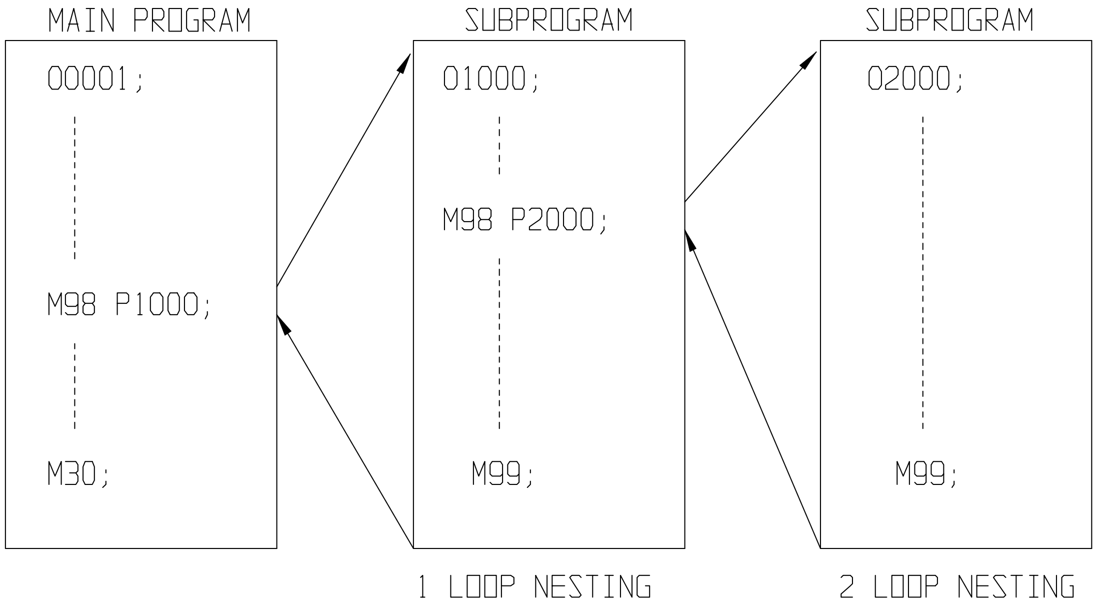
Note: Pocket clearing is used to remove stock from a part. If too many passes are programmed, the surface may be violated. It is best to use the M97 (pocket clear) to remove several passes and then use other means to clear the remaining material.

Subprogram call (M98)

Subprogram terminate (M99)

The subprogram call command (M98) can be used to execute any program residing in memory from another program. After the called program is executed, the control will then transfer back to the calling program one block after the M98 command.

When the main program calls a subprogram, it is regarded as a one loop nest. A two loop nesting can be executed as shown below.



When used with an L___ command, an M98 command can call a subprogram repeatedly. An L___ command can specify up to 999 repetitions of a subprogram.

Nesting with up to 50 loops is allowed.

M2 can be used instead of M99. If a subprogram ends without an M2 or M99 it will return to the calling program as if an M2 or M99 was encountered.

Preparation of subprogram

A subprogram is the same as any other program.

Specifying M99 at the end of a subprogram is optional. If the program was called by an M98, an M02, M30 or M99 will return. Subprograms are entered into memory in the same fashion as normal programs.

Subprogram execution

A subprogram is executed when called by the main program or another subprogram. A subprogram call has the following format.

M98 PXXXX LXXX

Where	PXXXX	=	subprogram number
And	LXXX	=	number of times the subprogram is to be repeated

Example: M98 P0002 L5
M98 P2 L5
Call 2 L5

This command reads, *call subprogram number 2 five times*.

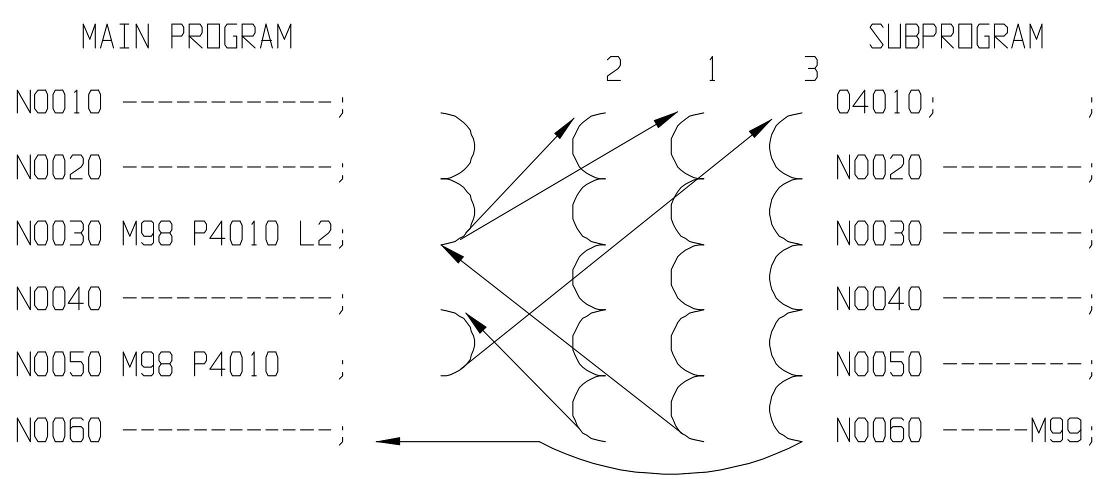
When the loop number is omitted, the subprogram is run once.

A subprogram call command and move command can be specified in the same block.

Example: X1 M98 P0200

In this example the subprogram 200 is called after completing movement in the X axis direction.

The execution sequence of a main program which calls a subprogram is as follows.



When the subprogram is called by another subprogram, it is executed in the same sequence as shown in the above example.

Notes on Subprograms

Note 1: *If the subprogram number specified cannot be found, a 603 error* "program O#### does not exist" *message is displayed.*

Note 2: *A subprogram call M98___ cannot be executed from MDI. In this case write a short program to call the subprogram.*

0XXXX
M98PXXX
M02 *Then execute it in the run mode.*

Note 3: *If a subprogram modifies the floating zero, (normally G92 X0 Y0 on the first block of the subroutine), the floating zero will be restored to the same offset value as it was when entering the subprogram. This can be useful for cutting the same pattern at different locations.*

Example:

0####	*O0002*
X1 Y2	*G92 X0 Y0*
M98 P2 (cut part at X1 Y2)	.
X3 Y5	.
	.
M98 P2 (cut part at X3 Y5)	.
M2	*M99*

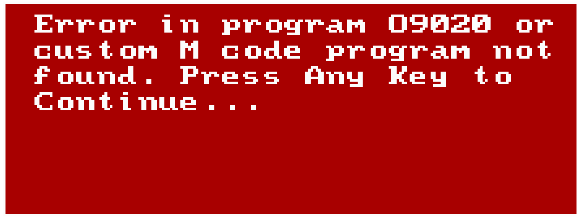
Note 4: *If you use the "call" statement, you can call subprograms with full DOS names.*

Example: Call C:\Test\Parts\XYZ.PRG

Custom M codes

Custom M codes can be created to execute a user-defined function. To set up the new custom M code first create a text program – numbered from 9020 to 9029 – that defines the new M code. Next, enter the number of the new custom code in the F3 (Power) parameter section after the newly created text program. A three-digit number can be used to number the newly created custom code.

The text program can reside in the RAM directory or in the parts directory. The program in the RAM directory holds precedence over the program in the parts directory. If you call any custom M or G code from within a custom code, it will execute its normal function. If a syntax error exists in a custom code, a message will be given on power-up.



Example 1: Set F3 (Power) parameter custom M code O9025 to 014 and enter a program into C:/RAM/O9025. An M14 execution is similar to a call to C:/RAM/O9025.

Example 2: Set F3 (Power) parameter custom M code O9027 to 3 and enter the following G program into C:/RAM/O9027.

Set 25	(To turn on a vacuum)
G4 F1	(To dwell for 1 second)
M3	(Turns the spindle on like a normal M3)

Parametric programming is similar to macro programming in that equations can be used to specify axis position rather than decimal numbers. The Centurion 6 does not restrict the use of parametrics to subroutines or macros. They may be used anywhere throughout a program. Parametric expressions may be used to specify M, G, F, and S functions. When a parametric expression is used for an axis position, it will first be evaluated and then cutter compensation will be applied. All the normal cutter compensation rules will apply to the evaluated point. When using parametric expressions in a program, the parameters that are used are the 100 user parameters discussed earlier on page 74, Section 2. Values generated by equations may be displayed on user parameter screens. Other listed system parameters may be used as input data to parametric equations, but under normal circumstances these parameters should not be changed.

Parametric reference

A parameter reference is specified by the letter "P" followed by a valid parameter number. When a parameter reference is used for a coordinate position it must be contained in brackets.

Example: X [P10]
Y [-P145]
Z [P2]

Parametric assignment command

Assigning is the most basic command in the use of parameters. The assignment character is an equal (=) sign.

Assignment statements replace the current value of a variable with a new value.

Example: P1 = 1.234

In this example the value 1.234 is assigned to parameter 1. Therefore both of the following commands would move to the same coordinate position.

X1.234 or X[P1]

Note: *If assigning parameters numbers greater than 499, the control needs to be in the data mode (G10). See the* APPENDIX *for a list of parameter numbers and their descriptions.*

PB byte parameter

Byte parameters generally hold values that are representative of the machine configuration. For example, the spindle range parameter PB50 will hold a value of 1 for low range, 2 for medium, and 3 for high. Byte parameter values cannot exceed 255.

Example: PB50=2

Note: *Byte parameters are not written in the verify mode. See the APPENDIX for a list of byte parameter numbers and their descriptions.*

Arithmetic operators

The following list shows the available arithmetic operators.

Operator	Operation
+	addition
-	subtraction
*	multiplication
**	exponent
/	division
DIV	integer division
MOD	remainder

Note: The value of A DIV B is the mathematical quotient of A/B with any fractional portion or remainder dropped.

Examples:		
	3/2 = 1.5	*3 DIV 2 = 1*
	24/5 = 4.8	*24 DIV 5 = 4*
	72/8 = 9.0	*72 DIV 8 = 9*
	5.46/2.1 = 2.6	*5.46 DIV 2.1 = 2*

The MOD returns the remainder obtained by dividing its two numbers.

3 MOD 2 = 1 remainder = 1
24 MOD 5 = 4 remainder = 4
72 MOD 8 = 0 remainder = 0
3.57 MOD 2.1 = 1.47 Fract (3.57 ÷ 2.1 =.7)
(.7 x 2.1 = 1.47)

Relational operators

The following list shows the available relational operators.

Operator	Operation
EQ or =	equal
NE or <>	not equal
LT or <	less than
GT or >	greater than
LE or <=	less or equal
GE or >=	greater or equal

Function operators

A function call is specified by the function name (e.g. SIN, ATAN, . . .) followed by the function argument in brackets. When a function is used for a coordinate position it must be contained in brackets.

Examples: X [SIN [45]]
Y [ATAN[1/2]]
Z [SQRT[9]]

A function returns a value and can be used interchangeably anywhere a decimal value is accepted.

The functions supported are as follows.

Sine (SIN) returns the sine of the argument.
SIN [90] = 1

Cosine (COS) returns the cosine of the argument.

COS [180] = -1

Tangent (TAN) returns the tangent of the argument.
TAN [135] = -1

Arctangent (ATAN) returns the arctangent of the argument. The argument must be specified in fractional form (e.g. ½, 2/1, -5/6, ...).
ATAN [1/-1] = 135

Square root (SQRT) returns the square root of the argument.
SQRT [9] = 3

Absolute value (ABS) returns the absolute value of the argument.
ABS [-15] = 15

Integer (INT) returns the integer part of the argument.
INT [5.5099] = 5

Random (RND) returns an integer random number between 0 and its argument -1.
RND [25] will return an integer >=0 and <=24

Natural Logarithm (LN) returns the natural logarithm of the argument.
LN [10] = 2.3026

Exponential (EXP) returns the exponential of the argument.
EXP [1] = 2.7183

Rounds (ROUND) rounds a decimal value to an integer value.

Values halfway in-between are rounded up.

ROUND [2.3] = 2
ROUND [7.88] = 8
ROUND [1.5] = 2
ROUND [-1.5] = -2

Values exactly halfway between are rounded to the nearest even number.

ROUND [2.5] = 2
ROUND [3.5] = 4

Mathematic expressions

Any combination of the previously described expressions are made up of arithmetic functions.

Examples: X[SIN[P123]*COS[P124]]
Y[2.5+[P2/P3]*SQRT[P4]]
Z[[P2DIV3]+[P2MOD3]]

Conditional statements

The Centurion 6 supports two types of conditional statements. These statements are used to transfer control of a program from one point to another based on some condition generated in the program. These statements are the IF-THEN statement and the WHILE-WEND statement.

IF-THEN

The IF-THEN statement is a way of conditionally executing a block if the results of an expression evaluates to true. The expression must contain one of the relational operators which allows the expression to be reduced to either true or false. If the expression is true, the THEN portion of the IF statement is executed. If the expression is false, the next line after the IF-THEN statement is executed.

Example of General Form:

	[any]	(relational)	[any]	
If	[mathematical]	(operator)	[mathematical]	Then any action
	[expression]		[expression]	

N20 IF P1 LT P2 THEN GOTO 15
N21

or

N22 IF P1 < P2 GOTO 15
N21

The above two statements accomplish the same thing. If the statement is true, N15 is executed; if it is false, N21 is executed.

Examples: IF P1*P3/COS[P90] GE TAN[P6] THEN X1

IF P4/P3 LT P6 GOTO 25

IF P1 = P2 THEN P4 = P5 - P6

Multiple IF statements can be used to check for multiple conditions.

Example: IF P36<5 THEN IF P1<>0 THEN M5
Defined, this means if P36 is less than 5 and P1 does not equal 0, shut the spindle off.

Note: The word THEN *is optional in all cases.*

WHILE-WEND

The second type of conditional statement is the WHILE-WEND statement. A WHILE statement contains an expression that controls the repeated execution of the blocks contained between the WHILE and WEND statements.

The expression controlling the repetition must contain one of the set of relational operators which allows the expression to be reduced to either true or false. The expression is evaluated before the contained blocks are executed. The contained blocks are executed repeatedly as long as the expression is true. If the expression is false at the beginning the blocks are not executed.

Example:

```
N20   WHILE [[P2*P3]/COS[P6]] LT P2
N21   P6 = P6 + 1
N22   Y[P2] Z[P3]
N23   X[P6]
N24   X1 Y0 Z0
N25   WEND
N26   M30
```

In this example lines N20 thru N25 will be repeated until the WHILE expression becomes false. Then line N26 will be executed instead of N21.

Note: Nested WHILE loops are allowed.

Transfer statements

Transfer statements transfer control from one section of a program to another. They are unconditional transfers in that when the statement is executed, control always transfers. The GOSUB/RETURN, and CALL statements return control to the N+1 block after they are finished, and the GOTO statement transfers control to the specified block without a return.

GOTO statement

The statement N### defines a label. GOTO's/GOSUB's can branch or transfer control to blocks containing these labels. A GOTO statement transfers progam execution to the block prefixed by the block label referenced in the GOTO statement.

GOTO 30 (The next block executed is block N30.)

Note: If there is more than one N30 program, it will transfer to the first N30.

CALL statement

A CALL statement transfers control to any program residing in the CNC's memory. Upon completion of the called program or an M2, M99, or M30, control is returned to the main program at the block immediately following the CALL statement.

The CALL format is as follows.

CALL	XXXX Program Number	LXXX Loop Count (optional)

If the L is omitted, the called program will be executed once. The call statement is the same as an M98.

Note: It is not necessary for the sub-program to be called in a separate program. You can call to an O#### in the same file. This eliminates the need to use the extract programs parameter.

Example:

X1 Y1
•
•
Call 2 (call to O0002)
•
•
M2 (the M2 here is optional)
O0002 (O#### in the same file has precedence over a separate O#### file.)
X2 Y2
•
•
M99 (the M99 here is optional)

GOSUB and RETURN

A GOSUB transfers program execution to the block number specified in the GOSUB statement. Execution will continue until a block containing a RETURN statement is encountered. The RETURN will transfer control back to the block immediately following the GOSUB statement. To use a GOSUB statement, the called block number must be part of the same program. Generally the subroutines are at the end of the main program.

The GOSUB format is as follows.

GOSUB	XXXX	LXXX
	Line #	Loop Count (optional)

If the L is omitted the GOSUB routine will be executed once.

```
N1
N2
N3
N4 GOSUB 100                    Main Program
N5
 .
 .
 .
N90 M30
N100
N101
 .


 .                              Subroutine
 .
N200
N201 RETURN
N202
```

When the GOSUB is executed in N4 the program will jump to N100 and start executing until N201 is reached. At N201 control will transfer to N5 and lines N5 thru N90 will be executed. The M30 will terminate the main program and keep lines 100 thru 202 from being executed again.

Notes on Call Statement

Note 1: Subprogram call allows nesting 50 levels deep.
Note 2: See page 294, Section 2 for information on using G92 in subprograms.
Note 3: Call to full DOS name programs are allowed.

Example: CALL S:/XYZ/PRG.X

Computational functions

1. Tangent Arc TANA
2. Tangent Line TANL
3. 3 Point Circle Generate CGEN

The above three functions can be used anywhere throughout a program to solve various intersection problems. These functions receive input data in parameters P90 thru P96, and they return the answer in parameters P80 through P85. The answers can then be used in line and circle commands to produce the desired results. The format for these three functions follows.

General Format for TANA, TANL

Input Parameters

P90 = X1 center of arc 1
P91 = Y1 center of arc 1
P92 = R1 radius of arc 1
P93 = X2 center of arc 2
P94 = Y2 center of arc 2
P95 = R2 radius of arc 2
P96 = R3 radius of tangent arc (use in TANA only)

Calculated Output Parameters

P80 = Xs X starting point of tangent arc or line
P81 = Ys Y starting point of tangent arc or line
P82 = Xe X end point of tangent arc or line
P83 = Ye Y end point of tangent arc or line
P84 = Xt X center of tangent arc (TANA case only)
P85 = Yt Y center of tangent arc (TANA case only)

The general form for a tangent arc or line function is as follows.

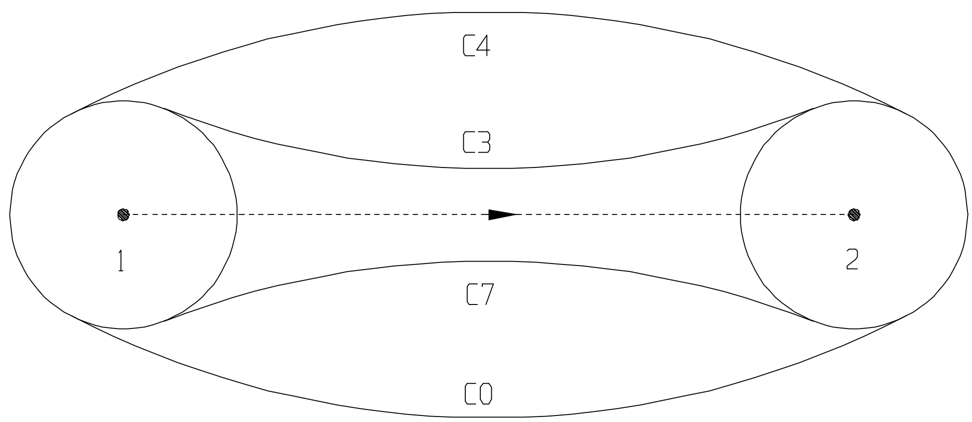
TANA C# or TANL C#

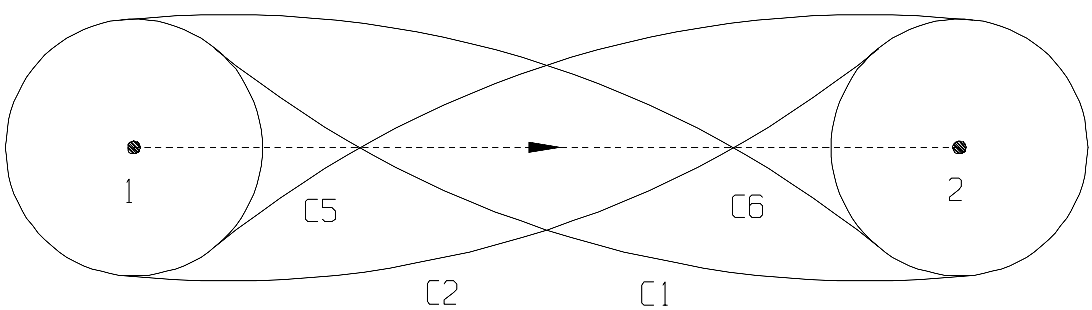
is a number 0 through 7 in the tangent arc case, and 0 through 3 in the tangent line case. This number selects one of the eight possible solutions of the TANA or one

of the four solutions of the TANL. The values of C# are defined as the tangent point being to the right or left of a line connecting the centers of the arcs and center of the connecting arc, when facing in the direction of tool movement. See the following diagrams for illustration of tangent solutions.

TANA Cases

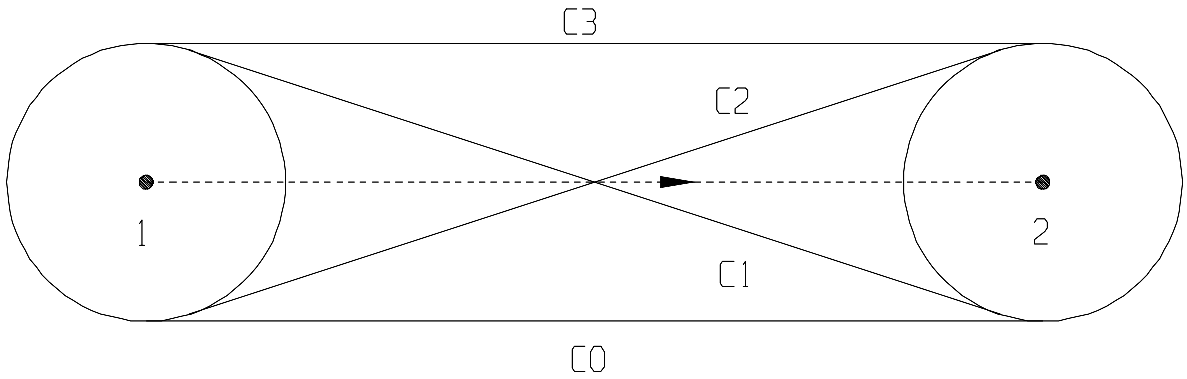
	1st	2nd	Center
C0 =	Right	Right	Left
C1 =	Left	Right	Left
C2 =	Right	Left	Left
C3 =	Left	Left	Left
C4 =	Left	Left	Right
C5 =	Right	Left	Right
C6 =	Left	Right	Right
C7 =	Right	Right	Right





TANL Cases

	1st	2nd
C0 =	Right	Right
C1 =	Left	Right
C2 =	Right	Left
C3 =	Left	Left



<u>Sample Program Using TANA or TANL</u>

N1	P90=0	XC of arc 1
N2	P91=0	YC of arc 1
N3	P92=1.5	radius of arc 1
N4	P93=5	XC of arc 2
N5	P94=4	YC of arc 2
N6	P95=2	radius of arc 2
N7	P96=5	radius of tangent arc (not used for tangent line)
N8	TANA C3 or TANL C3	
N9	G2 R1.5 XC0 YC0 X[P80] Y[P81]	

TANA or TANL calculated end points

N10 G3 R5 XC[P84] YC[P85] X[P82] Y[P83]

(TANA calculated center and end points)

or

N10 G1 X[P82] Y[P83]

(TANL calculated end point)

N11 G2 R2 XC5 YC4 X Y
end end

The circle generate function will calculate the center and radius of an arc through any three non-co-linear points. The general format for the CGen function is as follows.

Input Parameters

P90=X1 P91=Y1	coordinates of first point
P92=X2 P93=Y2	coordinates of second point
P94=X3 P95=Y3	coordinates of third point

Output Parameters

P80=XC P81=YC	center of calculated circle
P82=R	radius of calculated circle

Sample Program using CGen

```
N1   P90=0 P91=0  first point
N2   P92=1 P93=1  second point
N3   P94=2 P95=0  third point
N4   CGEN
N5   G1 X0 Y0     position to start of arc
N6   G17 G2 R[P82] XC[P80] YC[P81] Xe____Ye____
```

The CGEN function can be used anytime throughout the program to calculate the radius and center of an arc. These calculations can then be used in a normal arc command along with trig help, chamfer, corner round, extend back, and any other function available.

Note: The arc direction is returned in P97 so the block N6 may be substituted with G17 G[P97] R[P82] XC[P81] YC[P80] Xe___ Ye___

Text command

The Centurion 6 has a lettering command which can be used to engrave serial numbers or other descriptions. The text cycles must be loaded by setting the “Load text cycles” parameter. The control must be rebooted after setting the parameter. The one-to-one letter size is 1" x 1" and is in a block letter font. All these letters can be scaled or rotated to achieve the desired size and orientation. The depth of cut is contained in P141, the Z plunge feedrate in P145, and the clearance plane in P140. A typical program would be as follows.

N1	P140=.1	Clearance of .1
N2	P141=-.2	Depth of cut .2 inches
N3	P145=5	Plunge feedrate of 5 ipm
N4	G1 F10	XY feedrate of 10 ipm
N5	S1000 M3	Spindle on CW
N6	X-4 Y.5	Position of first letter
N7	Text [MILLTRONICS MFG]	Desired text
N8	M30	End program

This program will write "MILLTRONICS MFG" at a depth of -.2" in 1" by 1" block letters starting at a position of X-4 Y.5.

The text cycle can also be used to engrave parameter values, times and dates.

Example:

```
P28 = -6.77777
TEXT [N=#28[34]]
```

This will engrave "N=-006.7778" (3 leading digits, 4 trailing). If the leading and trailing fields are left blank, the default format from the machine setup parameters is used.

Successive part numbers can be engraved using the following scheme:

TEXT[#23[30]]	Engraves the value of P23, 3 places to the left, 0 to the right)
P23=P23+1	(increment part counter)

When the leading and trailing is 0, the ascii value of the parameter is engraved.

Example:

```
P1 = 35 (Ascii for '#')
P2 = 123
TEXT [PART #100] #2[30]]
```

This will engrave "PART #123".

When the leading and trailing = -1, the time is engraved (the parameter # is ignored).

Example: TEXT [TIME IS #1 [-1]] will engrave "TIME IS 016:46:51".

When the leading and trailing = -2 the date is engraved (the parameter # is ignored).

Example: TEXT [DATE IS #1[-2]] will engrave "DATE IS 02/11/08"

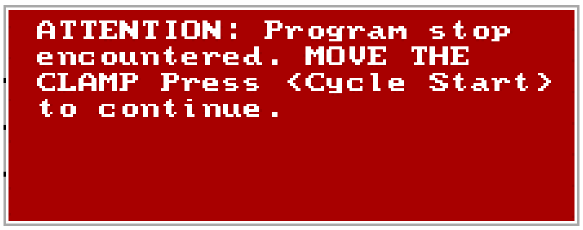
Miscellaneous Commands

Spaces Spaces can be used anywhere within the program. For example, Z1.234 can be written as Z 1 . 23 4 if desired.

Blocks Blocks without any information are allowed.

Comments Comments are any text enclosed in parentheses and they are ignored by the control. Comments can be anywhere in a program or in a block. When a comment is on a block with an M0 (block stop), M1 (optional stop), or M6 (tool change), the comment will be displayed in the message window.

Example: M0 (MOVE THE CLAMP)



Example: M6 (3/8" DRILL)

PRINT Print is used to print text to the screen. PRINT can be used to display text, parameter values, times, dates, etc.

Example: P87=32.45
PRINT [N=#87] shows "N=32.4500"

#n[LT] #n[LT] displays parameter n with L leading digits and T trailing digits.

Example: P100=1.235 P101=2.87656
PRINT [P100=#100[04] P101=#101[33]] shows
"P100=1.2350 P101=002.877"

If the leading and trailing fields are left blank, the default leading and trailing format for the machine setup parameters is used.

When LT=0, exceptions apply.

If LT=0, the ASCII value of the parameter is shown.

Example: P1=72 (ASCII H)
P2=105 (ASCII i)
PRINT [#1[00]#2[00]] shows "Hi"

If LT=-1, the parameter is ignored. The time is shown in format 0hh:mm:ss.

If LT=-2, the parameter is ignored. The date is shown in format yy/mm/dd.

Example: PRINT [Time=#1[-1] Date=#1[-2]] shows "Time=012:51:52 Date=97/10/23"

POPEN The POPEN command opens a file or RS-232 for output using the DPRNT command.

POPEN P0 The POPEN P0 command opens a file for output. The file to open is specified in the MISC parameters. If the file name in this parameter is blank, the output will be written to REPORT.DAT.

POPEN P1 The POPEN P1 command opens serial port 1 for output.

POPEN P2 POPEN P2 opens serial port 2 for output.

PCLOS PCLOS will close a serial port that has been opened using POPEN.

DPRNT DPRNT outputs text to a file or RS-232 port which is specified by the POPEN command.

Example: DPRNT [PLEASE CLEAR THE WORK AREA]

#n writes the value of a parameter

Example: DPRNT [X#208 Y#209 Z#210] outputs the current X, Y,and Z positions to a file or an RS-232 port.

#n[LT] will output parameter n with L leading and T trailing digits.

Example: P100=1.235 P101=2.87656
DPRNT [P100=#100[04] P101=#101[33]] outputs
"P100=1.2350 P101=002.877"

If the leading and trailing fields are left blank, the default leading and trailing format for the machine setup parameters is used.

If LT=00, the ASCII value of the parameter is shown.

Example: P1=40 (ASCII ( )
P2=72 (ASCII H)
P3=105 (ASCII i)
P4=41 (ASCII))
PRINT [#1[00]#2[00]#3[00]#4[00]] outputs (Hi)

If LT=-1, the parameter is ignored. The time is shown in format 0hh:mm:ss.

If LT=-2, the parameter is ignored. The date is shown in format yy/mm/dd.

Example: DPRNT [Time=#1[-1] Date=#1[-2]] shows "Time=012:51:52 Date=97/10/23"

DPRNT can be used to generate report files relating to parts production, parts inspections, digitizing information, etc.

INPUT The INPUT statement is used for data input from the front panel.

Example: INPUT (X START POSITION) P1

The operator will be prompted to input data.

The operator can use the data displayed by pressing the ENTER key. If ESC is pressed during an input statement, the program will be terminated.

The HDW command can be used to enter the handwheel mode during a program. A comment may be added to prompt the operator during the HDW command. HDW is ignored when verifying a program.

Example: HDW (TOUCH THE TOOL TO THE PART) prompts the operator with the following message.

The operator presses the <enter> key after handwheeling the machine. If <esc> is pressed during handwheeling, the program will be terminated.

To manually set and clear input/output pins, refer to the following.

CLR Clear an output pin.

Example: CLR X7 (clears X output #7)

SET Set an output pin.

Example: SET Z3 (sets Z output #3)

PULSE0 Pulses an output pin.

Example 1: PULSE0 Z10 (clears Z output #10, delays for the number of milliseconds specified by the MISC parameter PULSEx pulse delay(ms) then sets output Z10)

Example 2: PULSE0 X2 P3.5 (clears X output #2 and delays for 3.5 seconds, then sets X output 2)

PULSE1 Pulses an output pin

Example: PULSE1 X5 (sets X output #5 delays for the number of number of milliseconds specified by the MISC parameter PULSEx delay(ms) then clears output X5)

STO STO is similar to the SET command, except the outputs are consecutive. There are twelve outputs on each axis.

Example: STO20 (sets Y output #8 or the 20th output)

CLO CLO is similar to the CLR command, except the outputs are consecutive.

Example: CLO25 (clears Z output #1 or the 25th output)

PIN PIN refers to a pin. X axis are 1000's Y axis are 2000's, Z axis are 3000's, etc.

PIN	X Axis	Y Axis	Z Axis	4th Axis	5th Axis	6th Axis
in1	1000	2000	3000	4000	5000	6000
in2	1001	2001	3001	4001	5001	6001
in3	1002	2002	3002	4002	5002	6002
in4	1003	2003	3003	4003	5003	6003
in5	1004	2004	3004	4004	5004	
in6	1005	2005	3005	4005	5005	
in7	1006	2006	3006	4006	5006	
in8	1007	2007	3007	4007	5007	
in9	1008	2008	3008	4008	5008	
in10	1009	2009	3009	4009	5009	
in11	1010	2010	3010	4010	5010	
in12	1011	2011	3011	4011	5011	
out1	1012	2012	3012	4012	5012	6012
out2	1013	2013	3013	4013	5013	6013
out3	1014	2014	3014	4014	5014	6014
out4	1015	2015	3015	4015	5015	6015
out5	1016	2016	3016	4016	5016	
out6	1017	2017	3017	4017	5017	

out7	1018	2018	3018	4018	5018
out8	1019	2019	3019	4019	5019
out9	1020	2020	3020	4020	5020
out10	1021	2021	3021	4021	5021
out11	1022	2022	3022	4022	5022
out12	1023	2023	3023	4023	5023

Note: PIN statements can be used in conditional statements such as IF-THEN and WHILE-WEND.

Examples:

```
IF PIN[2017] EQ 1 THEN PB50=1
WHILE PIN[1005] NE 0
M62
M63
WEND
P5=P3 + PIN[2011]
```

IPIN

IPIN refers to an input pin. The argument is the input number. IPIN [32] is the 32nd input. There are 64 inputs. IPIN [32] is Z input #9. IPIN [17] is the same as PIN [2005] or Y input #6. The IPIN statements can be used in conditional statement or to assign variables.

OPIN

OPIN refers to an output pin. The argument is the output number. OPIN [14] is the 14th output. There are 64 outputs. OPIN [40] is 4th axis output #4. OPIN [3] is the same as pin [1003] or x axis input 4. The OPIN statements can be used in conditional statements or to assign variables.

Parts Inspection and Digitizing Commands

PROBE1 PROBE1 can be used with any axis move to command an interrupted move. The move will be terminated when an input goes low.

Example: PROBE1 F.01 X1.3 Y-2
The X and Y move toward 1.3 and -2 respectively until the probe contacts the part.

PROBE2 PROBE2 may be used with any axis move to command an interrupted move. The move will be terminated when an input goes high.

Example: PROBE2 F.01 X1.3 Y-2
The X and Y move toward 1.3 and -2 respectively until the probe releases from the part.

Note: The input used by the probe commands is selectable in the miscellaneous parameters.

Back line

The back line function may be used on any line command. This function reverses the direction of the programmed line. Back line is normally used when you know the end point of the line and not its starting point. The end point is programmed and the line is extended backward to the start point. When using this function all Trig Help functions are still valid.

Example 1:

```
X6 Y2
X5
X4 Y1.5
G2 R1 XC3 YC1.5 AB270
G1 X1 Y2 back C0 W145
X0
```

Back - extend back from (1,2)
C0 - use the arc intersection closest to (1,2)
W145 - extend the line from (1,2) at an angle of 145°

Example 2:

```
X6 YZ
X5
X4 Y1.5
G2 R1XC3YC1.5 AB270
G1 X1 YZ BACK C2 W145
X0
```

C2 - use the farthest intersection

Example 3:

```
X6 Y2
X5
X4 Y1.5
X1 Y2 BACK C0 W-35
X0
```

Note: This example uses a back line between two lines to program an unknown point.

MOD

MOD is used to shift an axis position. It is generally used for rotary axis to obtain a positive position between 0 and 360 degrees. It can be useful after a rotary axis has made several revolutions in the same direction.

Examples:

G0 A750
MOD A360 (A axis position is now 30°)

G0 A-100
MOD A360 (A axis position is now 240°)

G0 A-500
MOD A360 (A axis position is now 220°)

G0 A437
MOD A20 (A axis position is now 17°)

G0 A33.285
MOD A2.1 (A axis position is now 1.785°)
33.285 / 2.1 = 15.85
15.85 remainder is .85
*.85 * 2.1 = 1.785*

Note: Using the MOD command also shifts the software limits. This is desirable for most rotary axis, but is undesirable for linear axis.

ORIGIN

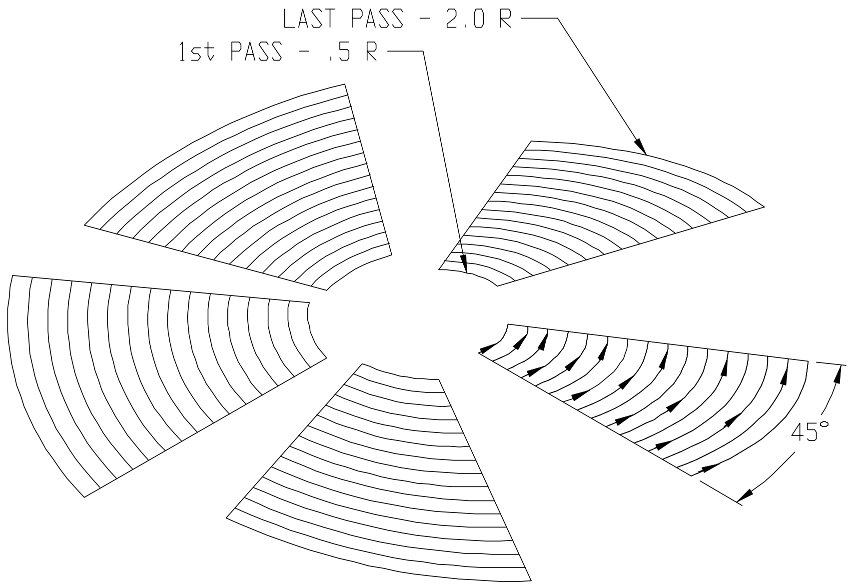
Origin is another way to shift axis coordinates. It shifts the axis coordinates by the operator amount.

Examples:

437 ORIGIN 20 = 417
33.285 ORIGIN 2.1 = 31.185
361 ORIGIN 360 = 1
-500 ORIGIN 360 = -860
950 ORIGIN 360 = 590

Note: The origin command also shifts the software limits.

The following illustrates the parametric program for cutting five 45° segments of a fan blade.




```
P2=0
N2
P1=.5
P140=.1
G31
N1
G0 X[P1] Y0
G1 Z0
G3 R[P1] AA0 AB45 Z[[.5-P1]/5]
G31
P1=P1+.1
IF P1 LE 2 GOTO 1
P2=P2+72
G68 AA[P2] I0 J0
IF P2 LT 360 GOTO 2
```

In the above example, the tool makes sixteen passes for each blade starting at X.5 to X2 in .1" steps. Z is 0 on the entire front edge of the blade and drops to 0 on the first pass and to -.3" on the last pass. Each blade sweep is 45 °. The N1-GOTO 1 loop is for each fan blade. The N2-GOTO 2 loop rotates each blade by an additional 72 °. P1 controls the sweep radius and Z depth. P2 controls the rotation angle.

Sample Program Using Some Special Statements

```
(Outside digitizing program. Assumes the center is 0,0.)
INPUT (Diameter) P1
INPUT (Z depth) P2
INPUT (Angle increment) P3
TI M6 (Probe)
H43 H1 D1
G0 X [P1/2+.5] Y0 (Move past the diameter +.5)
Z[P2] (To the Z depth)
P81=7 (Set byte parameter special flags to shut off trig help and cutter
        compensation)
POPEN P0 (Open file for output. Set report file to c:/parts/O0100.)
DPRNT [(Dig file for #1 diameter @ Z#2)]
DPRNT [S1500 M3]
DPRNT [X#208 Y#209]
DPRNT [Z#210]

P4=0 (Degree counter)
N1
G1 F20 (PROBE1) X0 Y0 (Move toward the center until probe touches)
DPRNT [X#208 Y#209] (Write current X and Y dimensions out to file)

PROBE 2 XC0 YC0 R[P1/2+.5] AB[P4] (Move away from the part until
        probe releases.)
P4=P4+P3 (Increment degrees)
XC0 YC0 R[SQRT[P208*P208+P209*P209]] AB[P4] (Move over)
PROBE 2 XC0 YC0 R[P1/2+.5] AB[P4] (Move away just in case we are
        touching.)
IF P4<=360 GOTO 1 (Do 360 °)
G0 Z1 (Z up)

DPRNT [Z1]
DPRNT [M5]
```

Sample Output From the Parametric Program

```
(Dig file for 4.0000 diameter @ Z-1.0000)
S1500 M3
X2.5000Y0.0000
Z-1.0000
X1.0500Y0.0000
X1.0360Y0.0906
X1.1030Y0.1945
X1.1301Y0.3028
X1.0525Y0.3831
X0.9516Y0.4437
```

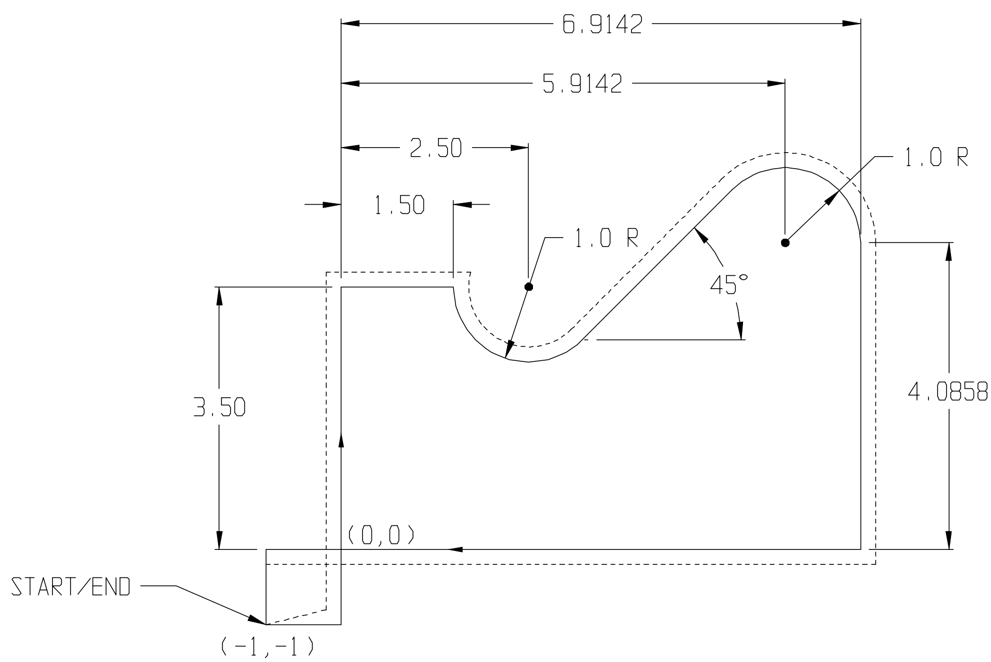
```
.
.
.
X1.0149Y-0.3694
X1.0301Y-0.3028
X1.0242Y-0.1806
X1.0560Y-0.0924
X1.13000Y-0.0000
Z1
M5
```

The following sample programs illustrate a variety of programming problems and show possible solutions to these problems using the Centurion 6 control. The program given for each sample part is by no means the only solution for that sample part.

Each sample part discussion begins with a drawing of the part, and then it gives the standard EIA (G and M codes), followed by an EIA program explanation, and finally a conversational program of the sample part. The sample part drawings for milling will include the start and end points of the programmed moves, the direction of tool travel, and the tool path indicated by dashed lines running parallel to the edge of the part. The drawings also indicate where the floating zero is set for each part. Feeds and speeds for end mills will assume a 3/8" cutter in aluminum. Other tools will be specified. Before cutting any of these sample parts, the operator must set the floating zero and tool offsets.

When entering the mill cycle geometry of a conversational program the last selection at the bottom of the line and arc screens is the end option. The end option allows the programmer to select the round corner or chamfer feature at that point in the program. If neither selection is desired the end option is skipped.

Sample 1



EIA Program Sample 1

```
N1    G0 G17 G20 G32 G40 G50 G69 G80 G90
N2    T1 M6
N3    X-1 Y-1 S3000 M03
N4    G43 H1 Z.1 M08
N5    G01 Z-.375 F5
N6    G41 D1 X0 F25
N7    Y3.5
N8    X1.5
N9    G3 R1 AA180 AB-45
N10   G1 R.S AB45
N11   G2 R1 XC5.9142 YC4.0858 AB0
N12   G1 Y0
N13   X-1
N14   G40 Y-1
N15   G0 Z.1 M9
N16   M5
```

Explanation of EIA Program Sample 1

N1 Selects rapid, XY plane, inch, and Z to tool change position; cancels cutter compensation, scaling, rotation, and canned cycles; selects absolute dimensioning. (All programs default G17, G40, G50, G69, G80, and G90. G20 is modal.)

N2 Tool change #1

N3 Positions to X-1 Y-1 and turns spindle on (3000 rpm)

N4 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N5 Feeds Z-.375 at 5 ipm

N6 Selects left cutter compensation, calls tool #1's "D" offset, and feeds to X0 Y-1 at 25 ipm
Note: The cutter compensation will ramp on *during this move.*

N7 Line move to X0 Y3.5

N8 Line move to X1.5 Y3.5

N9 CCW arc 1" radius starting at 180° , ending at -45°

N10 Line move using an estimated end point described using polar coordinates; angle 45° radius .5

N11 CW arc 1" radius absolute center of XC5.9142 YC4.0858 and an end angle of 0°

N12 Line move to Y0

N13 Line move to X-1 Y0

N14 Turn off cutter compensation during move to X-1 Y-1

N15 Rapids Z to .1 and turns off coolant

N16 Turns off spindle

Conversational Program Sample 1

Event 0 of 11
Program Setup

Program name	[SAMPLE 1]
Dimensions	[Absolute]
Units	[English]
Work Coordinate	[---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 11
Tool Change

Tool	[Change]
Tool Change Position	X[ ]
	Y[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

```
Event 2 of 11
Tool Pierce - Start Mill Cycle

 Z Pierce Feedrate        [5  ]
 Return Point             [Clearance]
 Clearance                [.1   ]
 Final Z Depth            [-.375  ]
 1st Z Depth              [-.375  ]
 Z Increment              [1    ]
 X Pierce Point           X[0    ]
 Y Pierce Point           Y[0    ]
 Compensation             [Auto Left]

 Options [---]
---------------------------------------------------
Event 3 of 11
Mill Geometry - Line

 Feedrate                 F[   ]
 Coordinates              [Cartesian]

 X axis                   X[    ]
 Y axis                   Y[3.5   ]
 Z axis                   Z[    ]

 End                      [---]

 Extend Back              [Off ]
---------------------------------------------------
Event 4 of 11
Mill Geometry - Line

 Feedrate                 F[   ]
 Coordinates              [Cartesian]

 X axis                   X[1.5   ]
 Y axis                   Y[    ]
 Z axis                   Z[    ]

 End                      [---]

 Extend Back              [Off ]
---------------------------------------------------
```

Event 5 of 11
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CCW]
Center	[Polar]
Arc Radius	R[1]
Start Angle	AA[180]
End Point	[Polar]
End Angle	AB[-45]
	Z[ ]
End Option	[---]

Event 6 of 11
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Polar]
Plane	[XY]
Type	[Current]
Length	R[.5]
End Angle	AB[45]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

```
Event 7 of 11
Mill Geometry - Arc
 Plane                         [XY]

 Feedrate                      F[   ]
 Direction                     [CW]
 Center                        [Abs Center]
 Arc Radius                    R[1    ]
 Arc Center                    XC[5.9142 ]
                               YC[4.0858 ]

 End Point                     [Polar]
 End Angle                     AB[0    ]
                               Z[    ]

 End Option                    [---]
----------------------------------------------------
Event 8 of 11
Mill Geometry - Line

 Feedrate                      F[   ]
 Coordinates                   [Cartesian]


 X axis                        X[    ]
 Y axis                        Y[0    ]
 Z axis                        Z[    ]

 End                           [---]

 Extend Back                   [Off ]
----------------------------------------------------
Event 9 of 11
Mill Geometry - Line

 Feedrate                      F[   ]
 Coordinates                   [Cartesian]

 X axis                        X[0    ]
 Y axis                        Y[    ]
 Z axis                        Z[    ]

 End                           [---]

 Extend Back                   [Off ]
----------------------------------------------------
```

Event 10 of 11
Tool Retract
End Mill Cycle

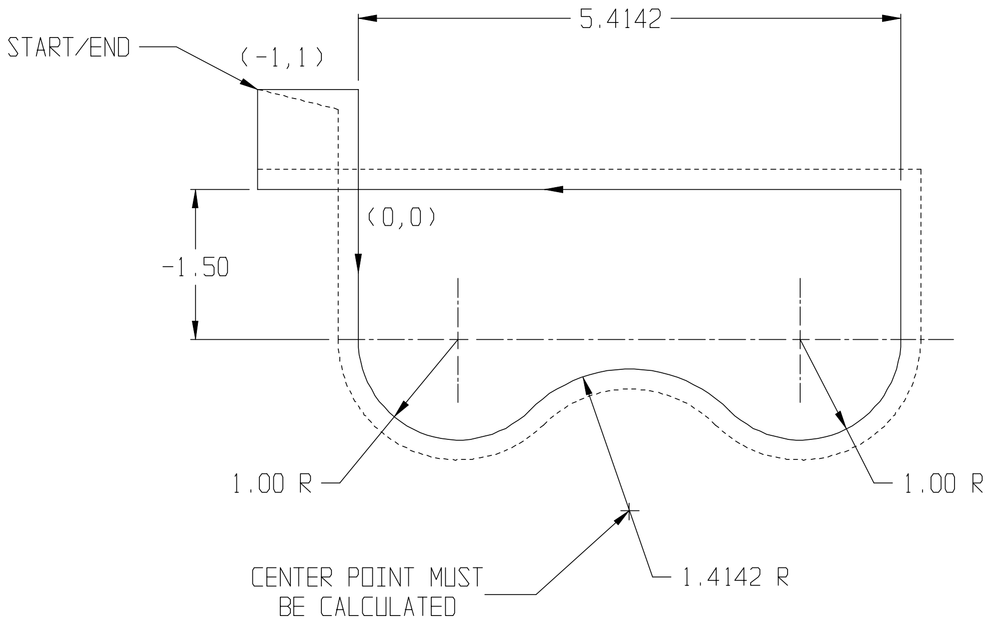
Point on part after tool retract [Auto]

Event 11 of 11
End of Program

Spindle off [Yes]
Coolant off [Yes]
Z to Toolchange [No]

X Position (home relative) [ ]
Y Position (home relative) [ ]

Sample 2A



EIA Program Sample 2A

```
N1   T1 M6
N2   G0 X-1 Y1 S3000 M3
N3   G43 H1 Z.1 M8
N4   G1 Z-.375 F5
N5   G42 D1 X0 F25
N6   Y-1.5
N7   G3 XC1 YC-1.5 AB-45 R1
N8   G2XC2.7071 YC-3.2071 AB-45 R1.4142
N9   G3 XC4.4142 YC-1.5 AB0 R1
N10  G1 Y0
N11  X-1
N12  G40 Y1
N13  G0 Z.1 M9
N14  M05
```

Explanation of EIA Program Sample 2A

N1 Tool change #1

N2 Rapid position to X-1 Y1; turns spindle on CW (3000 rpm)

N3 Calls tool #1's "H" offset and positions Z to .1; turns on coolant

N4 Feeds Z-.375 at 5 ipm

N5 Selects right cutter compensation, calls tool #1's "D" offset, and moves to X0 at 25 ipm
Note: Cutter compensation will "ramp on" during this move.

N6 Line move to X0 Y-1.5

N7 CCW arc 1" radius using an XC1 YC-1.5 and an estimated end angle of -45°

N8 CW arc 1.4142 radius using an XC2.7071 YC-3.197 and an estimated end angle of 45°
Note: The center point must be calculated prior to programming.

N9 CCW arc 1" radius using an CX4.4142 YC-1.5 and an end angle of 0°

N10 Line move to X5.4142 Y0

N11 Line move to X-1 Y0

N12 Turn off cutter compensation, move to X-1 Y1

N13 Rapid Z axis to .1, turns off coolant

N14 Turns off spindle

Conversational Program Sample 2A

Event 0 of 10
Program Setup

Program name	[SAMPLE 2]
Dimensions	[Absolute]
Units	[English]
Work Coordinate	[---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 10
Tool Change

Tool	[Change]
Tool Change Position	X[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

```
Event 2 of 10
Tool Pierce - Start Mill Cycle

 Z Pierce Feedrate          [5  ]
 Return Point               [Clearance]
 Clearance                  [.1   ]
 Final Z Depth              [-.375  ]
 1st Z Depth                [-.375  ]
 Z Increment                [1    ]

 X Pierce Point             X[0    ]
 Y Pierce Point             Y[0    ]
 Compensation               [Auto Right]

 Options [---]
---------------------------------------------------
Event 3 of 10
Mill Geometry - Line

 Feedrate                   F[   ]
 Coordinates                [Cartesian]

 X axis                     X[    ]
 Y axis                     Y[-1.5  ]
 Z axis                     Z[    ]

 End                        [---]

 Extend Back                [Off ]
---------------------------------------------------
Event 4 of 10
Mill Geometry - Arc
 Plane                      [XY]
 Feedrate                   F[   ]
 Direction                  [CCW]
 Center                     [Abs Center]
 Arc Radius                 R[1    ]
 Arc Center                 XC[1    ]
                            YC[-1.5  ]
 End Point                  [Polar]
 End Angle                  AB[-45   ]
                            Z[    ]

 End Option                 [---]
---------------------------------------------------
```

Event 5 of 10
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CW]
Center	[Abs Center]
Arc Radius	R[1.4142]
Arc Center	XC[2.7071]
	YC[-3.197]
End Point	[Polar]
End Angle	AB[-45]
	Z[ ]
End Option	[---]

Event 6 of 10
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CCW]
Center	[Abs Center]
Arc Radius	R[1]
Arc Center	XC[4.4142]
	YC[-1.5]
End Point	[Polar]
End Angle	AB[0]
	Z[ ]
End Option	[---]

Event 7 of 10
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 8 of 10
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 9 of 10

Tool Retract
End Mill Cycle

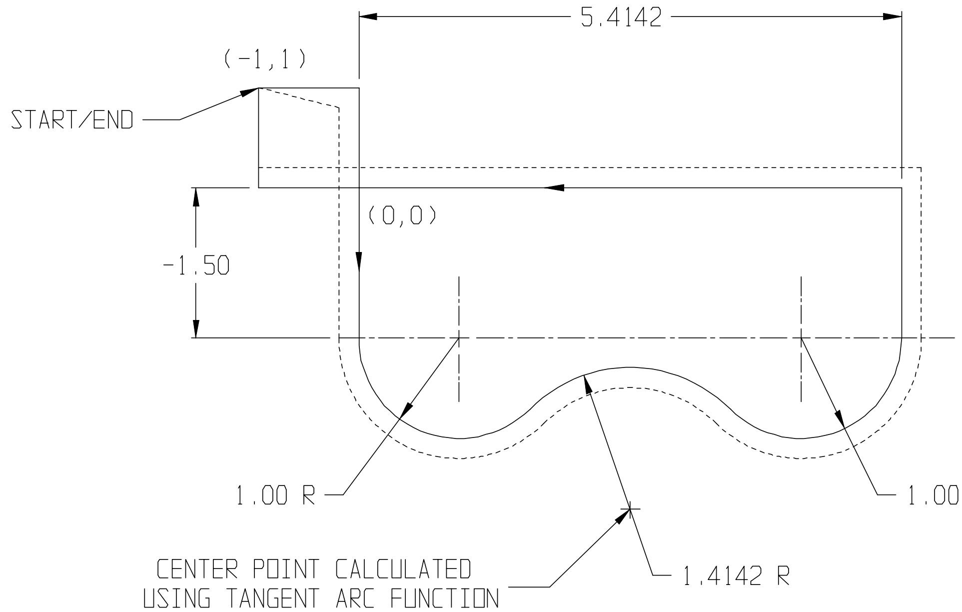
Point on part after tool retract
[Auto]

Event 10 of 10
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

Sample 2B

Same part as sample 2A but programmed using tangent arc function.



EIA Program Sample 2B

```
N1    T1 M6
N2    G0 X-1 Y1 S3000 M3
N3    G43 H1 Z.1 M8
N4    G1 Z-.375 F5
N5    G42 D1 X0 F25
N6    Y-1.5
N7    P90 = 1
N8    P91 = -1.5
N9    P92 = 1
N10   P93 = 4.4142
N11   P94 = -1.5
N12   P95 = 1
N13   P96 = 1.4142
N14   TANA C7
N15   G3 XC1 YC-1.5 AB-45 R1
N16   G2 XC[P84] YC[P85] R[P96] X[P82] Y[P83]
N17   G3  R1 XC4.4142 YC-1.5 AB0
N18   G1 Y0
N19   X-1
N20   G40 Y1
N21   G0 Z.1 M9
N22   M5
```

Explanation of EIA Program Sample 2A

N1 Tool change #1

N2 Rapid position to X-1 Y1; turns spindle on CW (3000 rpm)

N3 Calls tool #1's "H" offset and positions Z to .1; turns on coolant

N4 Feeds Z-.375 at 5 ipm

N5 Selects right cutter compensation, calls tool #1's "D" offset, and moves to X0 at 25 ipm.
Note: Cutter compensation will ramp on *during this move.*

N6 Line move to X0 Y-1.5

N7,N8 Are the X,Y center point of the first arc

N9 Radius of first arc

N10, N11 Are the X,Y center point of the second arc

N12 Radius of second arc

N13 Radius of tangent arc

N14 Tangent arc function

N15 First arc with end point

N16 Tangent arc with calculated center and end point

N17 CCW arc 1" radius using an XC4.4142 YC-1.5 and an end angle of 0°

N18 Line move to X5.4142 Y0

N19 Line move to X-1 Y0

N20 Turn off cutter compensation, move to X-1 Y1
Note: Cutter compensation will "ramp off" during this move.

N21 Rapid Z axis to .1, turns off coolant

N22 Turns off spindle

Conversational Program Sample 2B

Event 0 of 9
Program Setup

Program name	[SAMPLE 2B]
Dimensions	[Absolute]
Units	[English]
Work Coordinate	[---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 9
Tool Change

Tool	[Change]
Tool Change Position	X[ ]
	Y[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

Event 2 of 9
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[0]
Y Pierce Point	Y[0]
Compensation	[Auto Right]

Options [---]

Event 3 of 9
Mill Geometry - Line

Feedrate	F[25]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[-1.5]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

```
 Event 4 of 9
 Connect two arcs with tangent line or arc
 in the Plane                  [XY]

Mill First arc in direction    [CCW]
           R1                  [1    ]
           XC1                 [1    ]
           YC1                 [-1.5  ]

Second Arc for computation is:
           R2                  [1    ]
           XC2                 [4.4142 ]
           YC2                 [-1.5  ]

Exit 1st arc                   [Right]
Enter 2nd arc                  Right]
Connect with                   [an Arc]
Center to the                  [Right]
Radius                         [1.4142 ]
Arc Direction                  [CW]
-------------------------------------------------
 Event 5 of 9
 Mill Geometry - Arc
  Plane                        [XY]

  Feedrate                     F[   ]
  Direction                    [CCW]
  Center                       [Abs Center]
  Arc Radius                   R[1    ]
  Arc Center                   XC[4.4142 ]
                               YC[-1.5  ]
  End Point                    [Polar]
  End Angle                    AB[0    ]
                               Z[    ]

  End Option                   [---]
-------------------------------------------------
```

Event 6 of 9
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 7 of 9
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

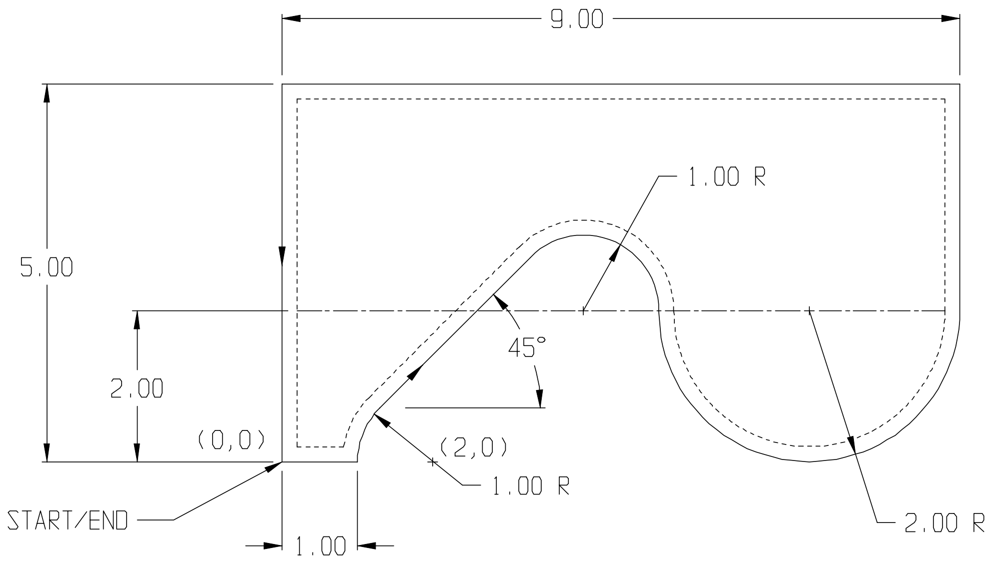
Event 8 of 9
Tool Retract
End Mill Cycle

Point on part after tool retract
[Auto]

Event 9 of 9
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

Sample 3A



EIA Program Sample 3A

N1	T1 M6
N2	G41 D1 S3000 M03
N3	G65 X0 Y99
N4	G0 Y0
N5	G43 H1 Z.1 M8
N6	G1 Z-.375 F5
N7	X1 F25
N8	G2 XC2 YC0 AB135 R1
N9	G1 AB45 R.5
N10	G2 XC4 YC2 X5 Y2 R1
N11	G3 XC7 YC2 X9 R2
N12	G1 Y5
N13	X0
N14	Y0
N15	G65 X99
N16	G40
N17	G0 Z.1 M9
N18	M5

Explanation of EIA Program Sample 3A

N1 Tool change #1

N2 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N3 Sets a "point before pierce" of X0 Y99
Note: Machine does not move to this position.

N4 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N5 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N6 Feeds Z-.375 at 5 ipm

N7 Line move to X1 Y0 at 25 ipm

N8 CW arc 1" radius using an XC2 YC0 and an end angle of AB135

N9 Line move using an estimated end point and described polarly, angle 45 radius .5"

N10 CW arc 1" radius using an XC-4 YC2 and an end point of X5 Y2

N11 CCW arc 2" radius using an XC7 YC2 and an end point of X9 Y2

N12 Line move to X9 Y5

N13 Line move to X0 Y5

N14 Line move to X0 Y0

N15 Establishes a "point after pierce" of X99 Y0
Note: Machine does not move to this position.

N16 Turns off cutter compensation

N17 Rapids Z to .1, turns off coolant

N18 Turns off spindle

Conversational Program Sample 3A

Event 0 of 12
Program Setup

Program name	[SAMPLE 3A]
Dimensions	[Absolute]
Units	[English]
Work Coordinate	[---]

Setup Notes:

[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 12
Tool Change

Tool	[Change]
Tool Change Position	X[ ]
	Y[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

Event 2 of 12
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[0]
Y Pierce Point	Y[0]
Compensation	[Left]
	[Cartesian]
X Before Pierce	X[0]
Y Before Pierce	Y[99]

Options [---]

Event 3 of 12
Mill Geometry - Line

Feedrate	F[25]
Coordinates	[Cartesian]
X axis	X[1]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 4 of 12
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CW]
Center	[Polar]
Arc Radius	R[1]
Start Angle	AA[180]
End Point	[Polar]
End Angle	AB[135]
	Z[ ]
End Option	[---]

Event 5 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Polar]
Plane	[XY]
Type	[Current]
Length	R[.5]
End Angle	AB[45]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 6 of 12
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CW]
Center	[Abs Center]
Arc Radius	R[1]
Arc Center	XC[4]
	YC[2]
End Point	[Absolute]
	X[5]
	Y[2]
	Z[ ]
End Option	[---]

Event 7 of 12
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CCW]
Center	[Abs Center]
Arc Radius	R[2]
Arc Center	XC[7]
	YC[2]
End Point	[Absolute]
	X[9]
	Y[ ]
	Z[ ]
End Option	[---]

Event 8 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[5]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 9 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 10 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 11 of 12
Tool Retract
End Mill Cycle

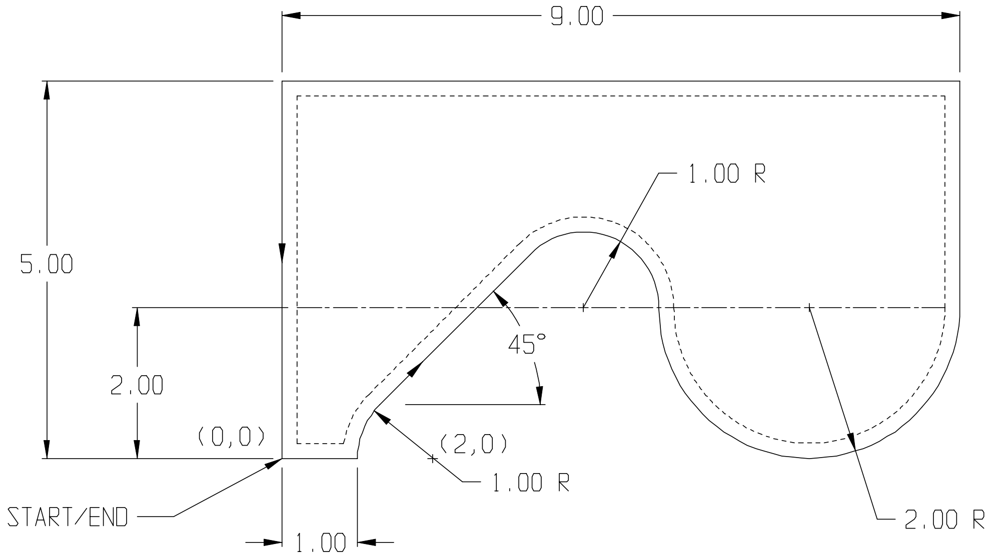
Point on part after tool retract
[Cartesian]
X[99]
Y[0]

Event 12 of 12
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

Sample 3B

Same part as sample 3A but programmed using tangent line function.



EIA Program Sample 3B

```
N1    T1 M6
N2    G41 D1 S3000 M3
N3    G65 X0 Y99
N4    G0 Y0
N5    G43 H1 Z.1 M8
N6    G1 Z-.375 F5
N7    X1 F25
N8    P90=2
N9    P91=0
N10   P92=1
N11   P93=4
N12   P94=2
N13   P95=1
N14   TANL C3 (See TANL explanation for values of C.)
N15   G2 XC[P90] YC[P91] R[P92] X[P80] Y[P81]
N16   G1 X[P82] Y[P83]
N17   G2 XC4 YC2 X5 Y2 R1
N18   G3 XC7 YC2 X9 Y2 R2
N19   G1 Y5
N20   X0
N21   Y0
N22   G65 X99
N23   G40
N24   G0 Z.1 M9
```

N25 M5

Note: Lines N8 thru N13 could be written as follows:
N9 P90=2 P91=0 P92=1 P93=4 P94=2 P95=1

Explanation of EIA Program 3B

N1 Tool change #1

N2 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N3 Sets a "point before pierce" of X0 Y99
Note: Machine does not move to this position.

N4 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N5 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N6 Feeds Z-.375 at 5 ipm

N7 Line move to X1 Y0 at 25 ipm

N8, N9 Center of the first arc

N10 Radius of the first arc

N11, N12 The center of the second arc

N13 Radius of the second arc

N14 Tangent line function

N15 First arc with calculated end points

N16 Line tangent to both arcs

N17 CW arc 1" radius using an XC7 YC2 and an end point of X5 Y2

N18 CCW arc 2" radius using an XC7 YC2 and an end point of X9 Y2

N19 Line move to X9 Y5

N20 Line move to X0 Y5

N21 Line move to X0 Y0

N22 Establishes a "point after pierce" of X99 Y0
Note: Machine does not move to this position.

N23 Turns off cutter compensation

N24 Rapids Z to .1, turns off coolant

N25 Turns off spindle

Conversational Program Sample 3B

Event 0 of 11
Program Setup

Program name [SAMPLE 3B]

Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 11
Tool Change

Tool [Change]
Tool Change Position X[ ]
Y[ ]

Tool Number T[1]
Tool Description [ ]
Next Tool Number [ ]
Spindle Speed S[3000]
Spindle Restart [CW]
Coolant [Flood]

Event 2 of 11
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[0]
Y Pierce Point	Y[0]
Compensation	[Left]
	[Cartesian]
X Before Pierce	X[0]
Y Before Pierce	Y[99]

Options [---]

Event 3 of 11
Mill Geometry - Line

Feedrate	F[25]
Coordinates	[Cartesian]
X axis	X[1]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 4 of 11
Connect two arcs with tangent line or arc
in the Plane [XY]

Mill First arc in direction [CW]
R1 [1]
XC1 [2]
YC1 [0]

Second Arc for computation is:
R2 [2]
XC2 [4]
YC2 [2]

Exit 1st arc [Left]
enter 2nd arc [Left]
Connect with [a Line]

Event 5 of 11
Mill Geometry - Arc
Plane [XY]
Feedrate F[ ]
Direction [CW]
Center [Abs Center]
Arc Radius R[1]
Arc Center XC[4]
YC[2]

End Point [Absolute]
X[5]
Y[2]
Z[ ]

End Option [---]

Event 6 of 11	
Mill Geometry - Arc	
Plane	[XY]
Feedrate	F[ ]
Direction	[CCW]
Center	[Abs Center]
Arc Radius	R[2]
Arc Center	XC[7]
	YC[2]
End Point	[Absolute]
	X[9]
	Y[ ]
	Z[ ]
End Option	[---]

Event 7 of 11	
Mill Geometry - Line	
Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[5]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 8 of 11	
Mill Geometry - Line	
Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 9 of 11
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

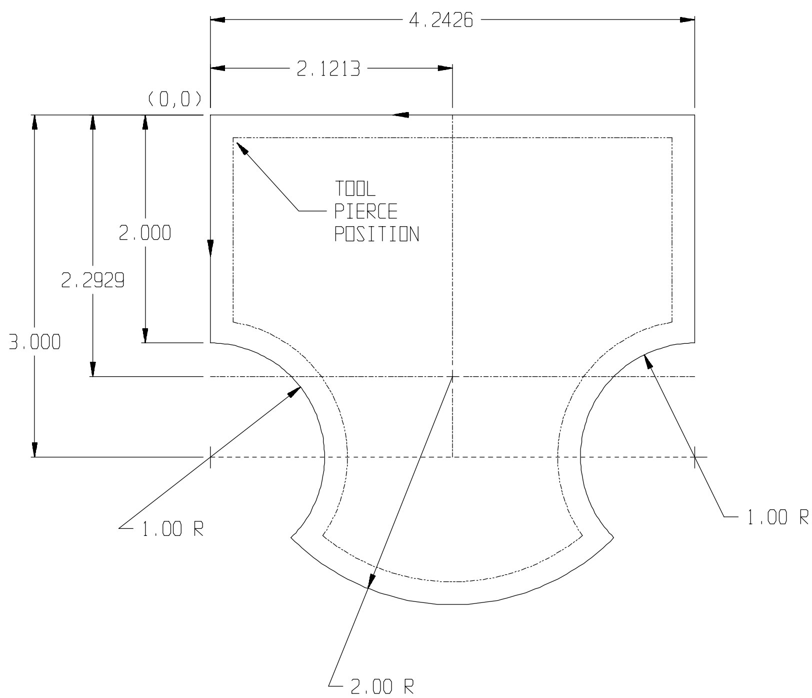
Event 10 of 11
Tool Retract
End Mill Cycle

Point on part after tool retract
[Cartesian]
X[99]
Y[0]

Event 11 of 11
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

Sample 4A



EIA Program Sample 4A

```
N1   T1 M6
N2   G41 D01 S3000 M3
N3   G65 X99 Y0
N4   G0 X0
N5   G43 H1 Z.1 M8
N6   G1 Z-.375 F5
N7   Y-2 F25
N8   G2 XC0 YC-3 AB-45 R1
N9   G3 XC2.1213 YC-2.2929 AB-45 R2
N10  G2 XC4.2426 YC-3 AB90 R1
N11  G1 Y0
N12  X0
N13  G65 Y-99
N14  G40
N15  G0 Z.1 M9
N16  M5
```

Explanation of EIA Program Sample 4A

N1 Tool change #1

N2 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N3 Establishes a "point before pierce" of X99 Y0
Note: Machine does not move to this position.

N4 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N5 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N6 Feeds Z-.375 at 5 ipm

N7 Line move to X0 Y-2 at 25 ipm

N8 CW arc 1" radius using an XC0 YC-3 and an estimated end angle of -45°

N9 CCW arc 2" radius using an XC2.1213 YC-2.2929 and an estimated end angle of -45 °

N10 CW arc 1" radius using an XC4.2426 YC-3 and an end angle of 90°

N11 Line move to X4.2426 Y0

N12 Line move to X0 Y0

N13 Establishes a point after retract of X0 Y99
Note: Machine does not move to this position.

N14 Cancels cutter compensation

N15 Rapids Z to .1, turns coolant off

N16 Turns spindle off

Conversational Program Sample 4A

```
Event 0 of 10
Program Setup

 Program name            [SAMPLE 4A     ]

 Dimensions              [Absolute]
 Units                   [English]
 Work Coordinate         [---]

Setup Notes:
[                ]
[                ]


[                ]
[                ]
[                ]
[                ]
--------------------------------------------------
Event 1 of 10
Tool Change

 Tool                    [Change]
 Tool Change Position    X[   ]
                         Y[   ]

 Tool Number             T[1 ]
 Tool Description        [         ]
 Next Tool Number        [ ]
 Spindle Speed           S[3000]
 Spindle Restart         [CW]
 Coolant                 [Flood]
--------------------------------------------------
```

Event 2 of 10
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[0]
Y Pierce Point	Y[0]
Compensation	[Left]
	[Cartesian]
X Before Pierce	X[99]
Y Before Pierce	Y[0]

Options [---]

Event 3 of 10
Mill Geometry - Line

Feedrate	F[25]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[-2]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

```
Event 4 of 10
Mill Geometry - Arc
 Plane                    [XY]
 Feedrate                 F[  ]
 Direction                [CW]
 Center                   [Abs Center]
 Arc Radius               R[1   ]
 Arc Center               XC[0   ]
                          YC[-3  ]
 End Point                [Polar]
 End Angle                AB[-45  ]
                          Z[   ]

 End Option               [---]
---------------------------------------------------
Event 5 of 10
Mill Geometry - Arc
 Plane                    [XY]
 Feedrate                 F[  ]
 Direction                [CCW]
 Center                   [Abs Center]
 Arc Radius               R[2   ]
 Arc Center               XC[2.1213 ]
                          YC[-2.2929 ]
 End Point                [Polar]
 End Angle                AB[-45  ]
                          Z[   ]

 End Option               [---]
---------------------------------------------------
Event 6 of 10
Mill Geometry - Arc
 Plane                    [XY]
 Feedrate                 F[  ]
 Direction                [CW]
 Center                   [Abs Center]
 Arc Radius               R[1   ]
 Arc Center               XC[4.2326 ]
                          YC[-3  ]
 End Point                [Polar]
 End Angle                AB[90  ]
                          Z[   ]

 End Option               [---]
---------------------------------------------------
```

Event 7 of 10
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 8 of 10
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 9 of 10
Tool Retract
End Mill Cycle

Point on part after tool retract
[Cartesian]
X[0]
Y[-99]

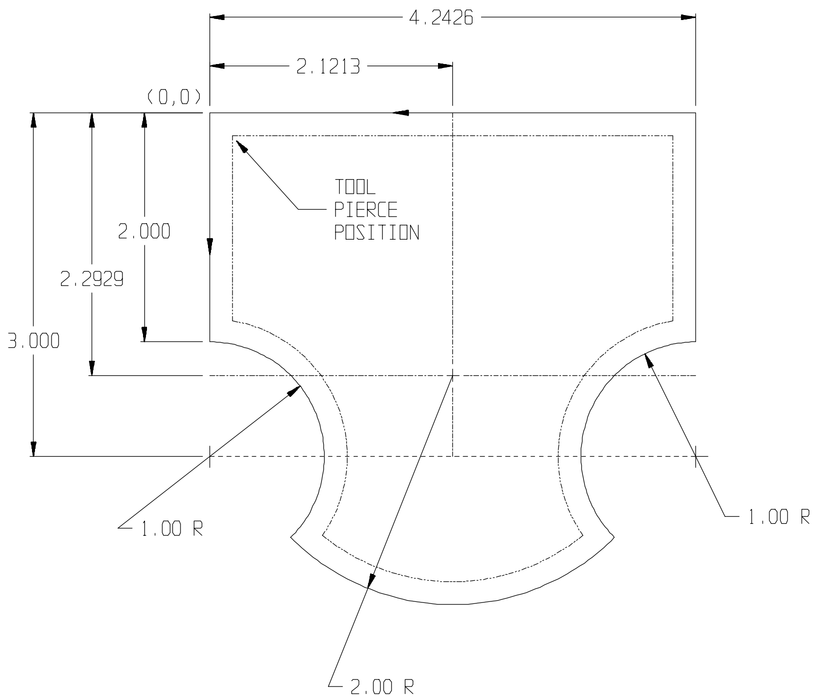
Event 10 of 10
End of Program

Spindle off [Yes]
Coolant off [Yes]
Z to Toolchange [No]

X Position (home relative)[ ]
Y Position (home relative)[ ]

Sample 4B

Programming arc using 3 point circle generate. Points X1, X2, X3 are the points used to program each arc.



EIA Program Sample 4B

```
N1    T1 M6
N2    G41 D1 S3000 M03
N3    G65 X99 Y0
N4    X0
N5    G43 H1 Z.1 M8
N6    G1 Z-.375 F5
N7    Y-2 F25
N8    P90=0
N9    P91=-2
N10   P92=1
N11   P93=-3
N12   P94=0
N13   P95=-4
N14   CGEN
N15   G2 XC[P80] YC[P81] R[P82] AB300
N16   P90=.1213
N17   P91=-2.2929
N18   P92=2.1213
```

```
N19    P93=-4.2929
N20    P94=4.1213
N21    P95=-2.2929
N22    CGEN
N23    G3 XC[P80] YC[P81] R[P82] AB300
N24    P90=4.2426
N25    P91=-4
N26    P92=3.2426
N27    P93=-3
N28    P94=4.2426
N29    P95=-2
N30    CGEN
N31    G2 XC[P80] YC[P81] R[P82] X[P94] Y[P95]
N32    G1 Y0
N33    X0
N34    G65 Y-99
N35    G40
N36    G0 Z.1 M9
N37    M5
```

Explanation of EIA Program 4B

N1 Tool change #1

N2 Selects left cutter compensation, activates tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N3 Establishes a "point before pierce" of X99 Y0
Note: Machine does not move to this position.

N4 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N5 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N6 Feeds Z-.375 at 5 ipm

N7 Line move to X0 Y-2 at 25 ipm

N8-N13 Are the coordinates of 3 points on the first circle

N14 Calculates circle based on the 3 points
N15 Arc command which moves to the calculated points

N16- N21 Are the coordinates of 3 points on the second circle

N22 Calculates second circle based on the 3 points

N23 Arc command which moves to the calculated points

N24- N29 Are the coordinates of 3 points on the third circle

N30 Calculates third circle based on the 3 points

N31 Arc command which moves to the calculated points

N32 Line move to X4.2426 Y0

N33 Line move to X0 Y0

N34 Establishes a point after retract of X0 Y99
Note: Machine does not move to this position.

N35 Cancels cutter compensation

N36 Rapids Z to .1, turns coolant off

N37 Turns spindle off

Conversational Program Sample 4B

Event 0 of 10
Program Setup

Program name [SAMPLE 4B]
Dimensions [Absolute]
Units [English]
Work Coordinate [---]

Setup Notes:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 10
Tool Change

Tool [Change]	
Tool Change Position	X[ ]
	Y[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

Event 2 of 10
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[0]
Y Pierce Point	Y[0]
Compensation	[Left]
	[Cartesian]
X Before Pierce	X[99]
Y Before Pierce	Y[0]
Options [---]	

```
 Event 3 of 10
 Mill Geometry - Line

  Feedrate                     F[25  ]
  Coordinates                  [Cartesian]

  X axis                       X[    ]
  Y axis                       Y[-2   ]
  Z axis                       Z[    ]

  End                          [---]

  Extend Back                  [Off ]
---------------------------------------------------
 Event 4 of 10
 Three Point Circle Generator
  Plane                        [XY]
  Direction                    [CW]
X1 [0    ]    Y1 [-2   ]
X2 [1    ]    Y2 [-3   ]
X3 [0    ]    Y3 [-4   ]

  Use X3,Y3 as End Point       [No]
  End Angle                    [300   ]
---------------------------------------------------
 Event 5 of 10
 Three Point Circle Generator
  Plane                        [XY]
  Direction                    [CCW]
X1 [.1213  ]    Y1 [-2.2929 ]
X2 [2.1213 ]    Y2 [-4.2929 ]
X3 [4.1213 ]    Y3 [-2.2929 ]

  Use X3,Y3 as End Point        [No]
  End Angle                    [300   ]
---------------------------------------------------
 Event 6 of 10
 Three Point Circle Generator
  Plane                        [XY]
  Direction                    [CW]
X1 [4.2426 ]    Y1 [-4   ]
X2 [3.2426 ]    Y2 [-3   ]
X3 [4.2426 ]    Y3 [-2   ]

  Use X3,Y3 as End Point        [Yes]
---------------------------------------------------
```

Event 7 of 10
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 8 of 10
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

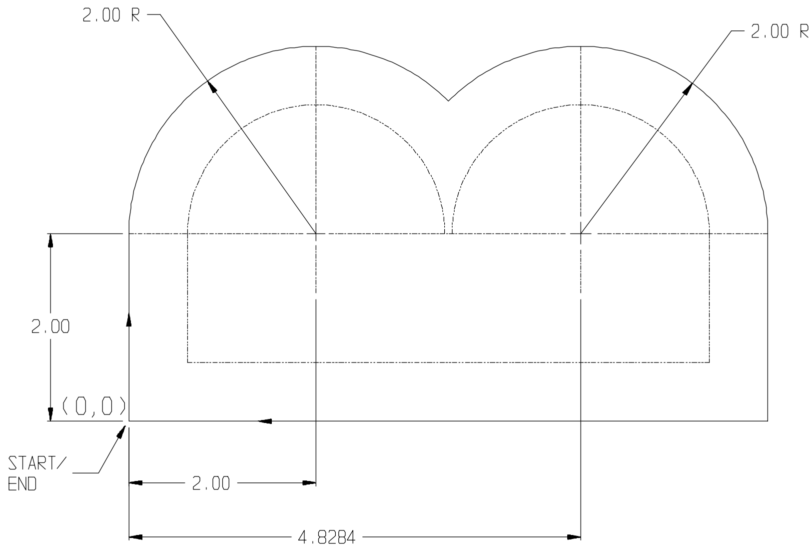
Event 9 of 10
Tool Retract
End Mill Cycle

Point on part after tool retract
[Cartesian]
X[0]
Y[-99]

Event 10 of 10
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

Sample 5



EIA Program Sample 5

```
N1    T1 M6
N2    G42 D1 S3000 M03
N3    G65 X99 Y0
N4    G0 X0
N5    G43 H1 Z.1 M8
N6    G1 Z-.375 F5
N7    Y2 F25
N8    G2 XC2 YC2 AB45 R2,R.0001
N9    X5 YC2 AB0 R2
N10   G1 Y0
N11   X0
N12   G65 Y99
N13   G40
N14   G0 Z.1 M9
N15   M5
```

Explanation of EIA Program 5

N1 Tool change #1

N2 Selects right cutter compensation, calls tool #1's "D" offset, and turns on spindle CW (3000 rpm)

N3 Establishes a "point before pierce" of X99 Y0
Note: Machine does not move to this position.

N4 Sets a "pierce point" of X0 Y0; moves to its compensated point as established by the previous block

N5 Calls tool #1's "H" offset, positions Z to .1, and turns on coolant

N6 Feeds Z-.375 at 5 ipm

N7 Line move to X0 Y2 at 25 ipm

N8 CW arc 2" radius using an XC2 YC2, an estimated end angle of 45° degrees, and an end option (round corner) of .0001" radius
Note: The end option of a .0001 radius forces an intersection between the arcs.

N9 CW arc 2" radius using an XC4.8284 YC2 and an end angle of 0 degrees

N10 Line move to X6.8284 Y0

N11 Line move to X0 Y0

N12 Establishes a point after retract of X0 Y99
Note: Machine does not move to this position.

N13 Turns off cutter compensation

N14 Rapids Z to .1, turns coolant off

N15 Turns spindle off

<u>Conversational Program Sample 5</u>

```
Event 0 of 9
Program Setup
 Program name            [SAMPLE 5     ]

 Dimensions              [Absolute]
 Units                   [English]
 Work Coordinate         [---]

Setup Notes:
[              ]
[              ]
[              ]
[              ]
[              ]
[              ]
--------------------------------------------------
```

Event 1 of 9
Tool Change

Tool	[Change]
Tool Change Position	X[ ]
	Y[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

Event 2 of 9
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[0]
Y Pierce Point	Y[0]
Compensation	[Right]
	[Cartesian]
X Before Pierce	X[99]
Y Before Pierce	Y[0]

Options [---]

Event 3 of 9
Mill Geometry - Line

Feedrate	F[25]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[2]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 4 of 9
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CW]
Center	[Abs Center]
Arc Radius	R[2]
Arc Center	XC[2]
	YC[2]
End Point	[Polar]
End Angle	AB[45]
	Z[ ]
End Option	[Round Corner]
Radius	[.0001]

Event 5 of 9
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CW]
Center	[Abs Center]
Arc Radius	R[2]
Arc Center	XC[4.8284]
	YC[2]
End Point	[Polar]
End Angle	AB[0]
	Z[ ]

End Option [---]

Event 6 of 9
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 7 of 9
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 8 of 9
Tool Retract
End Mill Cycle

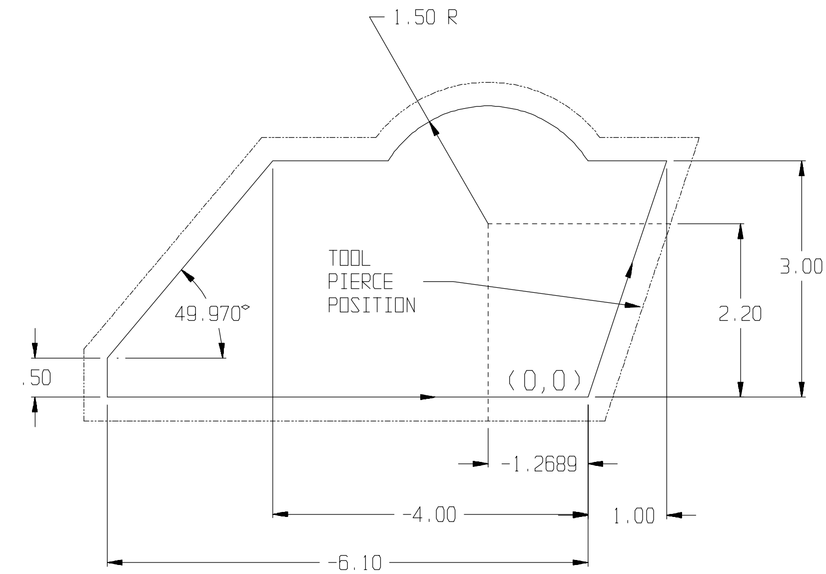
Point on part after tool retract

	[Cartesian]
	X[0]
	Y[99]

Event 9 of 9
End of Program

Spindle off	[Yes]
Coolant off	[Yes]
Z to Toolchange	[No]
X Position (home relative)	[ ]
Y Position (home relative)	[ ]

Sample 6



EIA Program Sample 6

```
N1    T1 M6
N2    G42 D1 S3000 M3
N3    G65 X0 Y0
N4    X.5 Y1.5
N5    G43 H1 Z.1 M8
N6    G1 Z-.375 F5
N7    X1 Y3 F25
N8    X0
N9    G3 XC-1.2689 YC2.2 X-3 R1.5
N10   G1 X-4
N11   X-6.1 Y.5
N12   Y0
N13   X0
N14   X.5 Y1.5
N15   G65 X1 Y3
N16   G40
N17   G0 Z.1
N18   M30
```

Explanation of EIA Program Sample 6

N1 Tool change #1

N2 Selects right cutter compensation, activates #1's "D" offset, and turns on spindle CW (3000 rpm)

N3 Establishes a "point before pierce" of X0 Y0
Note: Machine does not move to this position.

N4 Establishes a "pierce point" of X.5 Y1.5; moves to its compensated point as established by the previous block

N5 Calls tool #1's "H" offset, positions Z to .1, and turns coolant on

N6 Feeds Z-.375 at 5 ipm

N7 Line move to X1 Y3 at 25 ipm

N8 Line move to an estimated end point of X0

N9 CCW arc 1.5" radius using an XC-1.2689 YC2.2 and an estimated end point of X-3

N10 Line move to X-4 Y3

N11 Line move to X-6.1 Y.5

N12 Line move to X-6.1 Y0

N13 Line move to X0 Y0

N14 Line move to X.5 Y1.5

N15 Establishes a "point after retract" of X1 Y3
Note: Machine does not move to this position.

N16 Turns cutter compensation off

N17 Rapids Z to .1

N18 Ends program, turns spindle and coolant off

Conversational Program Sample 6

Event 0 of 11
Program Setup

Program name	[SAMPLE 6]
Dimensions	[Absolute]
Units	[English]
Work Coordinate	[---]

Setup Notes:

[ ]
[ ]
[ ]
[ ]
[ ]
[ ]

Event 1 of 12
Tool Change

Tool	[Change]
Tool Change Position	X[ ]
	Y[ ]
Tool Number	T[1]
Tool Description	[ ]
Next Tool Number	[ ]
Spindle Speed	S[3000]
Spindle Restart	[CW]
Coolant	[Flood]

Event 2 of 12
Tool Pierce - Start Mill Cycle

Z Pierce Feedrate	[5]
Return Point	[Clearance]
Clearance	[.1]
Final Z Depth	[-.375]
1st Z Depth	[-.375]
Z Increment	[1]
X Pierce Point	X[.5]
Y Pierce Point	Y[1.5]
Compensation	[Auto Right]

Options [---]

Event 3 of 12
Mill Geometry - Line

Feedrate	F[25]
Coordinates	[Cartesian]
X axis	X[1]
Y axis	Y[3]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 4 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 5 of 12
Mill Geometry - Arc

Plane	[XY]
Feedrate	F[ ]
Direction	[CCW]
Center	[Abs Center]
Arc Radius	R[1.5]
Arc Center	XC[-1.2689]
	YC[2.2]
End Point	[Absolute]
	X[-3]
	Y[ ]
	Z[ ]
End Option	[---]

Event 6 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[-4]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 7 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[-6.1]
Y axis	Y[.5]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 8 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[ ]
Y axis	Y[0]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 9 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[.5]
Y axis	Y[1.5]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

Event 10 of 12
Mill Geometry - Line

Feedrate	F[ ]
Coordinates	[Cartesian]
X axis	X[0]
Y axis	Y[ ]
Z axis	Z[ ]
End	[---]
Extend Back	[Off]

```
---------------------------------------------------
 Event 11 of 12
 Tool Retract

 End Mill Cycle

  Point on part after tool retract
                                   [Auto]
---------------------------------------------------
 Event 12 of 12
End of Program

 Spindle off                      [Yes]
 Coolant off                      [Yes]
 Z to Toolchange                  [No]

 X Position (home relative)   [    ]
 Y Position (home relative)   [    ]
```

Sample 7

This sample program uses the rotary axis.

The "A" axis is programmed in decimal degrees in XXX.XXX format and performs linear interpolation with the X, Y, and Z axes.

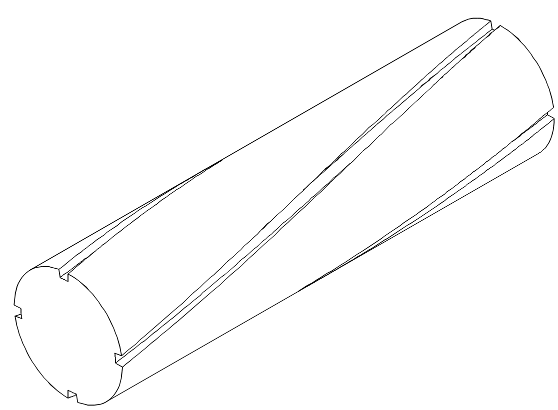
The feedrate for the rotary axis is specified in degrees per minute divided by 10.

Example: G1 A90 F18.0

In the above example, A will feed to 90° at a rate of 180 DPM (degrees per minute).

Example of Rotary Axis Programming Sample 7

03119	A axis moves - Cut 4 helical slots
G20 G90	
N1 G0 Z.1	Position Z above the work piece
N2 X0 Y0 A0	Position X, Y, and A to the start point
N3 G1 F20 Z-.1	Feed Z to depth
N4 F15 X-5 A90	XA interpolation move
N5 G0 Z.1	Position Z above the work piece
N6 X0	Position X back to the start point
N7 G1 F20 Z-.1	Second slot
N8 F15 X-5 A180	
N9 G0 Z.1	
N10 X0	
N11 G1 F20 Z-.1	Third slot
N12 F15 X-5 A270	
N13 G0 Z.1	
N14 X0	
N15 G1 F20 Z-.1	Fourth slot
N16 F15 X-5 A360	
N17 G0 Z.1	
N18 X0	
(End of 03119)	



Error Messages

001 ***Invalid function number***
Note what just occurred and call for technical support. A call was made to a non-existent DOS function.

002 ***File not found***
File name specified as OLD does not exist. Try MENU.

003 ***Path not found***
The drive or subdirectory specified does not exist.

004 ***Too many open files***
Check Config.sys for FILES=20.

005 ***File access denied***
The file may be read only or is on a write-protected disk.

006 ***Invalid file handle***

007 ***Memory control blocks destroyed***

008 ***Insufficient memory***
The program being loaded is too large to fit into memory. A program being run or verified is too large to run in the control's memory. Try DNC mode.

009 ***Invalid memory block address***

010 ***Invalid environment***

011 ***Invalid format***

012 ***Invalid file access code***

013 ***Invalid data***

015 ***Invalid drive specified***
Drive specified does not exist.

016 ***Cannot remove current directory***
Note what just occurred and call for technical support.

017 ***Cannot rename across drives***

018 ***No more files***

100 Disk read error

An attempt was made to edit a file that has been corrupted in some way, perhaps loss of power while editing, or an error 101 occurred while editing. Try a different file to see if the problem is specific to one particular file. If this is the case, the program must be recreated.

Note: This error occurs when trying to edit conversational files that were created with SLS software.

101 Disk write error - Parts memory is full

To avoid this error, remove programs from memory when you are done using them (store on a floppy). Also, watch the amount of memory available as you are programming. Deleting some programs from the parts memory will free up space for additional programs.

Procedure to recover from ERROR 101

(Disk write error: Parts memory full)

Type in commands that are shown in CAPITAL letters, followed by the ENT key.

1) ENTER (C:/RAM should be displayed)
2) CD.. (change directory)
3) CD PARTS (change to the parts directory)
4) DIR (list PART files)
5) DEL O####(#### is your part number)

NOTE: You can recreate your O#### file from your conversational (P####) file by reposting it.

6) Power the machine OFF, then ON again, to RESET the control. Use UTIL-FILES-ERASE to erase programs that are no longer needed.

NOTE: You can check how much parts memory is available by using UTIL-INFO to look at the information page.

102 File not assigned

103 File not open

104 File not open for input

105 File not open for output

106 Invalid numeric format

150 ***Disk is write-protected***
Check the write protect tab on the floppy disk that is being used.

151 ***Unknown unit***

152 ***Drive not ready***
Check to see that there is a disk in the floppy drive.

153 ***Unknown command***

154 ***CRC error in data***
The disk being read is corrupted.

155 ***Bad drive request structure length***

156 ***Disk seek error***
Check the cabling from the control to the floppy drive.

157 ***Unknown media type***

158 ***Sector not found***

159 ***Printer out of paper***

160 ***Device write fault***

161 ***Device read fault***

162 ***Hardware failure***
Improper format on the floppy disk. Try another disk or try reformatting the same disk. Verify that floppy cables are properly connected.

163 ***Sharing Violation***

200 ***Division by zero***

201 ***Range check error***

202 ***Stack overflow***

203 ***Heap overflow - Insufficient RAM memory***
Run the DOS command CHKDSK to determine the amount of RAM that is available on the system. If possible, unload unnecessary device drivers that are loaded on the PC before starting the CNC.

<u>Steps to take to avoid ERROR 203</u>
(Heap overflow: Insufficient RAM memory)

If text cycles or canned cycles are being loaded and not being used, turn them off. See PARMS-SETUP-MISC.

1) (F7) PARMS
2) (F9) CTRL
3) Move cursor to Load Text Cycles
4) Toggle to "No"
5) (ESC)
6) (ESC) to the main menu.
7) Power the machine OFF, then ON, again.

If you are running a large program, try running it through the RS-232 RUN mode.

204 Invalid pointer operation

205 Floating point overflow

206 Floating point underflow

207 Invalid floating point operation

208 TP Overlay Error

209 TP Overlay Error

210 TP Object Error

211 TP Object Error

212 TP Object Error

213 TP Object Error

214 TP Object Error

300 Program already exists
An attempt was made to use a program name already in use. Use a different name.

301 Invalid program number
Valid program numbers are 1 through 9999.
* Simulator program numbers are 1 through 15.

302 No programs to select from

This error may occur anytime a menu is being created for file selection when there are no files. There may be an unformatted disk in the floppy drive. Parts memory may be empty.

303 *Problem saving program(s) to disk*
There is no floppy disk in the disk drive. The floppy disk may not have room to store additional files. There may be an unformatted disk in the drive.

304 *Problem loading program(s) from disk*
Disk was removed from floppy drive after setting files.

305 *Not formatted for conversational. Try text editor.*

306 *Conversational system has been corrupted*

307 *Illegal event number*
Event number in conversational program is negative.

308 *Invalid tool number*
Tool number is less than zero or greater than 99.

309 *Can't copy or rename a file to itself*
Try using a different file name.

310 *File not formatted for conversational or parameters*
Problem receiving a file via RS-232. Check to see if the proper file was sent.

311 *Parameter file not valid*
Problem receiving parameter file via RS-232. Check to see if the parameters were sent.

312 *Insufficient parts storage*
Parts memory is full. Try erasing some programs from the parts directory (UTILS-FILES-ERASE).
Program on floppy disk may be too large to fit into the control's memory.

313 *Insufficient storage for compression, unable to post the file*
Erase some programs from the parts memory.

314 *Insufficient storage, post has been aborted*
Erase some programs from the parts memory.

315 *Out of storage space on the floppy*
Floppy disk is full or has too many programs. There is a limit of 224 files that can be stored in the root directory on a floppy disk. A sub-directory can be created on

the floppy disk and the floppy path changed to save files to the sub-directory. This allows full use of the disk space.

316 Not enough storage to create a new file
There is not enough parts space to create a new conversational program. Erase unnecessary programs to free up parts space.

317 Simulator allows only 15 events

319 Send file aborted

320 Invalid PLC program.
One of the ACRO.HEX (or NCB.HEX) files in the RAM directory is corrupt.

321 Error in program O.
A custom MCODE or customer GCODE has an error.

322 or custom G code program not found.

323 or custom M code program not found.

324 Invalid path or program not found for M6 tool change macro.
The tool change macro may have an error in it.

400 Home required
The machine must be homed before any axis movement can take place on the machine, i.e. MDI, JOG, HDW, etc.

401 X axis software limit overtravel
402 Y axis software limit overtravel
403 Z axis software limit overtravel
404 A axis software limit overtravel
405 B axis software limit overtravel
406 C axis software limit overtravel

These errors are a result of the axis reaching the programmed limits of travel.

When jogging or handwheeling and an axis limit is reached, the control will only allow movement in one direction.

In a program the error may be encountered. The G92 and/or G54 parameters may need to be modified.

Tool length offsets are also a potential cause of Z axis overtravel.

407 X axis excess error condition
408 Y axis excess error condition
409 Z axis excess error condition
410 A axis excess error condition
411 B axis excess error condition
412 C axis excess error condition

These errors are caused by the axis not being able to keep up with the programmed move at the programmed speed.

Does the error occur during rapid moves only? Y__ N__
If so, check bus voltage and rapid feed parameters.

Do any of the drive cards have red LED's lit? Y__ N__
If so, which light on which card? VOL, GF, RMS, or SG

Other causes might be too heavy of a cut, worn tool, low ∀15 volts, accel/decel parameters, or drive card failure.

413 Attempted to move into safe zone

415 Can't establish DNC link while program is running
The program being run must be halted before the DNC link can be established.

416 Out of position, arc end point is not on the arc

417 Can't edit parameters while program is running
The program must be halted before editing parameters.
Is the program in block mode or feedhold?

418 Controller Card Error

448 Air Pressure Error. Check air hose connection

449 ERROR 449: Emergency stop: Axis drive fault. Reset servo drives or power down.

450 ERROR 450: Emergency stop condition. Reset drives or press <ESC> to clear this message.

451 Lube fault, Check Waylube Level
The float switch on the autolube pump is indicating that oil needs to be added to the autolube tank.

452 Tool not found in auto tool sequence
Check UTIL-TLCHG-SLOTS for that tool number.

453 Tool pot not up during turret movement
Check to see if the POT UP switch is functioning as it should be.

454 Not at tool change position
Try commanding a G32 before the M6 command.

455 ATC arm is not out, axis movement not allowed

456 Can't process auto tool change after switching to manual

457 Move length too great for control (split into smaller moves)

458 X axis is in the manual mode. Push reset to enable it

459 Y axis is in the manual mode. Push reset to enable it

460 Feedrate too large for control.

500 Last softkey pressed is not supported at this time

501 Illegal address "#" encountered
The character within quotes " " is not a valid address, such as X, Y, Z, R, G, etc. The block where the error occurred is shown in the block display. Check that block for the invalid address.

502 Undefined canned cycle

503 Return without gosub
Refer to page 328, Section 5 about gosub and return.

504 Coincident points, the start point and end point are the same on an arc without a center
The start point and end point are the same on an arc without a center.

505 Radius too small to span given points
Start and end points are more than "R" distance apart.

507 After compensation, line to arc lacks intersection

509 After compensation, arcs lack intersection

515 Unexpected file size
The control read a file that was not the correct size.

517 Parameter out of range
Parameter number is less than zero. For parameter numbers greater than 699 you must use data mode (G10, G11).

518 Illegal program statement
Command in program statement is not considered valid.

519 Feedrate out of range
The programmed feedrate is beyond the "maximum feedrate" parameter value in the machine setup parameters. The program feedrate may be negative.

520 Spindle speed out of range
The programmed spindle speed is beyond the "spindle range" parameter in the machine setup parameters. The programmed spindle speed may be negative.

521 Negative arc radius
An attempt was made to generate a negative arc radius.

522 Negative polar radius
A polar radius must be specified as a positive value.

523 Illegal tool number
Valid T numbers are 0 - 99.

524 Illegal radius number
Valid D numbers are 0 - 99.

525 Illegal length number
Valid H numbers are 0 - 99.

526 Invalid access code
The access code does not match that which is loaded in the machine setup parameters.

527 Invalid access level
Valid levels are 0 - 4.

529 Duplicate address' encountered
The same address was found twice on the same block, such as X0 Y0 X.5.

530 Colinear line to line in round corner

531 Colinear line to arc in round corner

532 Colinear arc to line in round corner

533 Colinear arc to arc in round corner

535 Chamfer length is < 0
Chamfer length must be a positive number.

536 Can't chamfer and round the same corner
Choose either chamfer or round corner.

537 Can't chamfer to or from arcs

538 Loop counter out of range
The maximum number of loops for a call is 999.

539 Dwell time out of range
Probably a negative number was specified. The maximum dwell time is 999999999 seconds.

540 Illegal dwell time "#" encountered
Try G4 F##.####; specify X, P, or F after G4.

541 No axis moves are allowed on a G31 or G32 block
G31 and G32 are intended to move Z only. Relocate X and Y moves to another block.

542 G30 Illegal return to reference parameter on G30 block
Should be P2, P3, or P4 for second, third, and fourth reference point.

543 Illegal G10 statement

544 Too many digits in number
The number of digits used is beyond what the address is expecting. For example, G100 should only be two digits.

545 Illegal K value for number of holes
K must be a number between 1 and 1000.

546 Nested calls or gosubs too deep
Probably a program is calling itself. Nest limit is 50 for program calls.

547 Comment not closed
Always use "( )" (parentheses) in pairs for program comments.

549 Unrecognized G code
G code encountered is not recognized by the control.

550 ***Bad numeric format***
Expecting a numeric value, or a parameter value enclosed within [ ], after an address X, Y, Z, R, etc.

551 ***Multiple decimal points***
Multiple decimal points were detected within one numeric value.

552 ***Missing "]"***
Always use square brackets in pairs.

553 ***Missing "["***
Always use square brackets in pairs.

554 ***Tangent function overflow***
Trying to find the tangent of a number close to 90E

555 ***Missing "/"***
Arctan "ATAN" syntax is P## = ATAN[#/#].

556 ***Negative SQRT argument***

557 ***Unknown function***

560 ***Illegal relational operator***

567 ***Unresolved call***
Program being called does not exist (Call ####).

568 ***Unresolved goto or gosub***
N#### does not exist in the program (Goto ####).

569 ***The tool is too large to cut inside the arc "Compensated radius is too small"***
Eliminate the arc, or use a smaller tool.

570 ***The tool is too large to cut inside the arc "1st compensated radius in arc to arc is < 0"***
Eliminate the arc, or use a smaller tool.

571 ***The tool is too large to cut inside the arc "2nd compensated radius in arc to arc is < 0"***
Eliminate the arc, or use a smaller tool.

572 ***Pocket clear is not in a Start/End mill cycle***
-WHILE WEND loop-
Use START at the beginning of the mill cycle and END at the end of the mill cycle.

573 ***Round wall is not in a Start/End mill cycle***
-WHILE WEND loop-
Use START at the beginning of the mill cycle and END at the end of the mill cycle.

574 ***Round wall radius will not span 1st Z depth and final Z depth***

575 ***Tapered wall is not in a Start/End mill cycle***
-WHILE WEND loop-
Use START at the beginning of the mill cycle and END at the end of the mill cycle.

576 ***Z increment is 0***

577 ***Input statements must precede axes moves***

578 ***Undefined text cycle***
Character specified in a text command is not supported.
Load Text Cycles parameter may not be set in PARMS-CTRL.

579 ***Compensated arcs do not intersect***

580 ***Invalid floating point operation. The argument passed to the LN function was zero or negative.***

581 ***Invalid floating point operation. The Operand passed to the "#" function was zero or negative***

588 ***Can't set tool offsets for tool #0***

600 ***Can't nest Start/End mill cycles -WHILE WEND loops-***
Do not start a mill cycle within a mill cycle.

601 ***Missing WHILE statement***
May be an end mill cycle without a start mill cycle.

602 ***Missing WEND statement***
May be a start mill cycle without an end mill cycle.

603 ***Program N#### does not exist***
Program being called as a subprogram does not exist. Check to see if the program called is in the memory.

605 Can't modify dry run status while program is running
Program must be halted before changing dry run status.
Try HALT-DRY-RESUME.

606 Program N#### is empty
Text program being run or verified is empty. Try editing and reposting the conversational file.

607 Can't exit DNC run mode while program is running
The DNC mode must be halted before exiting.

608 'P' expected in M98 block

609 G20/21 not allowed in DNC Fast

610 Position move required after this rotation

611 Feed must be specified in every block while inverse time mode is active.

666 Quill movement detected, reposition the quill to it's original position.

Digitizing Errors

800 Illegal probe block
Syntax error in guidance file. Check the block format. Blocks must start with an X, Y, or Z which is to be followed by one or two P commands.

801 Missing end of pick
Pick boundary is not closed. Input file did not end with a pick boundary definition block.

802 Reversed scan segment
Multiple scan segment started that would cause the scan direction to change in the middle of the current scan.

803 Missing end of scan
Pick segment terminated before the end of scan was defined. There should be at least two scan definition blocks between each pick boundary.

804 Reversed pick segment
Multiple pick segment started that would cause a change in pick direction. Use multiple probe segments if this is the desired intention.

805 ***Invalid probe setup***
Input file does not start with a comment containing three asterisks. Also, the following three blocks should be X, Y, Z, or Y, X, Z depending on scan plane.

806 ***Scan origin expected***
Multiple pick segment started without defining the start of the scans within that segment.

807 ***Probe file not found***
Could not find the selected input file.

808 ***Setup not selected***
Tried to probe without selecting both the input file and the output mode from the probe setup screen.

809 ***Bad Z limits encountered***
The max Z height is less than the max Z depth value.

810 ***Stuck digitizing probe***
Digitizing may have reached max Z height and part contact is still detected. If the probe is not actually touching the part, the probe may require maintenance. If it is touching the part, then the part has to be lowered or the max Z height should be increased.

812 ***Input out of range***

900 ***RS232 overrun error (The system sending data may not have the same baud rate as the CNC.)***
Check RS-232 baud rate parameter in PARMS-CTRL.

901 ***RS232 parity error (The system sending data may not have the same parity as the CNC.)***
Check RS-232 parity parameter in PARMS-CTRL.

902 ***RS232 framing error (Remote system and CNC may not have the same line settings or a loose cable.)***
Check line settings in PARMS-CTRL for baud rate, parity, and stop bits.

903 ***RS232 break detected (RS232 cable may be loose.)***
Check cabling and connectors for good contact.

950 ***Obsolete controller software for this system***

951 ***Obsolete acroloop software for this system***

952 ***Obsolete keyboard software for this system***

953 Obsolete bit access for ncb controller.

954 Control not detected.

955 Interface not detected.

Byte Parameters

000	Parameter File Version
016	Cartesian Inch Leading Digits
017	Cartesian Inch Trailing Digits
018	Cartesian Metric Leading Digits
019	Cartesian Metric Trailing Digits
020	Angle Inch Leading Digits
021	Angle Inch Trailing Digits
022	Angle Metric Leading Digits
023	Angle Metric Trailing Digits
024	Spindle Inch Leading Digits
025	Spindle Inch Trailing Digits
026	Spindle Metric Leading Digits
027	Spindle Metric Trailing Digits
028	Feed Inch Leading Digits
029	Feed Inch Trailing Digits
030	Feed Metric Leading Digits
031	Feed Metric Trailing Digits
040	RS-232 Change Line-Feed to Carriage Return Linefeed In
041	Send Carriage returns and Linefeeds at the end of each RS-232 block.
042	Key-Board code
045	Machine Version
046	Force Front Panel Spindle
047	Minimum Parts Space
048	use G92 instead of G54 in the Hand-Wheel and Jog screens
049	Extract programs with O##### in RS-232 and Files Load
050	Spindle Range
051	Number of ATC Pockets
053	Resolve Programs in DNC (for GOTOs)
055	Disable 417 Errors
057	Door Open Switch
058	Use Small Soft Keys
059	Load Tool Offsets
064	Initial Units Inch or Metric
065	Number of Axis
068	Spindle Axis
069	Tool Change Type
070	Run Rapid 100%
071	Dry Run Rapid 100%

072	Spindle on in Dry Run
073	Tool Table Diameters\Radius
075	Load Engraving Cycles
076	Load Canned Cycles
077	Check Spindle up to Speed
078	Check Spindle Zero Speed
079	G18 is ZX or XZ plane
080	Cad Type DXF or CDL
081	Special Flags
082	Offset Round and Tapered Walls
083	Full Dos File Names
085	Primary Serial Port
086	COM1 Baud Rate;
087	COM1 Data/Stop Bits and Parity
088	COM2 Baud Rate;
089	COM2 Data/Stop Bits and Parity
090	Tape Start Character
091	Tape Stop Character
092	RS-232 Buffer Size
093	Secondary Serial Port
094	RS-232_End of file character
096	Power up with Block Skip On
097	Power up with Optional Stop On
099	Display Multiple Block Lines
100-109	Post M-code Table
110-119	Custom G-code Table
120-129	Custom M-code Table
130-145	M6 Macro File Name
146	Drive on Menus
160-175	Report File Name
190	Check Air Pressure
192	Run in 2 or 3 Axes mode
193	Boot to 2 or 3 Axis mode
194	Old Front Panel System
195	Way Lube Timer
196	Update Skill Level
197	Hand Wheel
198	Skill Level
199	Manual Option
200	ATC Clockwise Coast
201	ATC Clockwise Brake
202	ATC Counter Clockwise Coast
203	ATC Counter Clockwise Brake
204	Electronic Spindle Gears
206	Show current position in E-Stop
209	Spindle Dac Value

212	Second Hand-Wheel Axis
213	Probe Axis
214	Probe Input
215	Cranking Factor (for hand-wheeling thru a program)
218	Serial Key board
219	European Code
220	Door Open Axis
221	Door Open Input
222	Door Over-Ride Axis
223	Door Over-Ride Input
227	End of Cycle Axis = 227;
228	End of Cycle Output = 228;
229	Yaskawa Axis Drives
230	Third Hand-Wheel Axis
231	Inverse Feed-rate in 1/Seconds or 1/Minutes
232	Video Mode
233	Screen Blank Back Light
234	Enable Three Hand-Wheels when Conversational Programming
275-291	Foreign Extension
294	Blocks of Look Ahead in Fast DNC
295	Reverse Spindle in Back-Gear
296	Check control over temperature
298	Remove tool lengths from graphics
320-335	Machine Type
338-353	Mechanical Version
354-369	Electrical Version
370	Ignore Rapids in Auto-Scale
371	Safe Zone on at Power Up
374	New Remote Hand-Wheel
375	Sub Work-Coordinate System
376-391	Machine Serial Number
392	High Tool Change on Pick and Place
396	Z Axis Scale Feed-back
397	Auto Rotary Brake
398	Sort Directories by Name
399	Manual Pocket Clear Tool Number
400	Ignore Tool Radius to Large Errors
404	Front Panel Type
530	Auger On Time
532	Wash-Down on Time
533	Wash-Down off Time
534	Check Tool Door Open
535	Spindle Cool Down Time
536	Spindle Lube Cycle Time
537	Yaskawa2 Drives
541	Check Drawbar Prox

542	Yaskawa M5 drive
543	Machine State 0=Nothing 1=Verifying 2=Program Running
544	Check Spindle in Gear

Real Parameters

Great care must be taken when writing to any parameters other than the User Parameters P00-P99.

P00-P99	User Parameters
P100-P111	Probe Parameters
P112-P139	Unassigned
P140	R plane
P141	Final Z depth
P142	Z initial level
P143	Z increment
P144	First Z depth
P145	Z feedrate
P146	Peck up increment
P147	Peck clearance
P148	Dwell 1
P149	Dwell 2
P150	Pocket radius
P151	X pocket dimension
P152	Y pocket dimension
P153	XY finish stock
P154	Z finish stock
P155	Cut width
P156	Bolthole radius
P157	Bolthole start angle
P158	# holes in 360 degrees
P159	# holes to be made in Bolthole
P160-P170	Unassigned
P171	Holes to be made in drill cycles "K"
P172	Mirror position axis 1 (X)
P173	Mirror position axis 2 (Y)
P174	Mirror position axis 3 (Z)
P175	Mirror position axis 4 (A)
P176	Mirror position axis 5 (B)
P177	Mirror pos. optional axis
P178	Mirror pos. optional axis
P179	Mirror pos. optional axis
P180	Scale position axis 1 (X)
P181	Scale position axis 2 (Y)
P182	Scale position axis 3 (Z)
P183	Scale posiiton axis 4 (A)
P184	Scale position axis 5 (B)
P185	Scale position opt. axis
P186	Scale position opt. axis
P187	Scale position opt.axis
P188	Scale factor axis 1 (X)
P189	Scale factor axis 2 (Y)
P190	Scale factor axis 3 (Z)
P191	Scale factor axis 4 (A)
P192	Scale factor axis 5 (B)
P193	Scale factor opt. axis
P194	Scale factor opt. axis
P195	Scale factor opt. axis
P196	Rotate I position
P197	Rotate J position
P198	Angle of rotation
P199	Auto Routines Plunge / Ramp
P200	Previous position (X)
P201	Previous position (Y)
P202	Previous position (Z)
P203	Previous position (A)
P204	Previous position (B)
P205	Previous pos. opt. axis
P206	Previous pos. opt. axis
P207	Previous pos. opt. axis
P208	Current position (X)
P209	Current position (Y)
P210	Current position (Z)

P211	Current position (A)
P212	Current position (B)
P213	Current pos. opt. axis
P214	Current pos. opt. axis
P215	Current pos. opt. axis
P216	Previous machine (X)
P217	Previous machine (Y)
P218	Previous machine (Z)
P219	Previous machine (A)
P220	Previous mach. opt. axis
P221	Previous mach. opt. axis
P222	Previous mach. opt. axis
P223	Previous mach. opt. axis
P224	Current machine (X)
P225	Current machine (Y)
P226	Current machine (Z)
P227	Current machine (A)
P228	Current machine (B)
P229	Current machine opt. axis
P230	Current machine opt. axis
P231	Current machine opt. axis
P232	Work offset axis 1 (X)
P233	Work offset axis 2 (Y)
P234	Work offset axis 3 (Z)
P235	Work offset axis 4 (A)
P236	Work offset axis 5 (B)
P237	Work offset optional axis
P238	Work offset optional axis
P239	Work offset optional axis
P240	Tool offset axis 1 (X)
P241	Tool offset axis 2 (Y)
P242	Tool offset axis 3 (Z)
P243	Tool offset axis 4 (A)
P244	Tool offset axis 5 (B)
P245	Tool offset optional axis
P246	Tool offset optional axis
P247	Tool offset optional axis
P248	Arc radius
P149	Arc I value
P250	Arc J value
P251	Feedrate
P252	Dwell
P253	Spindle speed
P254	Temporary X position
P255	Temporary Y position
P256	Temporary Z position
P257	Temporary A position
P258	Temporary axis position
P259	Temporary axis position
P260	Active tool number
P261	Active tool radius
P262	Active tool length
P263	Active radius offset number
P264	Active length offset number
P265	Canned cycle active
P266	Zone status
P267	Unit ratio
P268	Pending tool
P269	Unused
P270	Temporary I
P271	Temporary J
P272	Temporary K
P273	Third axis rotation angle
P274-P282	Rotation table
P283-P285	Unused
P286-P292	Rotation coordinates
P293-P299	Unassigned
P300	Modal 00
P301	Modal 01
P302	Modal 02
P303	Modal 03
P304	Data mode
P305	H offset direction
P306	Interpolate
P307	Coordinates
P308	Active plane
P309	Cutter compensation
P310	Canned cycle
P311	Dimension
P312	Feed unit
P313	Spindle unit
P314	Spindle direction
P315	Linear unit

P316	Scale
P317	Rotate
P318	Mirror
P319	Work system
P320	Primary
P321	Secondary
P322	Tertiary
P323	Return plane
P324	Tapping
P325	Custom code
P326	Custom M-code
P327	Custom G-code
P328	Feed per Rev (wait for marker)
P329	Feedrate override Lock
P330	Spindle override Lock
P331-P334	Unassigned
P335	Hard Tap Fudge Factor
P336-P343	Unassigned
P344	Inverse Time Feedrate
P345	Bore Type
P346	G76 Orient Bore Angle
P347	G76 Orient Bore Distance
P348-P349	Unassigned
P350-P361	Used in tool setting software
P362-P363	Unassigned
P364	Draw Bar Delay used in tool change macros
P365	Feed per Minute Feedrate
P366	Feed per Rev Feedrate
P367-P368	Unassigned
P369	Job time
P370	True tool number
P371-P399	Unassigned
P400	Work G92 axis 1 (X)
P401	Work G92 axis 2 (Y)
P402	Work G92 axis 3 (Z)
P403	Work G92 axis 4 (A)
P404	Work G92 axis 5 (B)
P405	Work G92 optional axis
P406	Work G52 axis 1 (X)
P407	Work G52 axis 2 (Y)
P408	Work G52 axis 3 (Z)
P409	Work G52 axis 4 (A)
P410	Work G52 axis 5 (B)
P411	Work G52 optional axis
P412	Work coordinate 1 (X)
P413	Work coordinate 1 (Y)
P414	Work coordinate 1 (Z)
P415	Work coordinate 1 (A)
P416	Work coordinate 1 (B)
P417	Work coord. 1 opt. axis
P418	Work coordinate 2 (X)
P419	Work coordinate 2 (Y)
P420	Work coordinate 2 (Z)
P421	Work coordinate 2 (A)
P422	Work coordinate 2 (B)
P423	Work coord. 2 opt. axis
P424	Work coordinate 3 (X)
P425	Work coordinate 3 (Y)
P426	Work coordinate 3 (Z)
P427	Work coordinate 3 (A)
P428	Work coordinate 3 (B)
P429	Work coord. 3 opt. axis
P430	Work coord. 4 (X)
P431	Work coord. 4 (Y)
P432	Work coord. 4 (Z)
P433	Work coord. 4 (A)
P434	Work coord. 4 (B)
P435	Work coord. 4 opt. axis
P436	Work coordinate 5 (X)
P437	Work coordinate 5 (Y)
P438	Work coordinate 5 (Z)
P439	Work coordinate 5 (A)

P440	Work coordinate 5 (B)
P441	Work coord. 5 opt. axis
P442	Work coordinate 6 (X)
P443	Work coordinate 6 (Y)
P444	Work coordinate 6 (Z)
P445	Work coordinate 6 (A)
P446	Work coordinate 6 (B)
P447	Work coord. 6 opt. axis
P448	Tool change offset (X)
P449	Tool change offset (Y)
P450	Tool change offset (Z)
P451	Tool change offset (A)
P452	Tool change offset (B)
P453	Tool chg. offset opt.axis
P454	Positive safe zone (X)
P455	Positive safe zone (Y)
P456	Positive safe zone (Z)
P457	Positive safe zone (A)
P458	Positive safe zone (B)
P459	Positive safe zone opt.
P460	Negative safe zone (X)
P461	Negative safe zone (Y)
P462	Negative safe zone (Z)
P463	Negative safe zone (A)
P464	Negative safe zone (B)
P465	Negative safe zone opt.
P466	Text Rotation Angle
P467-P473	Unassigned
P474	Current Max Feedt
P475	R_LogBeatsSkip
P476-P494	Unassigned
P495	ToolChangeSpacing
P496-P509	Unassigned
P510	StartBlock
P511	EndBlock
P512	CutIncrement
P513	RufCutType
P514	Start Mill Options
P515	RufCutDepth
P516	Island
P517-P698	Unassigned
P699	Parts Counter
P700-P789	Unassigned
P790	Spindle range 1
P791	Spindle range 2
P792	Spindle range 3
P793	Spindle Range 4
P794	Spindle Range 5
P795	Spindle Range 6
P796	Spindle Range 7
P797	Spindle Range 8
P798	Max Graph Size
P799-P811	Unassigned
P812	Block Rate
P814	Cad Epsilon
P815-P816	Unassigned
P817	Spindle Encoder PPU1
P818	Actual Spindle RPM
P819-P839	Unassigned
P840	Pulse Delay
P841	Max Spindle RPM
P842	Tapping Ramp Low Gear
P843	Spindle Acceleration Ramp
P844	Spindle Deceleration Ramp
P845-P849	Unassigned

P850	ADC Sample
P851	ADC Scale
P852	ADC Value
P853	ADC Trigger
P854	Tapping Ramp High Gear
P855	Spindle Encoder PPU2
P856-P892	Unassigned
P893	Soft Start Delay
P894	Clamped Feedrate
P895	Cranking Max ipm
P896	Handwheel Encoder PPU
P897	Cranking Minutes Per Turn
P898	Screen Blank Time
P899	Digitizing Enable
P900-P962	Unassigned
P963	Max Corner Deviation
P964	Unassigned
P965	SpindleScale (fraction of full spindle dac)
P966	Minimum Block Time (seconds, min time when not in fast DNC)
P967	Unassigned
P968	Minimum Linear Feed AccDec
P969	Minimum Rotary Feed AccDec
P970	Minimum Linear Rapid AccDec
P971	Minimum Rotary Rapid AccDec
P972	Quill Epsilon
P973	Front Panel Base Port
P974	Quill Scale PPU
P975-P999	Unassigned
P1000-P1099	Axis 1 Address (X)
P1100-P1199	Axis 2 Address (Y)
P1200-P1299	Axis 3 Address (Z)
P1300-P1399	Axis 4 Address (A)
P1400-P1499	Axis 5 Address (B)
P1000	Axis address label (X)
P1001	Pulses per unit (X)
P1002	Home position (X)
P1003	Home direction (X)
P1004	Positive limit (X)
P1005	Negative limit (X)
P1006	Maximum feed (X)
P1007	Dry run feed (X)
P1008	Rapid velocity (X)
P1009	Rapid acc/dec (X)
P1010	Home sequence (X)
P1011	Velocity toward home (X)
P1012	Velocity away from home (X)
P1013	Velocity toward marker (X)
P1014	Encoder multiplier (X)
P1015	Slow jog velocity (X)
P1016	Slow jog acc/dec (X)
P1017	Rapid jog velocity (X)
P1018	Rapid jog acc/dec (X)
P1019	In position (X)
P1020	G00 unidirectional (X)
P1021	G60 unidirectional (X)
P1022	Backlash (X)
P1023	Excess error (X)
P1024	Rotary=0 Linear=1 (X)
P1025	English Lead (X)
P1026	English Trail (X)
P1027	Metric Lead (X)
P1028	Metric Trail (X)
P1029	Jog Key Direction (X)
P1030-P1040	Unassigned (X)
P1041	Home Marker Switch (X) (Home Switch = 0, Marker = 1)
P1042	G28 reference point (X)

P1043	G30 reference point 2 (X)
P1044	G30 reference point 3 (X)
P1045	G30 reference point 4 (X)
P1046	Max Handwheel Error (X)
P1047-P1048	Unassigned (X)
P1049	Feed Back (X)
P1050	Invert Handwheel (X) (non-zero = invert handwheel direction)
P1051	Gain Proportional (X)
P1052	Gain Velocty (X)
P1053	Gain Acceleration (X)
P1054	Gain Handwheel (X)
P1055	Feed S-Curve Acc\Dec (X)
P1056	Rapid S-Curve Acc\Dec (X)
P1057-P1099	Unassigned (X)
P1100-P1199	Same as above only for (Y) axis
P1200-P1299	Same as above only for (Z) axis
P1300-P1399	Same as above only for (A) axis
P1400-P1499	Same as above only for (B) axis
P1500	Unassigned
P1501	Software Version
P1502-P1503	Unassigned
P1504	Turret Pocket Count
P1505-P1554	Unassigned
P1555	Current Pocket
P1556-P1595	Tool Slots for swing arm style tool changers
P1596-P1598	Unassigned
P1599	Spindle Orient Delay
P1600-P1607	Thermo Degrees per Volt
P1608-P1615	Thermo Inches per Volt
P1616-P1623	Thermo Inverse Growth per Degree
P1624-P1631	Thermo Position Sensor
P1632-P1639	Thermo Voltage Reference
P1640-P1647	Thermo Position Reference
P1648-P1699	Unassigned
P1700-P1799	Security Codes
P1800-P1999	Unassigned
P2000-P2099	Tool Radii 0..99
P2100-P2199	Tool Lengths 0..99