CENTURION 1
THREE AXIS SLS

PROGRAMMING AND OPERATIONS MANUAL

Version 1.7
May 1, 1995

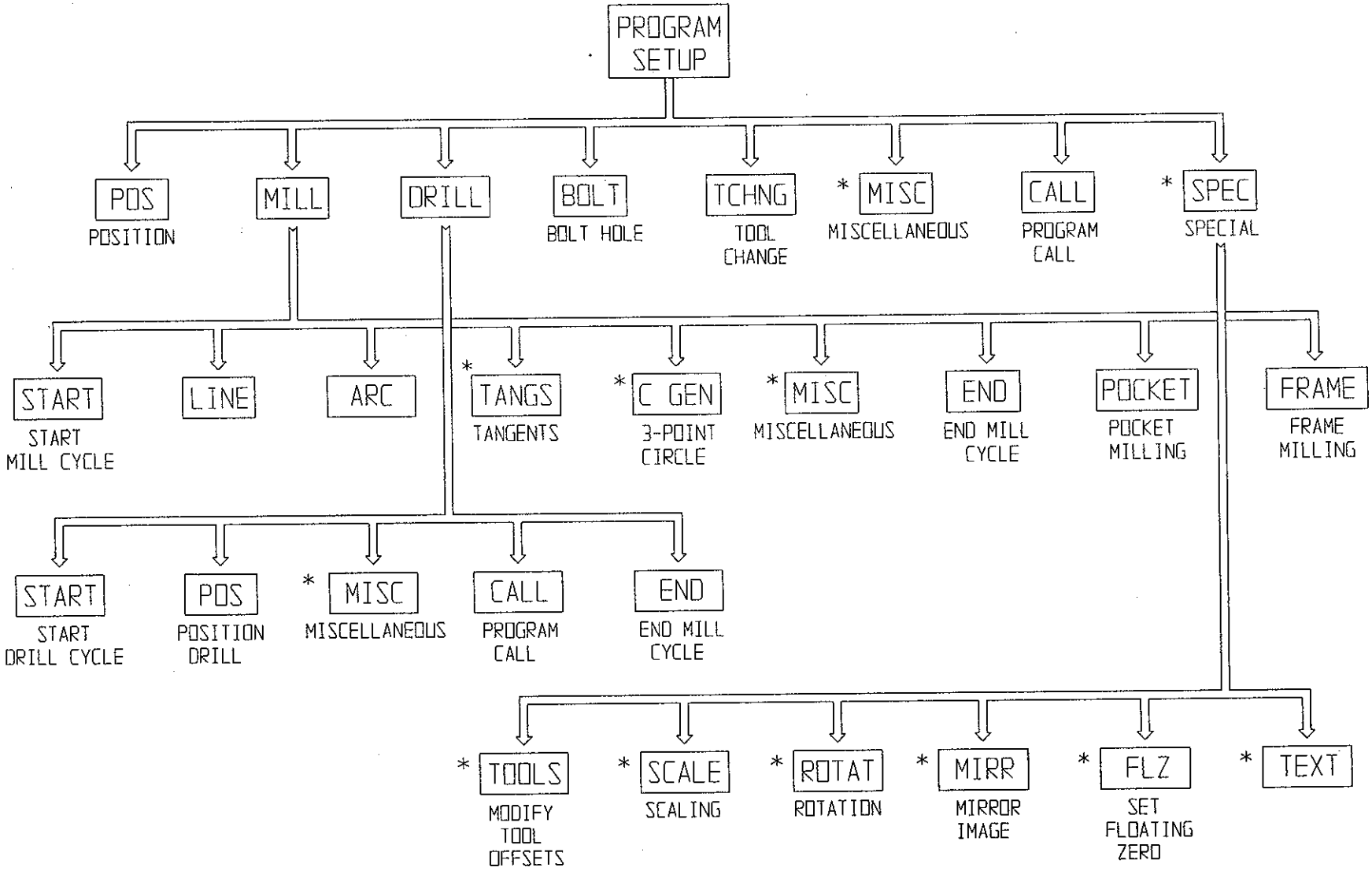
Milltronics Manufacturing Company
1400 Mill Lane
Waconia, MN 55387

MANUAL REVISIONS

1.61	01-18-95	Programming Changes to Sample Programs
1.7	05-01-95	Add SLS

TABLE OF CONTENTS

CONVERSATIONAL FLOWCHART



'*' INDICATES THIS FEATURE IS NOT AVAILABLE IN SKILL LEVEL 1.

INTRODUCTION

You have just purchased a Centurion 1 control from Milltronics Manufacturing. This control is specifically designed to meet the needs of one or two part runs with the capabilities of handling a hundred or greater part production. The invention of our Skill Level Select (SLS) software allows the control to run in one of four different modes. These modes are as a two or three axes system with a simplified (Skill Level 1) or advanced (Skill Level 2) instruction set.

During a power up sequence the Centurion 1 will ask if you would like to run the machine as a two or three axes machine. If you choose two axes the CNC will lock the third axis in position and change all the machine's instructions and autoroutines to two axes. In the three axes mode Z will be activated and the control instructions will change to accommodate the third axis. Either of these modes can be run in either of the two skill levels. The skill level can be changed at any time but it is generally best to have it set before starting a program.

Once a program is written in Level 1 or 2 it will stay in that level independent of the skill level of the control at the time of editing. If the control is in Skill Level 2 and you are editing a Level 1 program, only Level 1 questions and commands will be active; however, any new program will be written with Level 2 commands. There is a setup parameter which can be set to automatically change Level 1 programs to Level 2 if they are being edited in Level 2. This may be desired if it is necessary to add a Level 2 command to a Level 1 program. To change from one skill level to another the following key sequence is needed:

ESC to the main screen and then push:

F10	→	F10	→	F9	→	F3	→	ESC
UTIL		PARM		CTRL		TOGL		
						(Select Level One or Two)		(To Return to Main Screen)

Another operator aid that the control provides is an on-line help mode. The HELP feature consists of pictures and text and is available anytime the HELP soft key is present on the screen. By depressing the HELP key a description or picture of the inputs needed for the current screen will pop up. Pressing the ESC key will exit the HELP screen and return back to the current screen.

The Centurion 1 has been designed to replace the conventional digital readout, sine plate, and rotary table options used on a manual mill, while still retaining the ease of use of a manual machine and adding the powerful features of a CNC machine.

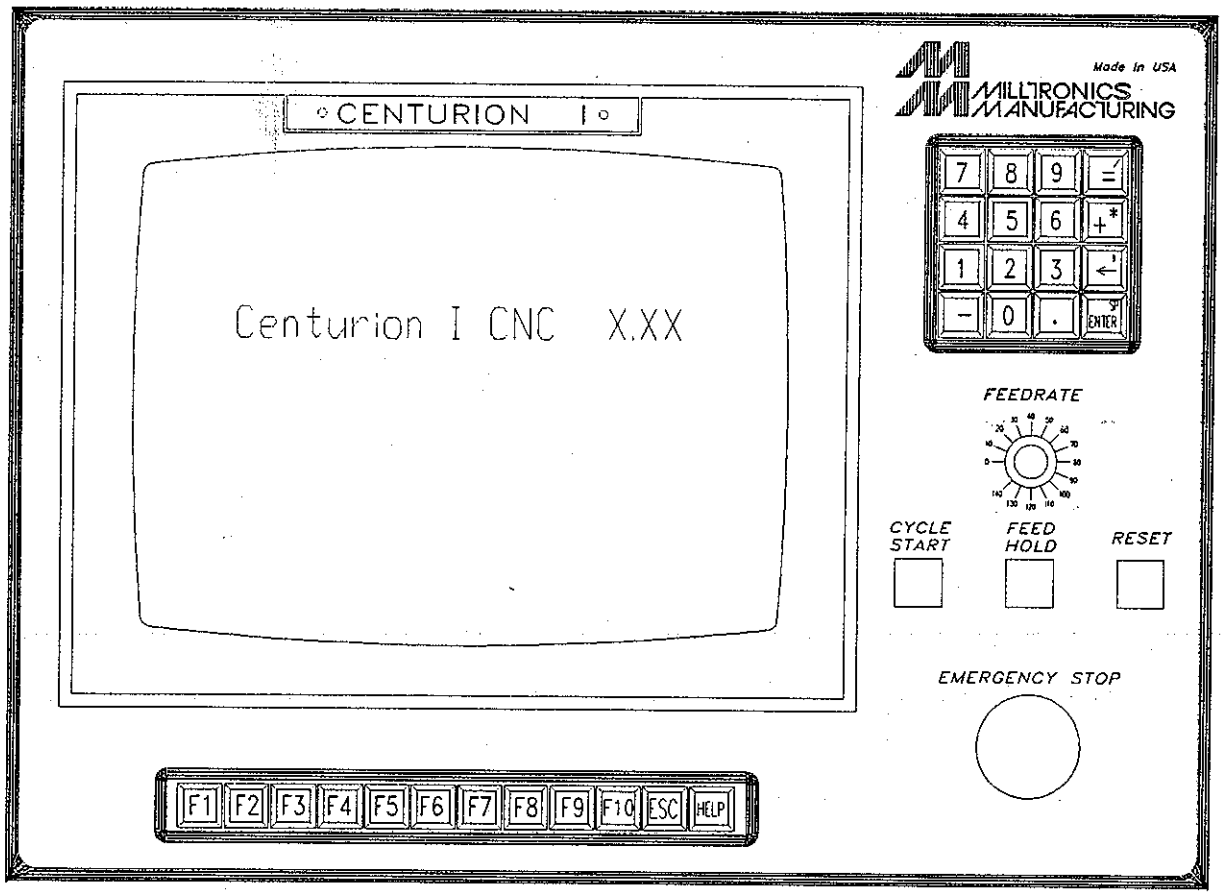
- The powerful control acts like a digital readout during manual operation.

- In the Skill Level 1 two axis mode the simplified control requires no operator training and can be run without a manual by simply reading the on-line help prompts. In this mode the machine closely resembles a mill with a digital readout, sine plate, and rotary table. This mode is well suited to simple 1 to 10 part runs.

- In the Skill Level 1 three axis mode the control requires a little more knowledge but still could be run by a knowledgeable operator with very little or no training. The X and Y programming remain the same, but some knowledge of tool setting is necessary. This level would be used for longer runs of simple parts.

- In the Skill Level 2 two axis mode the Centurion 1 turns into a full featured CNC as powerful as any CNC found on today's machining centers. This mode would be used for larger production runs of complicated parts.

- Through out this manual • *Skill Level 2* denotes functions that are only available in Skill Level 2.



Chapter 1

Control Configuration

The pendant station has a CRT to display operator instructions and part programs, a numeric and G and M code keypads, function keys for operator input, and miscellaneous switches to control the drives, axes motion, and program start/stop. The function of each element is as follows:

Emergency Stop Push Button

When pushed, all axes and spindle motors will immediately stop and a "MOTORS OFF" message will appear on the CRT.

Reset

The RESET push button turns on the axes motors allowing the CNC to move the machine when commanded. When pushed, "MOTORS ON" will appear on the CRT.

Spindle Enable

This button must be pushed once after either an EMERGENCY STOP or POWER OFF to enable the spindle CW, CCW switch.

Cycle Start

Depressing this button starts any machining sequence controlled by the CNC.

Feedhold

Feedhold stops all axes motion. It may be pushed at any time during a machining sequence. To restart the motion, CYCLE START needs to be pushed.

Feedrate Override

The FEEDRATE OVERRIDE switch varies the feedrate from 0 to 180% of the programmed feedrate.

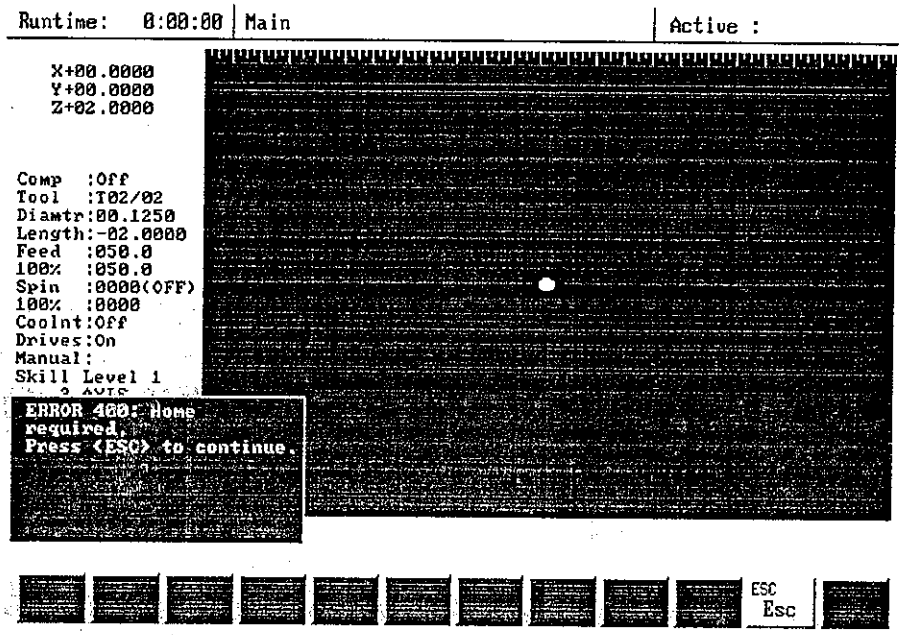
Function Keys, Numeric, and G and M Code Keypads

All menu-driven control functions and programming are accessed through these keys. Keys F1 through F10 are used to select the menu functions of the control, while the numeric keypad is used for data entry. Normally the G and M code

keypad would only be used for editing programs dumped from a CADCAM system.

14" CRT

The CRT is used to display positions, part programs and operator messages. After power has been turned on and the control has initialized itself, the following screen should appear:



This is the main screen and all operations will start and end on this screen. There are five main areas to this display.

History Line is across the top of the display. Runtime of the current part will be displayed on the left and the active program number is shown on the right. The sequence of keys that were pushed to reach the current screen are shown in the middle.

Current Operational Information is the data field down the left-hand portion of the screen. Axes positions and other current information needed when running a program are shown.

Operator Message Window appears in the lower left-hand corner of the screen. This area will normally be blank and is reserved for the operator message window. All actions which need to be taken by the operator will be displayed in this area.

Graphic Display Area is the largest area of the screen and will normally be used to show part graphics when running or verifying a program. Smaller data entry windows will also be displayed in this area when in data entry modes such as programming, MDI, tool offset and utility modes.

Function Menus are located at the bottom of the screen. These menus change as each screen is accessed. Pushing F1 through F10 selects the function listed with the function key. ESC moves back to the previous menu.

After power up the following function menu is displayed.

MAIN

F1 Home	F2 Jog	F3 Tool	F4 Run	F5 MDI	F6 Displ	F7 Unlck	F8 Prog	F9 Verf	F10 Util

This menu is displayed every time the Main screen is entered. Pushing the ESC key repeatedly will ultimately return you to the Main menu from any menu in the system.

Chapter 2

Homing

After initial power on the machine has to be referenced or homed before any automatic operation can be executed. Machines are generally equipped to home automatically to limit switches, or to home to the nearest encoder marker pulse. If the machine is not equipped with home switches, manually move the axis to the indicator arrows before initiating the home cycle. To initiate a home sequence, push ESC until the Main screen is reached. Then push F1 HOME. A message requesting that the CYCLE START button be pushed will appear on the screen.

Message Window

```
ATTENTION:  <CYCLE START>
to begin home sequence
```

Pushing CYCLE START will start the home sequence, and when it is finished the Main screen will return.

If your machine does not have indicator arrows on the Z axis, just line up the arrows on the X and Y axes before homing. Z axis will automatically home itself when the home sequence is initiated.

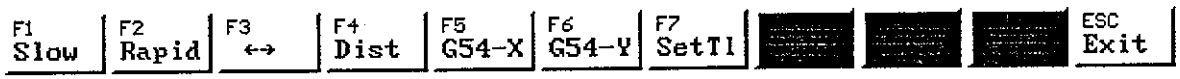
Chapter 3

Operating the Centurion 1 in Manual and Jog Modes

Manual Machining with Drives Off

Prior to doing manual machining with the X and Y axes, the Z axis should be positioned to its proper depth. This can be accomplished through use of the Jog function. Once Z has been positioned, the servo drives can be turned off and a brake will automatically hold Z in its current position.

Pushing in the EMERGENCY STOP button will turn the servo drives off. The axes can then be moved by the manual handwheels on the X and Y axes. The Centurion 1 display will keep track of the X and Y position just like a standard digital readout. The graphics display will follow all axes movements made by hand. By depressing the F2 JOG key on the main screen, a menu will be displayed which will allow the X and Y axes to be zeroed at any location on the table.

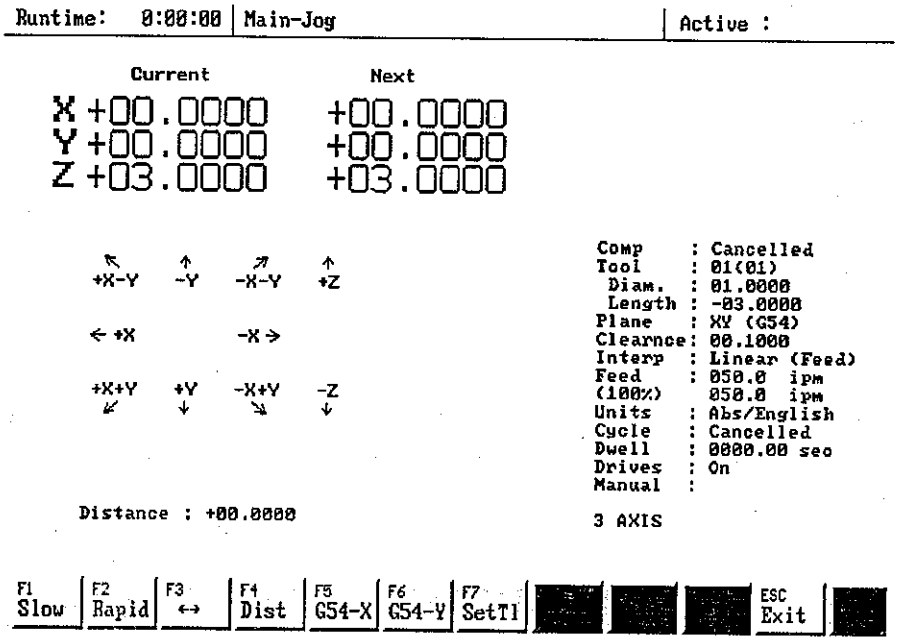


Pushing either F5 or F6 will zero out the control at the current table location.

Manual machining can be accomplished by simply zeroing out at a position, and then moving to a given coordinate, clamping the axis, and then doing a drilling or milling operation. This sequence can be repeated over and over until the part is completed.

Manual Machining Using Jog Mode

By selecting the Jog mode (F2), the Jog screen will appear:



If the drives are off (EMERGENCY STOP), the machine can still be moved by hand and the readout will keep track of the axes positions. With the drives on (RESET), the keys on the keyboard can be used to move the machine. Keys F1 through F4 are used to select the axis feedrate and distance to move. The slow and rapid keys (F1 and F2) increase or decrease the axis feedrate every time they are pushed. The new feedrate is displayed on the screen opposite the feed prompt. This will be the rate the axis will move at when it is selected. The feedrate override on the front panel can also be used to scale the jog feedrate in this mode. The scaled feedrate is displayed below the feed prompt. The F3 key selects one of two jog types. When the F3 ↔ key is lit the control is in the continuous jog mode. Whenever one of the direction arrow keys on the keyboard is held down, the axis will move. When it is released, movement will stop. In the incremental mode when F4 DIST is pushed, a prompt will ask for a specific distance to move. In this mode whenever one of the direction arrow keys is pushed, the axis will move in that direction a distance and feedrate equal to the number displayed in the "Dist" and "Feed" prompts.

Pushing F3 clears the distance prompts so a new number can be entered. The feedrate always remains at the last entered value. To enter a distance of 5.476, the following keystrokes would be needed.

F4 DIST
Type in 5.476
Enter

By pushing the 6 → key the table will move 5.476 inches to the right from its current location. By using this feature the mill can be used in a manner similar to a power feed. Parts can be faced or milled by simply moving the machine to the desired location manually, selecting Jog mode, entering the desired feedrate and distance, and selecting the desired direction.

Manual Machining Using Unlock Mode

The unlock feature is accessed off the Main screen by selecting the F7 UNLCK key (see screen on page 1-3. This feature allows the operator to electronically turn off either the X or Y servo drive. This allows manual movement of the selected axis while the other axis remains in servo hold. This ensures that the other axis does not move while machining is occuring.

When F7 UNLCK is pushed the following keys appear:

Pushing F9 or F10 will allow the selected axis to be moved manually. This axis will remain in the manual mode until the Reset key is depressed. At this time the servo will turn back on. The axes which are in manual will be displayed in the machine status window after the manual prompt.

Chapter 4

Definitions, Concepts, and Features of the Centurion 1

Before operating the Centurion 1, some basic programming conventions need to be discussed.

Axis Movement

X Axis Positive X motion occurs when the table moves to the left. Measurement on the part to the right of zero is positive.

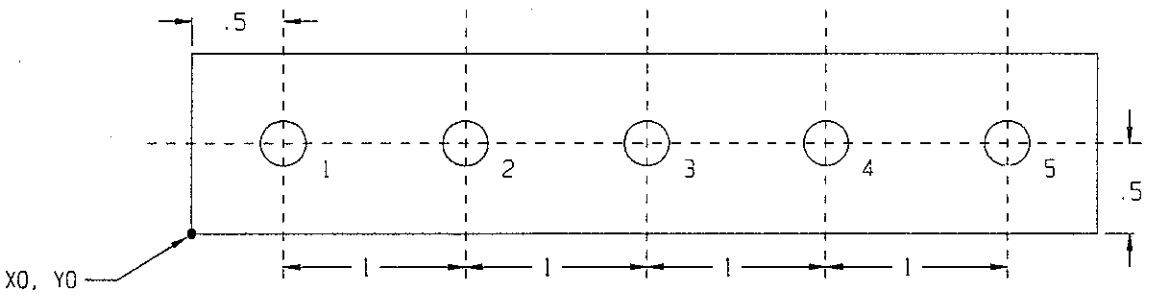
Y Axis Positive Y motion occurs when the table moves toward the operator. Measurement on the part above zero is positive.

Z Axis Positive Z motion occurs when the tool is moved up.

Coordinate System Definitions

Absolute/Incremental Dimensions

When using the absolute mode, all dimensions on the part are referenced to a zero or common point on the part. In the incremental mode, all dimensions are referenced from the current position of the machine.



The above part dimensioned using absolute coordinates:

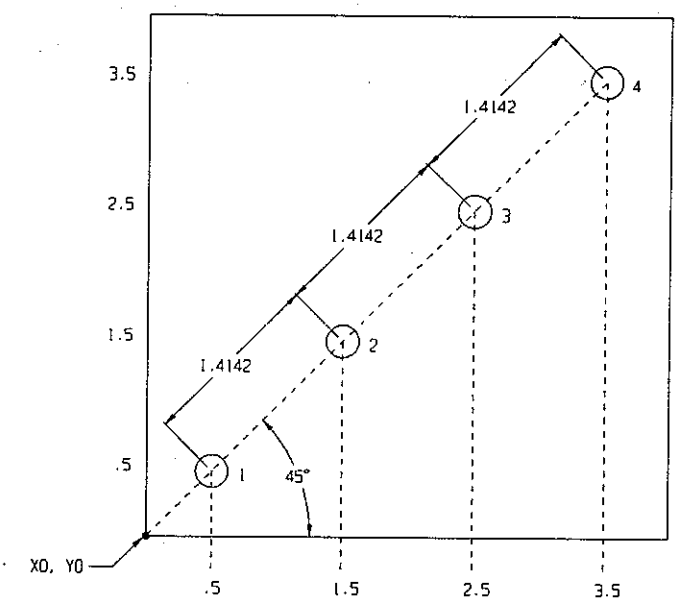
Hole 1	X.5 Y.5
Hole 2	X1.5
Hole 3	X2.5
Hole 4	X3.5
Hole 5	X4.5

The same part using incremental coordinates:

Hole 1	X.5 Y.5
Hole 2	X1
Hole 3	X1
Hole 4	X1
Hole 5	X1

Cartesian/Polar Coordinates

When using Cartesian coordinates, all dimensions are given as an X or Y point. When using polar coordinates, points are given as an angle and length from the current position of the machine.



Above part dimensioned using:

Absolute/Cartesian		Incremental/Cartesian	
Hole 1	X.5 Y.5	Hole 1	X.5 Y.5
Hole 2	X1.5 Y1.5	Hole 2	X1 Y1
Hole 3	X2.5 Y2.5	Hole 3	X1 Y1
Hole 4	X3.5 Y3.5	Hole 4	X1 Y1

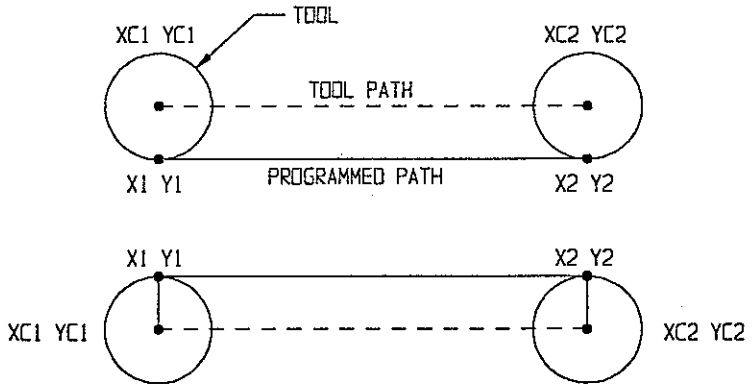
Polar

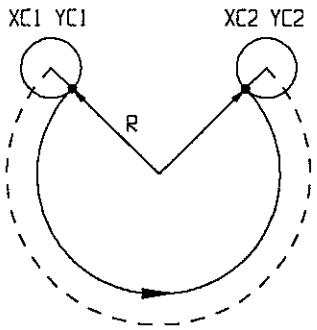
Hole 1 (length)R=.7071 AA=45°
Hole 2 (length)R=1.4142 AA=45°
Hole 3 (length)R=1.4142 AA=45°
Hole 4 (length)R=1.4142 AA=45°

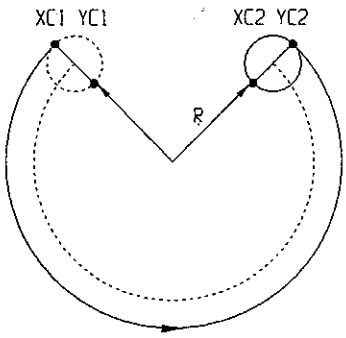
When using polar coordinates, the length to move is coded as an R and the angle is coded as AA.

Tool Diameter Compensation

Tool diameter compensation is the ability of the control to compensate for the diameter of the cutter being used. Using this feature allows print dimensions to be programmed instead of tool centerline dimensions. In the MDI mode the Centurion 1 will always offset the tool perpendicular to the start and end points of a line and circle (auto left or right compensation).

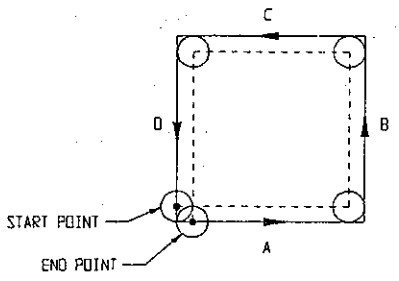




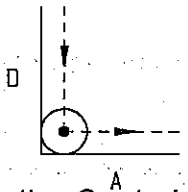


As can be seen in these preceding examples, there are two ways the cutter can displace itself for each line and circle. These two ways are defined as right and left compensation. Each time compensation is turned on, the Centurion 1 will need to know if it is to use right or left compensation. To make this determination, make believe you are the cutter and are walking around the part in the direction of the cut. Then determine if you would like the cutter on your right or left hand side, and make that compensation choice. As can be seen in the above examples, if the tool is moving from X1 Y1 to X2 Y2, left compensation puts the tool above the line and right is below. In the circle case, right compensation is on the outside of the circle and left is on the inside. Note that if you are moving from X2 Y2 to X1 Y1, right is above the line, left is below, and on the circle left is outside and right is inside.

In the Program mode the Centurion 1 uses a more general method of turning cutter compensation on and off. This is because as the shapes become more complicated, a simple displacement 90° to the start of a line or arc can cause problems. For instance, if one is trying to cut a square pocket, offsetting perpendicular to one of the starting sides will not work.

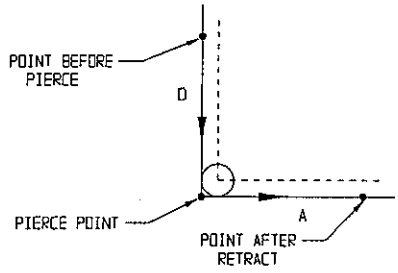


As can be seen, starting the tool perpendicular to the beginning of line A works for line A, but destroys line D, and stopping the tool perpendicular to the end of line D is fine for D but destroys line A.



The ideal way to start and stop the tool would be to have it be tangent to both lines A and D at the start and end points.

To enable the Centurion 1 to calculate this tool position, it needs to understand both the starting and ending corner shape. The Centurion 1 will ask for this data when cutter compensation is turned on or off. It does this by asking for a **Pierce Point** and **Point Before Pierce** when starting, and **Point After Retract** when stopping. Getting back to our example, these points are as follows:



The **Point Before Pierce** is simply any point on line D, and the **Pierce Point** has the coordinates of our starting corner. The **Point After Retract** is simply any point on line A.

If we give some dimensions to lines A and D, the starting and ending commands will look like:

	Point Before Pierce	X 2.0000 Y 3.0000
	Compensation	[LEFT]
	Pierce Point	X 2.0000 Y 1.0000
Ending commands would be:	Point After Retract	X 4.0000 Y 1.0000

As you can see from the example, the length of lines A and D as well as what happens between lines A and D, have no effect on the **Pierce Point**, **Point Before Pierce**, and the **Point After Retract**. The angle formed by lines A and D does affect the position of the tool starting and ending points. The following is a chart of all the possible starting and ending corner configurations and the position the tool will assume when cutter compensation is turned on. In these examples lines A and D will be assumed to be 1" long, and the angles formed by the lines are 30°, 90°, and 180°. As stated earlier, any angle is permissible and the length of the lines is immaterial. As the angle changes, only the X and Y coordinates of the **Point Before Pierce** and **Point After Retract** are affected.

Auto Compensation

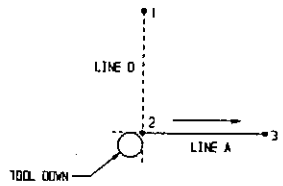
The auto cutter compensation mode is the same as cutter compensation in the MDI mode. Auto at the start of a program will offset the tool 90° to the right or left of the first line. Auto at the end of the program will stop the tool 90° to the right or left of the end of the last line of the program. This mode does not require the use of a point before pierce or point after retract and can be used anytime the part geometry allows, i.e. starting and finishing off the part on outside cuts.

In Skill Level 1 the only mode available is Auto.

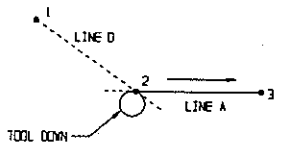
STARTING CORNER DEFINITION
Right Tool Compensation
Skill Level 2

Starting Corners

Point Before Pierce	X 0.0000
	Y 1.0000
Compensation	[Right]
Pierce Point	X 0.0000
	Y 0.0000

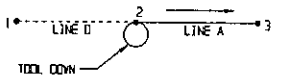


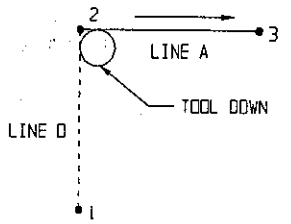
1 = Point Before Pierce
2 = Pierce Point
3 = First Move of Program



Point Before Pierce	X-.8660
	Y .5000
Compensation	[Right]
Pierce Point	X 0.0000
	Y 0.0000

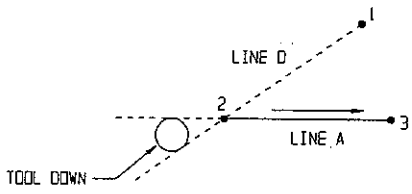
Point Before Pierce	X-4.0000
	Y 0.0000
Compensation	[Right]
Pierce Point	X 3.0000
	Y 0.0000





Point Before Pierce	X-1.0000
	Y-3.0000
Compensation	[Right]
Pierce Point	X -1
	Y -2

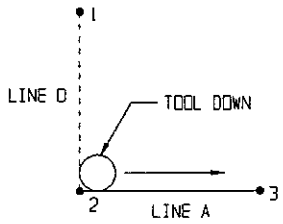
Point Before Pierce	X 1.8660
	Y 3.5000
Compensation	[Right]
Pierce Point	X 1
	Y 3



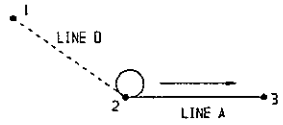
STARTING CORNER DEFINITION
Left Tool Compensation
Skill Level 2

Starting Corners

Point Before Pierce	X 1.0000
	Y 2.0000
Compensation	[Left]
Pierce Point	X 1.0000
	Y 1.0000

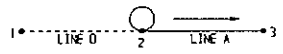


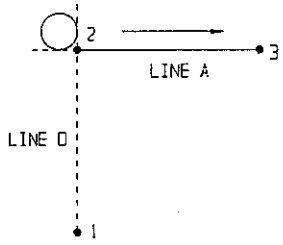
1 = Point Before Pierce
2 = Pierce Point
3 = First Move of Program



Point Before Pierce	X-1.8660
	Y-1.5000
Compensation	[Left]
Pierce Point	X-1.0000
	Y-2.0000

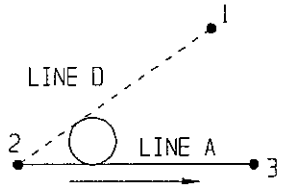
Point Before Pierce	X-1.3850
	Y 2.0000
Compensation	[Left]
Pierce Point	X - .3850
	Y 2.0000





Point Before Pierce	X 3.0000
	Y-2.0000
Compensation	[Left]
Pierce Point	X 3.0000
	Y-1.0000

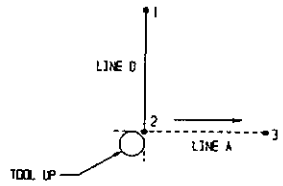
Point Before Pierce	X .8660
	Y .5000
Compensation	[Left]
Pierce Point	X 0
	Y 0



ENDING CORNER DEFINITIONS
Right Tool Compensation
Skill Level 2

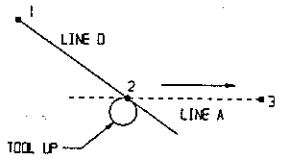
Ending Corners

Point After Retract X 1.0000 Y 0.0000



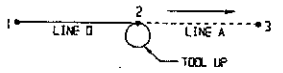
1-2 = Last Move of Program
2 = Tool Retract Position
3 = Point After Retract

Point After Retract X 1.0000 Y 0.0000

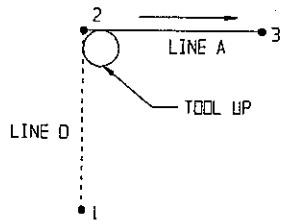


Tool Up Point in this example is always X0, Y0, although it can be any point within the travels of the machine.

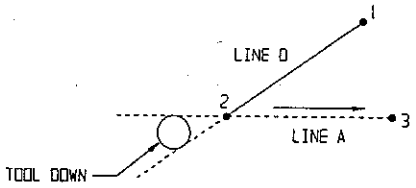
Point After Retract X 1.0000 Y 0.0000



Point After Retract X 1.0000 Y 0.0000



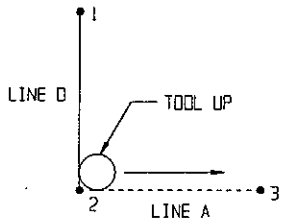
Point After Retract X 1.0000 Y 0.0000



ENDING CORNER DEFINITIONS
Left Tool Compensation
Skill Level 2

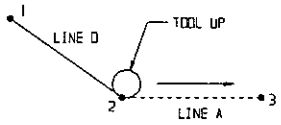
Ending Corners

Point After Retract X-1.0000
Y 2.0000



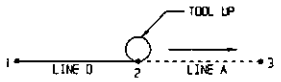
1-2 = Last Move of Program
2 = Tool Retract Position
3 = Point After Retract

Point After Retract X-1.0000
Y 2.0000

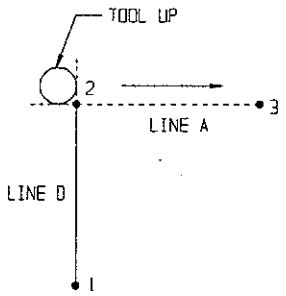


Tool Up Point in this example is always X-2, Y2, although it can be any point within the travels of the machine.

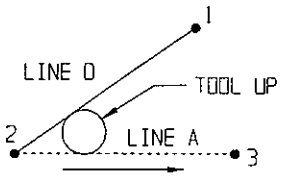
Point After Retract X-1.0000
Y 2.0000



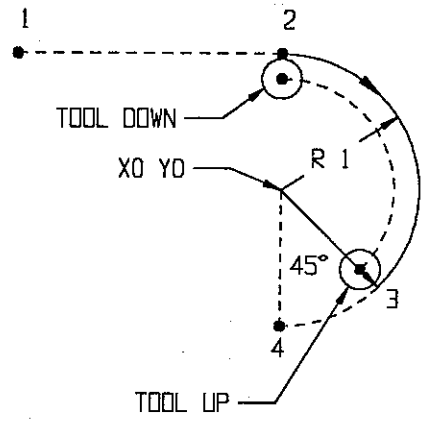
Point After Retract X-1.0000
Y 2.0000



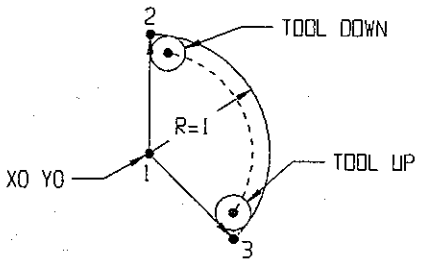
Point After Retract X-1.0000
Y 2.0000



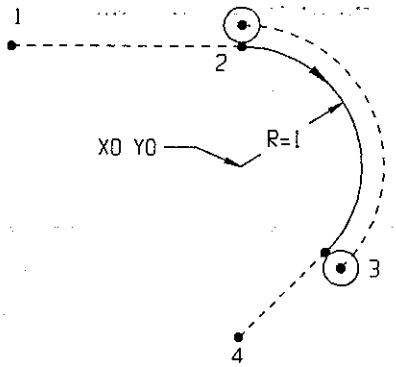
It is always preferable to start and stop cutter compensation on line moves. If this is not possible and you must turn cutter compensation on and off at an arc, always pick the Point Before Pierce and the Point After Retract either tangent to the arc or perpendicular to it. These are examples of starting and ending cutter compensation on arcs. The auto cutter compensation feature may also be used.



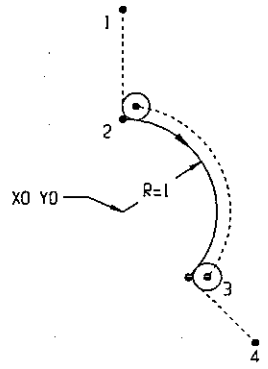
Point Before Pierce	X -1.5 Y 1.0000
Compensation	[Right]
Pierce Point	X 0.0000 Y 1.0000
Point After Retract	X 0.0000 Y -1.4142
[Auto Right]	Starts and stops the same



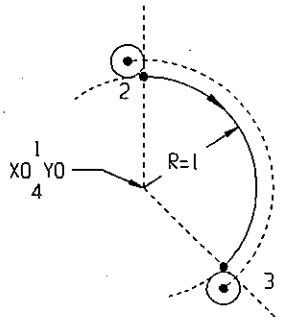
Point Before Pierce	X 0.0000 Y 0.0000
Compensation	[Right]
Pierce Point	X 0.0000 Y 1.0000
Point After Retract	X 0.0000 Y 0.0000



Point Before Pierce	X -1.5 Y 1
Compensation	[Left]
Pierce Point	X 0.0000 Y 1.0000
Point After Retract	X 0.0000 Y -1.4142
[Auto Left]	Starts and stops the same



Point Before Pierce	X 0.0000
	Y 2.0000
Compensation	[Left]
Pierce Point	X 0.0000
	Y 1.0000
Point After Retract	X 2.0000
	Y -2.0000



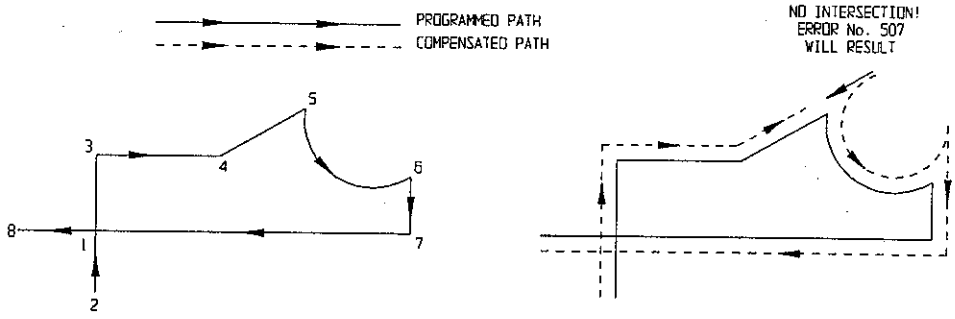
Point Before Pierce	X 0.0000
	Y 0.0000
Compensation	[Left]
Pierce Point	X 0.0000
	Y 1.0000
Point After Retract	X 0.0000
	Y 0.0000

The following is a discussion of some of the problems that can be encountered with cutter compensation and unusual geometries.

Because of the intersectional nature of the compensation package, there has to 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.

NOTE: All cutter compensation examples are shown without Z axis moves.

1 = Pierce Point
2 = Point Before Pierce
8 = Point After Retract



(a) Path of a cutter with 0" tool diameter

(b) Path of a cutter with non-zero tool diameter

Figure 7
Explanation of How Displaced Tool Paths
Cannot Have an Intersection

The solution to the above part is to introduce a 00.0001" chamfer or round corner at Point 5 between the non-intersecting surfaces.

1 = Pierce Point
2 = Point Before Pierce
9 = Point After Retract

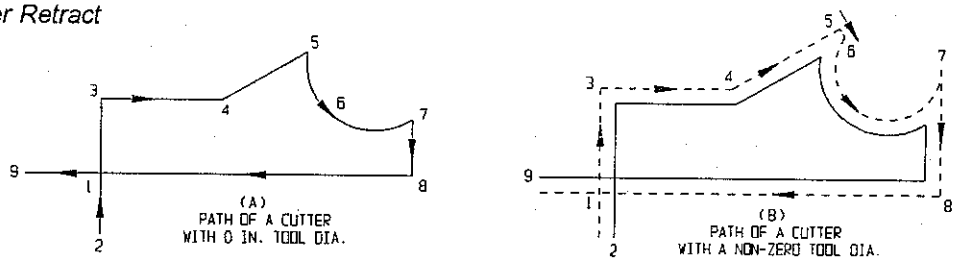
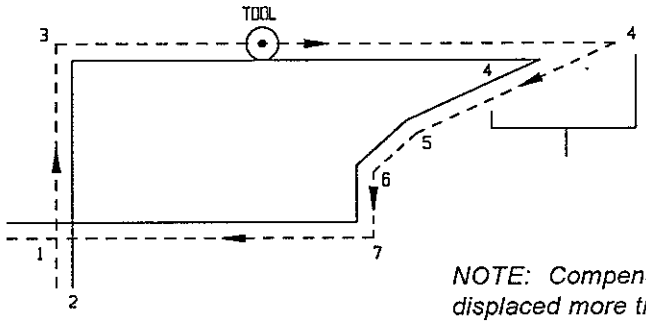


Figure 8
Explanation of How a 00.0001" Chamfer Should Be Introduced
To Solve a No-intersection Problem

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.

1 = Pierce Point
2 = Point Before Pierce
8 = Point After Retract



NOTE: Compensation point (4) is displaced more than the tool radius away from (4).

Figure 9
Outside "V" Cutter Compensation

Figure 10 shows how a 00.0001" chamfer added at Point (4) has saved an unnecessary departure.

1 = Pierce Point
2 = Point Before Pierce
9 = Point After Retract

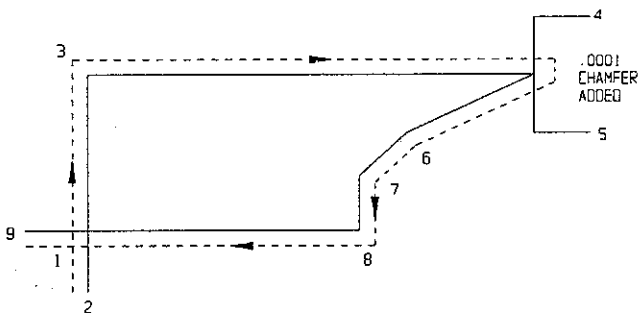


Figure 10
Outside "V" Cutter Compensation Solution

The following case shows how the compensated point for an inside "V" will stay away from the programmed point by more than the tool radius.

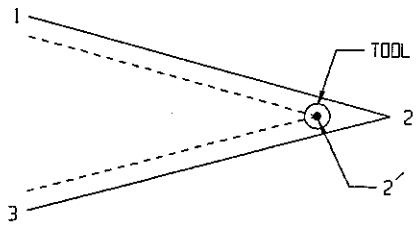


Figure 11
Inside "V" Cutter Compensation

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.

How to Determine How the Compensated Path Will Look

Step 1 Sketch actual part and label points in sequence.

2 = Pierce Point
1 = Point Before Pierce
11= Point After Retract

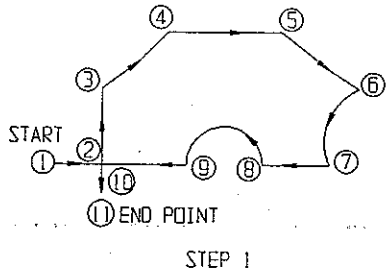


Figure 11.1
Compensation Exercise Step 1

Step 2 Sketch lines displaced tool radius away from part surface from Point (1) to Point 11.

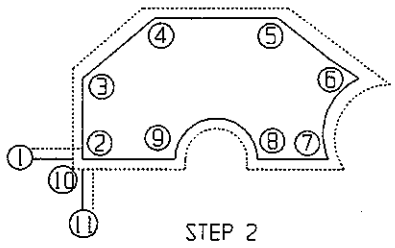


Figure 11.2
Compensation Exercise Step 2

Step 3 Check to see if 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 if the error is not corrected.

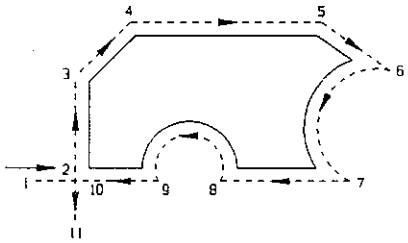


Figure 11.3
Compensation Exercise Step 3

Step 4 Point (1) is the Point Before Pierce and Point (11) is the Point After Retract. The tool never goes to these points, but it is useful to plot them as they determine where the tool actually starts and stops. The tool will come down at Point (2) and raise up at Point (10).

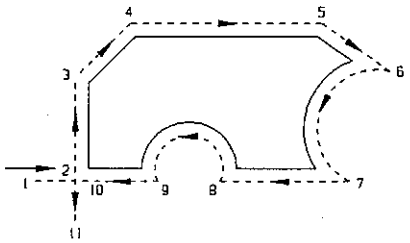


Figure 11.4
Compensation Exercise Step 4

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 starting at (2) will leave a little notch of uncut material. A similar case is obvious when the tool retracts at (10). There again the corner will be left uncut.

The solution is to rearrange the Point Before Pierce and the Point After Retract so that the corner is cut properly.

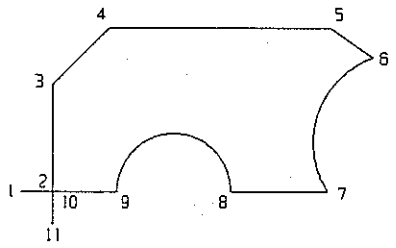
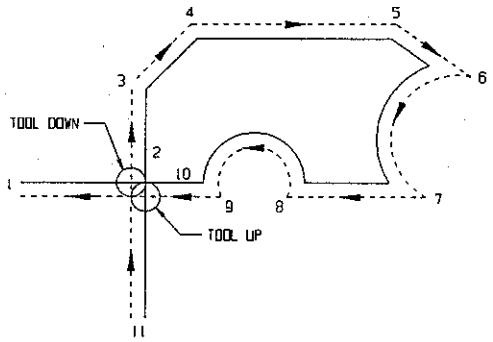


Figure 11.5
Compensation Exercise Step 5

Step 6 Note how we fixed this problem by simply making Point (11) the Point Before Pierce and Point (1) the Point After Retract.

11 = Point Before Pierce
2 = Pierce Point
10 = Last move of program
1 = Point After Retract



As can be seen from the starting and ending corner definitions beginning on page 4-6, what we have done is simply change the type of starting and ending corner from 90° to 180°.

A few more points are to be noted for the above example:

1. The starting and ending corners do not have to be the same. They could be any combination that is needed to properly cut the part.

2. The circle at (9) and (8) points to complications that will arise if the tool radius is increased indefinitely. As the radius is increased, (9) and (8) will keep moving closer to one another. For some radius value they will become identical. If the radius is increased further, the tool radius will have become too large to make that circle, and the system will give an error telling the operator that an intersection cannot be found at that line.

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.

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.

5. 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. Now 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 <u>smaller</u> tool radius, the part can be made under or <u>oversize</u>.

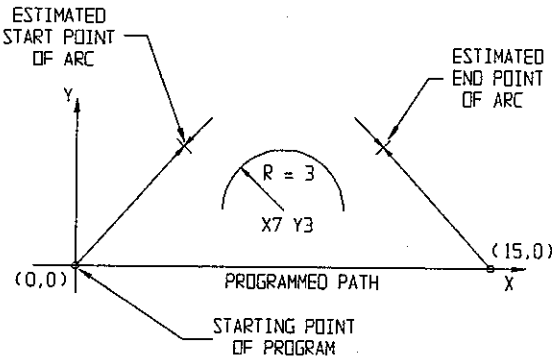
6. In general when using cutter compensation, no feature on the part can be smaller than the tool radius. This includes such things as slots, arcs and vees. If a part contains such features, they should be replaced by straight lines, cutter compensation should be turned off, or a smaller tool should be used.

<u>Trig Help</u>

Trig help will allow the programmer to estimate both the start and end points of any arc. The control will then 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

Program1

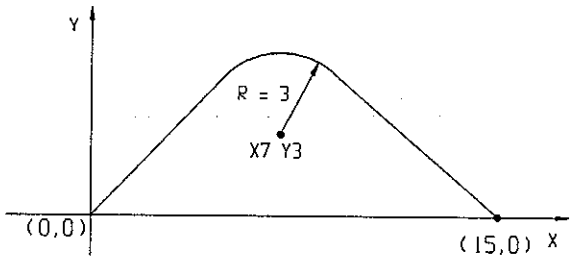


Line F20 X0 Y0

Line X2 Y6 (estimated start point of arc)

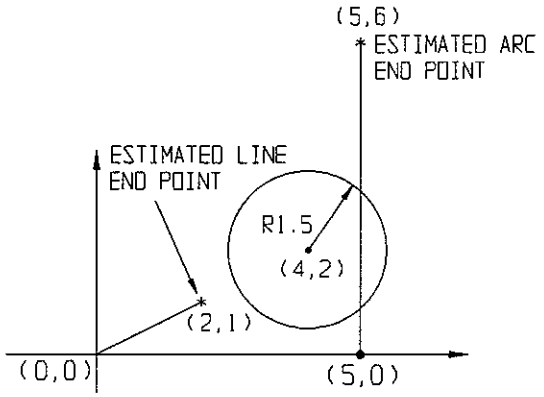
Arc CW R3
Center XC7 YC3
End point X12 Y6 (estimated end point of arc)

Line X15 Y0



Path Generated by Program 1

Program 2



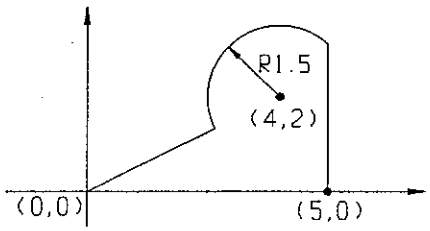
Programmed Path

Line F20 X0 Y0

Line X2 Y1 (estimated start point of arc)

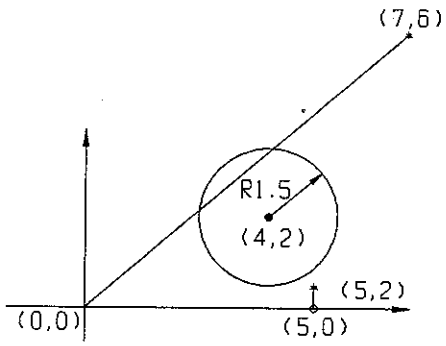
Arc CW R1.5
Center XC4 YC2
End point X5 Y6 (estimated end point of arc)

Line X5 Y0



Path Generated by Program 2

Program 3



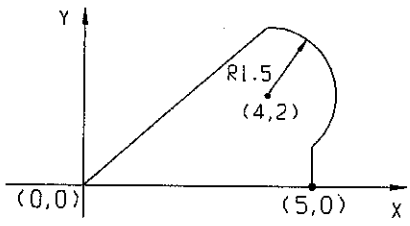
Programmed Path

Line F20 X0 Y0

Line X7 Y6 Cartesian (estimated start point of arc)

Arc CW R1.5
Center XC4 YC2
End point X5 Y.2 (estimated end point of arc)

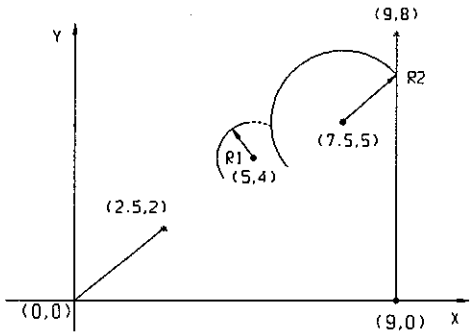
Line X5 Y0



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

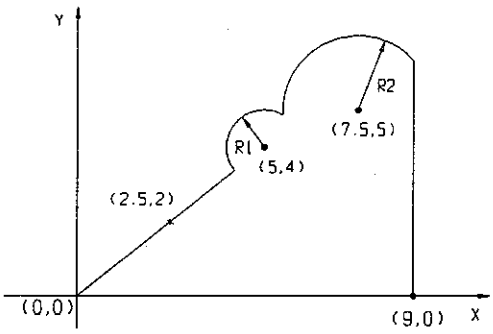
Line F20 X0 Y0

Line X2.5 Y2 (estimated start point of first arc)

Arc CW R1
Center XC5 YC4
End point X5 Y5 (estimated end point of first arc)

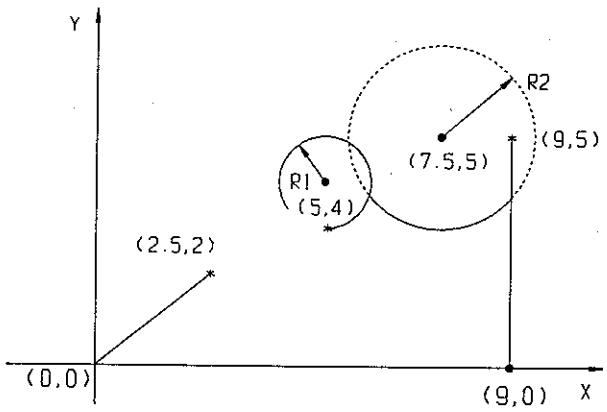
Arc CW R2
Center XC7.5 YC5
End point X9 Y8 (estimated end point of second arc)

Line X9 Y0



Path Generated by Program 4

Program 5



Programmed Path

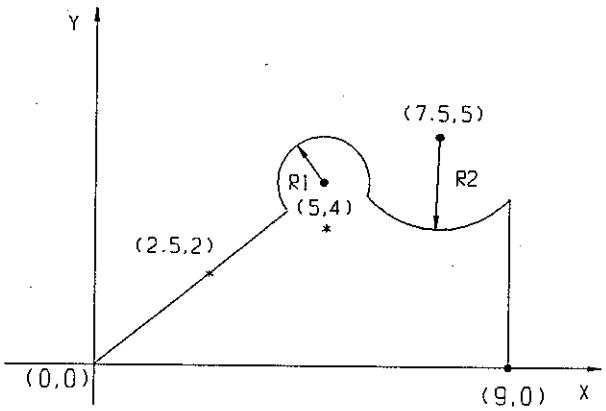
Line F20 X0 Y0

Line X2.5 Y2 (estimated start point of first arc)

Arc CW R1
Center XC5 YC4
End point X5 Y3 (estimated end point of first arc)

Arc CCW R2
Center XC7.5 YC5
End point X9 Y5 (estimated end point of second arc)

Line X9 Y0



Path Generated by Program 5

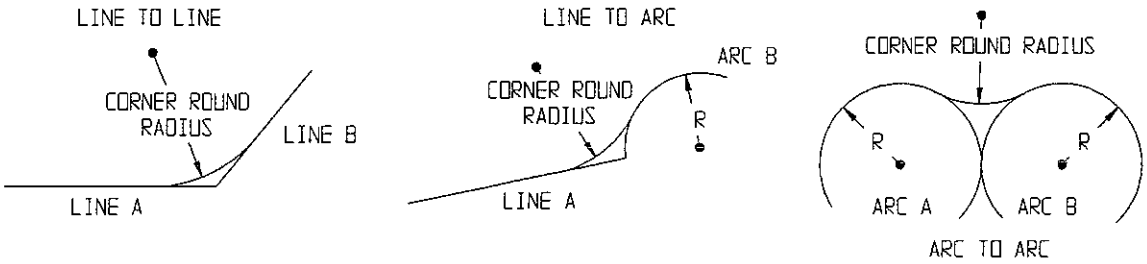
In general, when estimating arc-to-arc intersections the end points chosen should be on the arc. The easiest points to pick are one of the quadrant points (0°, 90°, 180°, 270°).

Things to Remember When Estimating Points

- Estimating should only be used with line/circle, circle/circle, and circle/line paths.
- For line/circle and circle/line, the start and end point estimates **must** 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 end points chosen should be on the arc. The easiest point to pick on an arc is at one of the quadrant points (0°, 90°, 180°, 270°).

Automatic Corner Round

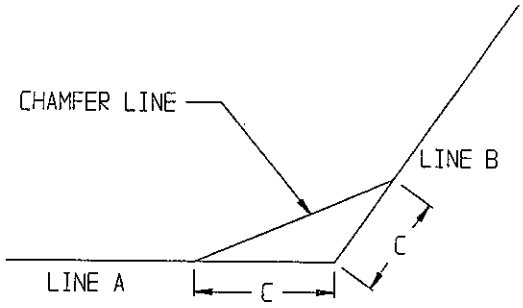
This function allows the operator to insert a tangent arc between any two lines, a line and an arc, and any two arcs. This option will appear at the bottom of every line and arc screen. The prompt at the bottom of the screen is End Option [ ]. To select this option, cursor to it and push the toggle key (F3) until Corner Round appears. By simply giving the control the desired blend radius, it will do all the calculations necessary to insert a blend radius. The following are some examples.



Automatic Chamfering

This function allows the operator to insert a chamfer between any two lines. It is one of the End Options and is accessed the same as the Corner Round. The following is an example of chamfering.

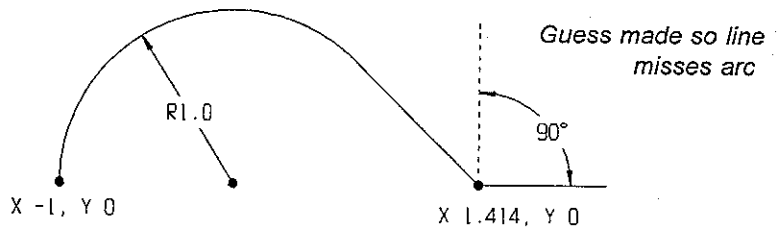
C is the distance entered when a chamfer is selected.



To Line Function on the Arc Screen

This function allows the operator to program a tangent or intersecting line from an arc to a point without calculating the tangency or intersection point on the arc. This function is accessed by pushing the toggle key (F3) when the cursor is at the end point of the arc selection. The following are some examples using this function.

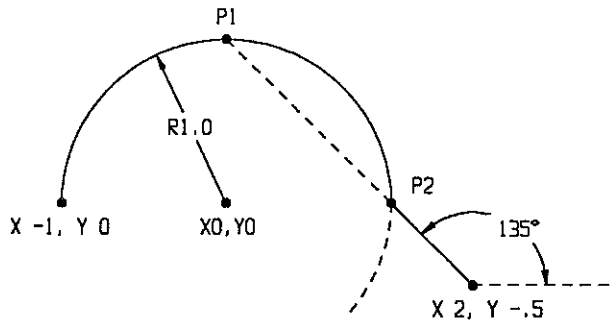
TANGENT LINE EXITING AN ARC



Program

Line	F25 Cartesian X-1 Y0		
Arc	CW R1 Center	XC0 YC0	
	End point to line	X1.414 Y0 A90°	(end point of line) (approximate angle of line to circle. If the angle causes the line to miss the arc, then a tangent will be calculated.)
Line	X1.414 Y0		

LINE AT GIVEN ANGLE
INTERSECTING AN ARC



Program

Line	F25 Cartesian		
	X-1 Y0		
Arc	CW R1		
	Center	XC0 YC0	
	End point	X2 Y-.5	(Short) P2 *or*
	To line	A135°	(Long) P1
Line	X2 Y-.5		

If the Short option is picked, the arc will go to P2 and then the line will start. If the Long option is chosen, the line will start at P1.

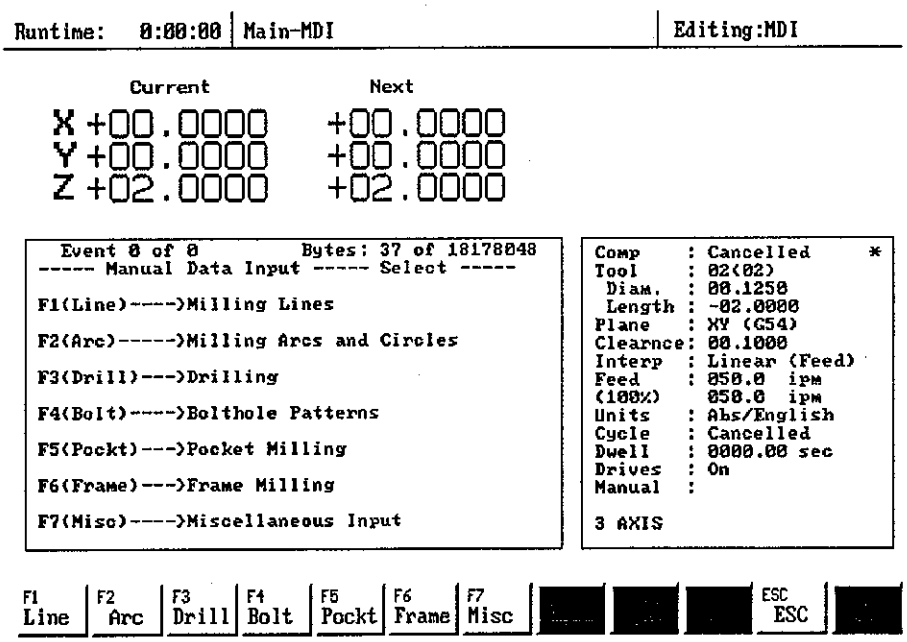
Chapter 5

Operating the Centurion 1 in MDI Mode

The MDI mode of the Centurion 1 was designed for the operator who just wants to walk up to the machine, **not** write a program, and do some simple machining operations. These operations include the following:

1. Milling Lines
2. Milling Arcs and Circles
3. Drilling
4. Bolt Patterns
5. Pocket Milling
6. Frame Milling
7. Miscellaneous

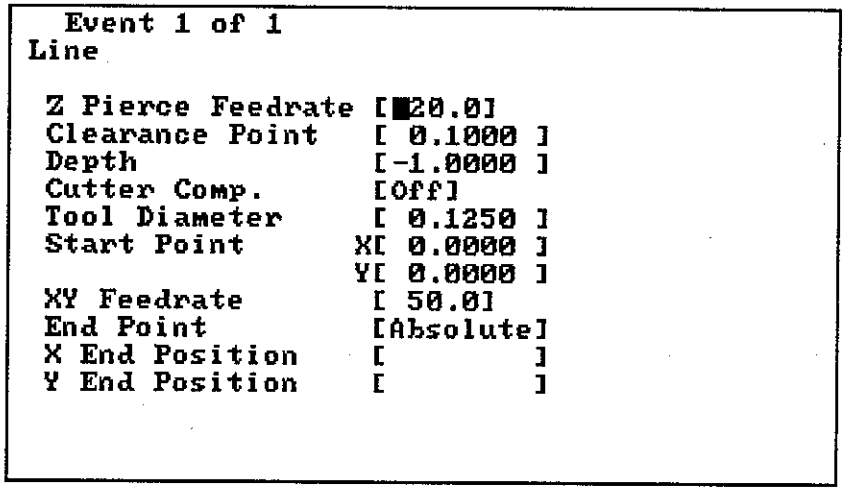
To enter the MDI mode, just select MDI from the main display screen (F5). If the control is not already at the main screen,just keep pushing the ESC key until this screen comes up. When MDI is selected the following screen will be displayed.



To execute one of the listed functions just push the corresponding Function (F) key. The following pages offer a detailed description of each MDI function.

F1 LINE Milling Straight and Angled Lines

When F1 LINE is selected the following window appears:



This selection would be used for milling operations.

1. The Z Pierce Feedrate field is used to specify the Z plunge feedrate of the tool into the work.

2. The Return Point entry normally is left at "Clearance", which means that after the cut has been finished, the tool will return to the value set for the clearance plane. If it is changed to "Initial", the tool will then return to the point it was at when the milling operation was started.

3. The Clearance value is usually positive (above the part) and is the dimension the tool will rapid to before the feed mode is initiated.

4. The Final Z Depth value is usually negative (into the part) and will be the depth of cut for the ensuing XY move.

5. Compensation and Tool Number fields would be used if it is desired to compensate for the cutter diameter during the milling operation. To use these fields, simply select whether you would like the tool on the right or left side of the part by using the F3 toggle key and enter the tool number being used. The diameter of the selected tool must be entered into the tool table prior to the start of the line cycle. If these fields are not used the centerline of the tool will follow the programmed path.

6. The Start Point field will always contain the current position of the machine. If this is the position the cut is going to start from, these entries can be skipped. However, if it is desired to move over to another point and then start the cut, that

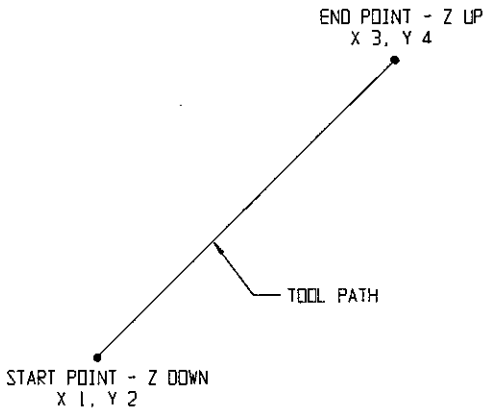
point's coordinates should be entered. The cut will always be made between the coordinates of the Start Point and End Point.

7. The Coordinates selection allows two different methods of entering the **End** Point. Pushing the F3 toggle key will cycle through each of the possibilities.

A. **Absolute** requires an X and Y position to describe the End Point.

Feedrate F[30.0]
Coordinates [ABSOLUTE]

X End Position X[3.0000]
Y End Position Y[4.0000]



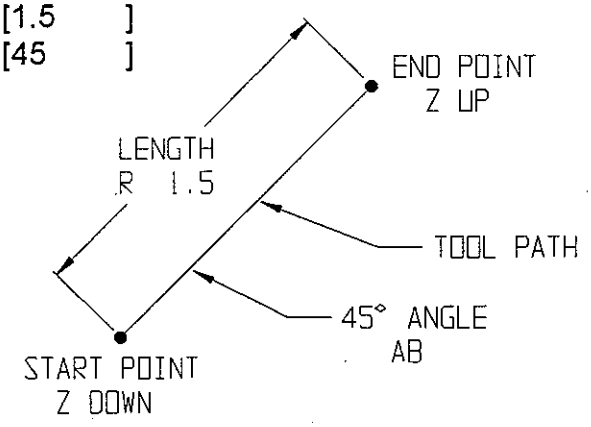
B. **Polar** requires an angle and a length to describe the End Point. .

Polar Type Current
End Point Display

Feedrate F[30.0]
End Point [POLAR]

Type [CURRENT]

Length of Line [1.5]
Angle of Line [45]



The feedrate is the same for all line types, and if cutter compensation is turned on, the tool will be offset to the right or left of the tool path by the radius of the cutter. After the desired coordinates are entered, push the Store Key (F1) to initiate the cutting sequence.

When the Store Key is pushed, the Centurion 1 will calculate the tool path and graphically display it on the CRT. It then prompts the operator with the following message:

```
ATTENTION: <Cycle Start>
to execute MDI command.
```

Make sure the spindle is on before pushing the Cycle Start button.

If the graphic display is the desired cut, push the Cycle Start button. The machine will now move to the start of the cut and rapid Z to the clearance plane, and then feed to the final Z depth.

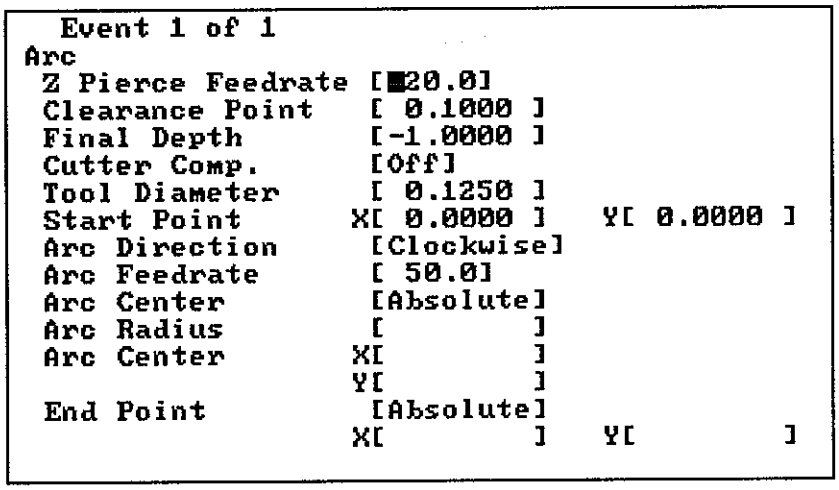
Once Z has reached the final Z depth the axis will move along the programmed path until the End Point is reached. At this point Z will rapid to either the clearance plane or the initial starting Z depth.

After Z has completed its retract the Centurion 1 will return to the MAIN-MDI menu and be ready for the selection of the next operation.

F2 ARC Arcs and Circles

When F2 ARC is selected the following window appears:

MAIN-MDI-ARC



This selection would primarily be used for milling operations involving arcs and circles. The Arc command works exactly like the line command except that the tool path described is an arc rather than a line.

1. The Z Pierce Feedrate field is used to specify the Z plunge feedrate of the tool into the work.

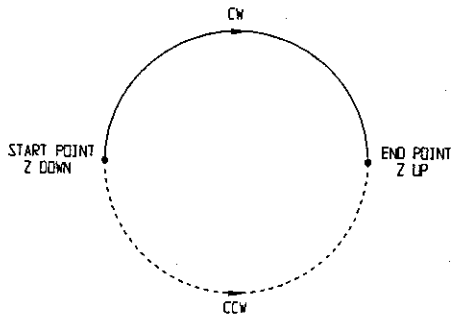
2. The Clearance value is usually positive (above the part) and is the dimension the tool will rapid to before the start of a cut and at the end of a cut.

3. The Final Z Depth value is usually negative (into the part) and will be the depth of cut for the ensuing XY move.

4. The Feedrate, Cutter Comp, Tool Diameter, and Start Point are all identical to the line command.

5. The **Start Point** field will always contain the current position of the machine. If this is the position the arc is going to start at, these entries can be skipped. However, if it is desired to move over to another point and then start the cut, that point's coordinates should be entered. The cut will always be made between the coordinates of the Start Point and End Point.

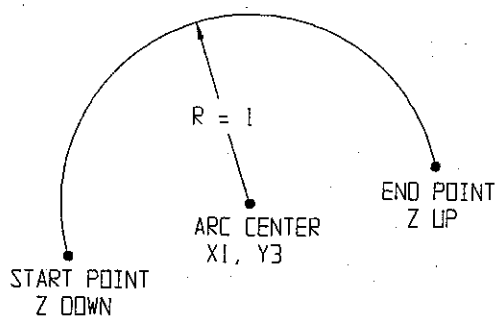
6. The **Arc Direction [CW/CCW]** field is changed with the F3 toggle key. The direction of the cut between the Start Point and End Point changes as follows:



7. The **Arc Center** field offers two different ways to describe where the center of the circle is located in Skill Level 1 and five ways in Skill Level 2.

A. Center [ABSOLUTE]

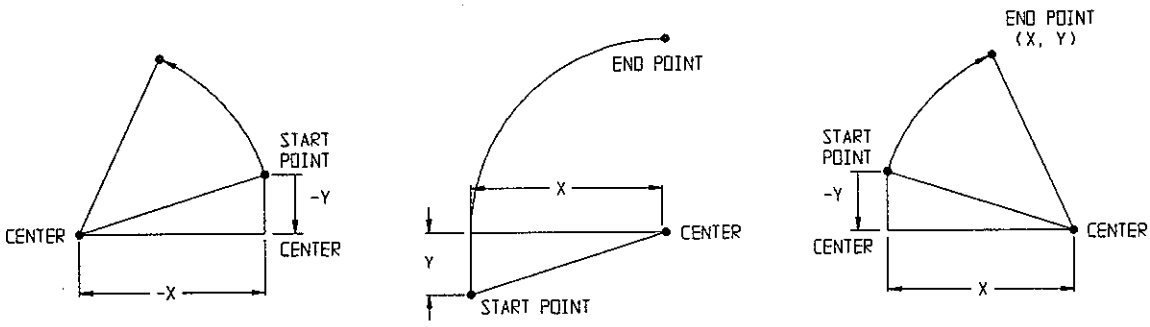
This method is used when the X and Y coordinates of the arc center are known.



B. Center [INCREMENTAL]
Skill Level 2 Only

This method defines the center of the arc relative to the Start Point of the arc. The arc center is defined by X center and Y center for the X and Y axes. The numerical value following X center or Y center is the distance from the start point to the arc center in the X or Y axis. X center and Y center are always incremental values.

The sign of X center and Y center depends on the relationship of the center to the Start Point as shown below:

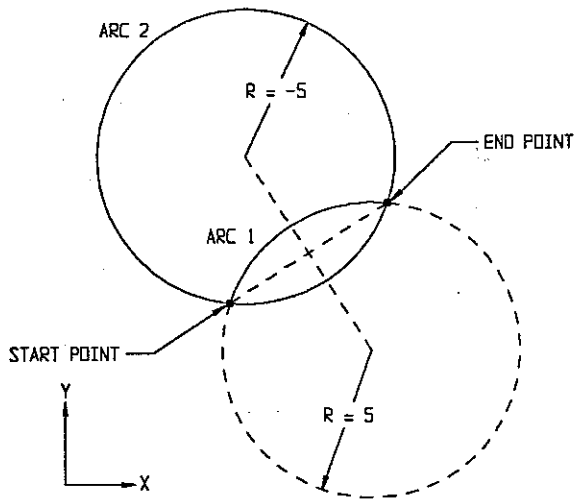


Programming with Circular Interpolation

C. Center [RADIUS ONLY]
Skill Level 2 Only

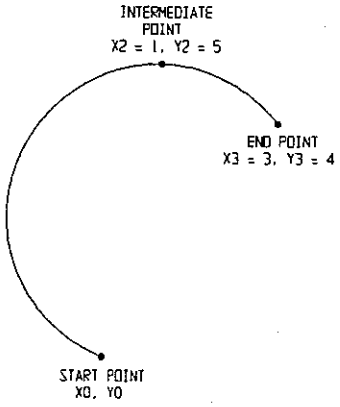
This format would be used when the Radius and the Start and End Points of the arc are known. When using this method there are two types of arcs which can be generated. One is less than 180°, and the other is greater than 180° as shown in the figure that follows. When the arc exceeds 180° the radius must be specified as a negative value. It should be noted that the Start and End Points may **not** have the same values.

For arc 1 (less than 180°), Radius R = [5.0000].
For arc 2 (greater than 180°), Radius R = [-5.0000].



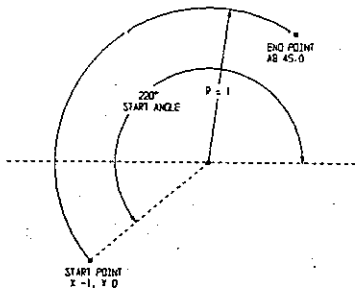
D. Center [3 Points]
Skill Level 2 Only

This would be used if the center and radius of the arc are not known, but the start, end, and one other point are known.

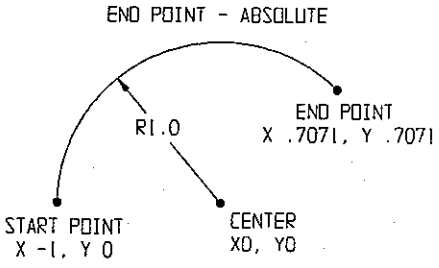


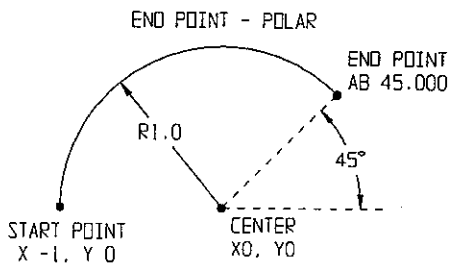
E. Center [Polar]

This would be used if the center is unknown, but the radius, start point, and start angle are known.

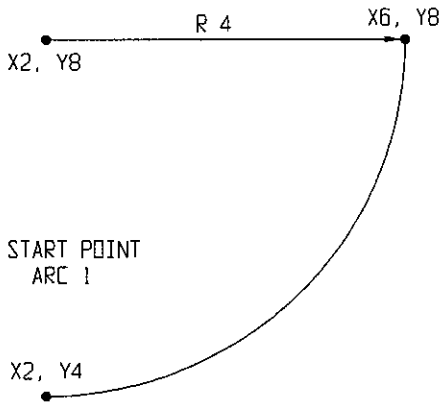


5. The **End Point** of an arc can be described one of two ways: either as an X and Y coordinate, or as a polar angle related to the center. To make a selection between the two types, just push the F3 toggle key at the End Point selection.





The following example shows the program to cut the same arcs using the five different ways to specify them. The methods used to program the arcs would depend on which dimensions are given on the part print.



Program 1 Center [ABSOLUTE]
End Point [ABSOLUTE] or [POLAR]
Skill Level 1 and 2

```
  Event 1 of 1
Arc
 Z Pierce Feedrate [ 20.0]
 Clearance Point   [ 0.1000 ]
 Final Depth       [-1.0000 ]
 Cutter Comp.      [Off]
 Tool Diameter     [ 0.1250 ]
 Start Point      X[ 0.0000 ]   Y[ 0.0000 ]
 Arc Direction     [CounterClockwise]
 Arc Feedrate      [ 50.0]
 Arc Center        [Absolute]
 Arc Radius        [         ]
 Arc Center       X[         ]
                  Y[         ]
 End Point         [Absolute]
                  X[         ]   Y[         ]
```

Program 2 Center [INCREMENTAL]
End Point [ABSOLUTE] or [POLAR]
Skill Level 2

```
  Event 1 of 1
Arc
 Z Pierce Feedrate [ 20.0]
 Return Point    [Clearance]
 Clearance Point[ 0.1000 ]
 Final Depth     [-1.0000 ]
 Compensation    [Off]
 Tool Diameter   [ 0.1250 ]
 Start Point    X[ 0.0000 ]  Y[ 0.0000 ]
 Arc Direction   [CCW] Arc Feedrate [ 50.0]
 Center          [Incremental]
 Inc Center     X[         ]
                Y[         ]

 End Point       [Absolute]
                X[         ]  Y[         ]
```

Program 3 Center [RADIUS]
Skill Level 2

```
  Event 1 of 1
Arc
 Z Pierce Feedrate [ 20.0]
 Return Point    [Clearance]
 Clearance Point[ 0.1000 ]
 Final Depth     [-1.0000 ]
 Compensation    [Off]
 Tool Diameter   [ 0.1250 ]
 Start Point    X[ 0.0000 ]  Y[ 0.0000 ]
 Arc Direction   [CCW] Arc Feedrate [ 50.0]
 Center          [Radius Only]
 Radius Length   [         ]     [Short]


 End Point       [Absolute]
                X[         ]  Y[         ]
```

Program 4 Center [3 POINTS]
Skill Level 2

```
  Event 1 of 1
Arc
 Z Pierce Feedrate [ 20.0]
 Return Point    [Clearance]
 Clearance Point[ 0.1000 ]
 Final Depth     [-1.0000 ]
 Compensation    [Off]
 Tool Diameter   [ 0.1250 ]
 Start Point    X[ 0.0000 ]  Y[ 0.0000 ]
                          Arc Feedrate [ 50.0]
 Center           [3 Point]
       Point2   X[         ]
                Y[         ]
       Point3   X[         ]
                Y[         ]
```

Program 5 Center [POLAR]
Skill Level 1 and 2

```
  Event 1 of 1
Arc
 Z Pierce Feedrate [ 20.0]
 Return Point    [Clearance]
 Clearance Point[ 0.1000 ]
 Final Depth     [-1.0000 ]
 Compensation    [Off]
 Tool Diameter   [ 0.1250 ]
 Start Point    X[ 0.0000 ]  Y[ 0.0000 ]
 Arc Direction   [CCW] Arc Feedrate [ 50.0]
 Center          [Polar]
 Radius Length   [4.0000  ]
 Start Angle     [270     ]

 End Point       [Absolute]
                X[         ]  Y[         ]
```

When the Store Key is pushed the Centurion 1 will calculate the tool path and graphically display the arc on the CRT. It will then prompt the operator with the same message listed on page 5-4.

F3 DRILL Drilling Cycles

In Skill Level 1 Return to Clearance is the only tool retract option. The initial selection is Level 2 Only.

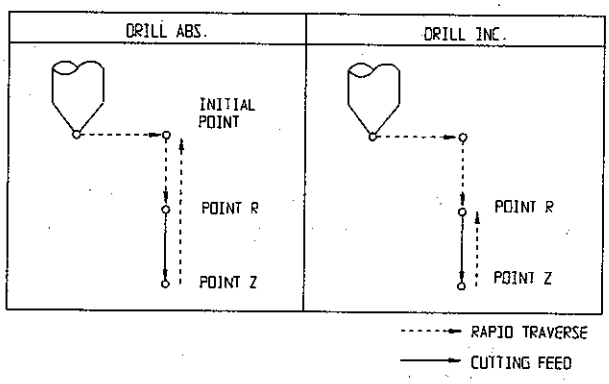
When F3 DRILL is selected, the following window will appear:

MAIN-MDI-DRILL

```
  Event 1 of 1
[Drill Cycle]
 Z Pierce Feedrate      [ 20.0]
•Return Point           [Clearance]
 Clearance Point        [ 0.1000 ]
 Final Depth            [-1.0000 ]


 Coordinates            [Absolute]
 X Position             [ 0.0000 ]
 Y Position             [ 0.0000 ]
```

Positions for drilling cycles can be specified in either polar or cartesian coordinate systems. The definitions of these coordinate points are the same as the line screens. See page 5-3 for a complete description. The only difference between the various drill cycles is the action of the Z axis. The following pages describe the Z movements for each of the drill cycles.

Drilling Cycle



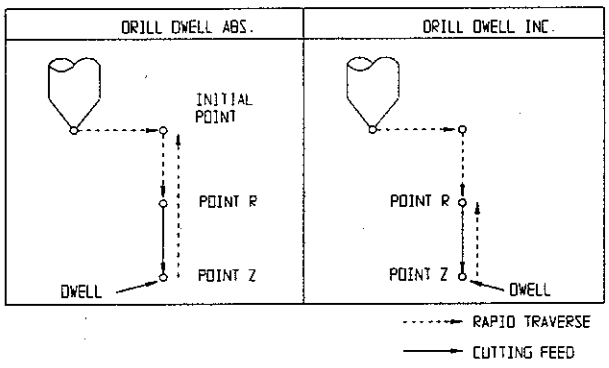
The Drill command specifies a drilling cycle. This cycle will do the following:

1. Rapids to point R
2. Feeds down to point Z.
3. Rapids to initial point or point R as determined by [Clearance] or [Initial] for the Return Point.

Upper Part of Drill Cycle Screen

```
  Event 1 of 1
[Drill Cycle]
  Z Pierce Feedrate       [ 20.0]
• Return Point            [Clearance]
  Clearance Point         [ 0.1000 ]
  Final Depth             [-1.0000 ]


  Coordinates             [Absolute]
  X Position              [ 0.0000 ]
  Y Position              [ 0.0000 ]
```

Drilling Cycle with Dwell



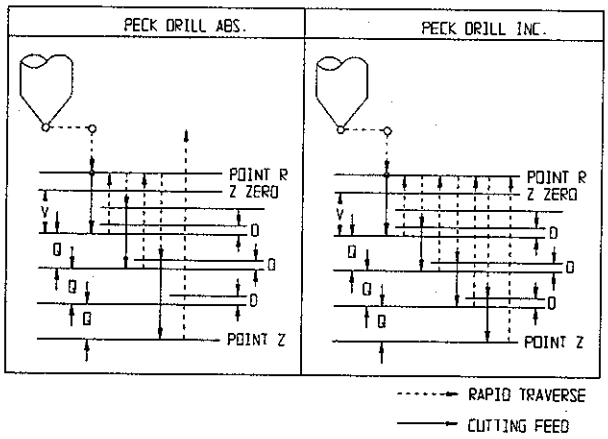
The Drill/Dwell command is similar to the Drill command; however, a specified Dwell is performed at the bottom of the hole. This cycle will do the following:

1. Rapids to point R.
2. Feeds down to point Z.
3. Dwells by XX seconds.
4. Rapids to initial point or point R as determined by the Return Point.

Upper Part of Drill/Dwell Screen

```
  Event 1 of 1
[Drill Cycle w/Dwell]
 Z Pierce Feedrate      [ 20.0]
*Return Point           [Clearance]
 Clearance Point        [ 0.1000 ]
 Final Depth            [-1.0000 ]
 Dwell Time in Sec      [  0.0   ]


 Coordinates            [Absolute]
 X Position             [ 0.0000 ]
 Y Position             [ 0.0000 ]
```

Peck Drilling Cycle



The Peck/Drill command specifies the Peck Drill cycle. This cycle will do the following:

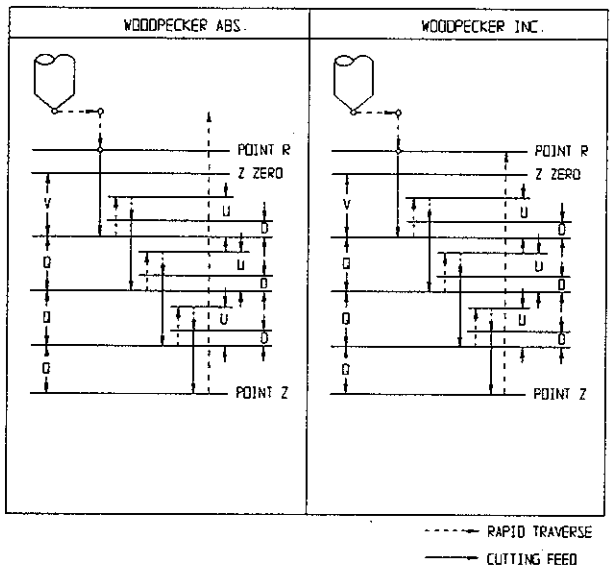
1. Rapids to point R.
2. Feeds to point V.
3. Rapids up to point R.
4. Rapids down to D value.
5. Feeds down by Q value or Z point (whichever is less).
6. Repeats steps 3-5 until point Z is reached.
7. Rapids to initial point or point R as determined by the Return Point.

Note: *The V code is optional; if left out, the first depth would equal R___ - Q___.*

Peck Drill Screen

```
  Event 1 of 1
[Drill/Peck Cycle]
  Z Pierce Feedrate      [ 20.0]
* Return Point           [Clearance]
  Clearance Point        [ 0.1000 ]
  Final Depth            [-1.0000 ]
  First Depth            [ 1.0000 ]
  Depth Increment        [-1.0000 ]

  Coordinates          [Absolute]
  X Position           [ 0.0000 ]
  Y Position           [ 0.0000 ]
```

Chip Breaker Drill Cycle



The Chip Breaker command specifies a high speed peck cycle. This cycle will do the following:

1. Rapids to point R.
2. Feeds down to point V.
3. Rapids up to 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.

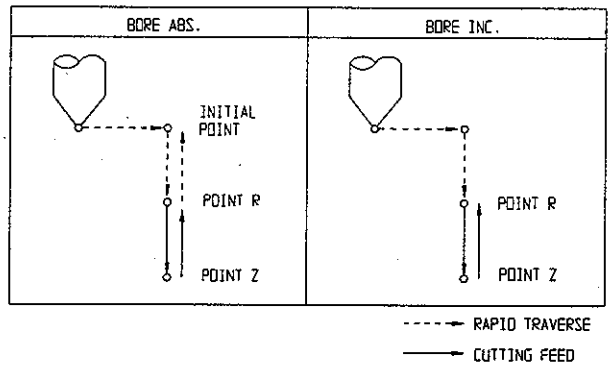
7. Rapids to initial point or point R as determined by G98/G99.

Note: The V code is optional; if left out, the first depth would equal R___ - Q___.

Chip Breaker Drill Screen

```
  Event 1 of 1
[Chip Breaker Drill Cycle]
  Z Pierce Feedrate       [ 20.0]
• Return Point            [Clearance]
  Clearance Point         [ 0.1000 ]
  Final Depth             [-1.0000 ]
  First Depth             [ 1.0000 ]
  Depth Increment         [-1.0000 ]

  Coordinates           [Absolute]
  X Position            [ 0.0000 ]
  Y Position            [ 0.0000 ]
```

Boring Cycle



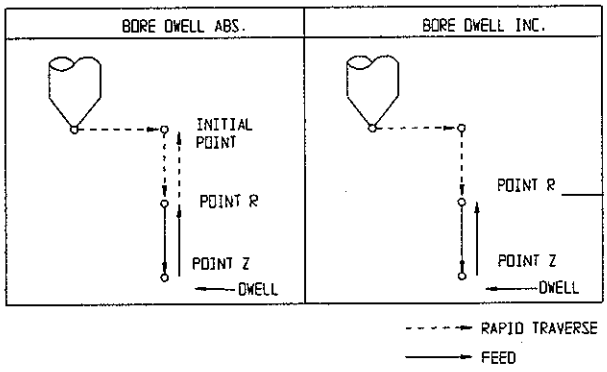
The Bore command specifies the boring cycle. At each axis position this cycle will do the following:

1. Rapids to point R.
2. Feeds to point Z.
3. Feeds to point R.
4. Rapids to initial point if specified by the Return Point code.

Bore Cycle Screen

```
  Event 1 of 1
[Bore Cycle]
  Z Pierce Feedrate       [ 20.0]
• Return Point            [Clearance]
  Clearance Point         [ 0.1000 ]
  Final Depth             [-1.0000 ]


  Coordinates             [Absolute]
  X Position              [ 0.0000 ]
  Y Position              [ 0.0000 ]
```

Boring Cycle with Dwell



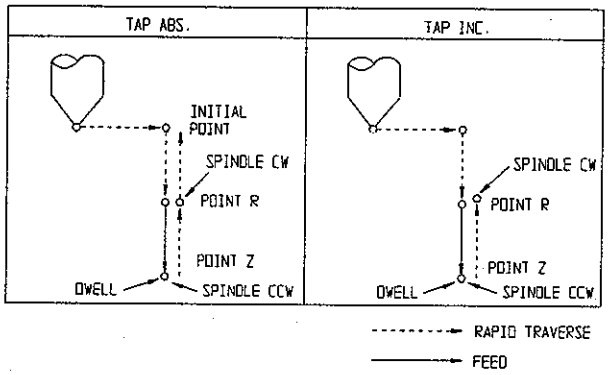
The Bore/Dwell command specifies the bore with dwell cycle. At each following axis position this cycle will do this:

1. Rapids to point R.
2. Feeds to point Z.
3. Dwells at bottom (specified by Dwell Time).
4. Feed to point R.
5. Rapids to initial point, if specified by the Return Point.

Bore with Dwell Screen

```
  Event 1 of 1
[Bore Cycle w/Dwell]
  Z Pierce Feedrate       [ 20.0]
* Return Point            [Clearance]
  Clearance Point         [ 0.1000 ]
  Final Depth             [-1.0000 ]
  Dwell Time in Sec       [  0.0   ]


  Coordinates             [Absolute]
  X Position              [ 0.0000 ]
  Y Position              [ 0.0000 ]
```

Tap Cycle



The Tap command specifies right hand tapping. At each following axis position this cycle will do this:

1. Rapids to point R.
2. Feeds to point Z.
3. Dwells before reversing (specified by the B code).
4. Reverses spindle (CCW).
5. Dwells after reversing (specified by the P code).
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 and the cycle does not stop until the end of the return operation when the feedhold is applied.*

TAPPING FEEDS AND SPEEDS

1/pitch = lead

RPM x lead = feedrate

Example: 1/4-20 tap, spindle rpm 400
1/20 = .05 (lead)
400 x .05 = 2- (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. Not all machines have an automatically reversing spindle. If the spindle does not automatically reverse, this cycle cannot be used.

Tap Cycle Screen

```
  Event 1 of 1
[Enable Tap Cycle]
 Z Pierce Feedrate      [ 20.0]
*Return Point           [Clearance]
 Clearance Point        [ 0.1000 ]
 Final Depth            [-1.0000 ]
 Dwell before Spdl Rev  [  0.0   ]
 Dwell after Spdl Rev   [  0.5   ]

 Coordinates          [Absolute]
 X Position           [ 0.0000 ]
 Y Position           [ 0.0000 ]
```

NOTES ON CANNED CYCLE SPECIFICATIONS:

Note 1: *The spindle must be turned on by the miscellaneous function before a canned cycle is specified.*

Misc. Spindle CW

.

.

.

Drill Cycle and Positions

.

.

.

Misc. Spindle Stop

Note 2: *If the block contains X, Y or Polar data, drilling is performed in canned cycle mode. If the block does not contain the X, Y or Polar data, drilling is not performed.*

Note 3: *If a following block contains a Z position by itself, drilling will not be performed. The Z axis will, however, rapid to this point. This can be used to manipulate a tool up-and-over obstructions without disabling the canned cycle.*

F4 BOLT Bolthole Circles

When F4 BOLT is selected, the following window will appear:

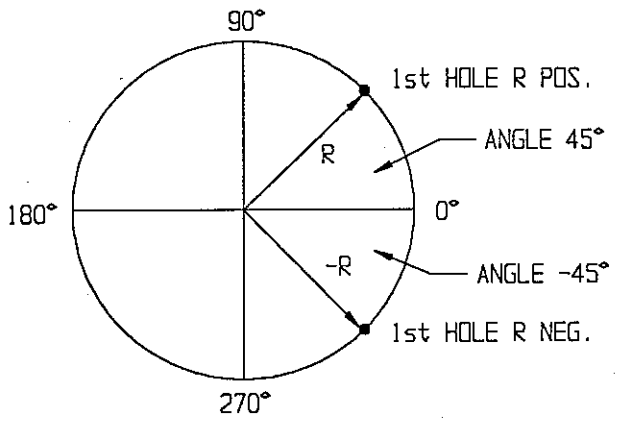
MAIN-MDI-BOLT

```
  Event 1 of 1
[Bolthole Drill Cycle]
 Z Pierce Feedrate      [ 20.0]
*Return Point           [Clearance]
 Clearance Point        [ 0.1000 ]
 Final Depth            [-1.0000 ]



 Bolthole Center    X[ 0.0000 ]
                    Y[ 0.0000 ]
 Bolthole Radius     [ 2.0000 ]
         Direction   [ClockWise]
 Angle to 1st Hole   [45.0000 ]
 # Of Holes To Be Made [  7]
 # Of Holes In 360 Deg [  9]
```

Pushing the F3 TOGL key at the first entry will select all the possible drill cycles that can be used with the bolthole routine. These cycles are described in detail on pages 5-12 thru 5-20.

Again, like the Arc and Line screens, the current position of the machine will be automatically filled in as the center of the bolt circle. If this is not the center, just type in the correct dimensions. The bolthole routine specifies all the hole positions based on the following 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 Centurion 1 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 radius is counterclockwise from the 3 o'clock position. A negative radius is clockwise from 3 o'clock.

Note: Holes will be drilled clockwise from the first position.



Definition of Starting Angle

F5 POCKT Pocket Milling of Circular and Rectangular Pockets

When F5 POCKT is selected, the following window appears:

MAIN-MDI-POCKT

```
  Event 1 of 1
[Circular Pocket Clear]
 Z Pierce Feedrate [ 20.0]
•Return Point      [Clearance]
 Clearance Point   [ 0.1000 ]
 Z Down Method     [Ramp]
 Final Depth       [-2.0000 ]
 First Depth       [-0.3000 ]
 Depth Increment   [ 0.3000 ]
 XY Feedrate       [ 30.0]Tool Diam[ 0.1250 ]
 Pocket Center    X[ 0.0000 ] Y[ 0.0000 ]

 Pocket Radius     [ 3.0000 ]
 Cut Width         [ 0.4000 ]
 XY Finish Stock   [ 0.0000 ]
 Compensation   [On] Cut Dir [CCW (Climb)]
```

The first choice on this screen determines which type of pocket will be executed. Again, the F3 TOGL key will make the selection between rectangular and circular pockets and clear, finish or facing cycles for these pockets.

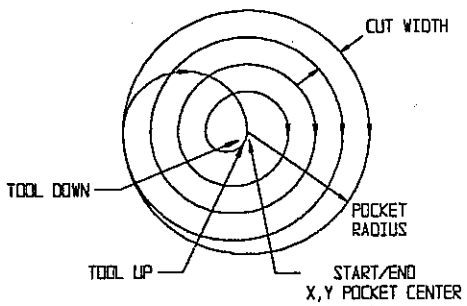
Once these selections are made, the screen will change to ask for the specific inputs necessary. A description of each cycle and its associated inputs follows.

Circle - Clear

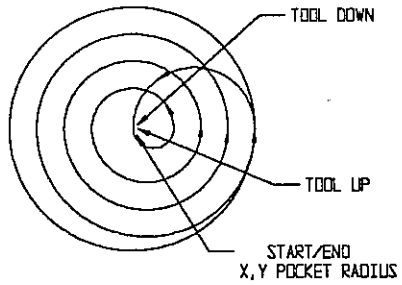
The Circular Pocket Clear menu will look as follows:

```
  Event 1 of 1
[Circular Pocket Clear]
 Z Pierce Feedrate [ 20.0]
•Return Point      [Clearance]
 Clearance Point   [ 0.1000 ]
 Z Down Method     [Ramp]
 Final Depth       [-1.0000 ]
 First Depth       [-1.0000 ]
 Depth Increment   [ 1.0000 ]
 XY Feedrate       [ 50.0]Tool Diam[ 0.1250 ]
 Pocket Center    X[1        ] Y[ 0.0000 ]

 Pocket Radius     [ 2.0000 ]
 Cut Width         [.25     ]
 XY Finish Stock   [.01      ]
 Compensation   [On] Cut Dir [CCW (Climb)]
```

The pocket generated will be the following shape. Simply fill in the correct dimensions for your pocket.



CW Circular Pocket Clearing



CCW Circular Pocket Clearing

The pocket center will be filled in with the current position of the machine and can be changed by simply typing over if necessary. If the XY finish stock is non-zero, the pocket will be smaller by that amount. If the tool compensation is on and the tool diameter is non-zero, the pocket will be cut to size taking the tool into consideration. The Z Down entry, Ramp or Plunge, will determine how Z moves to the next level.

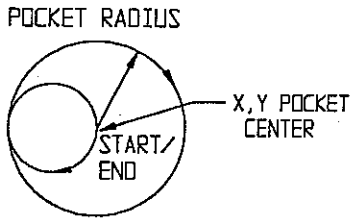
Circle - Finish Inside

The Circle Finish menu is as follows:

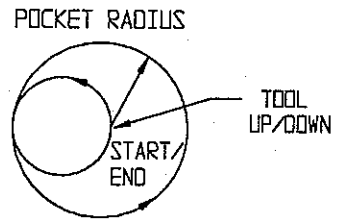
MAIN-MDI-POCKT

```
  Event 1 of 1
[Circular Finish Inside]
 Z Pierce Feedrate [ 20.0]
*Return Point      [Clearance]
 Clearance Point   [ 0.1000 ]
 Z Down Method     [Ramp]
 Final Depth       [-1.0000 ]
 First Depth       [-1.0000 ]
 Depth Increment   [ 1.0000 ]
 XY Feedrate       [ 50.0]Tool Diam[ 0.1250 ]
 Pocket Center    X[1        ] Y[ 0.0000 ]

 Pocket Radius     [ 2.0000 ]


 Compensation   [On] Cut Dir [CCW (Climb)]
```

The pocket generated will be the following shape. Just fill in the correct dimensions for your pocket.



Inside CW Finish Circle



Inside CCW Finish Circle

The pocket center will be filled in with the current position of the machine and can be changed by simply typing over if necessary. If the tool compensation is on and the tool diameter is non-zero, the pocket will be cut to size taking the tool into consideration. The Z Down Ramp or Plunge entry determines how the tool will move from one Z level to the next.

Rectangular - Clear

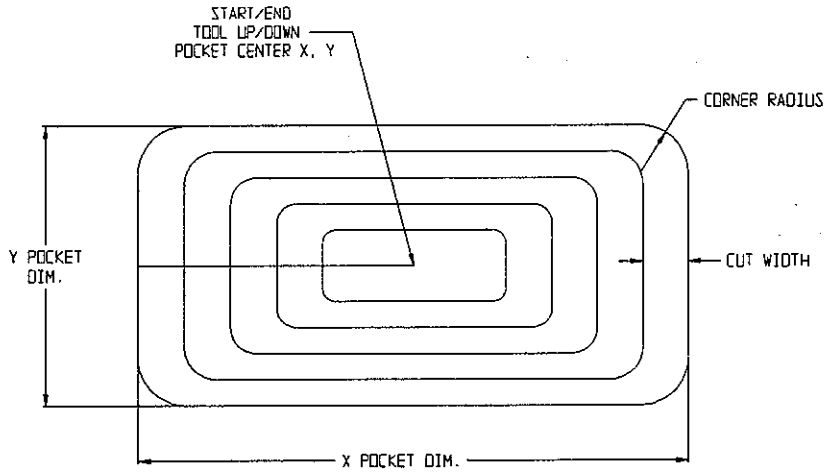
The Rectangular Pocket autoroutine menu will look like this:

MAIN-MDI-POCKT

```
  Event 1 of 1
[Rectangular Finish Inside]
 Z Pierce Feedrate [ 20.0]
*Return Point       [Clearance]
 Clearance Point    [ 0.1000 ]
 Z Down Method      [Ramp]
 Final Depth        [-1.0000 ]
 First Depth        [-1.0000 ]
 Depth Increment    [ 1.0000 ]
 XY Feedrate        [ 50.0]Tool Diam[ 0.1250 ]
 Pocket Center     X[2.0000  ] Y[ 0.0000 ]
 Pocket Dimension  X[ 4.0000 ] Y[ 4.0000 ]
 Corner Radius      [ 2.0000 ]


 Compensation    [On] Cut Dir [CCW (Climb)]
```

This 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 routine except that a rectangle with radiused corners is cut. The rectangular routines have the addition of X and Y pocket dimension. 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.

The pocket generated by this routine will have the following shape:



The finish stock, tool dimension and cutter compensation work the same for this pocket as they do for circular pockets.

Rectangular - Finish

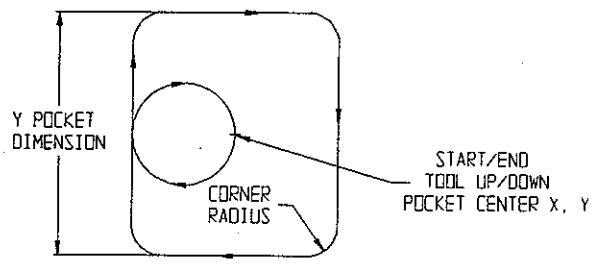
The Rectangular Finish Pocket autoroutine menu will look like this:

MAIN-MDI-POCKT

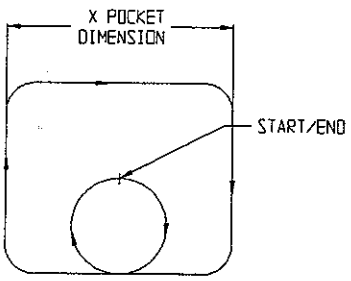
```
  Event 1 of 1
[Rectangular Finish Inside]
 Z Pierce Feedrate [ 20.0]
*Return Point       [Clearance]
 Clearance Point    [ 0.1000 ]
 Z Down Method      [Ramp]
 Final Depth        [-1.0000 ]
 First Depth        [-1.0000 ]
 Depth Increment    [ 1.0000 ]
 XY Feedrate        [ 50.0]Tool Diam[ 0.1250 ]
 Pocket Center     X[2.0000  ] Y[ 0.0000 ]
 Pocket Dimension  X[ 4.0000 ] Y[ 4.0000 ]
 Corner Radius      [ 2.0000 ]


 Compensation    [On] Cut Dir [CCW (Climb)]
```

This autoroutine is used to remove the finish stock left by the rectangular clear routine, or to remove some amount of stock in a single pass around the inside of a rectangle. The autoroutine works in an identical manner to the circular autoroutines. It starts at the center and makes one pass around the rectangle. The circle from the middle of the autoroutine to the outside edge will always be along the longest side of the pocket.

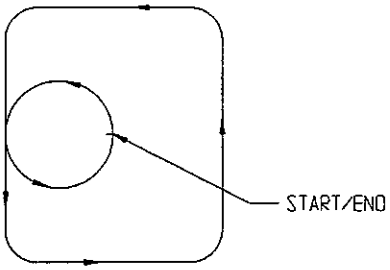
The pockets generated by this autoroutine have four possible tool paths depending on the length of the X and Y sides of the pocket.



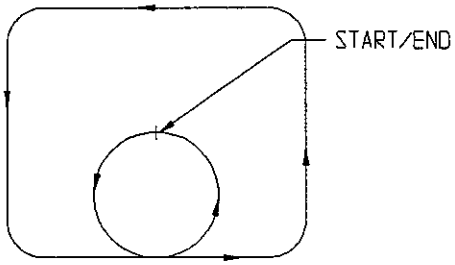
Inside CW Finish
Rectangular X≤Y



Inside CW Finish
Rectangular X>Y



Inside CCW Finish
Rectangular w/Circular Corners
X≤Y



Inside CCW Finish
Rectangular w/Circular Corners
X>Y

The cutter compensation, cut direction, and tool diameter work the same for this pocket as the circular pockets.

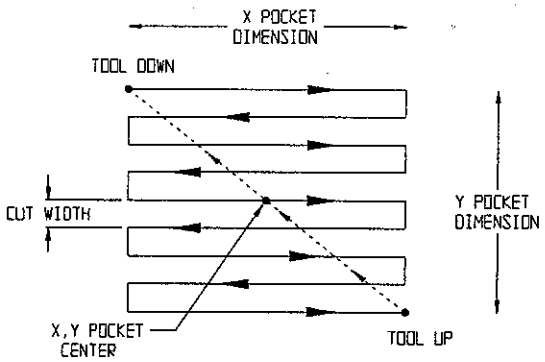
Rectangular - Face

The Rectangular Facing cycle would be used to mill the top of a block or boss. The facing screen looks as follows:

MAIN-MDI-POCKT

```
  Event 1 of 1
[Facing Cycle]
 Z Pierce Feedrate [ 20.0]
• Return Point      [Clearance]
 Clearance Point    [ 0.1000 ]
 Z Down Method      [Ramp]
 Final Depth        [-1.0000 ]
 First Depth        [-1.0000 ]
 Depth Increment    [ 1.0000 ]
 XY Feedrate        [ 50.0]Tool Diam[ 0.1250 ]
 Pocket Center    X[2.0000  ] Y[ 0.0000 ]
 Pocket Dimension X[ 4.0000 ] Y[ 4.0000 ]

 Cut Width          [.25     ]

 Compensation   [On]
```

The facing cycle will produce the following tool path:



Rectangular Facing Cycle

The cutter compensation and tool diameter work the same for this pocket as all other pockets.

F6 FRAME Frame Milling Cycles

When F6 FRAME is selected, one of the following two windows will appear:

MAIN-MDI-FRAME

```
  Event 1 of 1
[Circular Finish Outside]
 Z Pierce Feedrate   [ 20.0]
• Return Point        [Clearance]
 Clearance           [ 0.1000 ]
 Z Down Method       [Ramp]
 Final Depth         [-1.0000 ]
 First Depth         [-.09    ]
 Depth Increment     [.05     ]
 Tool Diameter       [ 0.1250 ]
 XY Feedrate         [ 50.0]
 Pocket Center      X[ 0.0000 ] Y[ 0.0000 ]

 Pocket Radius       [ 2.0000 ]
 Cut Direction [ClockWise (Climb)]
 Compensation  [On]
```

MAIN-MDI-FRAME

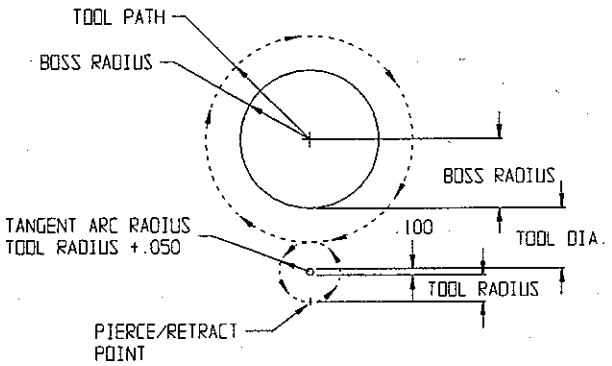
```
  Event 1 of 1
[Rectangular Finish Outside]
 Z Pierce Feedrate   [ 20.0]
•Return Point        [Clearance]
 Clearance           [ 0.1000 ]
 Z Down Method       [Ramp]
 Final Depth         [-1.0000 ]
 First Depth         [-.09    ]
 Depth Increment     [.05     ]
 Tool Diameter       [ 0.1250 ]
 XY Feedrate         [ 50.0]
 Pocket Center      X[ 0.0000 ] Y[ 0.0000 ]
 Pocket Dimensions  X[ 4.0000 ] Y[ 4.0000 ]
 Corner Radius       [ 2.0000 ]
 Cut Direction [ClockWise (Climb)]
 Compensation  [On]
```

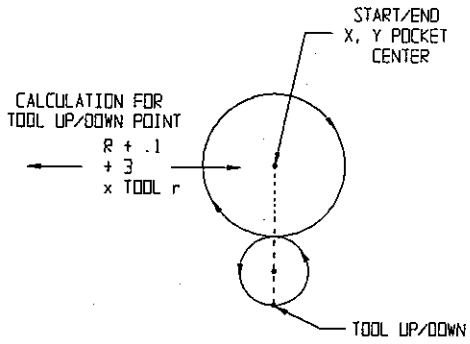
These autoroutines are basically the same except one will cut around the outside of a circle and the other cuts around the outside of a square or rectangle. The following figures describe the tool paths for these autoroutines.

Circular Finish Outside

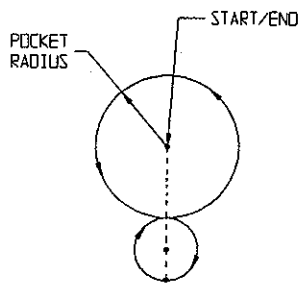
This autoroutine is identical in operation to the circular finish autoroutine except that it cuts the outside of a circular boss rather than the inside. Because it needs to position to the outside of the boss, it will use the following formula to calculate the distance from the center to the feed down point.

$$X_{\text{current position}} = \text{Circle radius} + .1 + [3 \times (\text{Tool Radius})]$$





Outside CW Finish Circle

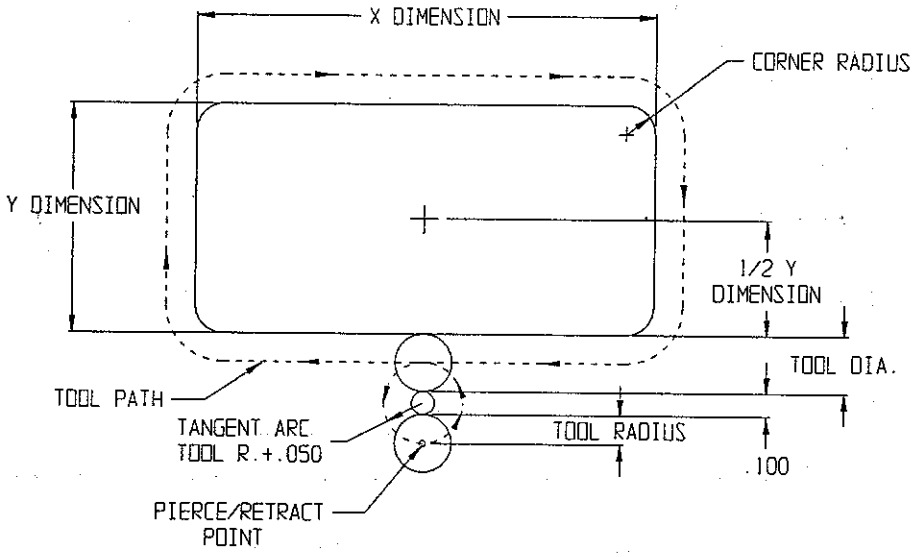


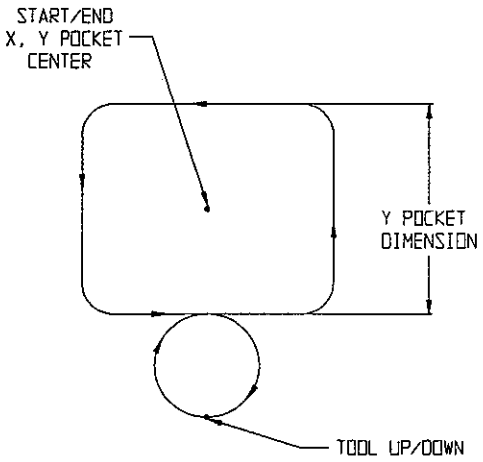
Outside CCW Finish Circle

Rectangular Finish Outside

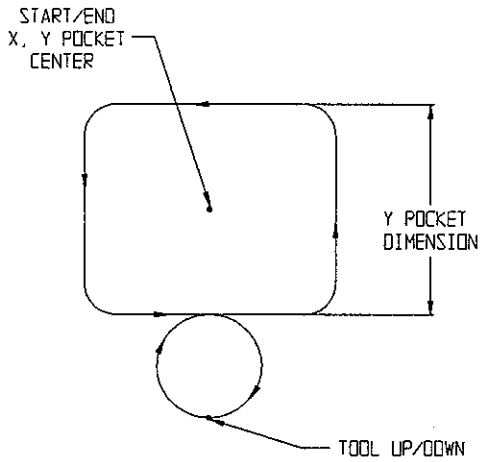
This autoroutine is used to remove finish stock in a single pass around the outside of a rectangular boss. The autoroutine works in an identical manner to the circular autoroutine. It starts in the center, makes a rapid move to the outside of the part, and then feeds the tool down. The formula the CNC 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)





Outside CCW Finish



Outside CW Finish

The cutter compensation, direction and tool diameter for these pockets are the same as any of the other autoroutines.

F7 MISC Miscellaneous Input

When F7 MISC is selected the following screen will appear:

```
  Event 1 of 1
Miscellaneous


>[                                    ]
>[                                    ]
>[                                    ]
>[                                    ]
>[                                    ]
>[                                    ]
>[                                    ]
>[                                    ]
>[                                    ]
```

This screen is primarily used to type in commands that are not covered in the conversational mode. However, if you would like to program in the M and G code mode rather than in conversational mode, you would use this screen. There are 10 blank lines available on this screen and any valid command can be entered. When the F1 Store Key and Cycle Start button are pushed, each line will be executed in order from the top of the screen to the bottom. A list of valid M and G codes follows.

G CODES

G00	Positioning Mode (Rapid)
G01	Linear Positioning Mode (Feed)
G02	CW Circular Interpolation
G03	CCW Circular Interpolation
G04	Dwell
G20	Inch Input
G21	Metric Input
G40	Cutter Compensation Off
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G43	Tool Length Offset Added
G44	Tool Length Offset Subtracted
G49	Cancel H Offset
G90	Absolute Dimension
G91	Incremental Dimension
G92	Set Floating Zero

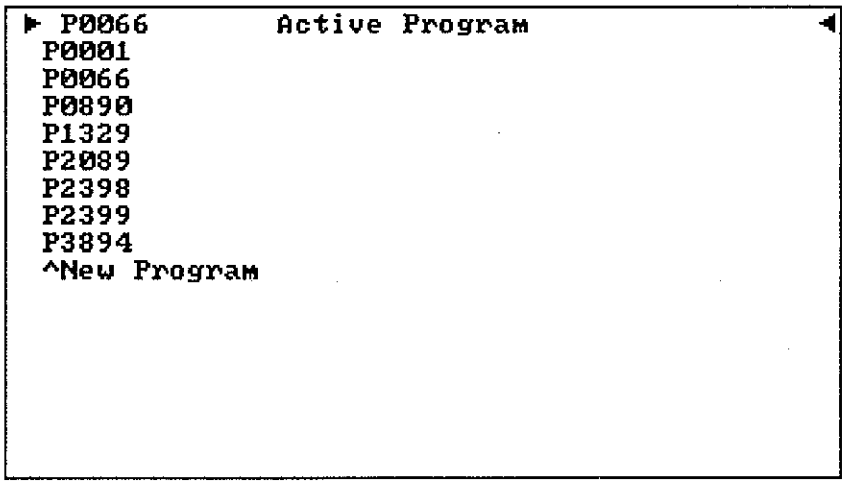
M CODES

M00	Program Stop
M01	Optional Stop
M02	End of Program
M03	End of Program Rewind

Chapter 6

Writing Programs for the Centurion 1

By selecting F8 PROG on the main menu, you enter the programming mode and the following window will appear in Skill Level 1 mode:



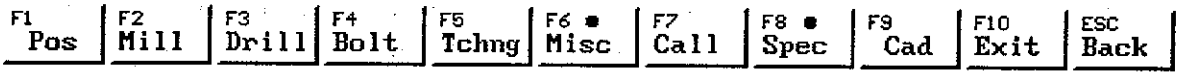
This window contains a listing of all the programs currently stored in the Centurion 1's memory. Using the arrow keys (F7 through F10), move the cursor to the program you wish to edit or look at. Once the cursor is at your desired program number, push the Enter Key or F5 and the selected program will be displayed. If you wish to work with the current program, push the Enter Key at the Active Edit position; if you wish to start a new program, cursor to the New Program selection and push Enter. When this is done the following message will be displayed in the Operator Status window. Just enter the new program number and push the Enter Key. The Centurion 1 will enter this program in the directory and bring up the Program Setup screen.

Operator Status Window

In Skill Level 2 selecting the F8 PROG key will display the following choices:


```
  Event 0 of 0
Program Setup
 Program name [                    ]

 Dimensions   [Absolute]
 Units        [English]

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

The Program Setup screen is always the first event of a program. This screen allows you to name the program and set the type of dimensioning the program is going to use. Once the screen is filled out the way you want, push the F1 STORE key. This will store the screen and bring up the main programming soft keys. These soft keys will look as follows and all the program functions needed to write a program will start from these keys.



The programming functions are the same as those used in MDI and the screen entries have the same meanings. In this section of the manual we will just show the window which appears when each function is accessed and then refer to the MDI section for a detailed explanation of each screen entry.

When operating in the program mode the Centurion 1 no longer needs the Start Point entry which appears on all the MDI movement screens. This is because the previous move establishes the Start Point for the next move and the Centurion 1 will automatically use those dimensions when needed. In the MDI mode there is no previous move which is stored in the control, and because of this a Start Point entry is required on each MDI screen. The following pages will show each of the program screens.

F1 POS

This selection would primarily be used for moving from one side of the part to another in rapid, or for a rapid move to drill a hole or move to the start of the part. End Point dimensions can be entered in one of five ways: Cartesian, Polar [Current], Polar [Angle], Polar [Incremental], or Polar [Absolute]. Refer to page 5-3 of the MDI section for a detailed explanation of these five modes. The five screens for these modes are as follows:

Position Screen Cartesian

```
  Event 1 of 1
Position

 Feedrate       [Rapid]
 Coordinates    [Absolute]


 X Position     [          ]
 Y Position     [          ]
 Z Position     [          ]
```

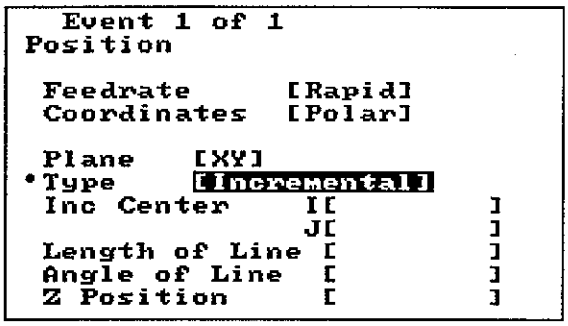
Position Screen Polar Current

```
  Event 1 of 1
Position

 Feedrate       [Rapid]
 Coordinates    [Polar]

 Plane    [XY]
•Type     [Current]


 Length of Line [          ]
 Angle of Line  [          ]
 Z Position     [          ]
```

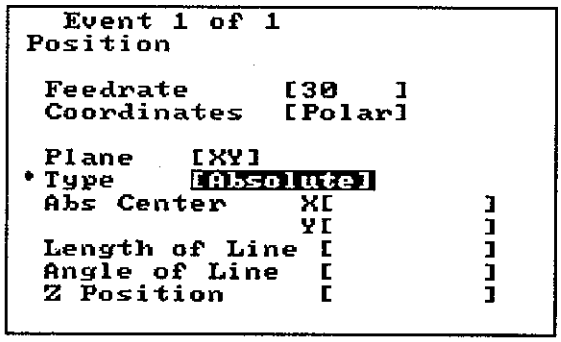
Position Screen Polar Angle

```
  Event 1 of 1
Position

 Feedrate       [30    ]
 Coordinates    [Polar]

 Plane    [XY]
• Type    [Angle]
 Start Angle    [          ]

 Length of Line [          ]
 Angle of Line  [          ]
 Z Position     [          ]
```

Position Screen Polar Incremental



Position Screen Polar Absolute

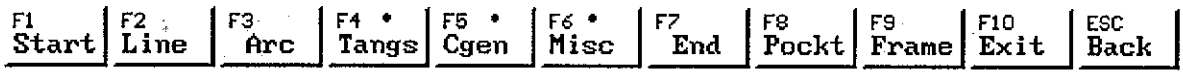


*Round, Chamfer, and Extend Back appear only when screens are accessed from the MILL-LINE screen.

The End [---] Option is available on line screens when in the program mode. This option allows the operator to select either a chamfer or round corner function at the end of each line move. To activate these selections, just push the F3 toggle key until the correct choice is on the screen. Then fill in the requested information and store the Position or Line Screens. For detailed information on these functions see page 4-23. Another function key which will appear on the menu line is F2 X Pos or Y Pos. This key will appear and disappear depending on which field the cursor is indicating as active. The purpose of this key is to enter the current X or Y position of the machine into the field indicated by the cursor. This function is particularly useful if you wish to manually move the machine around a part and enter the coordinates into a program as you go.

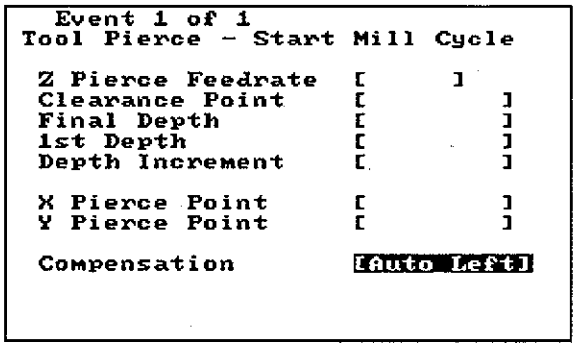
F2 MILL

The F2 MILL selection is used anytime milling is required. The following set of soft key selections will be displayed.

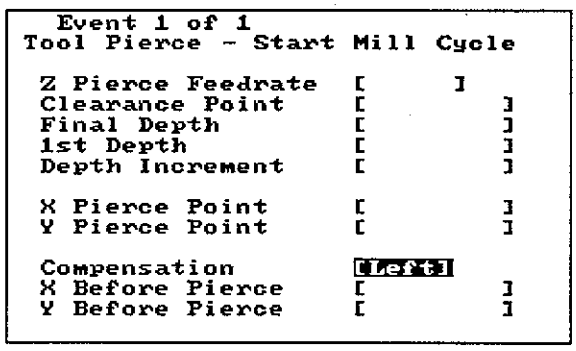


All milling cycles will normally start with F1 START mill cycle, then the Part Shape, and finish with F7 END mill cycle. The Start mill cycle screen has two forms depending on whether cutter compensation is on or off:

Cutter Compensation Off/Auto



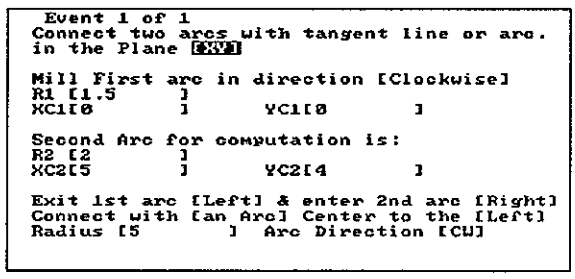
Cutter Compensation On Right/Left



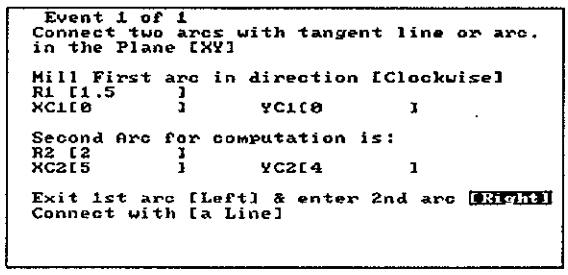
The purpose of these two screens is to initialize the mill cycle, select the type of tool compensation, specify the Start Point or Pierce Point and establish the Z depths and increments necessary to mill the part. If cutter compensation is to be used, prior to this screen a tool number needs to be selected. See page 6-12 for help in selecting a tool number. If cutter compensation is off, the Pierce Point is simply the point at which you would like to start cutting the part. Auto left or right cutter compensation only requires a pierce point. See page 4-3 for explanation. Cutter compensation right/left requires two points to determine the starting tool position. The Point Before Pierce and the Pierce Point are these two points, and their relationship to the part and starting tool position are described in detail on page 4-4.

F2 LINE is used to cut straight lines or angles in a part. A complete description of the line milling function can be found on page 5-2. If arcs or circles are to be cut, F3 ARC would be the selection. The screen selections and programming would be the same as those on page 5-6. The Arc screens in program mode also allow the Round, Chamfer, and Extend Back functions, just like the line screens. However, the arc only allows the round corner function. Reference page 4-23 for details on this function. The F4 TANGS function is a new function available only in the program mode. This function, when selected, will display one of the following two screens.

Tangent Arc



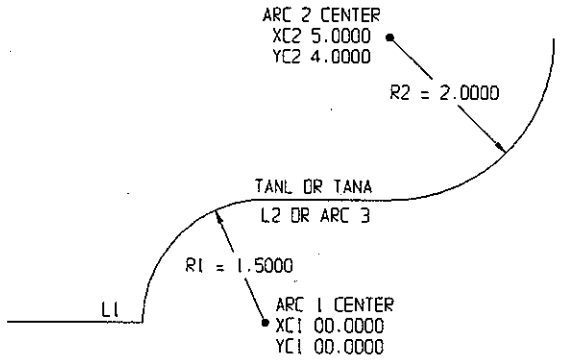
Tangent Line



The F4 TANGS (Tangents) 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 was done to enable a series of tangent lines or arcs to be programmed consecutively. Therefore, a TANL and TANA command would normally be 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 these values.



The general sequence for the above shape would be as follows:

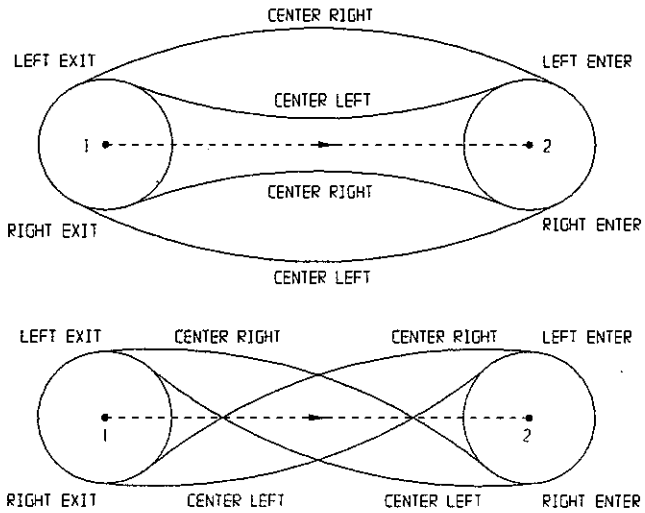
Event 1	Line L1
Event 2	Tangent line or arc function describing arc R1 and line L2 or arc 3
Event 3	Arc R2

In the above example the line would exit the first arc to the left and enter the second arc to the right of a line connecting the centers.

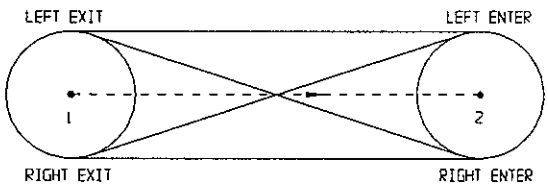
There are eight possible solutions to the Tangent Arc selection, and four solutions to the Tangent Line case. The figures below show the possibilities.

The left/right values are defined as the tangent point being to the right or left of a line connecting the centers of the arcs when facing in the direction of tool movement. See the following diagrams.

TANA CASES



TANL CASES



F7 END The End Function is always the last entry of a Mill Cycle. It is necessary to terminate the milling sequence, turn cutter compensation off, and also stop the mill cycle and retract the cutter. If a Mill cycle is started and not ended, an error will occur. In Skill Level 1 End simply ends the mill cycle and stops the tool, even with end of the last line or arc [Auto Mode]. In Skill Level 2 other ending functions are also available. The End Mill Cycle screens look like this:

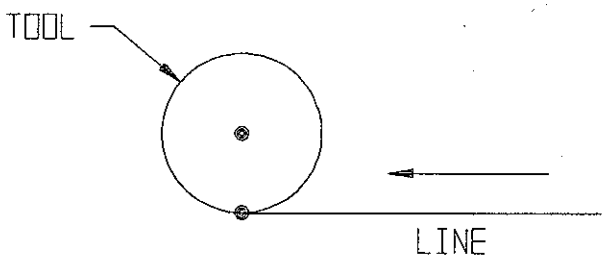
Skill Level 1

```
  Event 1 of 1

Tool Retract
End Mill Cycle
```

Skill Level 2

```
  Event 1 of 1
Tool Retract End Mill Cycle

 Point On Part
 After Tool Retract [Auto]

 Options [---]
```

The Point On Part after Tool Retract determines exactly where the tool will stop before the retract occurs. See the section beginning on page 4-4 for a detailed explanation. The auto selection will stop the center of the tool even with the end of the line or arc.



If the *Options* is selected (F3 TOGL), the following screen will appear:

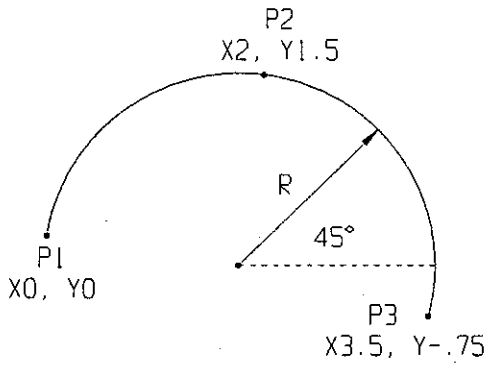
```
  Event 1 of 1
Tool Retract End Mill Cycle


 Point On Part
 After Tool Retract [To a Point]
                    X[          ]
                    Y[          ]
 Options [Clear this pocket]

 Number of passes   [    ]
 Cut Width per Pass [          ]

 Enter negative cut width for finish pass
```

This option allows clearing of an irregularly shaped pocket. If cutter compensation is turned on such that an inside cut is performed, each pass will cause the cutter to move in toward the center by the distance of the cut width. If cutter compensation dictates an outside cut, each pass will cause the cutter to move away from the part the cut width. Because of an infinite number of part geometries, some parts may not be completely cleared by this routine.

- F5 CGEN The F5 key selects the Circle Generate function. This function is used whenever three points on an arc are known and no other information is present. By selecting this function and entering three points, the CNC will calculate the arc which passes through them. The following screen will appear when this function is selected.

CGEN Function

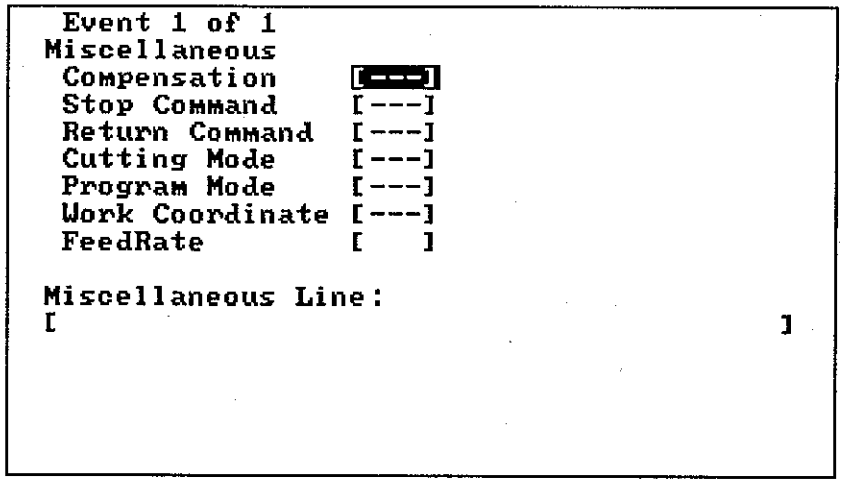
```
 Event 1 of 1
Three Point Circle Generator
    Plane [XY]

Point1  X[0        ]  Y[0        ]
Point2  X[2        ]  Y[1.5      ]
Point3  X[3.5      ]  Y[-.75     ]

    Use X3,Y3 as End Point [No]
    End Angle [45■      ]
```



There are two ways to end this arc; it can either stop at P3, X3.5 Y-.75, or it can stop at some end angle, such as 45°.

• F6 MISC In the program mode the MISC selection brings up the following window and is used to input special instructions into the program.

Miscellaneous Window



To select one of the above functions just move the cursor, using either the arrow keys or the Enter button, to the desired location. Then using the F3 toggle key select the desired option, move on to the next function, and select it. When all functions have been selected, push F1 Store to enter it into the program.

The Miscellaneous Line at the bottom of the screen is for insertion of special M or G codes which are not covered on the conversational screens.

F8 POCKT This programming function is identical to the pockets described in the MDI section on page 5-22. It is used mainly to rough and finish rectangular and circular pockets. Once this function is selected, all that is necessary to cut a pocket is to enter its length, width, and cut increment.

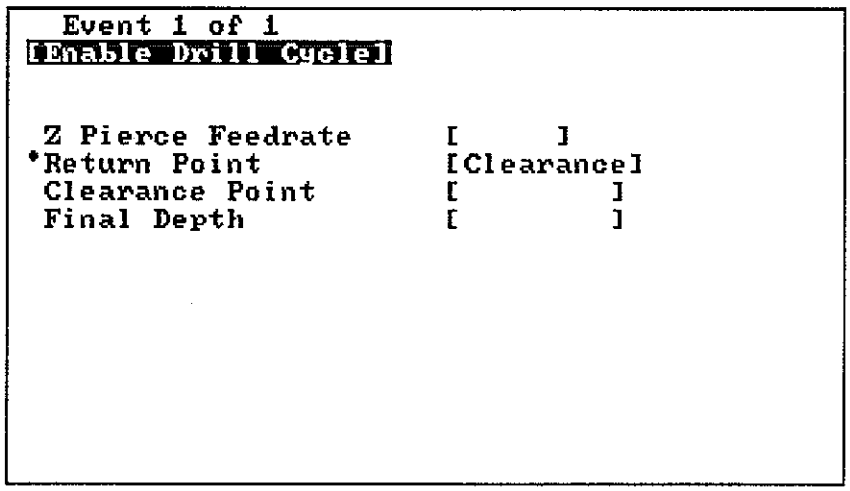
F9 FRAME This programming function is identical to the frame milling described in the MDI section on page 5-28. It would be used to rough and finish rectangular and circular bosses. Again, all that is necessary to make the boss is to enter its length, width and cut increment.

F3 DRILL

The drilling cycle on this machine will automatically insert a Rapid move between holes, stop, and then do the prescribed Z cycle to drill the hole and move the machine to the next hole.

When the Drill cycle is selected these soft keys will appear:

All drill cycles must be setup prior to execution and ended after the last hole. This is done with the F1 START and F5 END selections. When F1 START is selected the following screen will appear:



By pushing the F3 TOGL key when at the first entry, all the drilling routines can be accessed from this screen. The available cycles are Drill, Drill w/Dwell, Drill/Peck, Woodpecker Drill, Bore, Bore w/Dwell and Tap Cycle. As each different cycle is called, the screen inputs will change to reflect the necessary inputs for that cycle. A complete description of each cycle's inputs and operation can be found on pages 5-12 thru 5-20. The drill cycles in the program mode are identical to those in MDI but are designed to do multiple holes rather than a single hole. After the start screen is filled in and stored, the next selection would be F2 POS or F4 CALL. If only one operation is going to be done per hole, F2 POS would be next. This key simply calls up the position screens shown at the beginning of this chapter. A position screen for each hole location would be filled in. When all hole locations have been entered, F5 END would be selected and stored. The purpose of this screen is to terminate the drilling operation. If this function is not done, every subsequent move will result in a hole being drilled. If multiple operations are going to be done at each hole, i.e. spot drill, drill, or counter sink, the hole positions would be put in a subprogram. This subprogram would be called in place of the position moves in the above example. The subprogram could then be called three separate times once for spotting, drilling and counter sinking. See page 6-13 for Call statement.

A lone Z position without an X, Y command will cause Z to go up or down and stay there until the next X and Y command is executed. This is used to skip over clamps. A

Z position with an X and Y command will change the Z depth for this hole and all following holes.

F4 BOLT

The bolthole function can be inserted into a program anytime. The bolt pattern function in the program mode is similar to the MDI mode. The currently active tool length will be used. Therefore a tool change prior to the bolt cycle should be done if the correct tool is not already in the spindle. For a detailed explanation, see page 5-20.

F5 TCHNG

This selection is used to tell the Centurion 1 what tool diameter to use when running the program.

When a new tool needs to be put into 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 length and diameter 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 as follows:

Skill Level 1

```
  Event 1 of 1
Tool Change

 Tool Number       [■ ]
 Tool Description  [                              ]
 Tool Diameter     [          ](optional)

 Spindle RPM       [      ]
 Spindle Restart   [---]
 Coolant           [---]
```

Skill Level 2

```
  Event 1 of 1
Tool Change

* Tool [Change]
  Tool Change Position  XI5        ]
                        YI5        ]

  Tool Number       [3 ]
  Tool Description  [1/2" ENDMILL                ]

  Spindle RPM       [2000 ]
  Spindle Restart   [Clockwise]
  Coolant           [Flood]
```

* *Tool Call doesn't have a tool change position entry.*

This function must precede any sequence of moves using milling or drilling. If it does not, the Centurion 1 will use whatever tool length and diameter that were last used. The tool description line can be used to tell the operator what size tool to use and the RPM to run it at. This line is displayed in the operator message area when the tool call or change function is executed.

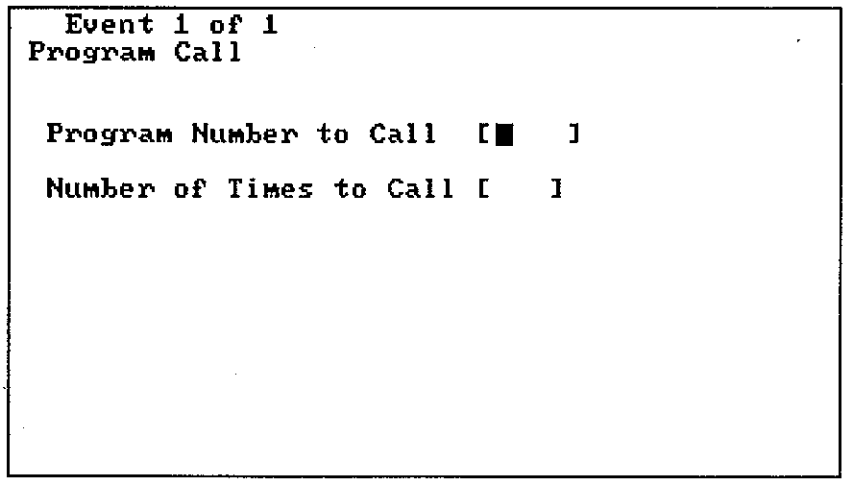
F6 MISC

This is just another menu to access the MISC functions window. Refer to page 5-31 for a detailed description.

F7 CALL

A CALL statement transfers control to any program residing in the CNC's memory. Upon completion of the called program, control is returned to the main program at the event immediately following the CALL statement.

The CALL screen is as follows:



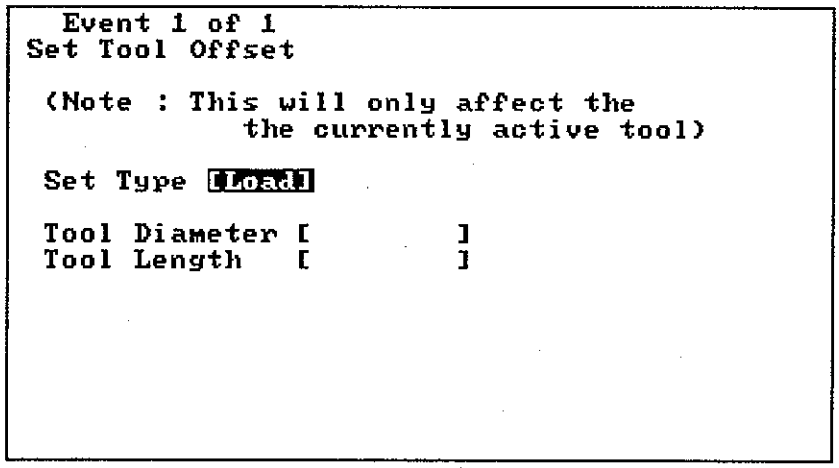
The Program Number specifies the program to be called. The number of loops specifies how many times to call the program. If it is left blank, the program will be called once.

F8 SPEC

This selection allows access to a number of special features of the Centurion 1. Any of these features can be used throughout a program and include things such as part scaling, rotation, and mirror image, as well as moving the zero point and cutting letters. The following are the menus and descriptions of each of the functions.

• F2 TOOLS

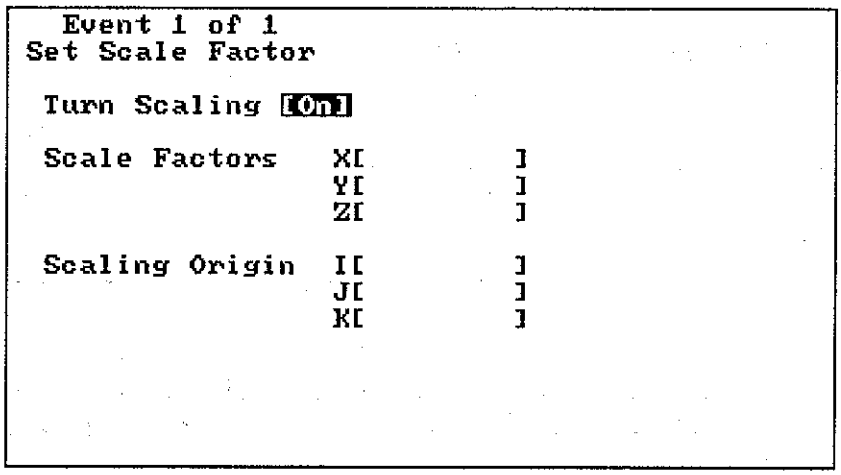
This function will modify or change the current tool offsets while the program is running. It would most commonly be used in conjunction with a program loop or subroutine call to increase or decrease the tool length or diameter. The following screen will appear when this function is called:



The load function replaces the old offset values with the new ones and the adjust will add or subtract the value from the current offset.

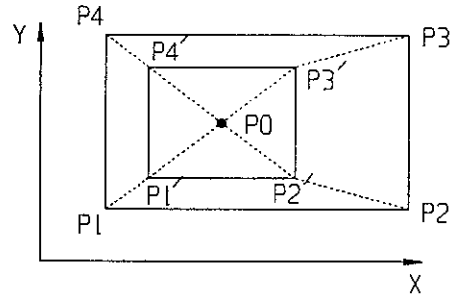
• F4 SCALE

SCALE (SPEC-SCALE)



The scaling center is set to 0 on power up.

X, Y and 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 cancelled by turning scaling [OFF].



This drawing shows the effect of scaling about a specific center point.

P1 - P4 Original program no scaling
P1'-P4' Scaled program
P0 Scaling center

NOTES ON SCALING

1. Once set, scaling remains in effect until cancelled.

2. If arcs are being scaled, the scale factors for the arc's axes must be equal.

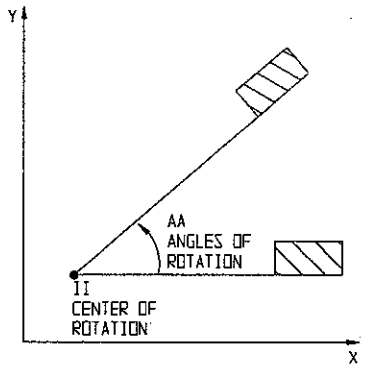
3. Scaling results are rounded down (.00009 = .0000). During scaling some moves may go to zero which could affect cutter compensation.

• F5 ROTATE

ROT (SPEC-ROT)

```
  Event 1 of 1
Set Rotation Angle

 Turn Rotation  [Adjust]

 Rotation Plane [XY]
 Rotation Angle  AA[          ]

 Rotation Origin  X[          ]
                  Y[          ]
```

Like scaling, part rotation is about some center point called the rotation origin. The coordinates of this origin will be assumed to be the floating zero coordinates unless otherwise specified. The rotation angle is the amount the part will be rotated and rotation will remain in effect until turned off.



Effect of Part Rotation

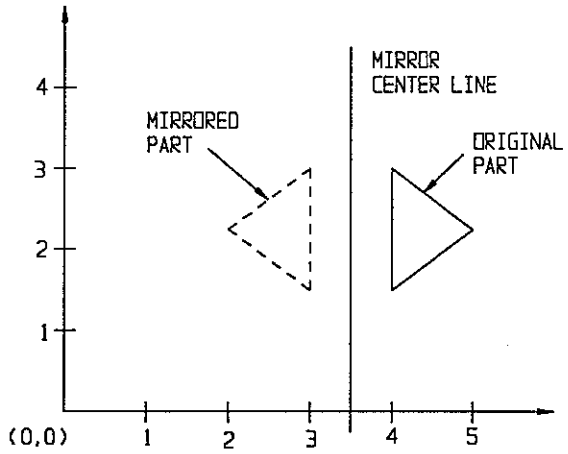
A part can be rotated in any of the major planes by simply toggling through the rotation plane selection entry.

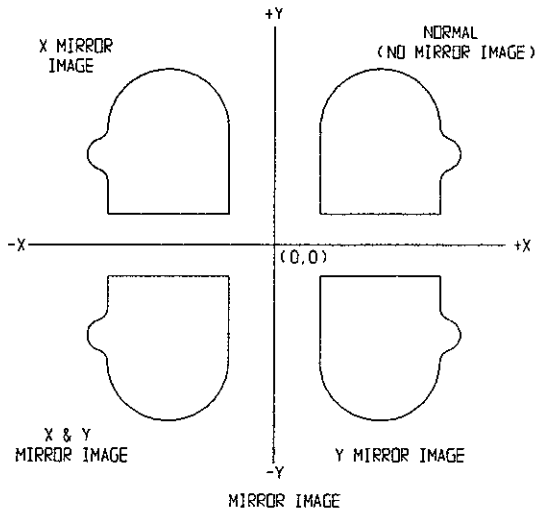
- F6 MIRROR

MIRR (SPEC-MIRR)

```
  Event 1 of 1
Set Mirror Image

 Turn Mirror Image [On]

 Mirror Position  X[3.5■    ]
                  Y[         ]
                  Z[         ]
```

The mirror image commands allow mirroring about any centerline. The mirror image centerline is not affected by either scaling or rotation. The dimension after the X, Y and Z axes specifies the distance of the mirror centerline from the current zero point. If the dimension is left blank, that axis will not be mirrored.





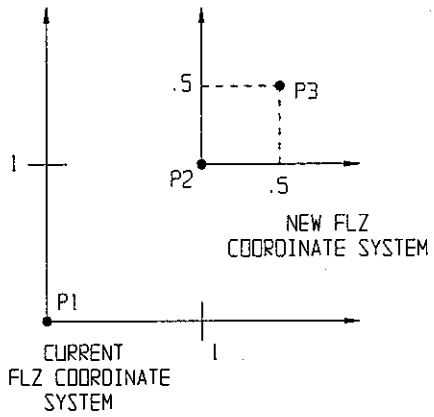
• F7 FLZ

FLZ (SPEC-FLZ)

```
  Event 1 of 1
Set Floating Zero

    Axis        X[0■       ]
                Y[0        ]
                Z[         ]
```


• *Skill Level 2*

This command establishes a zero reference so that the position of the tool becomes the programmed position.



If the machine is positioned at P2, which is a command of X1 Y1, and then a FLZ 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 FLZ X-1 Y-1 is commanded, the next time X.5 Y.5 is commanded the machine will position to P3.

- F8 TEXT

```
  Event 2 of 2
Text

 Z Feedrate  [■    ]
 Clearance   [       ]
 Text Depth  [       ]

 XY Feedrate [     ]
 X Text Size [       ] (1=1" Letters)
 Y Text Size [       ]

 Text String [                          ]
```

The Centurion 1 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 to yes, MAIN-UTIL-PARMS-CTRL. **The control must be rebooted after setting the parameter.** The one-to-one letter size is 1" by 1" and is in a block letter font. All these letters can be scaled or rotated to achieve the desired size and orientation. The Text Depth and Clearance determine how deep the letter is to be cut and how high the tool will pick up in between letters. A typical program would be developed as follows:

```
  Event 0 of 1
Program Setup
 Program name [SPEC-TEXT EXAMPLE■  ]

 Dimensions    [Absolute]
 Units         [English]

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

```
  Event 1 of 1
Position

 Feedrate      [Rapid]
 Coordinates   [Absolute]


 X Position     [0         ]
 Y Position     [0         ]
 Z Position     [          ]
```

Move position to start of first letter.

```
  Event 2 of 2
Text

 Z Feedrate  [5     ]
 Clearance   [.1       ]
 Text Depth  [-.06     ]

 XY Feedrate [18    ]
 X Text Size [1        ] (1=1" Letters)
 Y Text Size [1        ]

 Text String [ONE TO ONE SCALE          ]
```

Text written starting at X0 Y0.

```
  Event 3 of 3
Position

 Feedrate      [Rapid]
 Coordinates   [Absolute]


 X Position     [0         ]
 Y Position     [-1        ]
 Z Position     [          ]
```

Position move to next line.

```
  Event 4 of 4
Text

 Z Feedrate  [5     ]
 Clearance   [.1      ]
 Text Depth  [-.06    ]

 XY Feedrate [18    ]
 X Text Size [.5      ] (1=1" Letters)
 Y Text Size [.75     ]

 Text String [.5 TALL BY .75 WIDE      ]
```

Text written starting at X0 Y1.

```
  Event 5 of 5
Position

 Feedrate      [Rapid]
 Coordinates   [Absolute]


 X Position     [0        ]
 Y Position     [-2       ]
 Z Position     [         ]
```

```
  Event 6 of 6
Text

 Z Feedrate  [5     ]
 Clearance   [.1      ]
 Text Depth  [-.06    ]

 XY Feedrate [18    ]
 X Text Size [.25     ] (1=1" Letters)
 Y Text Size [.25     ]

 Text String [.25 WIDE BY .25 TALL     ]
```

Text written starting at X0 Y2.

```
  Event 7 of 7
End of Program

 Z to Toolchange  [No]
 Turn off the Spindle [No]


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

When exiting the Program mode the above screen will appear.

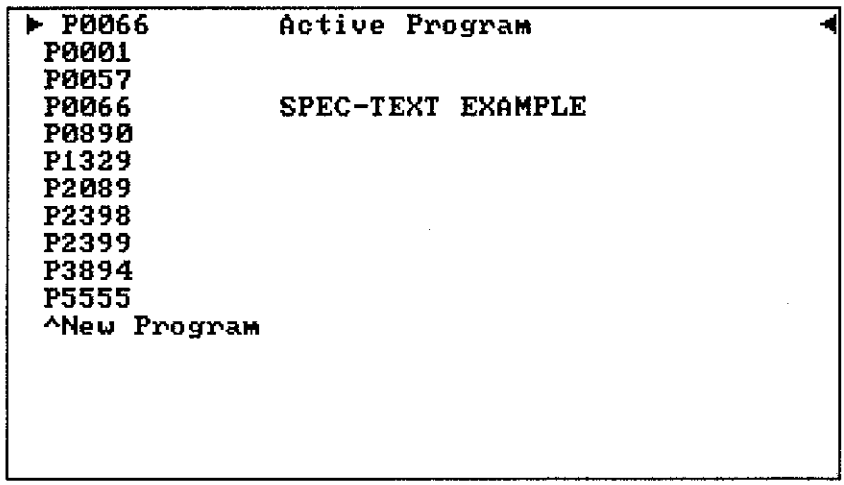
This is an optional screen and does not have to be filled in. The screen is designed to move the tool and table to an unload position and turn the spindle and coolants off. If this is not desired, leave the positions blank and the program will stop wherever the last move ended.

Chapter 7

Running the Centurion 1

The Run mode in the Centurion 1 allows the operator to run or rerun programs written at an earlier date. When the F4 Run key is selected the following window will appear:

Skill Level 1

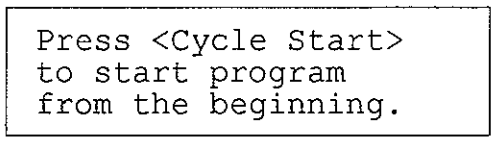


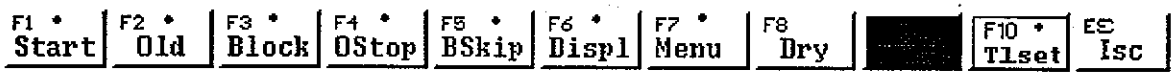
Skill Level 2

This is a list of all the programs currently residing in memory. To run a program, just use the arrow keys to move the cursor to the desired program and push F5 Enter or the Enter key. The F6 TOGL key will switch the menu display from just displaying the program numbers to displaying the comments typed in on the Program Setup screen.

When a program is selected the following menu will be displayed:

Operator's Message Window





At this point there are several options that can be activated before the program is run.

- F1 START When this function is activated in Level 2 it will start running the active program from the beginning.

- F2 OLD When this function is activated in Level 2 it will allow the operator to type in the desired program to be run.

- F3 BLOCK When this function is activated the machine will run one move at a time and stop. Pressing the Cycle Start button will execute the next line. This mode is used mainly to debug a program the first time it is run.

- F4 OSTOP When this function is activated the machine will stop at each OSTOP command in the program. OSTOP is entered through the MISC screen. Each time the Cycle Start button is pushed the program will run to the next OSTOP command.

- F5 BSKIP The Block Skip function will skip all blocks in the program that start with "/". The slash would have to be edited into the program off-line.

- F6 DISPL This function allows the graphics display to be changed prior to running or during the run.

- F7 MENU This function displays the current programs listed in memory and allows selecting of a new program to be run.

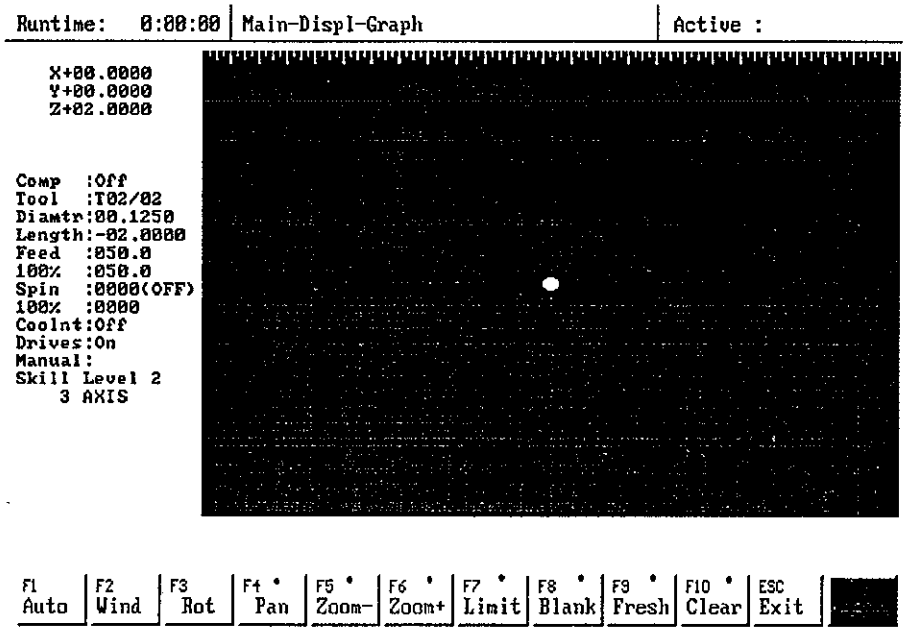
F8 DRY The Dry Run function increases all the feedrates in the program to 75 ipm. This feature is used when verifying a program to quickly run through the part geometry without cutting.

- F10 TLSET This soft key selects the tool presetting program. This program will request a tool # and then call that tool into the spindle (via tool changer or manual load). It then goes to the handwheel or jog mode. The operator touches the tool off on the part and pushes ENTER to enter the offset. The tool is then put away and the next tool is requested.

Chapter 8

The Centurion 1 Graphic Display

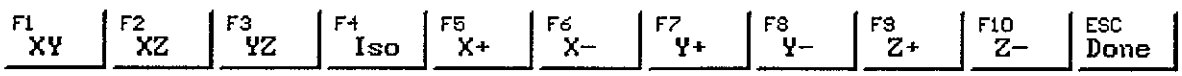
The F6 Display function allows the operator to change the graphic display in order to get a better view of the part.



The graphics on this control are full 3-D and will be displayed in the Graphics Area. 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 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.

F3 ROT Display Rotation

When F3 ROT is selected the following soft keys will be displayed:



The F1 XY, F2 XZ, F3 YZ and F4 ISO keys give the four standard rotations of a part: XY plane, XZ plane, YZ plane and isometric views. The orientation index in the lower right corner of the screen shows the current part orientation and rotates to show what the new orientation will be. Depressing the F1 thru F4 keys moves the orientation index

• *Skill Level 2*

to its new position, and then pushing the ESC key will cause the part display to rotate to its new position and the previous menus to reappear. The F5 thru F10 keys are used to infinitely rotate any of the selected axes. Again, as the key is pushed the orientation index rotates, indicating the orientation of the part display. When ESC is pushed the part will rotate to its new orientation. F5 and F6 rotate X axis ±, F7 and F8 rotate Y axis ±, and F9 and F10 rotate Z axis ±.

• F4 PAN

The F4 PAN key selects the Pan function which allows the operator to pan around a part.

The square cursor which appears on the screen can be moved around using the arrow keys F7 thru F10. To pan simply move the cursor to the point on the display which is desired to be at the center of the screen and push F5 ENTER or the ENTER key on the keyboard. The display will shift to its new position and panning can be started again. The ESC key will terminate the Pan function and drop back to the Graph screen.

F2 WIND

The F2 WIND key selects the window function which allows the operator to window in on a particular area of the part. The square cursor which appears on the screen can be moved around using the arrow keys F7 thru F10 (the same as Pan). To window, move the cursor to the first corner of the window area and hit F5 ENTER or the ENTER key. 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 and hit ENTER. The area enclosed in the box will now be displayed on the entire screen.

F1 AUTO

The F1 key selects the Auto Zoom function. This function automatically scales and centers any part on the screen. Normally an Auto has to be done after a part is rotated to get it back to the center of the screen.

• F5 ZOOM

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

- F6 ZOOM+

The F6 ZOOM+ key selects the Zoom+ function. This function increases the size of the part being displayed on the screen. Generally this function is used to enlarge a specific area of a part enabling one to see greater detail.

- F7 LIMIT

The F7 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 reduce the size of the machined areas of the part. The axis overtravel limits are set from the parameter screens. If the tool is programmed outside this box, an overtravel error will result.

- F8 BLANK

The F8 BLANK function key turns the CRT off until another key is pushed. If the machine is going to sit for a long period of time, the CRT should probably be blanked. There is also an automatic blanking function which will turn the screen off if no keys are pushed for five minutes. The time is selectable by parameter.

- F9 FRESH

The F9 FRESH function redraws the screen. If the machine has been running for a long time and the screen is cluttered, the screen can be redrawn.

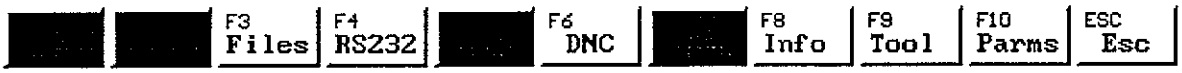
- F10 CLEAN

The F10 CLEAN function key clears the screen of all graphics. New graphics will be drawn as the machine moves.

Chapter 9

Using the Utilities Screens

The F10 UTIL button accesses the Utilities screen menus for the many non-programming functions of the control. When this function is selected the following menu will appear:

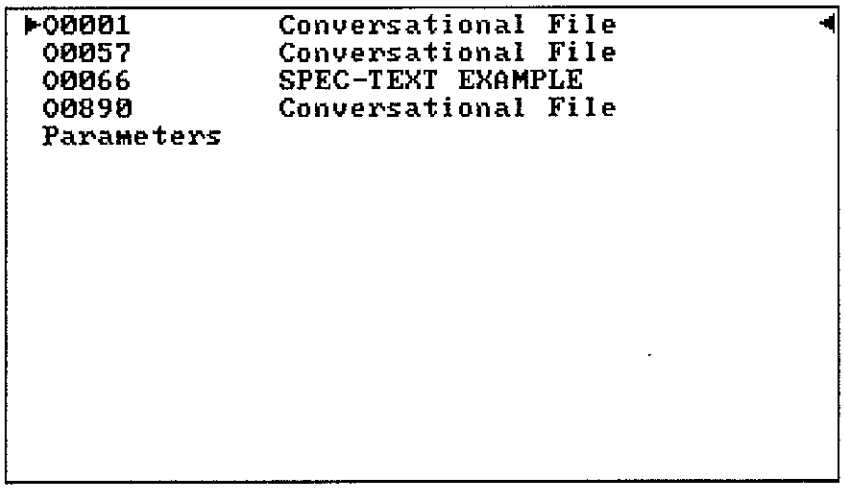


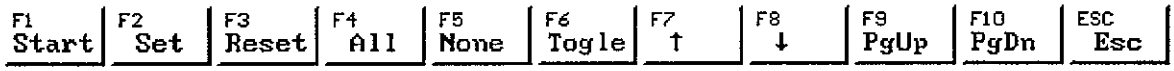
F3 FILES (MAIN-UTIL)

The F3 FILES selection from the main Utilities screen allows saving, loading, copying and renaming of files (programs). The following soft key selections become active:

F1 LOAD (MAIN-UTIL-FILES)

This function is used to load programs from the floppy disk into the control's program memory. When this function is selected the following window is displayed:





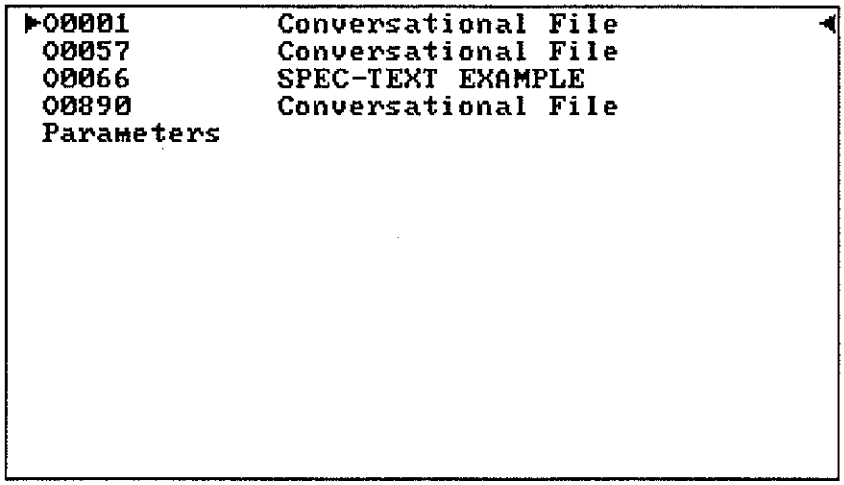
The edit window displays a list of the programs on the floppy drive, their length (in bytes), their creation date, creation time, and program name.

The sequence to load a program is first to select which program or programs are to be loaded. To do this, move the cursor using the arrow and page keys, F7 thru F10, to the desired program. Then push F2 SET to mark that program as one to be loaded. You will notice that this program number will be highlighted. This sequence can be repeated until all the programs which need to be loaded are selected. By pushing F1 START all the highlighted programs will be loaded into the control's memory. If the program being loaded already exists in the control, an operator message will ask if this program number should be skipped or replaced by the new program. The function keys for the Load sequence are identical in operation to those of the Save function and are as follows:

F1 START	Pressing this key will begin the transfer of the selected programs from program memory to floppy disk.
F2 SET	Selects the file at the cursor position to be loaded from or stored to the floppy disk.
F3 RESET	Unselects the file at the cursor position.
F4 ALL	Selects all programs on floppy disk or memory to be loaded or stored.
F5 NONE	Unselects all selected programs.
F6 TOGL	Changes menu display data.
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.

F2 SAVE (MAIN-UTIL-FILES)

This function is used to save programs from the control's program memory to the floppy disk. When this function is selected the following window and soft keys are displayed:



F1 Start	F2 Set	F3 Reset	F4 All	F5 None	F6 Togle	F7 ↑	F8 ↓	F9 PgUp	F10 PgDn	ESC Esc

The edit window displays a list of the programs stored in memory as well as the length (in bytes) of the programs, their creation date, creation time, and the program name. The F6 TOGL key selects whether all this data is displayed or just the program numbers. The following are descriptions of the soft keys used to save a program to the floppy disk. The sequence to save a program is first to select which program or programs are to be saved. To do this, move the cursor using the arrow and page keys, F7 thru F10, to the desired program. Then push F2 SET to mark that program as one to be saved. You will notice that this program number will be highlighted. This sequence can be repeated until all the programs which need to be saved are selected. By pushing F1 START, all the highlighted programs will be saved to the floppy disk. If the program being saved already exists on the floppy disk, an operator message will ask if this program number should be skipped or replaced by the new program. The function keys for the Save function are the same as for the Load function just discussed.

F3 RENAM (MAIN-UTIL-FILES)

This function allows renaming of an existing program. It operates identically to the load function. After the program is selected, F1 START is pushed and the control will ask for the new name of the selected programs.

F4 COPY (MAIN-UTIL-FILES)

This function will copy an existing program to a new program name, in other words, make a duplicate with a new name. The selection process is identical to the LOAD function.

F5 ▸RAM (MAIN-UTIL-FILES)

This function transferes a program from the floppy disk to the DNC buffer.

F9 ERASE (MAIN-UTIL-FILES)

This function is used to erase programs from the control's memory. The sequence to select which programs to erase are identical to that for the Load and Save functions just discussed. Once the programs are selected, push F1 START to erase.

F4 RS232 (MAIN-UTIL)

This key, located on the main Utilities screen, is used whenever it is desired to transfer a program from the RS232 port to memory. The following soft keys will appear:

F1 COM (MAIN-UTIL-RS232)

This key indicates which COM port the control is going to send or receive data on.

F5 SEND (MAIN-UTIL-RS232)

The Send option is used to send programs from the control's program memory to an off-line computer. The following soft keys appear:

F7 MENU (MAIN-UTIL-RS232-SEND)

The F7 MENU selection lists all the program in memory and uses the same selection keys as the LOAD function.

F1 Start	F2 Set	F3 Reset	F4 All	F5 None	F6 Toggle	F7 ↑	F8 ↓	F9 PgUp	F10 PgDn	ESC Esc

The way to select the program to be sent is the same as the selection process for Load and Save to the disk that were just discussed. If the program called *Parms* is selected, the control's system parameters will be sent out on the RS-232.

F1 START (MAIN-UTIL-RS232-SEND)

After the programs to be transfered have been selected with the F2 SET key, begin the process by pressing the F1 START key. The functions keys will change again.

F1 BEGIN (MAIN-UTIL-RS232-SEND)

Once the F1 START key is pushed the F1 BEGIN key will appear. The operator then needs to prepare the receiving computer and push F1 BEGIN to start the data transfer.

F6 REC (MAIN-UTIL-RS232)

The Receive function is used to receive programs from the RS-232 port into the control's memory. This window will appear:

The operators message window will prompt for a program number. Once an entry is made into either of these lines, a window will appear to display the incoming program as it is received. Receiving will continue until the ESC key is pushed. A program name has to be an O followed by up to a four digit number.

F7 ▸RAM (MAIN-UTIL-RS232)

This function transfers the data coming in on the RS232 to the DNC buffer. Because the buffer contains only one program at a time a program name is not required.

F6 DNC (MAIN-UTIL)

The DNC mode is used for running large programs. These programs are not loaded into the control's memory, so they cannot have goto's, while-wend loops, gosub's or calls.

When F6 DNC from the main Utilities screen is pressed, the function keys show the DNC modes. They are:

F3 FAST (MAIN-UTIL-DNC) and F4 RUN (MAIN-UTIL-RUN) are two DNC modes which can be selected to run programs. They are identical in operation and setup.

The FAST mode has a limited instruction set and is intended to run surface programs from a CADCAM system. DNC FAST should be used when it is necessary to run a large program or a program containing very short moves and requires no additional calculations (Trig help, cutter compensation). Also, the graphic display is turned off.

The F4 RUN mode allows all the features of the CNC to be used and is designed to run long CADCAM programs that do not have the majority of their moves less than .05" in length. Once one of the modes is selected the following soft keys are active:

F1 thru F6 choose the input mode of the file or programs to be DNCed.

F1 RS232 Brings the programs in thru the RS232 port.

F2 FILE This will ask for a program selection from the control's memory.

F3 DISK This will ask for a program selection from the floppy disk's directory.

F4 ANY The ANY selection is exactly the same as the FILE selection except it will execute files with full DOS names. The file command only executes programs with 0XXXX names.

F5 OLD Asks for a program number to be entered. The program must be in memory.

F6 RAM▸ This selection will execute the programs currently stored in the DNC buffer. The buffer can be loaded from either the RS232 or floppy disk.

F9 SKIP This function key allows entry of a skip count. The skip count is the number of program blocks to be skipped before the program begins execution. This allows execution to start anywhere in the program.

F10 MIRR This function will mirror the incoming DNC program in either the X or Y axis. The axis selection is accomplished via the F1 ±X or F2 ±Y soft keys.

ESC Abort Stops execution of the program.

F7 TERM (MAIN-UTIL-DNC)

The F7 TERM key activates the Remote Terminal Mode. It effectively is an MDI mode running from a remote RS-232 terminal. Once this mode has been entered, the control executes individual commands coming in from the communications port. All trig help and cutter comp look-ahead has been removed from this mode causing the commands to execute immediately.

Also, a new command, HDW, has been added to cause the terminal mode to pause and enter the handwheel mode. This was added to allow the remote terminal to request manual operation from the local machine operator, such as moving around a clamp or obstruction. This command should come only from a remote terminal into the RS232-TERM mode.

F9 VERF (MAIN-UTIL-DNC)

F9 VERF works identical to the F3 FAST and 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 except the option for part/path/both is not available.

Note: For F3 FAST, F4 RUN and F9 VERF in the RS232 DNC mode, a search for a tool number or block number can be done to start a program from a desired location, but cannot be reset to First after this search.

The following messages may be seen on the first line of the edit window when the DNC mode is entered.

-Waiting for DNC link . . .
No program has been selected from Disk.

-DNC link established . . .
The program listed in the active program box has been selected to DNC and a CYCLE START will begin execution.

-Executing DNC program . . .

The program is being executed.

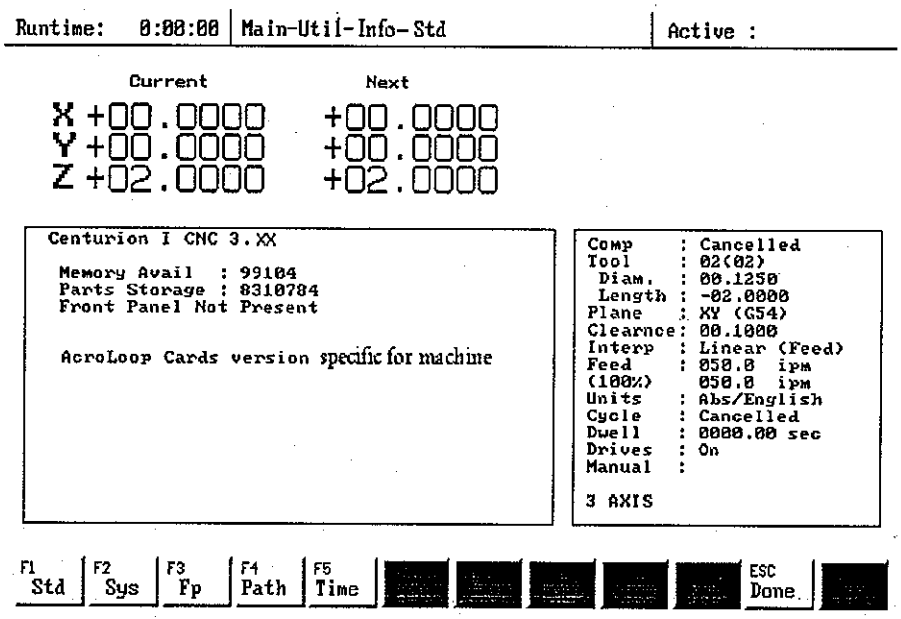
-Skipping DNC program . . .

The number of blocks entered in skip count is being skipped.

F8 INFO (MAIN-UTIL)

This screen gives general system information about the hardware and software versions in the control. The following two screens of information are available. The F1 and F2 keys select each of the two screens.

F1 STD



Memory available	gives the system memory available in bytes.
Parts storage	gives the amount of parts storage in bytes.
Front panel present	gives the front panel version code.
Controller card	gives the controller card version (v0206 is 2.06) and an error count (should be zero)

Acroloop information for X, Y and Z axis is as follows:

first the version, (v0214=2.14),
then an error count (should be zero),
then four bytes giving optional Acroloop programs loaded (the .HEX files).

These last four bytes are formatted as:

aamm ssrr, where:

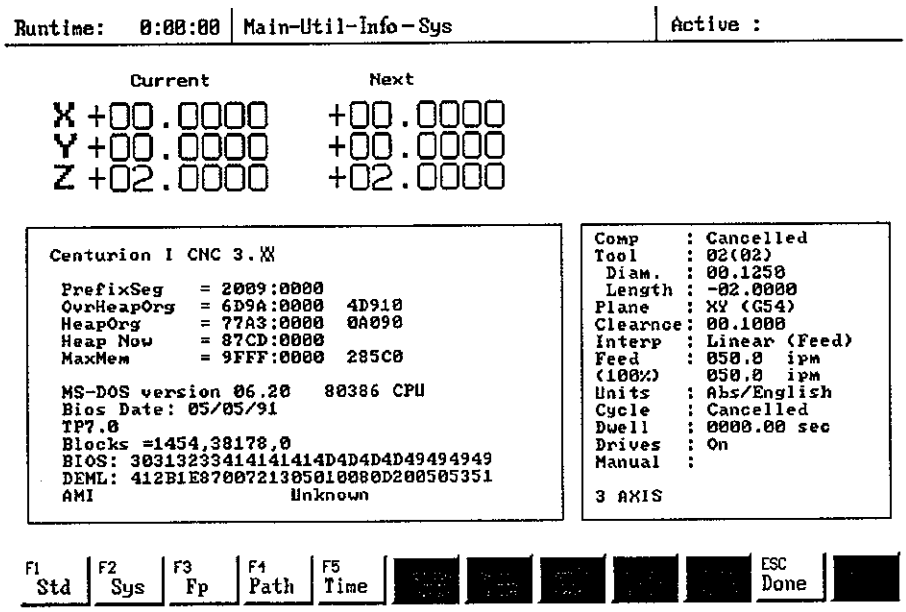
aa=axis (01=X, 02=Y, etc)

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)

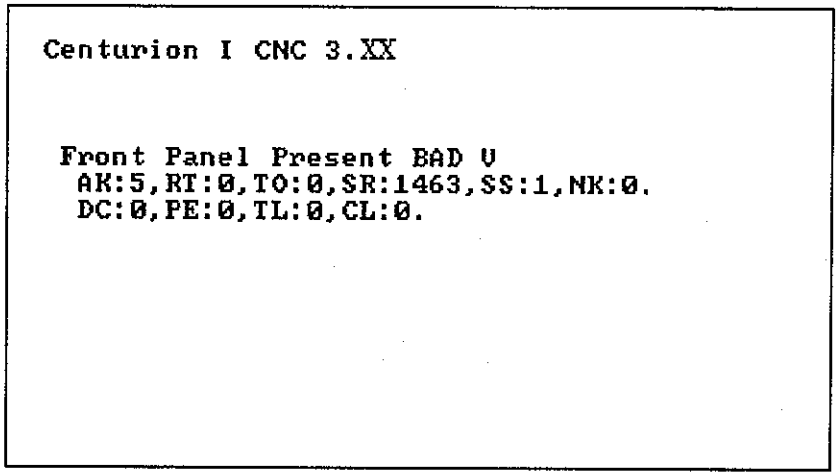
F2 SYS



This screen gives internal information about the system. Lines 1 through 5 show memory allocations to DOS, CNC Overlays, and the heap. Line 6 shows the MS-DOS version and whether the CPU is an 80286 or 80386. Lines 11 & 12 show the compiler and blocks pre-allocated for canned cycles, text cycles and custom M & 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)

This screen shows some front panel information.



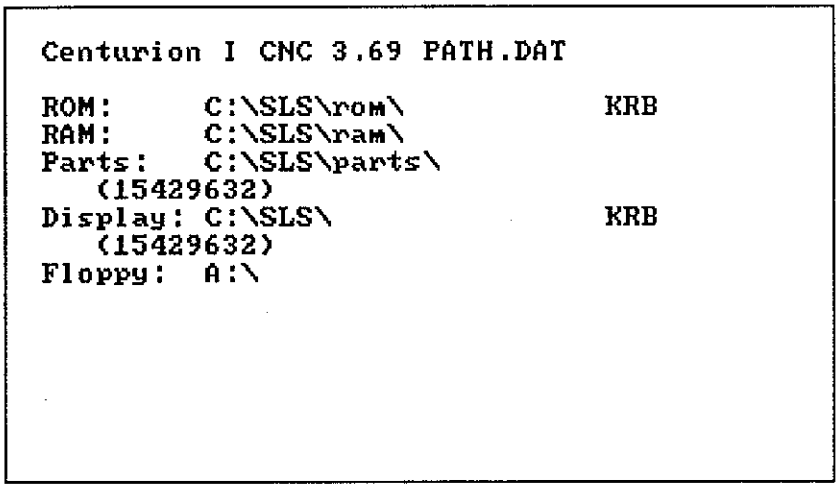
The first line gives the front panel version.

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)

This screen displays the paths and the volume label of each.

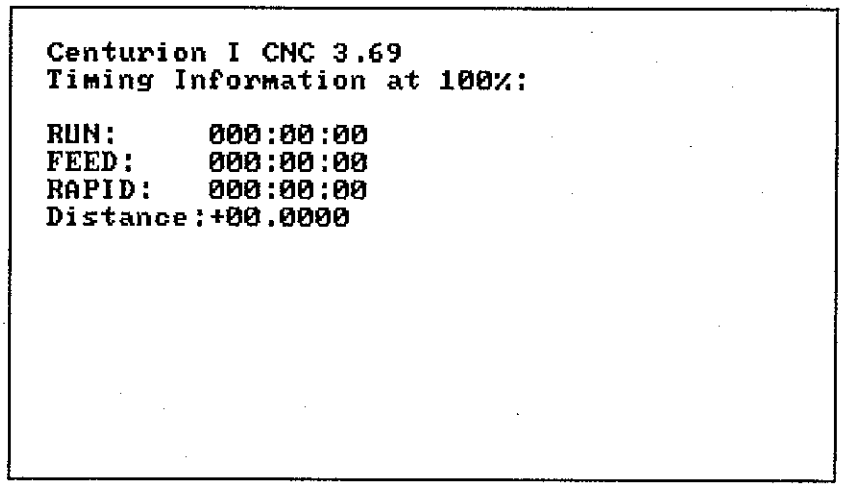


This screen shows the path file. In the standard order, these are the directories for ROM, RAM, Parts, Display and Floppy. Below the Parts directory is shown the available parts space in bytes. For the ROM, Parts and Display directories, the DOS volume ID for that drive is shown to the right.

NOTE: The path file is reloaded when this information is displayed. Therefore one can edit the path file in level 3, then reload it by using this command.

F5 TIME (MAIN-UTIL-INFO)

This screen shows timing information for the last program verified. It assumes 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.



F9 TOOL (MAIN-UTIL)

This function displays the tool table which contains the length and diameters of all the current tools. Pushing F1 EDIT allows entry of new values into the table. Normally the values in this table would be entered via the TOOL key in HWD and JOG.

F10 PARMS (MAIN-UTIL)

The F10 key on the UTIL screen brings up this parameter menu:

The F1 SETUP key asks for a validation code to access the machine setup parameters. The validation code and access levels are supplied by the machine tool builder and should be part of the system parameter setup sheet. Assuming the proper codes have been entered, after F1 SETUP is selected the following keys will appear:

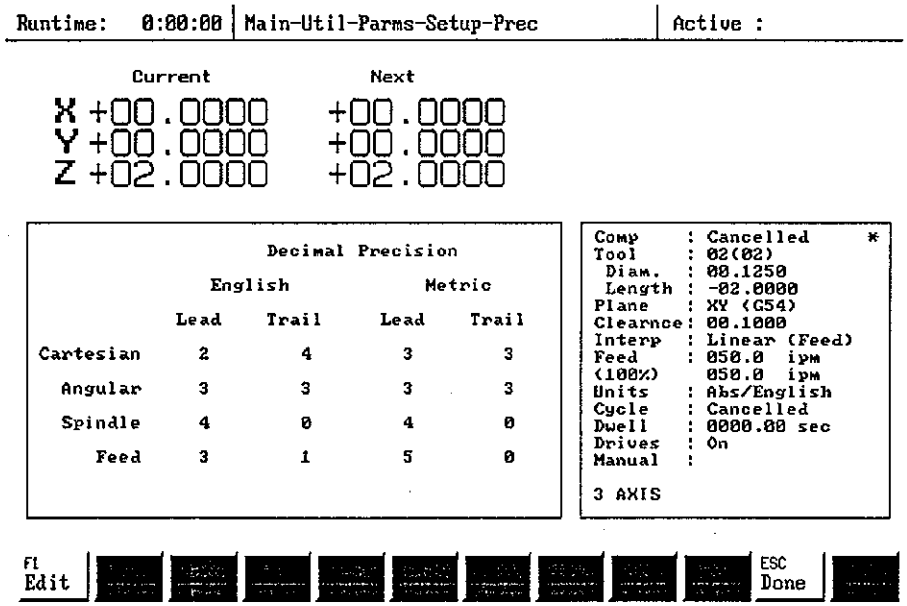
F1 Level	F2 Prec	F3 Power	F4 Axis	F5 Misc	F6 Ovrs	F7 BScrw		F8 DOS		ESC Done

F1 LEVEL (MAIN-UTIL-PARMS-SETUP)

This key allows re-entry of the validation code and access level.

F2 PREC (MAIN-UTIL-PARMS-SETUP)

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

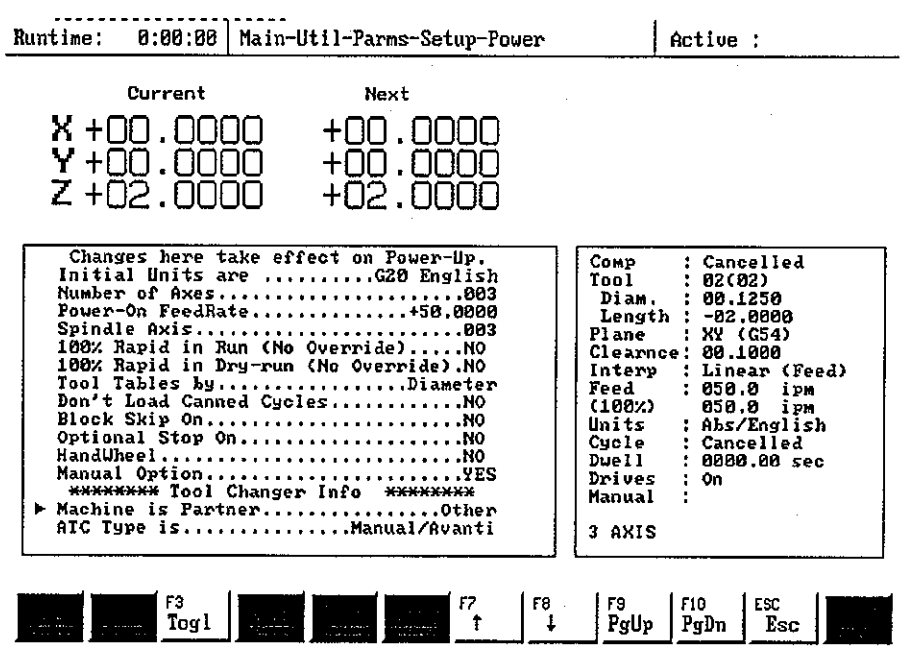


The above screen shows some typical settings for leading and trailing zeroes for the different coordinate systems. The number of leading and trailing zeroes is unlimited, but some practical limits do exist. If the numbers get too large they will not fit on the screen in their allotted space, and if they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

F3 POWER (MAIN-UTIL-PARMS-SETUP)

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

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



Initial Units are:	G20 English or G21 Metric depending on the feedback or screw type.
Number of Axes:	Can be 1 to 6.
Power-On Feedrate:	Can be any number up to the maximum feedrate.
Spindle Axis:	1 to 6 1=X, 2=Y, 4=A . . . etc
100% Rapid in Run: (No Override)	*NO* means the feedrate override will affect rapid in the run mode. *YES* means the feedrate override will not affect rapid in the run mode.
100% Rapid in Dry-Run: (No Override)	*NO* means the feedrate override will affect rapid in the dry run mode. *YES* means the feedrate override will not affect rapid in the dry run mode.
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

Load TEXT Cycles:	*NO* means Text Cycles will not be loaded *YES* means Text Cycles will be loaded, the text cycles take approximately 23K of RAM.
Don't Load Canned: Cycles	*NO* means Canned Cycles will not be loaded. *YES* means Canned Cycles will be loaded, the canned cycles (drill, pockets, frame etc.) take approximately 39K of RAM.

Custom M and G Code Tables

Custom Gcode 09010	
Custom Gcode 09011	
Custom Gcode 09012	
Custom Gcode 09013	
Custom Gcode 09014	
Custom Gcode 09015	Load the Gcode number into the
Custom Gcode 09016	program # that is the macro for
Custom Gcode 09017	that code.
Custom Gcode 09018	
Custom Gcode 09019	
Custom Mcode 09020	
Custom Mcode 09021	
Custom Mcode 09022	Load the Mcode number into the
Custom Mcode 09023	program # that is the macro for
Custom Mcode 09024	that code.
Custom Mcode 09025	
Custom Mcode 09026	Cannot have a custom Mcode for M0
Custom Mcode 09027	
Custom Mcode 09028	
Custom Mcode 09029	

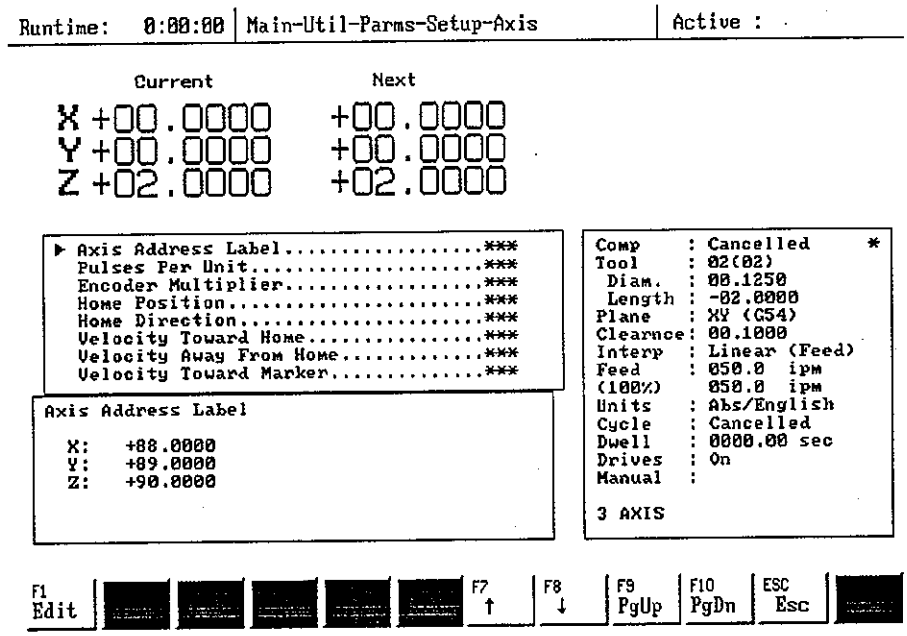
Use smaller soft keys: Yes/No

To change a parameter push F1 EDIT. A series of arrow keys will be displayed; simply use them to move the cursor to the desired parameter and type in the new number.

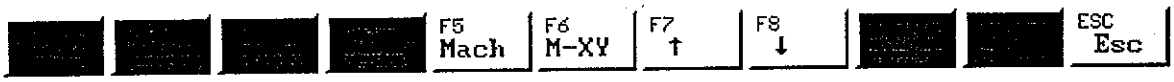
Note: After any power parameter is changed, the machine must be powered down, then up again. These parameters are only read on power up.

F4 AXIS (MAIN-UTIL-PARMS-SETUP-AXIS)

If the F4 Axis selection is pushed the following screen will be displayed.



Keys displayed in the Edit Mode:



The PGUP, PGDN and arrow keys move through the tables on the upper display. The lower display changes to display the data associated with the cursor position of the upper display. To edit the values in these tables the cursor is positioned via the arrow keys to the desired parameter in the upper screen. When the EDIT key is pushed the cursor will move to the lower screen. At this point again use 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, push the ESC key. The new value will be entered and the cursor will go back to the upper screen and the selection process can be started over. Some parameters can be related to the machine position. To edit or load these parameters use F5 MACH to load the X, Y, and Z positions in to the parameters or F6 M-XY for just X and Y 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
X + 10000.0000
Y + 10000.0000
Z + 10000.0000

The number of pulses the feedback gives per unit of travel.
English 1 = 10000 pulses
Metric 1cm = 1000 pulses

Encoder Multiplier
X 02.0000
Y 01.0000
Z 01.0000

Sets an internal multiplier on the number of pulses coming from the encoder.
Para. set 0 = 4* multiplication
Para. set 1 = 2* multiplication
Para. set 2 = 1* multiplication

Home Position
X +00.0000
Y +00.0000
Z +00.0000

The dimension assigned to the machine zero or home position.

Home Direction
X +00.0000
Y +00.0000
Z +00.0000

Defines the direction of rotation of the motor when a home is commanded.
CW = 00.0000
CCW = 01.0000

Velocity Toward Home
X 60.0000
Y 60.0000
Z 60.0000

Sets the feedrate at which an axis seeks the home limit switch

Velocity Away From Home
X 12.0000
Y 12.0000
Z 12.0000

Sets the velocity at which an axis feeds off the home limit switch

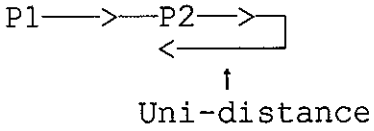
Velocity Toward Marker
X 06.0000
Y 06.0000
Z 06.0000

Sets the velocity at which an axis searches for the encoder marker pulse

Home Sequence
X 02.0000
Y 02.0000
Z 01.0000

The Home Sequence 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.

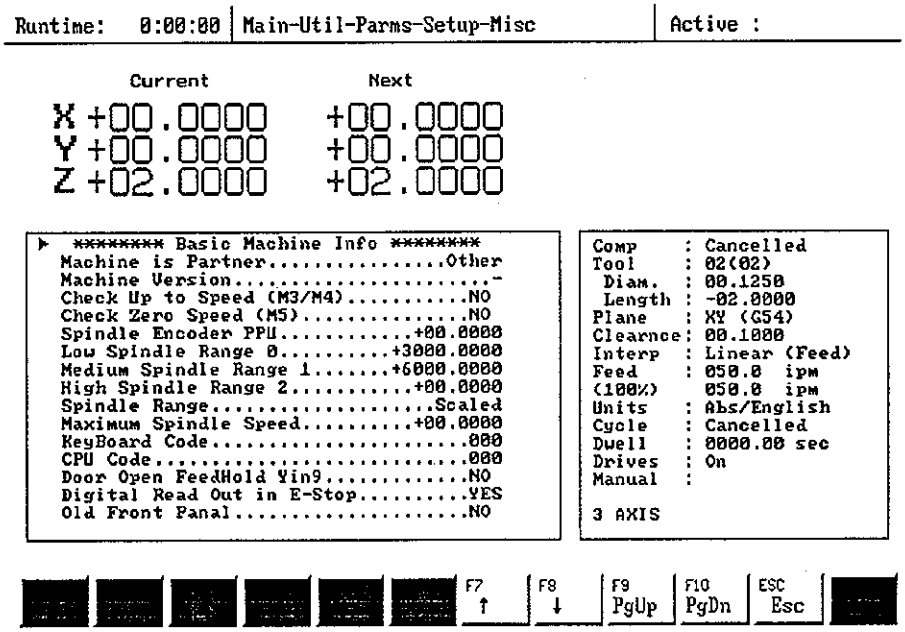
Positive Limit X 00.0000 Y 00.0000 Z 00.0000	Dimension from machine zero where the positive software limit occurs
Negative Limit X -31.0000 Y -18.0000 Z - 6.5000	Dimension from machine zero where the negative software limit occurs
Maximum Feed X 100.0000 Y 100.0000 Z 100.0000	Sets the maximum G01 feedrate 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 200.0000 Y 200.0000 Z 200.0000	Set the maximum G00 feedrate in inches per minute or mm per minute
Rapid Acc/Dec	40.000 The Acc/Dec constant is a number between 1 and 200 that determines the rate at which the axis velocity is stepped up. The smaller the number the longer the Acc/Dec times will be. Acceleration and deceleration in this control are linear ramps from one speed to another.
Max Jog Velocity	100.0000 Velocity in IPM
Jog Acc/Dec	40.0000 Parameter set the same as Rapid Acc/Dec
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.
In Position	Sets the distance in feedback

X 00.0000 Y 00.0000 Z 00.0000	units from the destination point where other axes will start their movement (inches or mm).
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. 
G60 Unidirectional X 00.0000 Y 00.0000 Z 00.0000	Same as G00 unidirectional except only active in a G60 block G60 X1 Y2
Backlash X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in inches or mm which the control will compensate for lost motion whenever an axis reversal takes place. Active in all modes.
Excess Error X 00.0000 Y 00.0000 Z 00.0000	Sets the distance in inches or mm the machine can lag behind the CNC before the CNC will shut the system down due to an excess following error condition. X 00.0000 = will never force an excess error
Rotary=0 Linear=1 X 01.0000 Y 01.0000 Z 00.0000	Sets whether an axis should be treated as circular or linear. In circular the feedrate is interpreted as degrees per minute rather than IPM. A circular axis also will roll over at 360 degrees.
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.

F5 MISC (MAIN-UTIL-PARMS-SETUP)

The F5 key brings up some miscellaneous setup parameters dealing with the spindle, RS-232 and M codes. When MISC is selected the following screen appears:



Keys displayed in the Edit Mode:

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

**********Basic Machine Info**********

Machine is Partner	Other, P1, P2, P3, P4, P5, P6, P7, P8, P9
Machine Version	H (H is the current model)
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 PPU	pulses per rev of spindle, used for hard tapping option
Low Spindle Range 0	max spindle speed for low range
Medium Spindle Range 1	max spindle speed for medium range
High Spindle Range 2	max spindle speed for high range
Spindle Range	which range the spindle is in Low, Medium, or High. Scaled assumes the range based on the programmed spindle speed and puts out the appropriate voltage.
Maximum Spindle Speed	Sets the maximum high range RPM. Could be less than max speed set in High Spindle Range 2 parameter.
Keyboard Code	015 for systems with a CAT900 card, 0 for others
CPU code	not used
Door Open Feedhold Y in 9	Activates door open/closed switch. Machine goes into feedhold when door is opened.
Digital Read Out in E-stop	Allows manual movement of machine and displays position as axes are moved.
Old Front Panel	Selects between old and new front panel types.
**********Software Options**********	
Max graph file size	8000 for standard systems, this limits the amount of graphics on the screen.
Minimum Parts Space	the system will go to the erase screen if the parts space is less than the parameter value is in KB
Lube Timer (min)	sets time between lube time messages
Screen Blank Time (min)	sets time before screen blanks if no keys are pushed
Special Flags	bit 1 (#1) will put a dot on the graphics screen when plotting the tool. bit 2 (#2) will shut off trig help bit 3 (#3) will shut off cutter compensation

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.
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
Full DOS File Names	If YES, eleven character DOS filenames (FILENAME.EXT) may be used in some circumstances.
Disable 417 Errors Load Tool Offsets Multiple Block Display	YES displays more than one line of the program when machine is running.

**********FastCam I 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 Mcode #0 Post Mcode #1 Post Mcode #2 Post Mcode #3 Post Mcode #4 Post Mcode #5 Post Mcode #6 Post Mcode #7 Post Mcode #8 Post Mcode #9	Mcodes listed here will be executed after all other operations within the block.

**********Additional Options**********

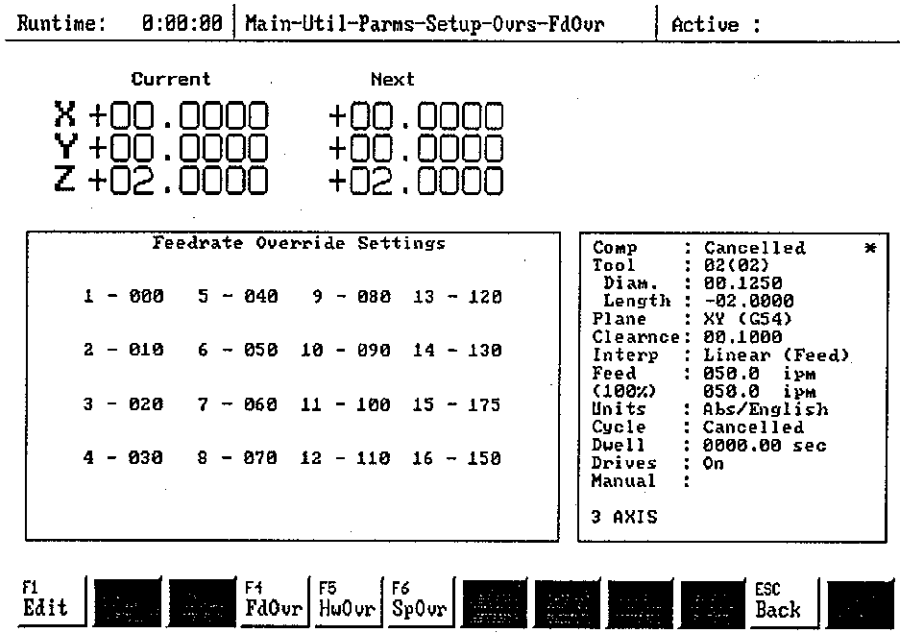
Report File	DOS file names to write DPRINT text to when using POPEN PO
PULSEX pulse delay (MS)	Msec pulse time used with PULSE0 and PULSE1

**********Spindle Power Monitoring**********

Spindle Power Raw	255	Value read from ADC, 0 to 255
Spindle Power Scale	200	Multiplier to obtain Spindle Power Value
Spindle Power Value (AMPS)		
Spindle Power Limit (AMPS)		
Actual Spindle Factor	60	

F6 OVRS (MAIN-UTIL-PARMS-SETUP)

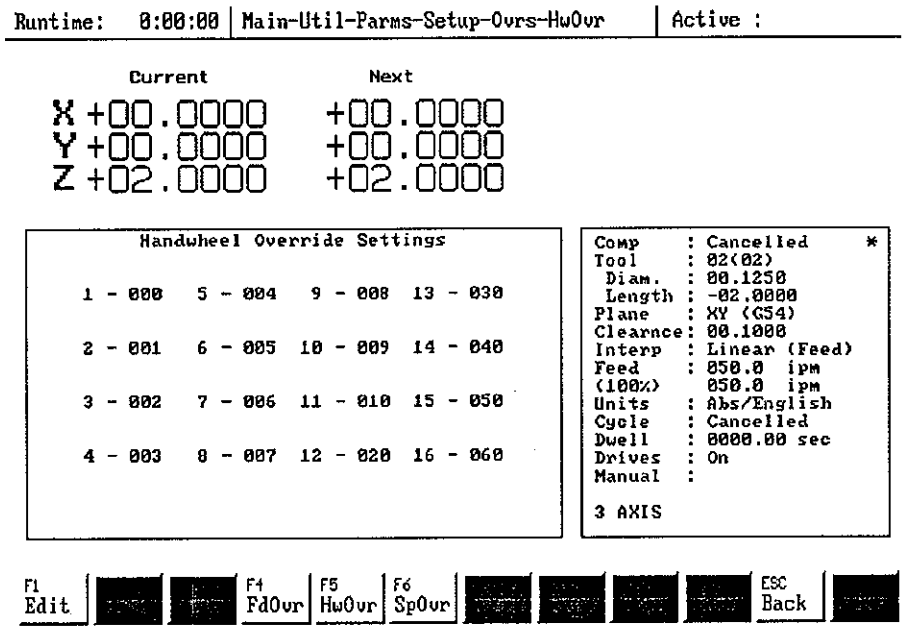
When F6 OVRS is selected the following screen will appear:



The control chooses the F4 selection as soon as you press the F6 OVRS (Overrides) key. The F4 FDOVR selection displays the feedrate override parameter settings. These settings determine which percentage will be used for each of the 16 feedrate override switch positions, i.e. entry 5-040 means 40% at 5th position, 1 is 0%, 2 is 10%, 3 is 20% . . . 16 is 150%. To edit the parameters, select EDIT and move the cursor to the value to be changed. Type in the desired change and hit ESC.

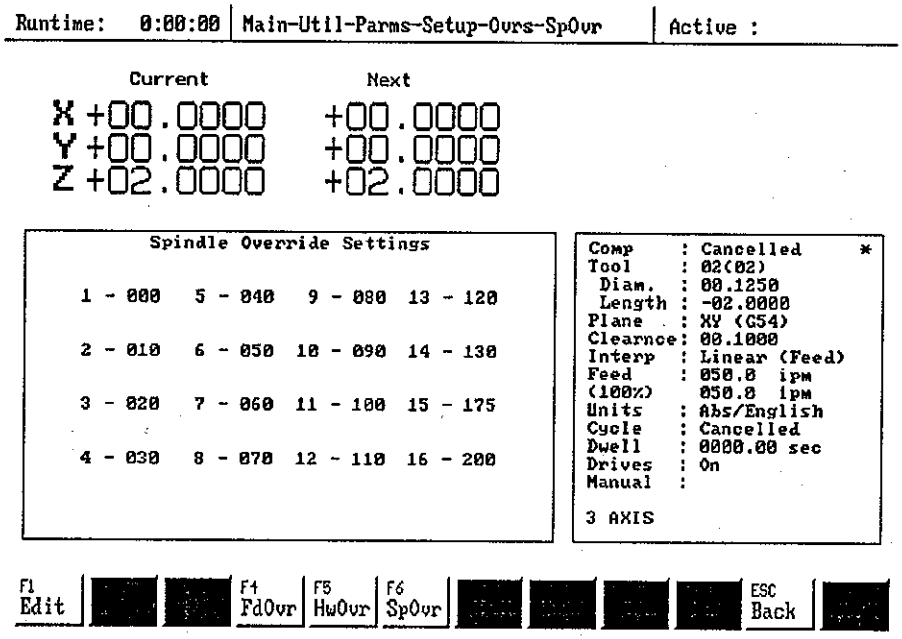
F5 HWOVR (MAIN-UTIL-PARMS-SETUP-OVRS)

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 the same way as the feedrate override parameters. The following screen displays the Handwheel Override Settings.

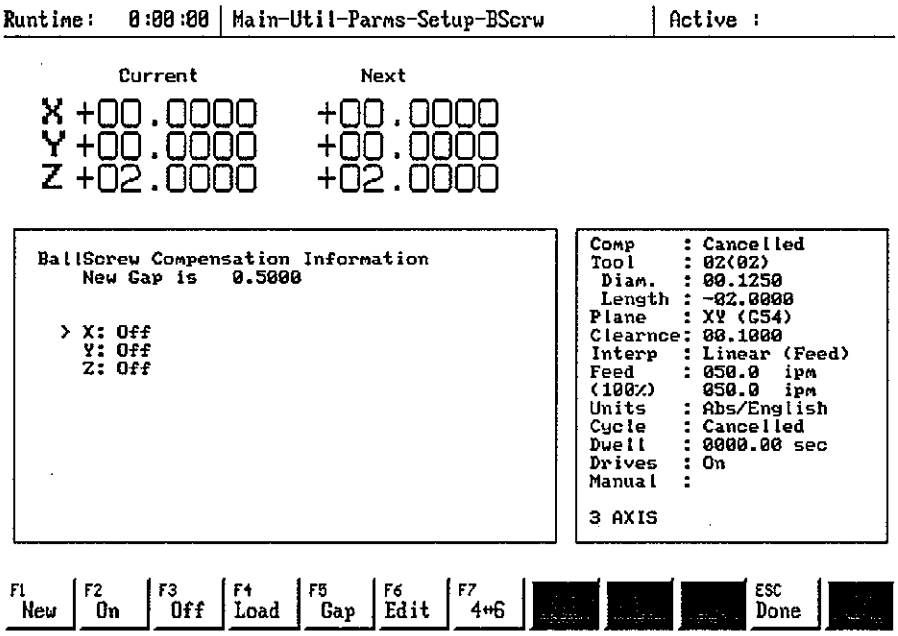


F6 SPOVR (MAIN-UTIL-PARMS-SETUP-OVRS)

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 the same as the feedrate override parameters. The spindle override screen is displayed below.



F7 BSCRW (MAIN-UTIL-PARMS-SETUP)



Ball Screw Compensation Table Creation help.

To select an axis, type the axis letter.

F1 NEW — This key creates a new ball screw table for the selected axis. The table will be created with an entry spacing equal to the GAP value which is usually .5". The table will have entries that cover the software limits of the selected axis. To enter values into the table select F6 EDIT.

F2 ON — Turns ball screw table on for the given axis. The last table loaded will be used.

F3 OFF — Turns ball screw table off for the given axis. OFF means no ball screw compensation for that axis.

F4 LOAD — The load function sends the current ball screw table to the axis controller. If that axis is in the ON state the new table will be active.

F5 GAP — Allows the GAP or table spacing to be changed. After it's changed F1 NEW will create a table with the new spacing.

F6 EDIT Displays the ball screw table for the selected axis. Values typed in are the ± deviation from the exact position.

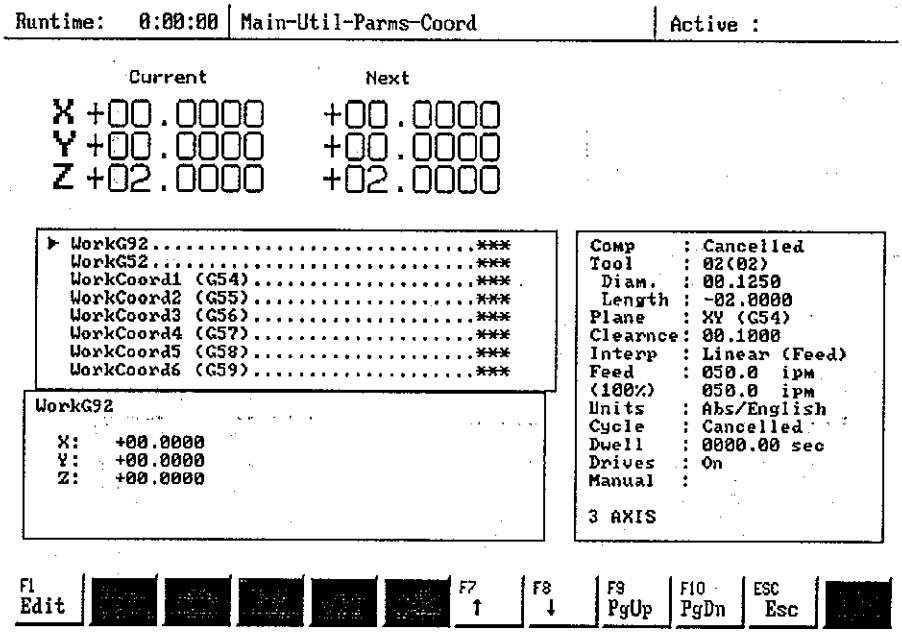
Example: exact position is 10.0000
actual position is 9.9995 table

F9 DOS (MAIN-UTIL-PARMS-SETUP)

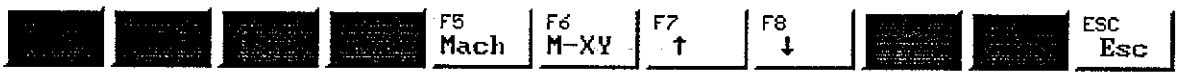
F9 DOS will exit the CNC software and go to a DOS prompt. Typing EXIT at the DOS prompt will return to the CNC software, or as will powering down and up.

F2 COORD (MAIN-UTIL-PARMS)

The F2 COORD key from 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, and then push the EDIT key and arrow to the desired axis. Type in the new values and push ENTER then EXIT. The following screen shows the G92, G52 and Work System parameters described in the G code section.



Keys displayed in the Edit Mode:



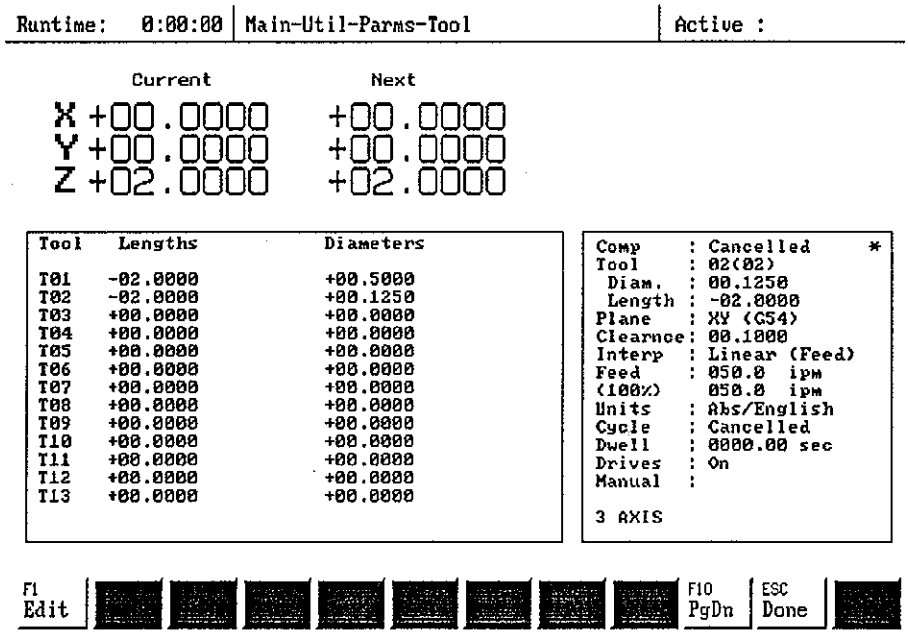
Operation of the Work Coordinate Systems, G92 and G52, was discussed in a different section. These parameters are positions relative to the machine zero and will become

the new zero point when they are used. The F5 MACH key in the edit mode enters the current machine position as the work coordinate zero point for X, Y and Z axes. The F6 M-XY enters the coordinates for X and Y axis only.

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 which the tool cannot enter.
Negative Safe Zone X 00.0000 Y 00.0000 Z 00.0000	If the tool is programmed into this cube an error will be displayed.
G28 Reference Point 1 G30 Reference Point 2 G30 Reference Point 3 G30 Reference Point 4	Are described in the Gcode section.

F3 TOOL (MAIN-UTIL-PARMS)

The F3 TOOL key brings up the following screen.



The editing on this screen is the same as on all the other PARMS screens. The F3 TOOL selection brings up both the tool length table and the radius table. A value typed into T04 Length is entered into H04 as well. This entry screen is mainly for

convenience as most tool offsets are entered through the tool setting programs. There are 99 sets of tool numbers and offsets which can be accessed.

Note: The control can be changed to use tool diameters or radius (see section on MISC parameters). When editing a tool offset the current value is displayed to the right of the value being edited, so small values can be added or subtracted from the offsets.

F9 CTRL (MAIN-UTIL-PARMS-CTRL)

This set of parameters is an extension of the program and machine setup parameters. Some of these parameters are duplicates but are presented here for convenience. * The two most important and commonly used parameters are the Skill Level and Update Lower Skill Levels.

**Note: These parameters can be changed anytime and do not need an access code. They are generally parameters that might need to be changed during normal operation of the machine.*

Skill Level	1/2. This parameter switches the control from its simple instruction set to the advanced set. It can be changed anytime **but not in the middle of a program.**
Update Lower Skill Levels	YES/NO. Yes will continuously update Level 1 programs to Level two. This is necessary if a Level 1 program needs Level 2 instructions. **However, once it has been updated it has to remain a Level 2 program and will need to be updated in Level 2.**
P146 Digitizing	Distance to retract chip brake cycle
P147 Digitizing	R plane distance for each peck cycle
P369 Job Time	This parameter contains the program run time.
Load Text Cycles	YES/NO. Loads lettering autoroutines into the program memory. Not loading text allows more program space for user programs.
Spindle Range	This parameter tells the control which gear range the spindle is in and scales the programmed RPM to agree with the spindle RPM. On automatic shifting machines this parameter is updated automatically. On manual shift machines the operator has to tell the control when the gear range has been changed.

Comm port, baud rate, parity, data bits and stop bits are communications parameters. See the RS-232 section for more information.

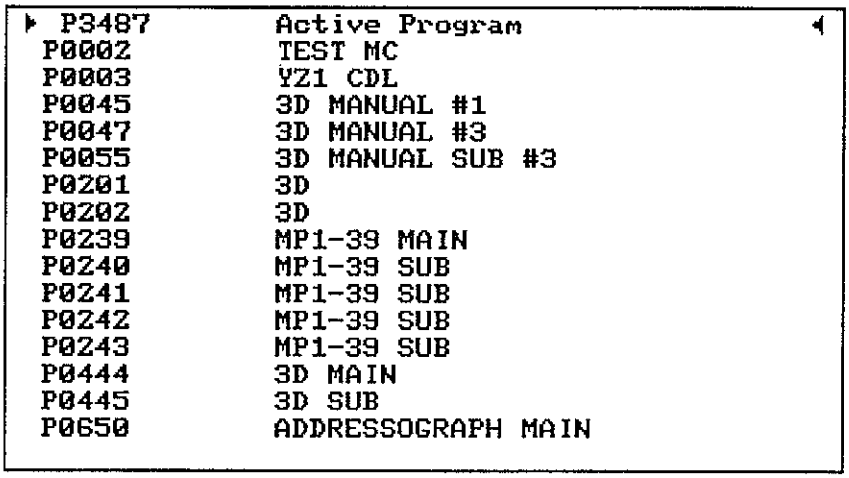
**********Serial Port Data**********

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, O/8/2
COM2 Baud Rate	110, 150, 300, 600, 1200, 2400, 4800, 9600, 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, O/8/2
Secondary Serial Port	NONE, COM1, or COM2
Tape Start Character	ASCII Value used sent at start of reading paper tape
Tape Stop Character	ASCII Value used sent at end of reading tape
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
RS-232 EOF character	ASCII value of character sent at end file sent on RS-232

Chapter 10

Graphic Verify

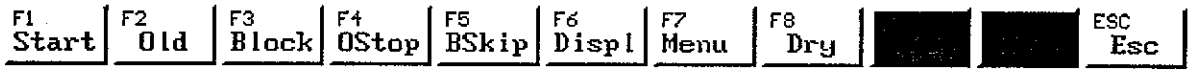
The Verify mode is used to graphically check a program. Verify is accessed by pushing the F9 VERF soft key from the main menu and the following window will appear if the control is in Skill Level 1 mode:



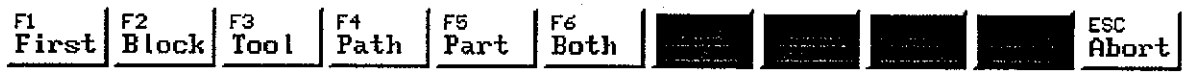
This window contains a listing of all the programs currently stored in the Centurion 1's memory. Using the arrow keys (F7 thru F10), move the cursor to the program you wish to verify. Once the cursor is at your desired program number, push the Enter key or F5 and the selected program will be displayed. If you wish to work with the current program just push the ENTER key at the Active Program position. When the program is selected, the following message and soft keys will be displayed:

Soft keys F3 and F8 have the same meaning and function in the Verify mode as they have in the Run mode. Refer to page 7-1 of the Run chapter for a more detailed description of these functions.

In the Skill Level 2 mode the following soft keys will be displayed:



These keys have the same meaning as their counterparts in the RUN mode. Refer to the chapter on *Running the Centurion 1* on page 7-1 for detailed explanations of the functions. In the Skill Level 1 mode the program has to be verified from the beginning as there are no other choices. In Skill Level 2 pushing F1 START will display the following choices to verify the program:



F1 FIRST Starts verifying the program from the beginning.

F2 BLOCK Asks for a block number and starts verifying from there.

F3 TOOL Asks for a tool number and verifies from it.

Soft keys F4 thru F6 select which type of graphic drawing will be displayed.

F4 PATH Displays on the screen only the tool centerline path. The dimensions displayed in the position display will have the tool diameter offsets included in them and is the actual position the machine will go to.

F5 PART Displays just the programmed part path without any tool offsets factored in.

F6 BOTH This selection displays both the part path and the tool path with offsets.

Once the part is displayed on the screen, manipulation of the graphic image is the same as in the Display mode. See the chapter on Display on page 8-1 for a full explanation.

Chapter 11

Machine Start-up and Programming Examples

Centurion 1 System Shell

The Centurion 1 system shell is a program that is used for various functions which are not supported by the Centurion 1 control software.

To enter the system shell, power up and when prompted, hit the F1 key to enter the system shell. *A:\> will be displayed.

These various functions are displayed using the F1 key:

ESC	Exit to CNC
F1	Display shell menu
F2	Setup SRAM
F3	Check SRAM
F4	Backup SRAM
F5	Set Paths
F6	Format Floppy
F7	Not Used
F8	[/]
F9	[:]
F10	[\]

F1 Displays shell menu.

F2 Setup SRAM. This allows reformat of SRAM drive (for memory expansion and so forth).

Note: All program and parameter files should be saved on floppy before format.

Format the SRAM drive (Y/N)?

Formats drive. Memory is cleared.

Create directories (Y/N)?

Creates necessary directories.

Default parameter file (Y/N)?

Defaults to standard parameter file from CNC software.

Default path file (Y/N)?

Defaults to standard path file from CNC software.

Blank ball screw tables (Y/N)?

X axis ball screw (Y/N)?
Positive home (Y/N)?
Y axis ball screw (Y/N)?
Positive home (Y/N)?
Z axis ball screw (Y/N)?
Positive home (Y/N)?

Blanks ball screw tables and sets positive home.

Begin SRAM setup (Y/N)?

Setup SRAM is performed.

F3 Check SRAM. This checks SRAM for lost or bad files.

*Erase *.CHK (Y/N)?*

Erases any lost or bad files found during SRAM check.

F4 Backup SRAM. This allows a total backup of the SRAM drive to a floppy disk.

F5 Set Paths. This option sets the drive and directory paths:

Hardware Used	No Disk	Floppy Disk
ROM Path	A:\ROM\	A:\ROM\
RAM Path	B:\RAM\	B:\RAM\
Parts Path	B:\PARTS\	B:\PARTS\
Graphics Path	C:\	D:\
Floppy Path	Z:\	C:\

F6 Format Floppy. This allows formatting of various types of floppy disks.

F8 [/] is used to enter a "/" character.

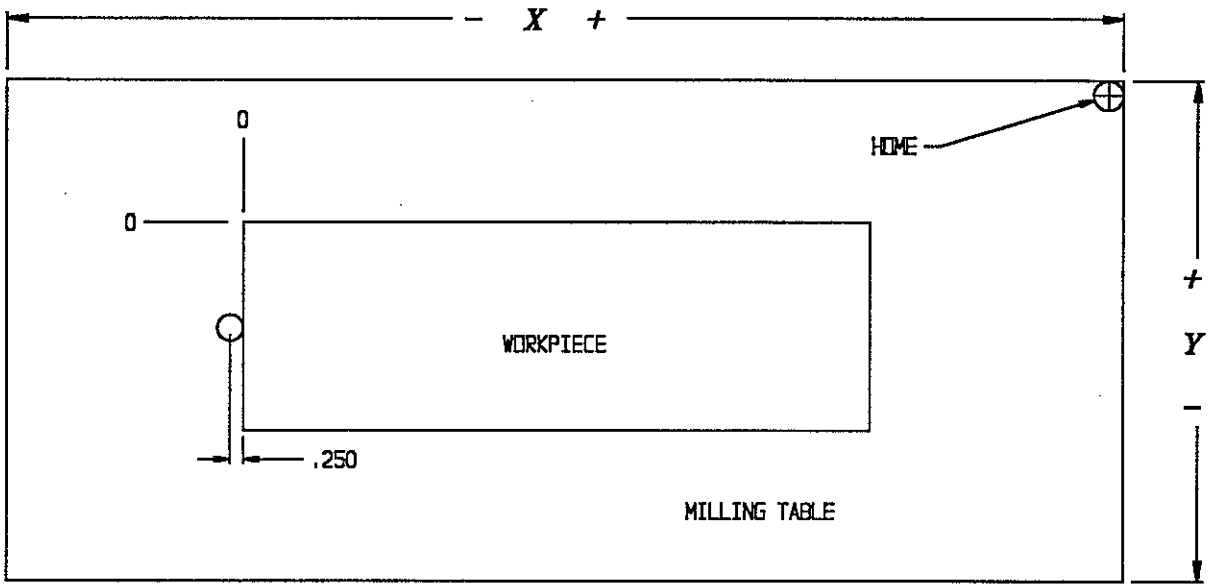
F9 [:] is used to enter a ":" character.

F10 [\] is used to enter a "\" character.

ESC Exits to the CNC.

Setting Part Zero (G54)

A work offset shifts the X, Y and Z 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 1/2" diameter edge finder in the X or Y axis, manually move or jog to the edge of the part. Then depress F5 G54-X or F6 G54-Y to set the floating zero for the appropriate axis. Next, to compensate for the edge finder's diameter, move Z up. Then move the X or Y axis back toward the part the distance of the edge finder radius and depress F5 FLZ-X or F6 FLZ-Y again.

At this point a new zero will be set. A command of X0 Y0 will move to the new zero point. This should place the center of the edge finder over the desired zero point on the part.

RS-232 Operation

COMMUNICATION PARAMETERS

Select F10 UTIL - F10 PARMS - F9-CTRL and F10 PGDN (twice) to set the following parameters:

COM/Port	1
Baud rate	1200 or 9600
Parity	Even
Data bits	7
Stop bits	2

Note: ***Both the computer and the machine control must have the same parameter settings.***

When communicating with RS-232 the receiving end must always be setup first.

To Receive:

1. Select F10 UTIL - F4 RS232 - F6 RECEV.
2. Enter the number for a new program, then [ENTER].
3. ESC terminates the Receive function.

To Send:

1. Select F10 UTIL - F4 RS232 - F5 SEND - F7 MENU.
2. Move the cursor to the program(s) to be sent and select F2 SET. After all programs are SET, select F1 START - F1 BEGIN.

Operational Key Sequences To Run And Program The CNC

HOME MACHINE

Main Power ON.

Press green RESET button to power up servos.

ESC To Main Menu.

F1 HOME

CYCLE START

SET WORK ZERO

ESC To Main Menu.

F2 for JOG, or EM STOP for manual.

Move X, Y & Z axes to program zero point. Z axis is normally left at Home.

In Jog Mode push F5 to set X zero, F6 to set Y zero.

In manual mode go to JOG, then F5 to set X zero, F6 to set Y zero.

ESC To Exit

TO RUN A PROGRAM

ESC To Main Menu

F4 Run

F8 Cursor to desired program.

F5 ENTER

Press CYCLE START to run program.

TO GRAPHIC VERIFY A PROGRAM

ESC To Main Menu

F9 VERF

F8 Cursor to desired program.

F5 ENTER

Press CYCLE START to begin Graphic Verify.

SCALE PART TO GRAPHIC WINDOW

F6 DISPL

F1 AUTO This will scale the part to fit the window.

TO ROTATE THE VIEW OF THE GRAPH

F6 DISPL

F3 ROT

Select Graph Plane (F1 thru F4) or (F5 thru F10) for user defined view

ESC To redraw new view.

F1 AUTO This will scale the part to fit the window.

TO DELETE A PROGRAM FROM MEMORY

ESC To Main Menu.

F10 UTIL

F3 FILES

F9 ERASE

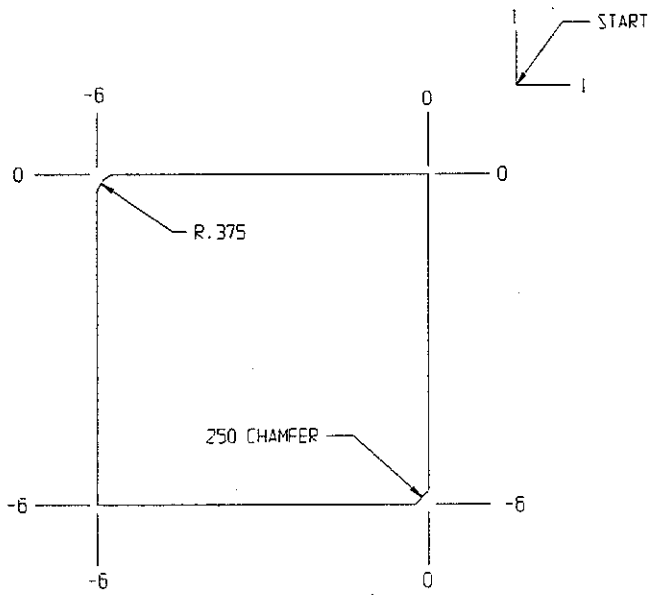
F7/F8 Move cursor up or down to select program number you wish to erase.

F2 SET

F1 START

ESC To return to Main Menu.

SAMPLE PROGRAM 1
Tool 1/2" End Mill



NOTE: ANYTIME YOU SEE * YOU MUST PRESS ENTER.

	F8 PROG	
	Cursor to New Program	
	F5	ENTER *
	Program Number: 1 ENTER	
Event 0	Program Setup	ENTER *
	Program Name	[MILL SQUARE W/RAD & CHAMFER] *
	• Dimensions (F3 TOGL)	*
	• Units (F3 TOGL)	[ENGLISH]
	Setup Notes	[ ]

• *Skill Level 2*

	F1(Will store this event)	
Event 1	F5 (TCHNG) Tool	[Change]
	Tool number	[1]
	Tool description	[1/2" mill Spindle 1500] *
	Tool diameter	[ ] (optional) *
	Spindle RPM	S [ ] *
	Spindle restart	[CW]
	Coolant	[Mist]
	F1 (Will store this event)	
Event 2	F2 (MILL)	
	F1 (START)	
	Z pierce feedrate	[20]
	Clearance point	[.1]
	Final depth	[-.5]
	1st depth	[-.2]
	Depth increment	[.2]
	X pierce point	X [1] *
	Y pierce point	Y [1] *
	Compensation (F3 TOGL)	[OFF] *
	F1 (Will store this event)	
Event 3	F2 (LINE)	
	Feedrate	[50] *

• *Skill Level 2*

	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[1] *
	Y position	[0] *
	Round or Chamfer	[NO] *
	F1 (Will store this event)	
Event 4	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[-6] *
	Y position	[ ] *
	Round or Chamfer (F3 TOGL)	[ROUND CORNER] *
	Radius	[.375] *
	F1 (Will store this event)	
Event 5	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[ ] *
	Y position	[-6] *
	Round or Chamfer	[NO] *
	F1 (Will store this event)	
Event 6	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *

• *Skill Level 2*

	X position	[0] *
	Y position	[ ] *
	Round or Chamfer (F3 TOGL)	[CHAMFER] *
	Length	[.25] *
	F1 (Will store this event)	
Event 7	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[ ] *
	Y position	[1] *
	Round or Chamfer	[NO] *
	F1 (Will store this event)	
Event 8	F4 (End Mill Cycle)	
	• Point on part after tool retract (F3 TOGL)	[To a point] X [1] * Y [1] *
	F1 (Will store this event)	
Event 9	F10 (EXIT)	
	F1 Edit	
	Z to tool change (F3 TOGL)	[YES]
	Spindle off (F3 TOGL)	[YES]
	X Position (home relative) Y Position (home relative)	[0] * [0] *
	(These two lines are optional; they would send the table to the Home position.)	
	* * * *	
	F1 (Will store this event or ESC will skip this event.)	

ESC (To exit conversational program)

ESC (To store program and return to Main Menu)

TO ADD CUTTER COMPENSATION TO THIS PROGRAM, CHANGE EVENT 2 AS FOLLOWS:

Event 2	F8	(PROG)	
	F5	(ENTER)	*
	F10	F10 Skip to Event 2	
	F1	(EDIT)	
	Z pierce feedrate		[20]
	Clearance point		[.1]
	Final depth		[-.5]
	1st depth		[-.2]
	Depth increment		[.2]
	X pierce point		X [1] *
	Y pierce point		Y [1] *
	• Compensation (F3 TOGL)		[RIGHT] *
	or		
	Compensation (Skill Level 1)		[AUTO RIGHT]
	• X before pierce		[1] *
	• Y before pierce		[2]
	F1	(Will store this event)	
	ESC		
	ESC		

• *Skill Level 2*

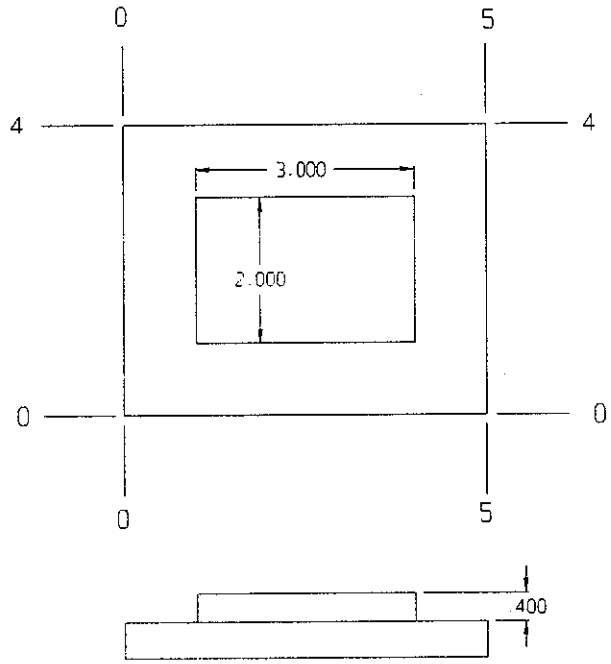
TO GRAPHIC VERIFY PROGRAM 1

F9 VERF

F5 ENTER Selects Program 1 or cursor to desired program *

CYCLE START

SAMPLE PROGRAM 2
ISLAND CLEAR
Tool 1/2" End Mill



NOTE: ANYTIME YOU SEE * YOU MUST PRESS ENTER.

Event 0	New Program Setup	ENTER *
	2 (For Program Number)	*
	Program Name	[ISLAND CLEAR DEMO] *
•	Dimensions (F3 TOGL)	[ABSOLUTE]
•	Units (F3 TOGL)	[ENGLISH]
	Setup Notes	[ ]
	F1 (Will store this event)	

• *Skill Level 2*

Event 1	F5 (TCHNG) Tool [Change]	
	Tool number	[1] *
	Tool description	[1/2 Mill Spindle 1200]
	Tool diameter	[ ] (optional)
	Spindle RPM	S [ ]
	Spindle restart	[CW]
	Coolant	[Mist]
	F1 (Will store this event)	
Event 2	F2 (Mill)	
	F1 (Start)	
	Z pierce feedrate	[20]
	Clearance point	[.2]
	Final depth	[-.400]
	1st Depth	[-.200]
	Depth increment	[.2]
	X pierce point	X [-.5] *
	Y pierce point	Y [1] *
	Compensation (F3 TOGL)	[AUTO RIGHT] *
•	X before pierce	X [1] *
•	Y before pierce	Y [2]
	F1 (Will store this event)	
Event 3	F2 (LINE)	

• *Skill Level 2*

	Feedrate	[20] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[4] *
	Y position	[1]
	Round or Chamfer	[NO]
	F1 (Will store this event)	
Event 4	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[ ] *
	Y position	[3] *
	Round or Chamfer	[NO] *
	F1 (Will store this event)	
Event 5	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	[1] *
	Y position	[ ]
	Round or Chamfer	[NO]
	F1 (Will store this event)	
Event 6	F2 (LINE)	
	Feedrate	[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *

• *Skill Level 2*

X position [ ] *

Y position [1] *

Round or Chamfer [NO]

F1 (Will store this event)

Event 7 F4 (End Mill Cycle)

• Point on part after tool retract X [2] *
Y [1] *

F1 (Will store this event)

F10 To exit Conversational Program

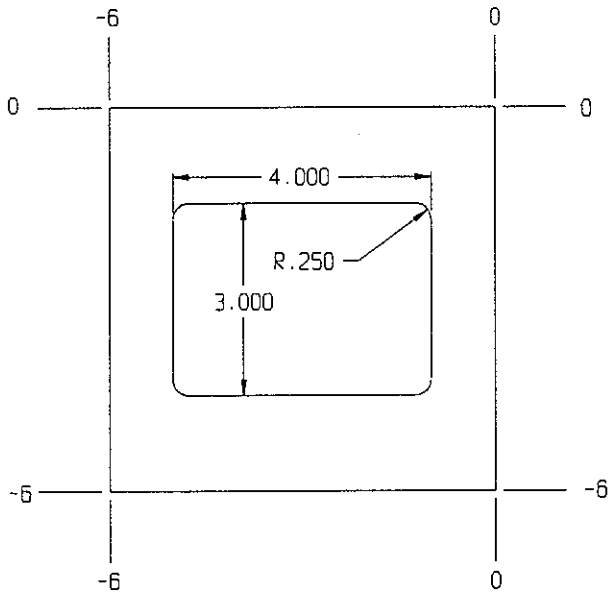
ESC

ESC To store program and return to Main Menu

Note: To move out .375 inches on each pass to clear further out, increase the tool diameter to .875 and run the program; then to 1.250 and run the program.

SAMPLE PROGRAM 3
SQUARE POCKET CLEAR
Tool 1/2" End Mill
.4 inches deep

NOTE: YOU MUST START POCKET CYCLE ON CENTERLINE OF POCKET.



NOTE: ANYTIME YOU SEE * YOU MUST PRESS ENTER.

Event 0	Program Setup	ENTER *
	3	*
	Program Name	[SQUARE POCKET CLEAR] *
	• Dimensions (F3 TOGL)	[ABSOLUTE]
	• Units (F3 TOGL)	[ENGLISH]
	F1 (Will store this event)	
Event 1	F5 (TOOL) Tool [Change]	

• *Skill Level 2*

	Tool number	[1] *
	• Tool change position	X [ ] Y [ ]
	Tool description	[1/2 Mill Spindle 1400]
	Spindle speed	S [ ]
	Spindle restart	[CW]
	Coolant	[Mist]
	F1 (Will store this event)	
Event 2	F1 (POS)	
	Feedrate (F3 TOGL)	[RAPID] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	X [-3] *
	Y position	Y [-3]
	F1 (Will store this event)	
Event 3	F2 (MILL)	
	F5 (POCKT)	
	F3 (TOGL) to	[RECTANGULAR POCKET CLEAR] *
	Z pierce feedrate	[5]
	• Return point	[Clearance]
	Clearance point	[.1]
	Z down method	[Ramp]
	Final depth	[-.400]
	1st depth	[-.200]

Depth increment		[.100]	
XY Feedrate		[20]	*
X + Y Pocket Center *(See Note 1.)*	X[	] Y[ ]	*
X + Y Pocket Dimension	X[4.0	] Y[4.0]	*
Corner Radius		[0]	*
Cut width (allow for overlap)		[.4]	*
XY Finish Stock		[0]	*
Compensation (F3 TOGL)		[ON]	
Cut direction (F3 TOGL)		[CW]	*

F1 (Will store this event)

F10 (To exit conversational program)

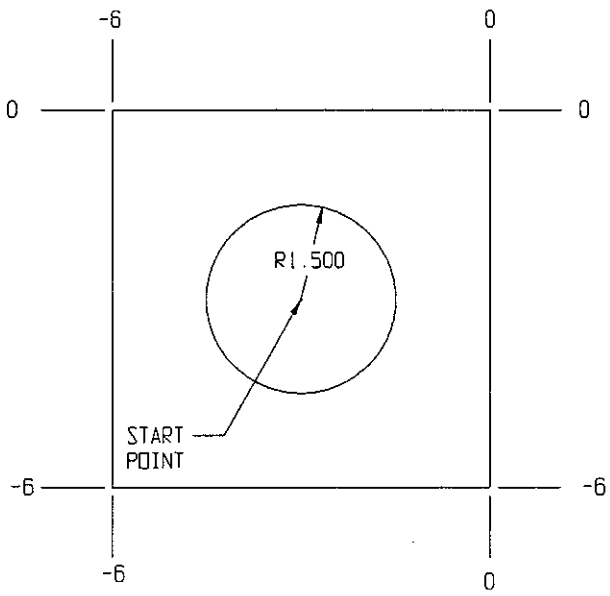
ESC

ESC (To store program and return to Main Menu)

Note 1: *If the XY pocket center dimension is left blank, the pocket will be centered about the current position of the machine.*

SAMPLE PROGRAM 4
CIRCLE POCKET CLEAR
Tool 1/2" End Mill

NOTE: YOU MUST START POCKET CYCLE ON CENTERLINE OF POCKET.



NOTE: ANYTIME YOU SEE * YOU MUST PRESS ENTER.

Event 0	Program Setup	
	Program Name	[CIRCLE POCK CLEAR] *
	• Dimensions (F3 TOGL)	[ABSOLUTE]
	• Units (F3 TOGL)	[ENGLISH]
	F1 (Will store this event)	
Event 1	F5 (TCHNG) Tool [Change]	
	• Tool change position	X [ ] Y [ ]

• *Skill Level 2*

	Tool number	[1]	*
	Tool description	[1/2 Mill 1550 Spindle]	
	Tool diameter	[ ] (optional)	
	Spindle RPM	[ ]	
	Spindle restart	[CW]	
	Coolant	[Mist]	
	F1 (Will store this event)		
Event 2	F2 (MILL)		
	F5 (POCKT)		
	F3 (TOGL) to	[CIRCLE POCKET CLEAR]	*
	Z pierce feedrate	[5]	
•	Return point	[Clearance]	
	Clearance point	[.1]	
	Z down method	[Ramp]	
	Final depth	[-.400]	
	1st depth	[-.200]	
	Depth increment	[.100]	
	XY Feedrate	[20]	*
	Pocket Center	X [-3] Y [-3]	*
	Pocket radius	[1.5]	*
	Cut Width	[.4]	
	XY finish stock	[.005]	*
	Compensation (F3 TOGL)	[ON]	

• *Skill Level 2*

Cut Direction (F3 TOGL) [CCW] *

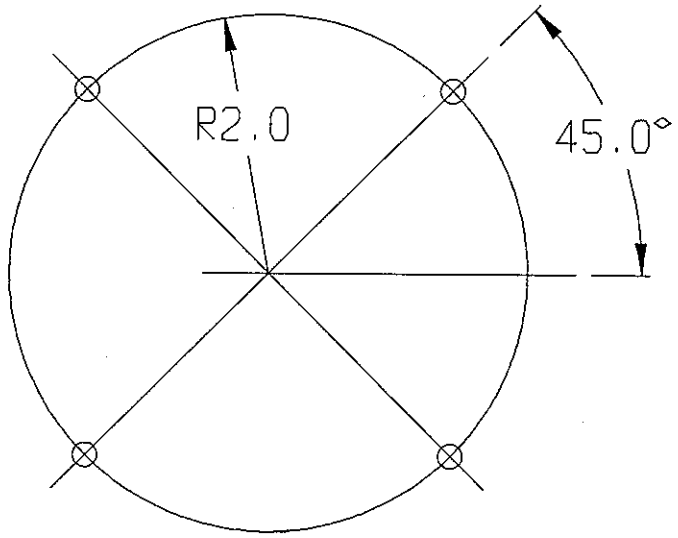
F1 (Will store this event)

F10 (To exit conversational program)

ESC

ESC (To store program and return to Main Menu)

SAMPLE PROGRAM 5
BOLT HOLE PATTERN



NOTE: ANYTIME YOU SEE * YOU MUST PRESS ENTER.

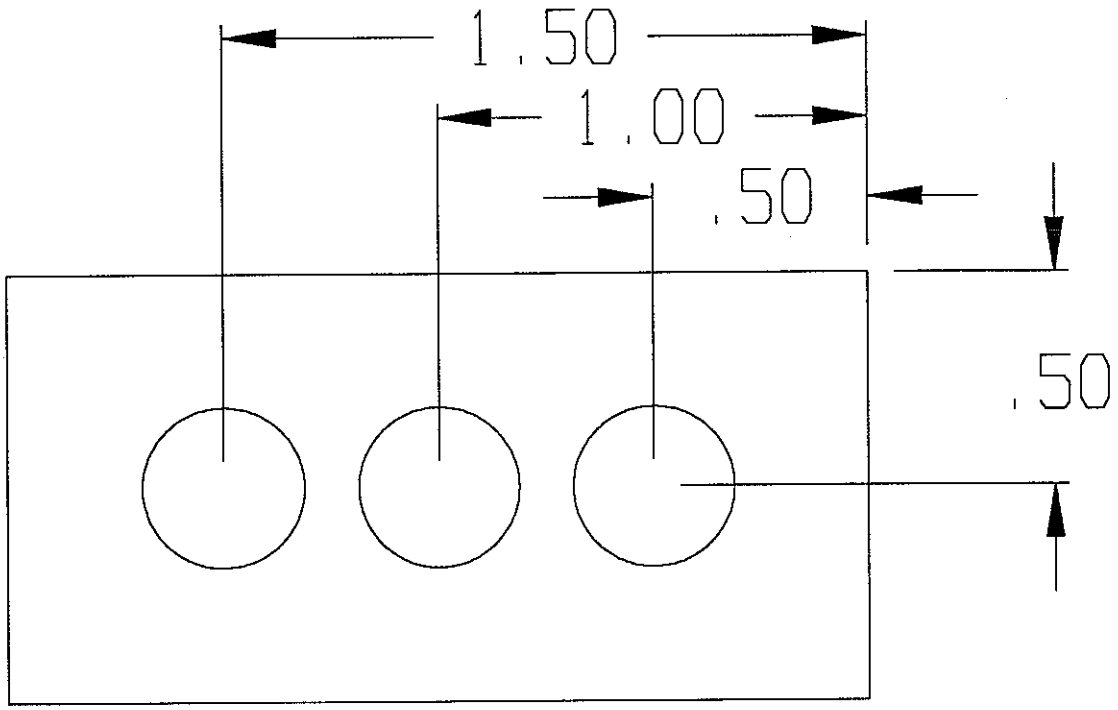
Event 0	Program Setup	
	Program Name	[BOLT HOLE PATTERN] *
	• Dimensions (F3 TOGL)	[ABSOLUTE]
	• Units (F3 TOGL)	[ENGLISH]
	F1 (Will store this event)	
Event 1	F5 (TCHNG) Tool [Change]	
	• Tool change position	X [ ] Y [ ]
	Tool Number	[1] *
	Tool Description	[1/4 Drill Spindle 1800] *

• *Skill Level 2*

	Tool Diameter	[ ] (optional)	
	Spindle RPM	S [ ]	*
	Spindle restart	[CW]	
	Coolant	[Mist]	
	F1 (Will store this event)		
Event 2	F4 (BOLT)		
	F3 TOGL to [Bolthole Drill Cycle]		
	Z pierce feedrate	F [10]	
	Return point	[Clearance]	
	Clearance point	R [.2]	
	Final depth	Z [-.8]	
	X Bolthole Center	X [0]	*
	Y Bolthole Center	Y [0]	*
	Bolthole Radius	[2] Dir [CW]	*
	Direction [CW/CCW]		
	Angle of First Hole	[45]	*
	# of holes to be made	[4]	*
	# of holes in 360°	[4]	
	F1 (Will store this event)		
	F10 (To Exit Conversational Program)		
	ESC		
	ESC (To store program and return to main menu)		

Note: If bolt center dimensions are left out, the bolt circle will be made about the current position of the machine.

SAMPLE PROGRAM 6
PECK DRILL CYCLE



NOTE: ANYTIME YOU SEE * YOU MUST PRESS ENTER.

Event 0	Program Setup	
	Program Name	[DRILL CYCLE DEMO] *
	• Dimensions (F3 TOGL)	[ABSOLUTE]
	• Units (F3 TOGL)	[ENGLISH]
	F1 (Will store this event)	
Event 1	F5 (TCHNG) Tool [Change]	
	• Tool change position	X [ ] Y [ ]

• *Skill Level 2*

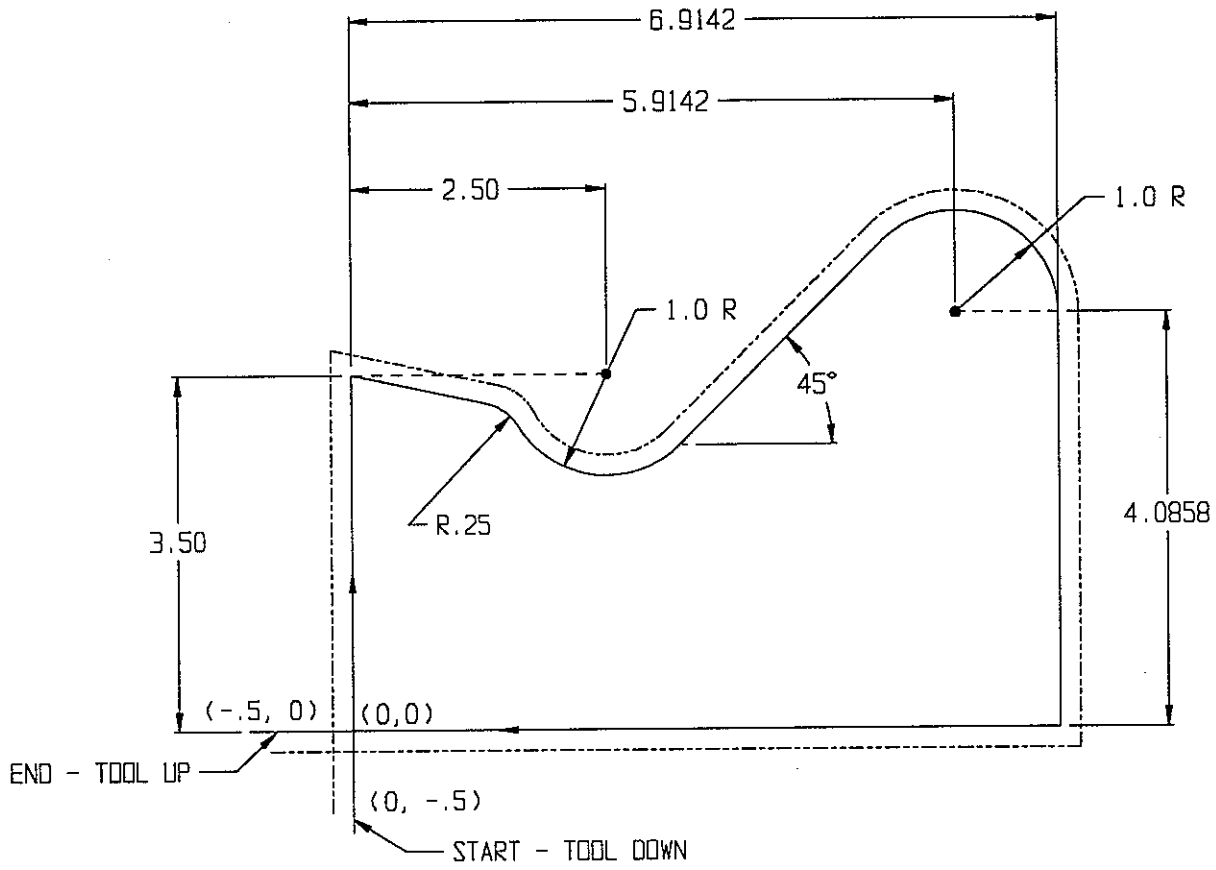
	Tool number	[1] *
	Tool description	[1/4 Drill Spindle 1850] *
	Tool diameter	[ ] (optional)
	Spindle RPM	S [ ] *
	Spindle restart	[CW]
	Coolant	[Mist]
	F1 (Will store this event)	
Event 2	F3 (DRILL)	
	F1 (START)	
	F3 (TOGL) [Enable Drill/Peck Cycle]	
	Z pierce feedrate	F [15]
•	Return point	[Clearance]
	Clearance point	R [.2]
	Final depth	Z [-.8]
	First depth	V [-.3]
	Peck Increment	Q [.2]
Event 3	F2 (POS)	
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	X [-.5] *
	Y position	Y [-.5]
	F1 (Will store this event)	
Event 4	F2 (POS)	
	Coordinates (F3 TOGL)	[ABSOLUTE] *

• *Skill Level 2*

	X position	X [-1] *
	Y position	Y [ ]
	F1 (Will store this event)	
Event 5	F2 (POS)	
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	X [-1.5] *
	Y position	Y [ ]
	F1 (Will store this event)	
Event 6	F5 (END)	
	F1 (Will store this event) Cancels Drill Cycle	
	F10 (To exit Conversational)	
	ESC	
	ESC (To store this program)	

SAMPLE PROGRAM 7
CONTOUR MILLING
.4 deep

This program MUST be programmed in the Skill Level 2 mode. Event 5 is not an option in Skill Level 1.



Event 0 Program Setup

Program Name [ ]

• Dimensions (F3 TOGL) [ABSOLUTE] *

• Units (F3 TOGL) [ENGLISH]

F1 STORE

• *Skill Level 2*

Event 1	F5	(TCHNG) Tool Change	
	•	Tool change position	X [ ] Y [ ]
		Tool number	[1] *
		Tool description	[1/2 Mill Spindle 1500] *
		Tool diameter	[ ] (optional)
		Spindle speed	S [ ] *
		Spindle restart	[CW]
		Coolant	[Mist]
	F1	(STORE)	
Event 2	F2	(MILL)	
	F1	(START)	
		Z pierce feedrate	[20]
		Clearance point	[.2]
		Final depth	[-.400]
		1st depth	[-.200]
		Depth Increment	[.2]
		X pierce point	[0] *
		Y pierce point	[0] *
		Compensation (F3 TOGL)	[AUTO LEFT] *
	•	X before pierce	[0]
	•	Y before pierce	[-1]
	F1	(STORE)	

• *Skill Level 2*

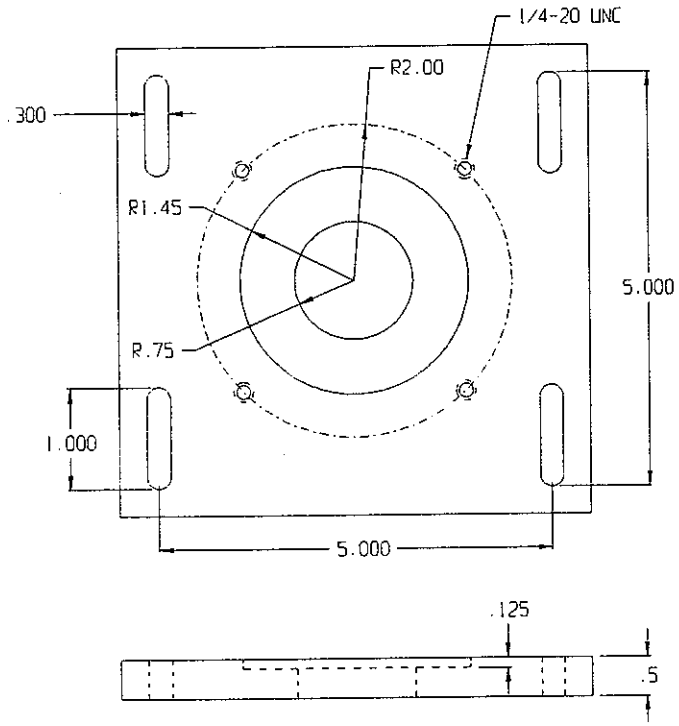
Event 3	F2 (LINE)			
	Feedrate			[20] *
	Coordinates (F3 TOGL)			[ABSOLUTE] *
	X position			[0] *
	Y position			[3.5]
	F1 (STORE)			
Event 4	F2 (LINE)			
	Feedrate			[ ] *
	Coordinates (F3 TOGL)			[POLAR] *
	• Plane (F3 TOGL)			[X Y]
	• Type (F3 TOGL)			[CURRENT] *
	Length of line			[1] *
	Angle of line			[-10] *
	Round or Chamfer (F3 TOGL)			[ROUND CORNER] *
	Radius			[.25] *
	F1 (STORE)			
Event 5•	F4 (TANGS)			
	Plane			[XY]
	Direction (F3 TOGL)			[CCW]
	R1			[1]
	XC1	[2.5]	YC1	[3.5]
	R2			[1]

• *Skill Level 2*

	XC2 [5.9142] YC2	[4.0858]
	Exit 1st Arc [RIGHT] * Enter 2nd Arc	[LEFT] *
	Connect with	[A LINE]
	F1 (STORE)	
Event 6	F3 (ARC) *(This event programs 2nd arc of TANG ARC functions.)*	
	Plane	[XY]
	Feedrate	F[ ] *
	Direction (F3 TOGL)	[CW] *
	Center (F3 TOGL)	[ABS CENTER] *
	Arc Radius	R[1] *
	Arc Center	XC[5.9142] * YC[4.0858] *
	End Point (F3 TOGL)	[POLAR] *
	End Angle	AB[0]
	Z position	[ ]
	Round Corner	[NO]
	F1 (STORE)	
Event 7	F2 (LINE)	
	Feedrate	F[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	X[6.9142] *
	Y position	Y[0] *
	Round or Chamfer	[NO] *

	Length	[.5]
	F1 (STORE)	
Event 8	F2 (LINE)	
	Feedrate	F[ ] *
	Coordinates (F3 TOGL)	[ABSOLUTE] *
	X position	X[-.5] *
	Y position	Y[0]
	Round or Chamfer	[NO]
	F1 (STORE)	
Event 9	F7 (END)	
	Point on Part After Tool Retract (F3 TOGL) [To a point]	X[-1] Y[0]
	Options [---]	
	F1 (STORE)	
	F10 (EXIT)	
	ESC	
	ESC (To Store Program and Return to Main Menu)	

SAMPLE PROGRAM 8
MOTOR BRACKET



In general, when programming a part with multiple operations at the same location or the same shape at different locations, it is best to create individual programs for these shapes. The above part has three of these cases: the slots, rabbit relief, and the bolt circle which needs to be spot drilled, drilled, and tapped. The tools for this job are:

Tool #1	1/4-20 tap drill
Tool #2	spot drill
Tool #3	1/2" end mill
Tool #4	1/4" end mill
Tool #5	1/4-20 tapping head

The following program is written on one of our programming sheets in a condensed form. Unlike the previous programs where every keystroke is described, in this format only the necessary data is written down. We always recommend writing a program down before trying to enter it into the control. It is not absolutely necessary to do this, but you will find that less mistakes are made and the total programming time to get the first part is less. The MDI mode is designed for running the simple one-time part which does not need a written program. However, if the part is too complicated or multiple parts are being made, a program should be written.

The previous part is one that could be done in MDI without a program if only a single part is being made it. If more than one is to be made, a program should be written.

Program Name **Main Prog. Motor Bracket**

Program No. **9**

Event #	Event Type	DATA
1	Tool Change	Tool #2, Spot drill, RPM 1500, Stop [Yes], Mist, CW
2	Drill	Clearance = .2, Z depth -.7, Z feed 15
3	Call	Program 11, Hole Locations
4	Bolt Drill	Center X0 Y0, Radius 2, 4 holes, Angle 1st 45°, Z depth -.2, Clear .1
5	Tool Change	Tool #1, 1/4-20 tap drill, RPM 1200, Stop [Yes], Mist, CW
6	Drill Peck	Z Feed 15, Clear .2, Z depth -.7, 1st Z -.3, Z increment .2
7	Call	Prog. 11, Hole Locations
8	Bolt Peck	Z feed 15, Clear .2, Z depth -.7, 1st Z -.3, Z increment .2, XC=0 YC=0, Radius 2
9	Tool Change	Tool Rad.=.125, Tool #4, 1/4 End Mill, RPM 1600, Stop [Yes], Mist, CW
10	Pos	X2 Y-2, Center of Slot 1
11	Call	Prog. 10, Slot 1
12	Pos	X2 Y2, Center of Slot 2
13	Call	Prog. 10, Slot 2
14	Pos	X-2 Y2, Center of Slot 3
15	Call	Prog. 10, Slot 3
16	Pos	X-2 Y-2, Center of Slot 4
17	Call	Prog. 10, Slot 4
18	Tool Change	Tool Rad.=.25, Tool #3, 1/2 End Mill, RPM 1200, Stop [Yes], Mist, CW
19	Pos	X0 Y0
20	Circle Clear	Radius 1.45, Finish 0, CW, Feedrate 2, Comp ON, Z depth -.125, Z 1st -.125, Z increment .2, Clearance .2, Ramp, Cut width .4
21	Circle Finish	Radius .75, Finish 0, CW, Feedrate 5, Comp ON, Z depth -.6, Z 1st -.3, Z increment .3, Clearance .2, Ramp, Cut width .3
22	Tool Change	Tool Rad.=.125, Tool #5, 1/4-20 tapping, RPM 600, Stop[Yes], Mist, CW
23	Bolt	Bore center X0 Y0, Radius 2, 4 holes, Angle 1st hole 45°, Z depth .8, Clear .2
24		
25		
26		
27		

Program Name Slots

Program No. 10

Event #	Event Type	DATA
1	Rect. Finish	X length .3, Y length 1, CW, Comp ON, Rad.=.15, Z feed 5, Clear .2, Z down [Plunge], Z final -.6, XY feed 10
2		Note: Do not fill in the pocket center dimension. The control will then
3		use the current position of the machine as the pocket center.
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		
18		
19		
20		
21		
22		
23		
24		
25		
26		
27		
28		
29		
30		

Program Name Hole Locations

Program No. 11

Event #	Event Type	DATA
1	Pos	X2 Y-2
2	Pos	Y2
3	Pos	X-2
4	Pos	Y-2
5	Pos	X0 Y0
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		
18		
19		
20		
21		
22		
23		
24		
25		
26		
27		
28		
29		
30		
31		

Program Name _____

Program No. _____

Event #	Event Type	DATA
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		
18		
19		
20		
21		
22		
23		
24		
25		
26		
27		
28		
29		
30		
31		

APPENDIX

CENTURION 1 SLS
ERROR MESSAGES

001 Invalid function number
002 File not found
003 Path not found
004 Too many open files
005 File access denied
006 Invalid file handle
007 Memory control blocks destroyed
008 Insufficient memory
009 Invalid memory block address
010 Invalid environment
011 Invalid format
012 Invalid file access code
013 Invalid data
015 Invalid drive specified
016 Cannot remove current directory
017 Cannot rename across drives
018 No more files

100 Disk read error
101 Disk write error - Parts Memory is full
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
151 Unknown unit
152 Drive not ready
153 Unknown command
154 CRC error in data. Diskette is Bad or it is not formatted
155 Bad drive request structure length
156 Disk seek error
157 Unknown media type
158 Sector not found
159 Printer out of paper
160 Device write fault
161 Device read fault
162 Hardware failure
163 Sharing Violation

200 Division by zero
201 Range check error
202 Stack overflow
203 Heap overflow - insufficient RAM memory
204 Invalid pointer operation
205 Floating point overflow
206 Floating point underflow
207 Invalid floating point operation
208, 209 TP Overlay Error
210..214 TP Object Error

300 Program already exists
301 Invalid program number

302 No programs to select from
303 Problem saving program(s) to disk
304 Problem loading program(s) from disk
305 Not formatted for conversational, try text editor
306 Conversational system has been corrupted
307 Illegal Event number
308 Invalid tool number
309 Can't copy or rename a file to itself
310 File not formatted for conversational or parameters
311 Parameter file not valid
312 Insufficient parts storage
313 Insufficient storage for compression, unable to post the file
314 Insufficient storage, post has been aborted
315 Out of storage space on the floppy
316 Not enough storage to create a new file
318 zoom factor is too large
319 Send file aborted

400 Home required
401..
406 Axis software limit overtravel
407..
412 Axis excess error condition
413 Attempted to move into safe zone
415 Can't establish DNC link while program is running
416 Out of position, arc end point is not on the arc
417 Can't edit parameters while program is running
418 Controller Card Error
451 Lube fault, Check Way lube Level
452 Tool not found in auto tool sequence
453 Tool pot not up during turret movement
454 Not at tool change position
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

500 Last soft key pressed is not supported at this time
501 Illegal address encountered
502 Undefined canned cycle
503 Return without gosub
504 Coincident points, the start point and end point are the same on an arc without a center
505 Radius too small to span given points
507 After compensation line to arc lacks intersection
509 Arcs lack intersection
517 Parameter out of range
518 Illegal program statement
519 Feedrate out of range
520 Spindle Speed out of range
521 Negative Arc Radius
522 Negative Polar Radius
523 Illegal Tool Number
524 Illegal Radius Number
525 Illegal Length Number
526 Invalid Access Code
527 Invalid Access Level

529 Duplicate address encountered
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
536 Can't chamfer and round the same corner
537 Can't chamfer to or from arcs
538 Loop counter out of range
539 Dwell time out of range
540 Illegal dwell time encountered
541 No axis moves are allowed on a G31 or G32 block
542 G30 Illegal return to reference parameter on G30 block
543 Illegal G10 statement
544 Too many digits in number
545 Illegal K value for number of holes
546 Nested calls or gosubs to deep
547 Comment not closed
548 M-Code out of range
549 Unrecognized G-Code
550 Bad numeric format
551 Multiple decimal points
552 Missing "]"
553 Missing "["
554 Tangent function overflow
555 Missing "/"
556 Negative SQRT argument
557 Unknown function
558 Assignment #2
559 Assignment #1
560 Illegal relational operator
567 Unresolved Call
568 Unresolved Goto or Gosub
569 The tool is too large to cut inside the arc "Compensated radius is too small"
570 The tool is too large to cut inside the arc "1st compensated radius in arc to arc is < 0"
571 The tool is too large to cut inside the arc "2nd compensated radius in arc to arc is < 0"
572 Pocket clear is not in a Start/End mill cycle -WHILE WEND loop-
573 Round wall is not in a Start/End mill cycle -WHILE WEND loop-
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-
576 Z increment is 0
577 Input statements must precede axis moves
578 Undefined text cycle {should display offending character}
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

600 Can't nest Start/End mill cycles -WHILE WEND loops-
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 does not exist
605 Can't modify dry run status while program is running
606 Program is empty
607 Can't exit DNC run mode while program is running
608 "P" expected in M98 block
609 G20/21 not allowed in DNC Fast.

800 Illegal probe block
801 Missing end of pick
802 Reversed scan segment
803 Missing end of scan
804 Reversed pick segment
805 Invalid probe setup
806 Scan origin expected
807 Probe file not found
808 Setup not selected
809 Bad Z limits encountered
810 Stuck digitizing probe
812 Input out of range

900 RS232 overrun error
901 RS232 parity error (The system sending data may be not have the same parity as the CNC)
902 RS232 framing error (Remote system and CNC may not have the same line settings or a loose cable)
903 RS232 break detected (RS232 cable may be loose)
950 Obsolete controller software for this system.
951 Obsolete acroloop software for this system.
952 Obsolete keyboard software for this system.