

ANILAM

**3000M CNC
Programming and Operations
Manual for
Three- and Four-Axis Systems**

Section 1 - CNC Programming Concepts

Programs.....	1-1
Axis Descriptions.....	1-1
X Axis	1-2
Y Axis	1-2
Z Axis	1-2
Defining Positions	1-2
Polar Coordinates.....	1-3
Absolute Positioning.....	1-3
Incremental Positioning	1-4
Tool-Length Offsets.....	1-5
Tool Diameter Compensation.....	1-6
Using Tool Diameter Compensation and Length Offsets with Ball-End Mills	1-10
Angle Measurement	1-10
Corner Rounding.....	1-11
Line-to-Line Corner Rounding	1-11
Line-to-Arc Corner Rounding.....	1-12
Arc-to-Arc Corner Rounding	1-12
Chamfering.....	1-13
Plane Selection	1-14
Arc Direction.....	1-15

Section 2 - CNC Console and Software Basics

Console	2-1
Keypad.....	2-1
Programming Hot Keys	2-2
Editing Keys	2-3
Manual Operation Keys	2-3
Operator Keys	2-4
Soft Keys (F1) to (F10).....	2-5
Off-line Keyboard (Optional).....	2-5
Software Basics	2-5
Pop-up Menus	2-5
Screen Saver.....	2-6
Switching Selections with the Toggle Key	2-6
Clear Key.....	2-6
Operator Prompts	2-6
ASCII Chart	2-6
Cursor and Highlight Functions	2-7
Entering Text	2-7
Typing Over and Inserting Letters and Numbers	2-7
Deleting Characters.....	2-8
Messages/Error Messages.....	2-8

Section 3 - Manual Operation and Machine Setup

Powering On the CNC.....	3-1
Shutting Down the CNC	3-1
Emergency Stop (E-STOP)	3-1
Performing an Emergency Stop	3-1
Activating/Resetting the Servos	3-2
(Re-)Starting the Spindle.....	3-2

Manual Mode Screen	3-3
Primary Display Area Labels	3-4
Secondary Display Area Labels.....	3-4
Position Display	3-5
Manual Machine Operation	3-6
Manual Mode.....	3-6
Auto Mode	3-6
Mode Settings	3-7
Activating Manual Mode Rapid or Feed	3-8
Setting a Feedrate	3-8
Adjusting Rapid Move Speed	3-8
Overriding the Programmed Spindle RPM.....	3-9
Absolute/Incremental Modes	3-9
Inch/MM Modes.....	3-9
Setting Absolute Zero	3-10
Defining Absolute Zero in X and Y Axes.....	3-10
Presetting the X- or Y-Axis	3-10
Fixture Offsets (Work Coordinate System)	3-11
Setting Tool Change Position	3-12
Locating Tool #0, Z0.....	3-12
Presetting the Z-Axis	3-13
Activating a Tool.....	3-13
Activating a Plane.....	3-13
Activating a Spindle RPM (Requires Programmable Spindle Option).....	3-14
Jog Moves.....	3-14
Changing the Jog Mode	3-15
Jogging the Machine (Conventional)	3-15
Jogging the Machine (Continuous)	3-16
Operating the Handwheel (Optional)	3-16
One-Shot Moves	3-17
Manual Data Input.....	3-17
Disengaging the Z-Axis Drive System.....	3-19

Section 4 - Writing Programs

Program Basics.....	4-1
Developing Part Programs	4-1
Writing Program Blocks.....	4-3
Using Graphic Menus	4-3
No Move Blocks	4-4
Programming an Absolute/Incremental Mode Change	4-4
Programming an Inch/MM Mode Change	4-4
Programming a Tool Change.....	4-5
Activating a Tool.....	4-5
Activating Tool-Diameter Compensation	4-6
Programming a Dwell	4-7
Programming a Return to Machine Zero	4-8
Programming Fixture Offsets.....	4-9
Resetting Absolute Zero (Part Zero).....	4-11
Programming a Plane Change	4-13
Programming a Feedrate Change	4-14
Programming a Spindle RPM	4-15

Straight Moves	4-16
Programming a Rapid Move	4-16
Programming a Line Move.....	4-17
Programming a Modal Move.....	4-17
Teach Mode (Programming from the Part).....	4-18
Line or Rapid Moves	4-19
Programming a Move Using XY Location, Radii, or Angles.....	4-20
Arcs.....	4-21
Selecting the Plane for an Arc	4-21
Programming an Arc Using an Endpoint and Radius	4-21
Programming an Arc Using the Center and Endpoint.....	4-23
Programming an Arc Using the Center and the Included Angle	4-25
Programming M-Code Blocks	4-27
Dry Run M-Codes.....	4-28
U-Axis Synchronization M-Codes.....	4-28

Section 5 - Programming Canned Cycles, Ellipses, and Spirals

Drilling Cycles	5-1
Basic Drill Cycle.....	5-2
Peck Drilling Cycle.....	5-3
Boring Cycle	5-4
Chip Break Cycle.....	5-5
Tapping Cycle.....	5-7
Drill Pattern.....	5-8
Bolt Hole Pattern	5-9
Thread Milling Cycle	5-11
Pocket Cycles	5-16
Facing Cycle.....	5-17
Rectangular Profile Cycle	5-19
Circular Profile Cycle	5-21
Rectangular Pocket Cycle	5-23
Circular Pocket Cycle	5-25
Frame Pocket Cycle	5-27
Hole - Mill Cycle.....	5-29
Irregular Pocket Cycle	5-31
Pockets with Islands	5-36
Subprograms.....	5-39
Situation: 1 (Repetitive Drilling Cycle)	5-39
Situation: 2 (Rough and Finish Cycles)	5-39
Subprogram Structure	5-39
Subprogram Example	5-39
Organizing Programs Containing Subprograms	5-40
Calling Subprograms from the Main Program.....	5-40
Ending Main Programs	5-40
Starting Subprograms.....	5-41
Ending Subprograms.....	5-41
Looping Subprograms	5-41
Rotating, Mirroring, and Scaling Subprograms (RMS).....	5-42
Ellipses and Spirals.....	5-43
Plane Selection.....	5-43
Programming an Ellipse	5-43
Programming a Spiral.....	5-45

Mold Cycles.....	5-47
Programming a Mold Rotation	5-47
Rotations Around X- and Y- Axes (Small Radius)	5-48
Rotations Around X- and Y- Axes (Large Radius)	5-52
Rotation Around the Z-Axis	5-53
Programming an Elbow Milling Cycle	5-55
Engraving, Repeat, and Mill Cycles	5-60
Engraving Cycle	5-60
Repeat Cycle	5-62
Mill Cycle	5-64
Probing Cycles.....	5-66
Tool Probe Cycles	5-66
Spindle Probe Cycles	5-83

Section 6 - Editing Programs

Activating the Program Editor.....	6-1
The Program Editor Screen.....	6-2
Saving Edits	6-3
Canceling Unsaved Edits	6-3
Deleting a Program Block.....	6-3
Inserting a Program Block.....	6-4
Editing a Program Block.....	6-4
Searching Blocks for Words or Numbers.....	6-4
Scrolling the Program Listing.....	6-4
Paging through the Program Listing	6-5
Jumping to First or Last Block in the Program.....	6-5
Using Comments.....	6-5
Writing a Comment Block	6-5
Commenting Out Existing Blocks	6-5
Canceling a Comment	6-6
Using Block Operations to Edit a Program.....	6-6

Section 7 - Viewing Programs with Draw

Draw Modes.....	7-1
Starting Draw	7-2
Draw Screen Description.....	7-3
Putting Draw in Hold	7-3
Canceling Draw	7-3
Draw Parameters	7-4
Text On or Off.....	7-4
Tool On or Off.....	7-5
Drawing Compensated Moves.....	7-5
Showing Rapid Moves.....	7-6
Setting Grid Line Type.....	7-6
Setting Grid Size.....	7-6
Putting Draw in Motion, Single-Step, or Auto Mode.....	7-7
Automatic Draw Restart.....	7-8
Erasing Display.....	7-8
Running Draw for Selected Blocks	7-8
Adjusting Draw Display	7-10
Fitting the Display to the Viewing Window.....	7-10
Halving Display Size.....	7-10
Doubling Display Size.....	7-10

Scaling the Display by a Factor	7-11
Zooming In	7-11
Erasing Display.....	7-11
Changing Draw Views	7-12
Selecting the View	7-12
Section 8 - Running Programs	
Selecting Programs for Running	8-1
Running a Program One Step at a Time	8-1
Single-Step Mode vs. Motion Mode.....	8-2
Holding or Canceling a Single-Step Run	8-2
Single-Step Execution of Selected Program Blocks	8-2
Switching from Single-Step to Auto	8-3
Auto Program Execution	8-3
Holding or Canceling an Auto Run	8-3
Starting at a Specific Block.....	8-4
Clearing a Halted Program.....	8-4
Using Draw while Running Programs.....	8-5
Parts Counter and Program Timer	8-6
Background Mode	8-7
Section 9 - Program Management	
Program Directory	9-1
Changing the Program Directory Display	9-2
Creating a New Program.....	9-2
Choosing Program Names	9-2
Loading a Program for Running	9-2
Selecting a Program for Editing and Utilities	9-3
Maximizing Program Storage Space	9-3
Program File Utilities	9-3
Displaying Program Blocks (Listing a Program)	9-4
Deleting a Program.....	9-4
Reading Disks in Floppy Drives (Logging to Other Drives).....	9-4
Marking and Unmarking Programs	9-5
Deleting Groups of Programs	9-6
Restoring Programs.....	9-6
Copying Programs to Floppy Disks	9-6
Renaming Programs.....	9-7
Printing Programs.....	9-7
Formatting Floppy Disks.....	9-7
Converting G-Code Programs to CNC Conversational Format	9-8
Checking Disks for Lost Data	9-14
Displaying System Information.....	9-15
Copying Programs from/to Unspecified Locations	9-16
Renaming Programs from/to Unspecified Locations	9-16
Printing from Floppy Drives	9-17
Section 10 - Tool Management	
Tool Page.....	10-1
Entering the Tool Page.....	10-1
Tool Page Description	10-2

Using the Tool Page.....	10-3
Finding Tools by Number	10-3
Changing Tool Page Values	10-3
Clearing a Tool (Whole Row).....	10-4
Clearing a Single Value	10-4
Adjusting a Single Value.....	10-4
Setting Tool-Length Offset.....	10-4
Automatically Setting Tool-Length Offsets from the Tool Page	10-5
Manually Setting Tool-Length Offsets from the Tool Page	10-5
Setting Tool-Length Offset for Ball-End Mills.....	10-5
Fixture Offsets	10-5
Setting RefProg Offset.....	10-6

Section 11 - Communication and DNC

Communication	11-1
Installing the RS-232 Cable.....	11-1
Accessing the Communication Package	11-2
Setting Communication Parameters.....	11-3
Selecting the Communication Port	11-3
Setting the Baud Rate	11-4
Setting Parity	11-4
Setting Data Bits.....	11-4
Setting Stop Bits.....	11-4
Software Settings	11-4
Setting Data Type.....	11-4
Testing the Data Link	11-4
Activating the Test Link Screen.....	11-5
Setting Test Link Display Modes	11-5
Testing the Link	11-6
Clearing the Receive Area.....	11-6
Sending a Program	11-6
Receiving a Program.....	11-7
Setting the Transmission and Receiving Display.....	11-7
Holding Transmission/Receiving Operations.....	11-7
Running in DNC	11-8
Using Data Control (DC) Codes	11-9
Using DC Codes In Receive Mode	11-10
Using DC Codes In Send Mode.....	11-10

Section 12 - Calculators

CNC Calculator Package	12-1
Math Calculator	12-1
Math Calculator Basics.....	12-2
Operations Involving Two Numbers.....	12-3
Math with a Column of Numbers	12-3
Using Parentheses	12-3
Using Additional Functions	12-4
Storing Numbers from the Math Calculator	12-5
Right Triangle Calculator.....	12-5
Activating the Triangle Calculator.....	12-5
Using the Triangle Calculator	12-6
Storing Right Triangle Calculator Results.....	12-6
Hiding the Right Triangle Calculator Screen.....	12-6

Geometry Calculator	12-7
Activating the Geometry Calculator	12-7
Geometry Calculator Screen	12-7
Using the Geometry Calculator.....	12-8
Point Templates.....	12-9
Line Templates.....	12-10
Circle Templates.....	12-11
Deleting Selected Elements	12-11
Deleting All Elements	12-11
Listing All Geometry Elements	12-12
Calculating the Distance between Two Elements.....	12-12
Last Position Recall.....	12-12
Recalling Values into a Program	12-13
Recalling Values from the Math Calculator.....	12-13
Recalling Values from the Right Triangle Calculator	12-14
Recalling Values from the Geometry Calculator	12-15
Recalling Values from One Calculator into Another.....	12-15

Section 13 - Off-line Software

Passwords.....	13-1
Exiting the Software	13-1
Windows Installation (Use Windows Installation Disk)	13-1
Running from Windows	13-2
Setting up the Icon.....	13-2
System Settings	13-3
Maximum Memory Allocated	13-3
Disabled Features	13-3
Using Soft Keys from a Keyboard	13-4
Keypad Equivalent Keyboard Keys	13-4
Editing with a Text Editor.....	13-7

Section 14 - Four-Axis Programming

Axis Types.....	14-1
Rotary Axis Programming Conventions.....	14-2
Non-Synchronous or Synchronous Auxiliary Axis	14-2
Programming Examples	14-3
Example 1: Drill (Sync-Off)	14-3
Example 2: Mill (Sync-On).....	14-5
Example 3: Mill (Sync-On).....	14-6

Section 15 - DXF Converter Feature

Requirements.....	15-1
Off-line Software.....	15-1
Machine Software.....	15-1
Entry to the DXF Converter	15-2
Creating Shapes.....	15-2
Contours.....	15-3
Drilling	15-3
CNC Code.....	15-3
Mouse Operations.....	15-4
DXF Hot Keys	15-5
Toggle Entity Endpoints (ALT + F)	15-5

DXF Soft Keys.....	15-6
Miscellaneous DXF Soft Key, F6.....	15-7
Output Menu Options.....	15-8
Shift X, Shift Y Descriptions.....	15-8
Convert Polyline Description.....	15-9
Display Menu Options.....	15-9
DXF Entities Supported.....	15-10
Drawing Entities Not Supported.....	15-10
Files Created.....	15-11
DXF Example.....	15-11
Unedited Conversational Program Listing.....	15-13
Edited Conversational Tool Path.....	15-14
Edited Conversational Program Listing.....	15-14
Using DXF for Pockets with Islands.....	15-16

Section 16 - CNC Software

Machine Software Installation.....	16-1
Software Option Kit Installation.....	16-1
Procedure.....	16-1
Using Soft Keys from a Keyboard.....	16-2
Keypad Equivalent Keyboard Keys.....	16-2
Making Jog Moves from a Keyboard.....	16-2
Index	Index-1

Section 1 - CNC Programming Concepts

Programs

This manual describes CNC programming and operations for 3000M three-axis systems.

A program is the set of instructions used by the CNC to direct machine movement. Each instruction is called a block and each block executes independently.

Programs are stored in the CNC's memory and accessed from the CNC's Program Directory. You can create, delete, copy, and rename programs in the CNC's Program Directory.

Axis Descriptions

The machine moves along its axes of motion. All movement along an axis is in either a positive or negative direction. Not all machines use the same system for identifying axes. The descriptions here are most commonly used for three axis mills. Refer to **Figure 1-1**.

NOTE: To keep directions straight when programming machine movements, consider tool motion rather than table motion. (When tool motion is positive, table motion in negative, and vice versa.)

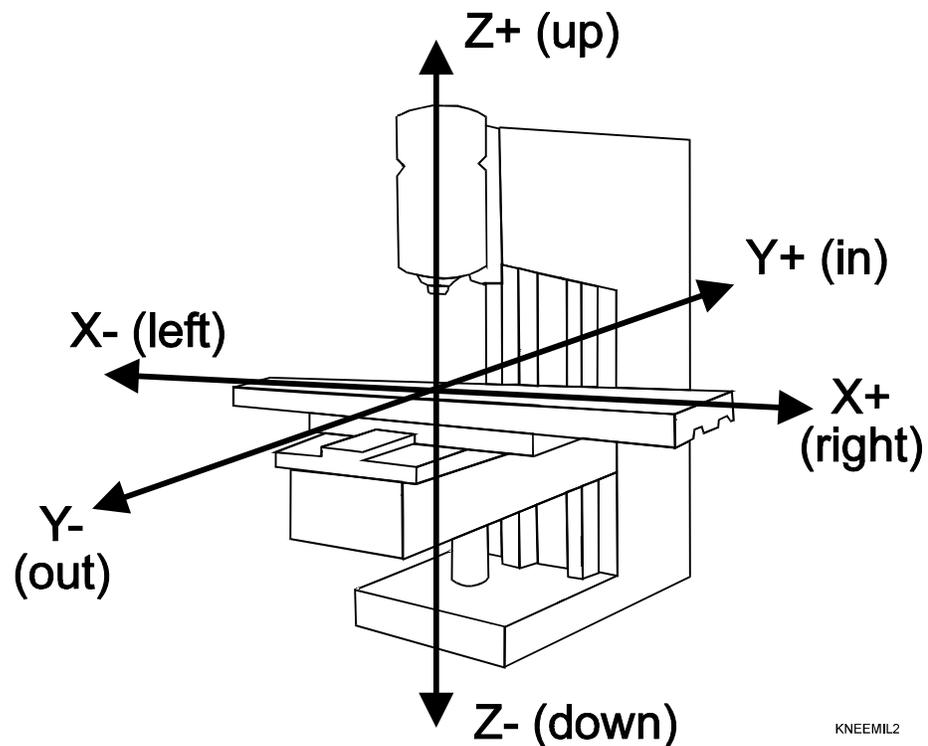


Figure 1-1, Mill Axes of Motion (Tool Motion Orientation)

X Axis

The table moves left and right along the X-axis. Positive motion is table movement to the left (tool, right); negative motion is table movement to the right (tool, left).

Y Axis

The table moves in and out along the Y-axis. Positive motion is table movement out (tool, in); negative motion is table movement in (tool, out).

Z Axis

In the Z-axis, the tool moves up and down on the spindle. Positive motion is tool movement up; negative motion is tool movement down (into the work).

Defining Positions

The intersection of the X, Y, and Z-axes is the reference point that defines most positions. This point is the X0, Y0, and Z0 position. Refer to **Figure 1-2**.

Most positions are identified by X, Y, and Z coordinates. A position two inches left, three inches back, and four inches up has the following coordinates:

X-2.0

Y3.0

Z4.0

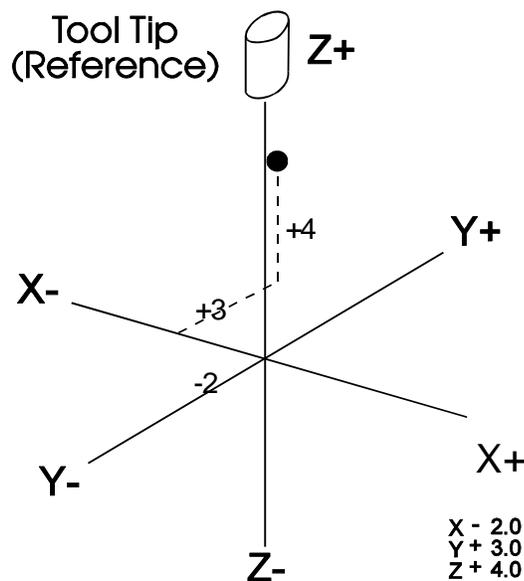
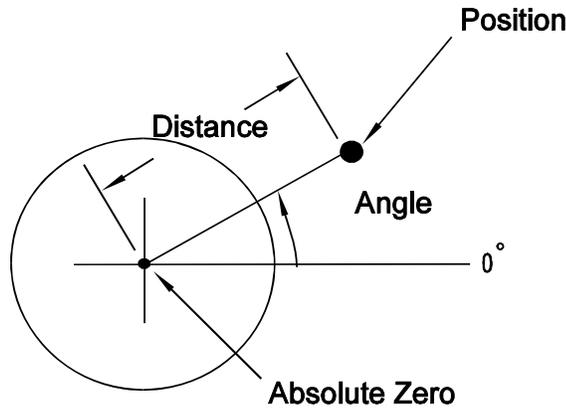


Figure 1-2, Locating Positions

Polar Coordinates

Polar Coordinates define points that lie on the same plane. Polar coordinates use the distance from the origin and an angle to locate points. Refer to **Figure 1-3**.



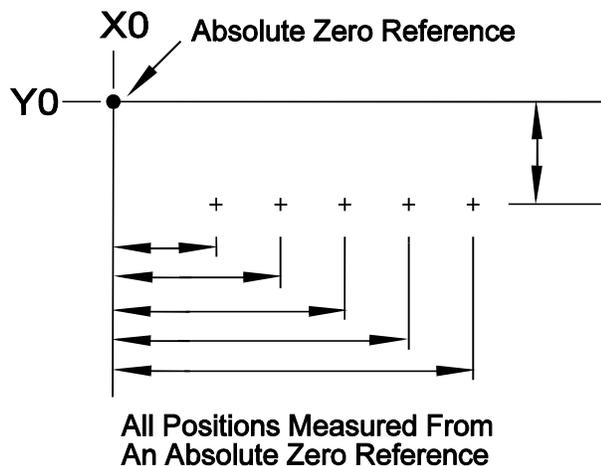
POLAR

Figure 1-3, Polar Coordinate System

Absolute Positioning

In the Absolute Mode, all positions are measured from the Absolute Zero Reference point. Absolute Zero is not a fixed position on the machine, but a point you select. Refer to **Figure 1-4**.

You can set the Absolute Zero Reference point (X0, Y0) anywhere. Usually the Absolute Zero Reference is set at a position that makes it easy use the dimensions from the blueprint. This is also called setting the Part Zero.



ABSOLUTE

Figure 1-4, Absolute Positioning

Incremental Positioning

Measure incremental moves from the machine's present position. This is convenient for performing an operation at regularly spaced intervals.

Refer to **Figure 1-5**.

NOTE: An incremental 0-inch/0-mm move will not make a position change.

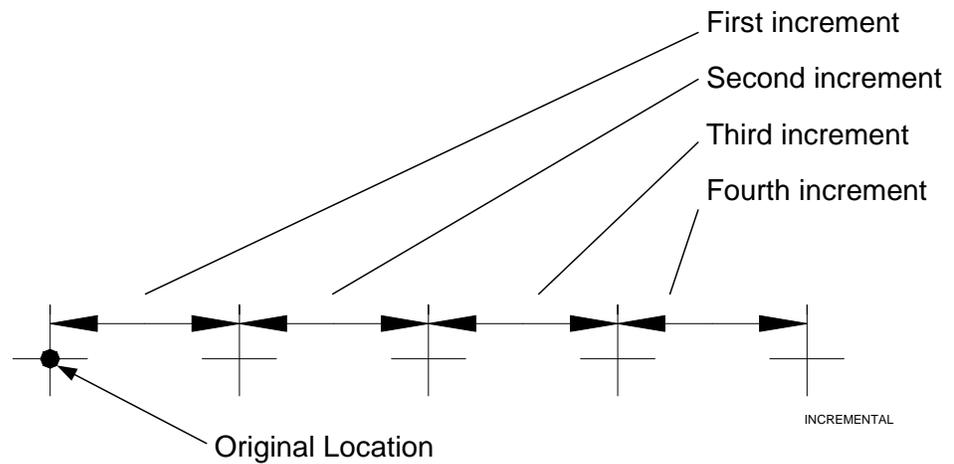


Figure 1-5, Incremental Positioning

Tool-Length Offsets

The operator sets the Z0 position of the quill, from which the CNC applies Tool-Length Offsets. Usually it is the fully retracted position of the quill.

NOTE: For machines without homing, it may be necessary to set machine home to make setting tool-length offsets easier. Either manually or by jogging take the Z-axis close to the top of travel. The servo must be turned on. Press **MDI (F7)**, press **Mill (F5)**, press **More (F7)**, curse down to **Home** and press **ENTER**, press **Z** the z will light up, press **Save**, press **Prev**, press **Exit**, and press **Start**. The Z-axis will change to zero.

Because tools differ in length, Z0 axis (Part Zero) is not set the same way as X0 or Y0. The tool-length offset is the distance from the tip of the tool to the top of the part. Enter a length offset for each tool in the Tool Page. (Refer to “Section 10 - Tool Management.”)

Tool-length offset is the distance from Z0 Tool #0 to the tip of the tool at the part Z0 (usually the surface of the work). Refer to **Figure 1-6**.

With tool-length offsets active, the Z-axis position display reads 0.00 when the active tool moves to Part Zero. Tool-length offsets simplify programming. To move to a position 0.5 inch into the work, program a move to a Z-.5 position.

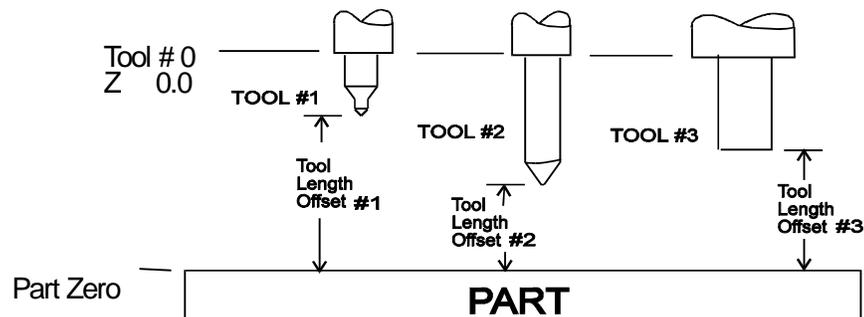


Figure 1-6, Tool-Length Offset

Tool Diameter Compensation

When tool compensation is not active, the CNC positions the tools center on the programmed path. This creates a problem when programming a part profile because the cutting edge is half a diameter away from the path. Use tool diameter compensation to overcome this problem.

NOTE: Be familiar with basic CNC principles before attempting to write compensated moves.

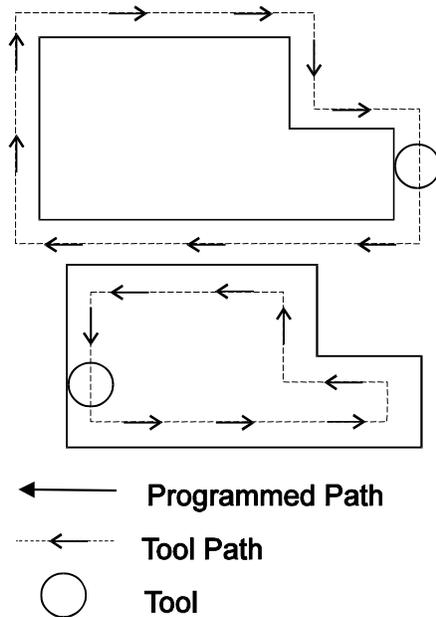
When tool compensation is active, the CNC offsets the tool by half a diameter to position the cutting edge of the tool on the programmed path.

This allows you to program the coordinates along the part profile without adjusting the path to compensate for tool diameter.

Most moves can be compensated. Specify right or left compensation. Right or left refers to the side of the path to which the tool offsets, viewed from behind the tool as it moves.

NOTE: Tool compensation should be used only with lines and arcs.

With left-hand tool compensation active, the tool offsets to the left of the programmed path (looking from behind the tool as it moves). Refer to **Figure 1-7**.



LHCOMP

Figure 1-7, Left-Hand Tool Compensation

With right-hand tool compensation active, the tool offsets to the right of the programmed path (looking from behind the tool as it moves). Refer to **Figure 1-8**.

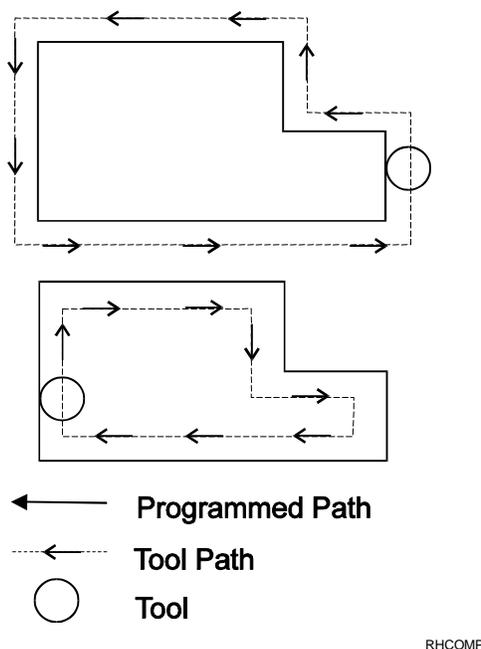


Figure 1-8, Right-Hand Tool Compensation

When the CNC encounters two consecutive, compensated moves, the tool follows the offset path for the first move until it reaches the offset path for the second move. Refer to **Figure 1-9**. The tool may intersect the offset path for the second move, either before or after the endpoint of the first move, depending on the geometry.

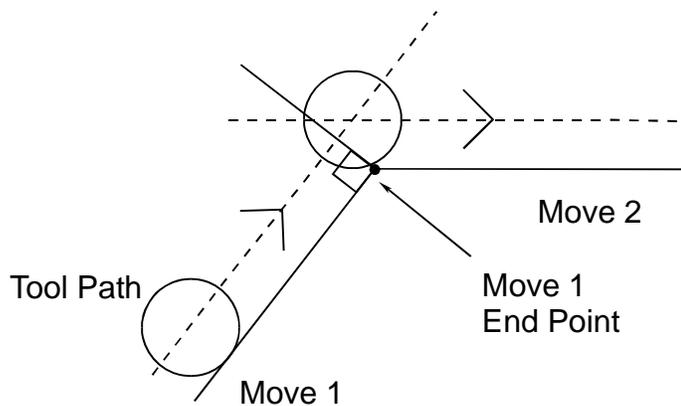


Figure 1-9, Consecutive Compensated Moves

The moves to and from compensated moves are called ramp moves. Ramp moves give the CNC time to position the tool. The ramp move must be at least half the active tool's diameter in length.

At the start of a ramp move, the tool centers on the programmed path. At the end of the ramp move (starting point of the compensated move), the tool centers perpendicular to the starting point, offset by half the tool's diameter. Refer to **Figure 1-10**.

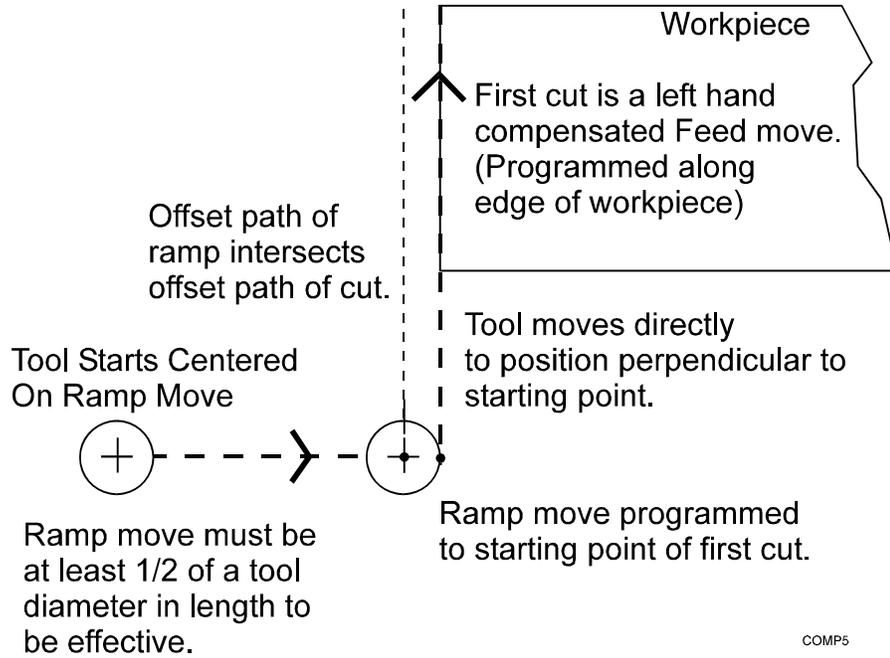
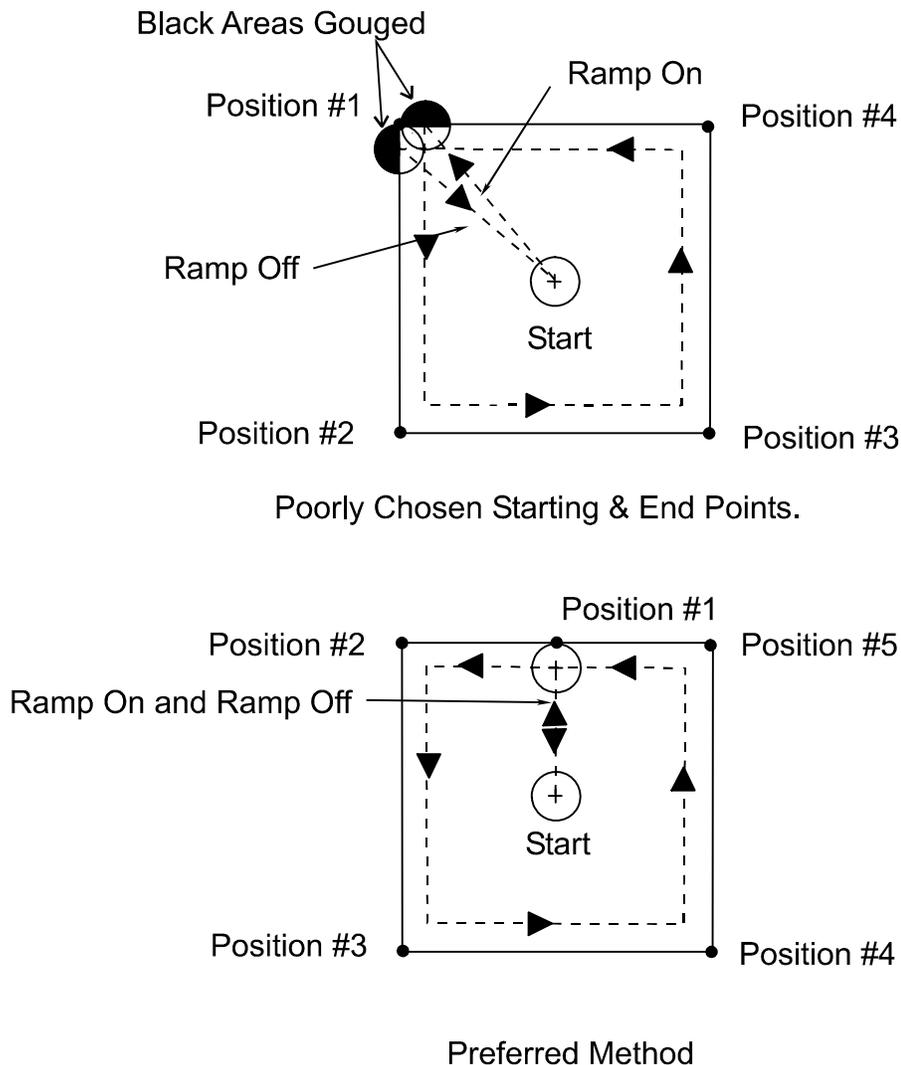


Figure 1-10, Ramping into a Compensated Move

Carefully consider how compensation will affect the position of the tool at the start and end of a move.

When a compensated move starts and stops in a corner, the tool gouges the work because the tool offsets to a position perpendicular to the endpoint. Begin ramp moves at the side to avoid gouging the workpiece. Refer to **Figure 1-11**.

NOTE: Use canned cycles to cut profiles and pockets, when possible. The CNC automatically selects ramp On/Off positions in a canned cycle.



COMP4

Figure 1-11, Ramp On/Off Choices for Milling Inside a Square

Using Tool Diameter Compensation and Length Offsets with Ball-End Mills

When using a ball-end mill to cut contoured surfaces, use tool diameter compensation and tool-length offset together, if at all. Unlike an end cutter, the tool-length offset for a ball-end mill is not set to the tip of the tool.

Set the tool-length offset for a ball-end mill half the tool's diameter back from the tip. Refer to **Figure 1-12**. For more details on how to set tool-length offsets, refer to "Section 10 - Tool Management."

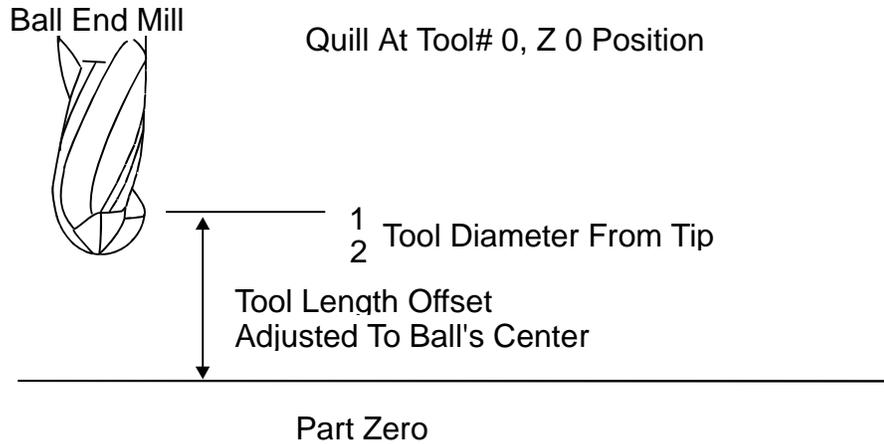


Figure 1-12, Setting Tool-Length Offset for Ball End Mill

Angle Measurement

Measure angles from the 3 o'clock position (0 degrees). Positive angles rotate in a counterclockwise direction; negative angles rotate in a clockwise direction. Refer to **Figure 1-13**.

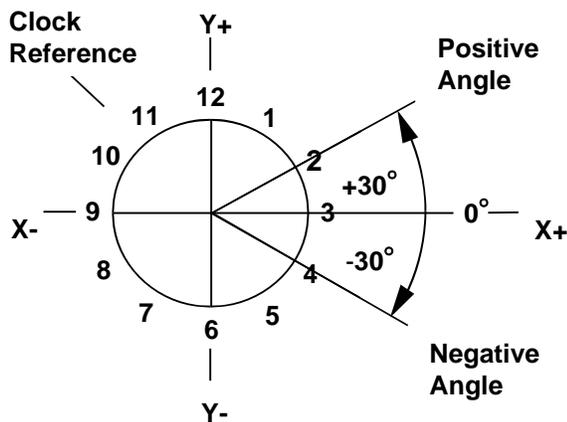


Figure 1-13, Absolute Angle Measurement

Corner Rounding

Corner rounding permits the operator to blend the intersection of consecutive moves.

To activate corner rounding, the operator keys a radius value (positive) into the **CornerRad** field of the first move. When the program runs, it blends the endpoint of the first move with the starting point of the second. The blend starts where the radius is tangent to the first move, and extends to where the radius is tangent to the second.

Use corner rounding between two lines or two arcs. Also use corner rounding between non-tangent line and arc moves.

Line-to-Line Corner Rounding

When the first move contains a **CornerRad** value, the CNC automatically finds the radius center and the tangent points necessary to calculate the tool path. The resulting tool path follows the solid line. Refer to **Figure 1-14**.

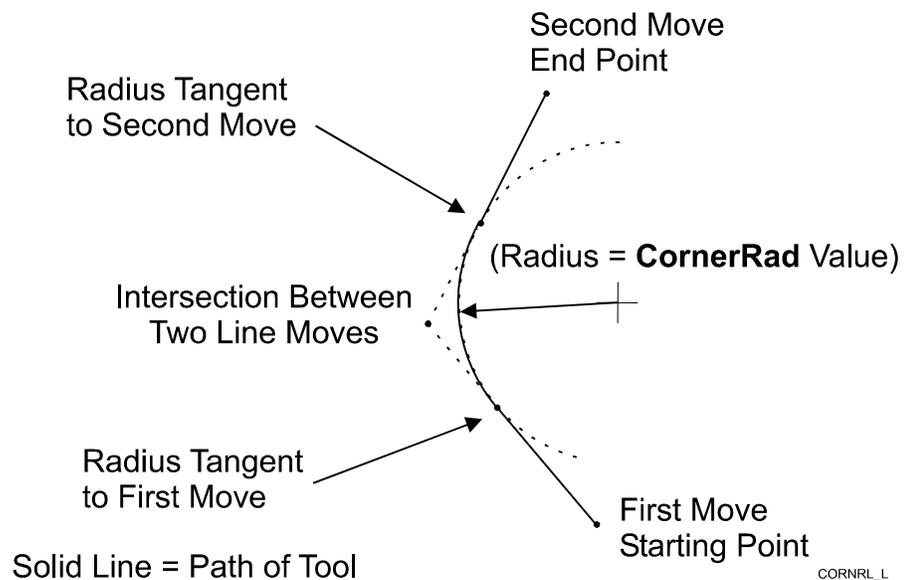
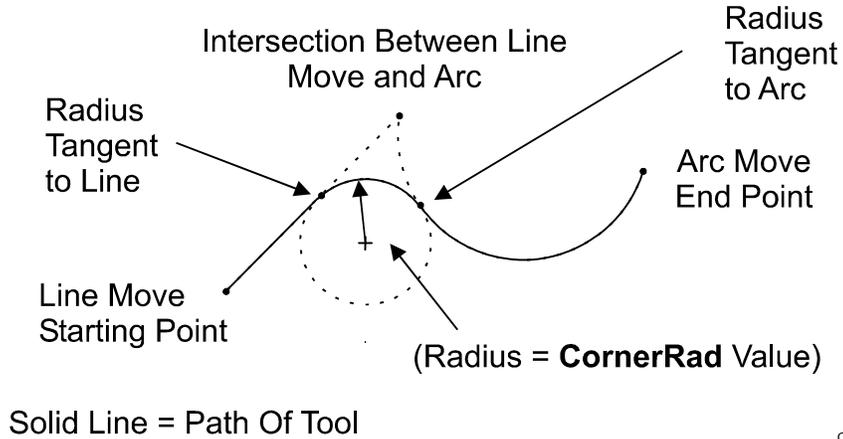


Figure 1-14, Line-to-Line Corner Rounding

Line-to-Arc Corner Rounding

When the first move contains a **CornerRad** value, the CNC automatically finds the radius center and the tangent points necessary to calculate the tool path. The resulting tool path follows the solid line. Refer to **Figure 1-15**.

NOTE: If the line move is already tangent to the arc move, the CNC ignores corner rounding.

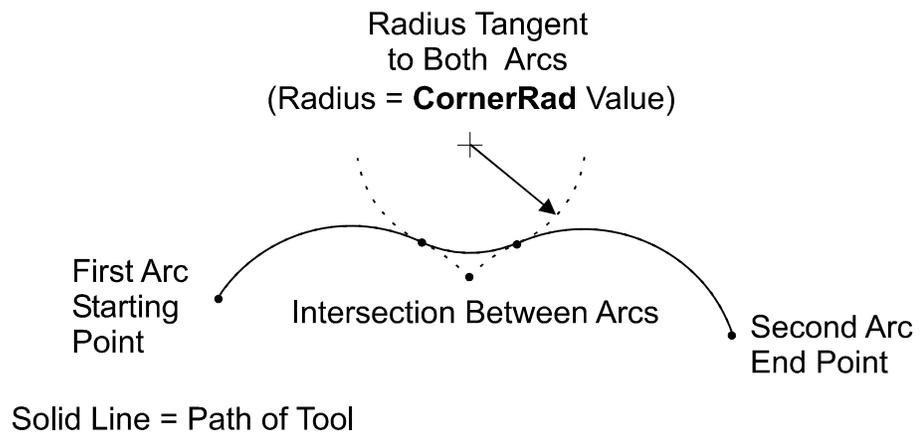


CORNRL A

Figure 1-15, Line-to-Arc Corner Rounding

Arc-to-Arc Corner Rounding

When a **CornerRad** value is programmed into the first move, the CNC automatically finds the radius center and the tangent points necessary to calculate the tool path. The resulting tool path follows the solid line. Refer to **Figure 1-16**.



CORNRA A

Figure 1-16, Arc-to-Arc Corner Rounding

Chamfering

Chamfer between two consecutive line moves. A chamfer starts at a specified distance before the endpoint of the first move and ends the same distance from the starting point of the second move. To program a chamfer move, enter a negative value into the **CornerRad** field of the first move. The entered value is the chamfer distance. The resulting tool path follows the solid line. Refer to **Figure 1-17**.

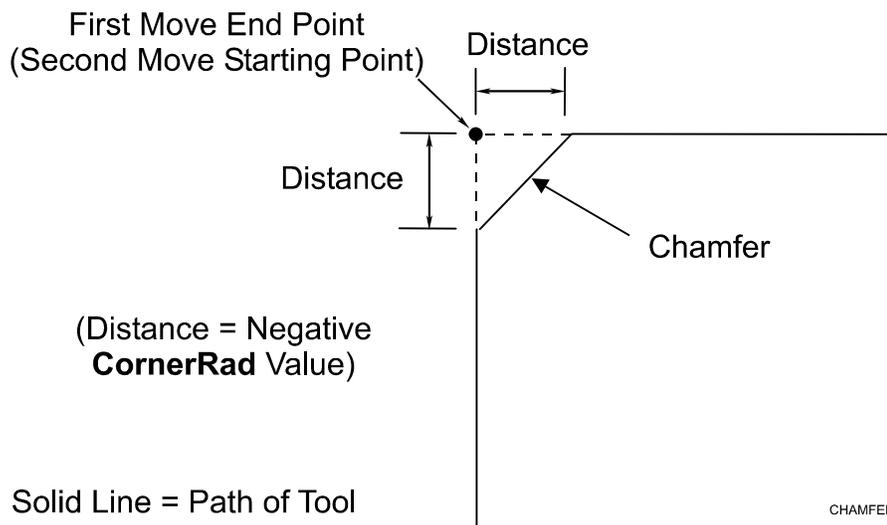


Figure 1-17, Chamfering

Plane Selection

Circular moves and tool diameter compensation are confined to the plane you select (XY, XZ, or YZ).

CAUTION: A plane viewed from the wrong side causes arc directions, angle references, and axis signs to appear reversed.

Refer to **Figure 1-18** for a description of the three available planes.

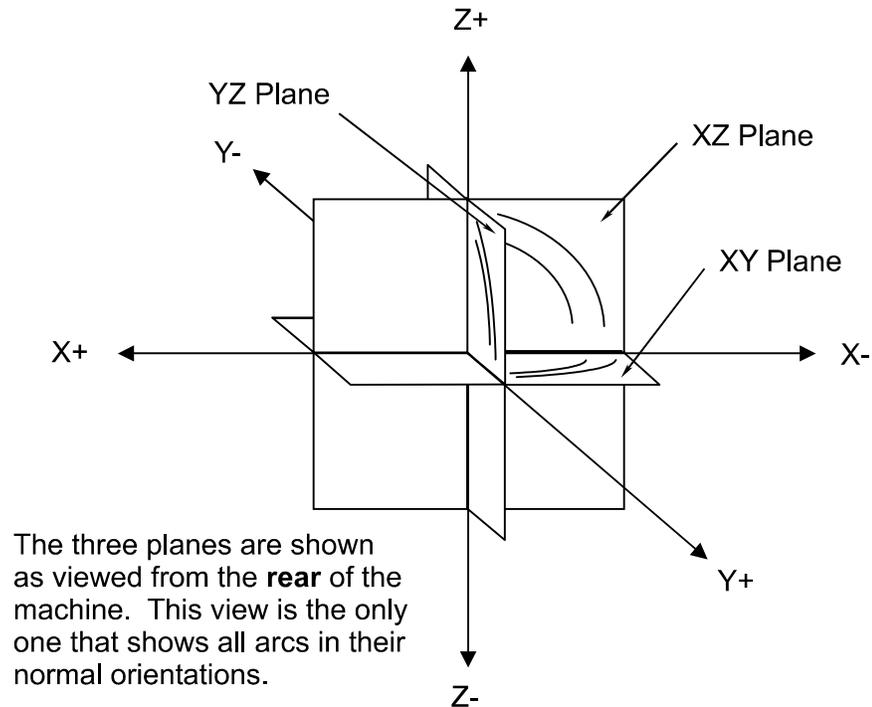
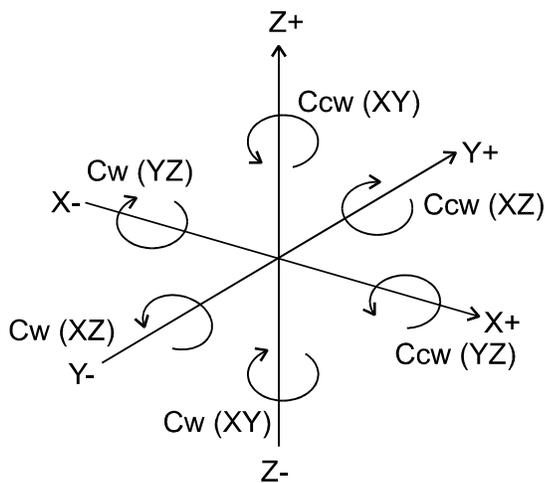


Figure 1-18, Plane Identification

Arc Direction

The standard rule is to view arc direction for a plane from the positive toward the negative direction along the unused axis. From this viewpoint clockwise (Cw) and counterclockwise (Ccw) arc directions can be determined. For example, in the XY plane, you view along the Z-axis, from Z+ toward Z-, to determine Cw/Ccw directions. The Cw/Ccw arc directions for each plane are shown in **Figure 1-19**.



ARCDIR

Figure 1-19, Clockwise and Counterclockwise Arc Directions

Section 2 - CNC Console and Software Basics

Console

The CNC console consists of a 12.1" color, flat-panel Liquid Crystal Display (LCD), a keypad to the right of the monitor, and soft keys below the monitor. Refer to **Figure 2-1**.

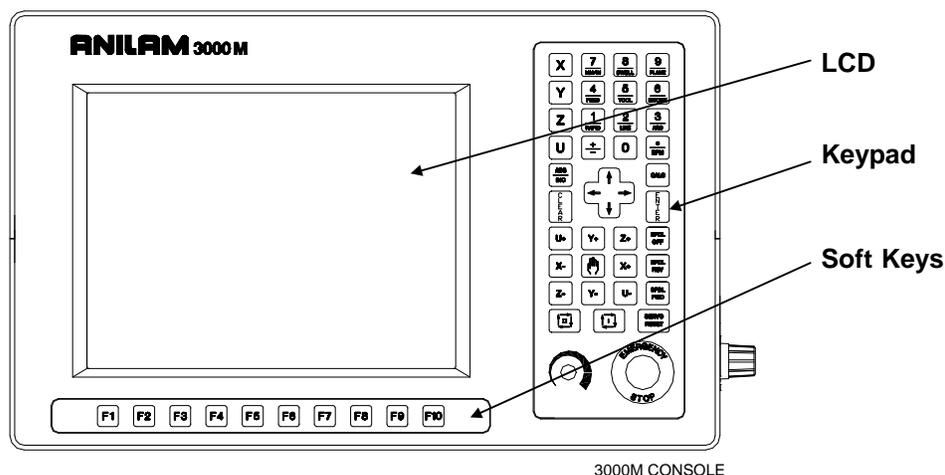


Figure 2-1, CNC Console

Keypad

Refer to **Figure 2-2**. The keypad to the right of the monitor has four types of keys:

- Programming Hot Keys
- Editing Keys
- Manual Operation Keys
- Operator Keys

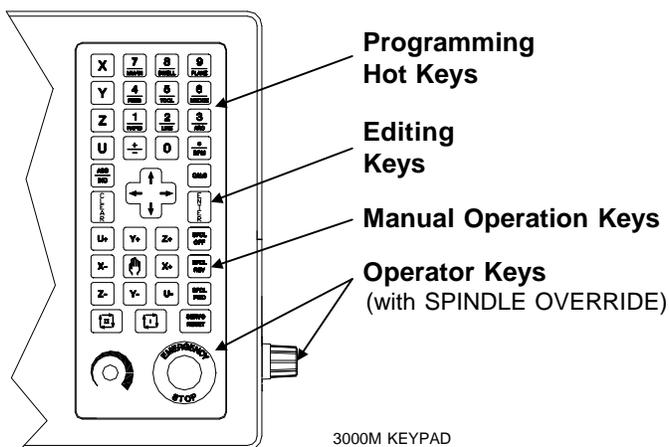


Figure 2-2, Keypad

Programming Hot Keys

Programming hot keys allow you to enter position coordinates and provide quick access to functions that speed up programming. They are active in the Edit and Manual Mode. Refer to **Table 2-1**.

Table 2-1, Programming - Hot keys

Label or Name	Key Face	Purpose
X		Selects X-axis for position inputs.
Y		Selects Y-axis for position inputs.
Z		Selects Z-axis for position inputs.
U		Selects U-axis for position inputs.
ABS/INC		Switches CNC between Absolute and Incremental Modes.
0		Zero / Switches comment asterisk in edit mode. Switches resolution display between program and Dist. To Go.
1/RAPID		One / Hot key for programming a Rapid move.
2/LINE		Two / Hot key for programming a Line move.
3/ARC		Three / Hot key for programming an Arc.
4/FEED		Four / Hot key for changing feedrate.
5/TOOL		Five / Hot key for programming a tool.
6/MCODE		Six / Hot key for programming an M-code.
7/UNIT		Seven / Hot key for switching between inches (Inch) and millimeters (mm).
8/DWELL		Eight / Hot key for programming a Dwell.
9/PLANE		Nine / Hot key for selecting a plane.
+/-		Sign change / Toggle hot key.
DECIMAL/RPM		Decimal point / Hot key for programming the spindle RPM.

(Continued...)

Table 2-1, Programming - Hot keys (Continued)

Label or Name	Key Face	Purpose
CALC		Calculators / Hot key to display the Select Type of Calculator: pop-up menu. See Figure 12-1, Calculator Selection Menu.

Editing Keys

Editing keys allow you to edit program blocks. These keys are located below the Programming Hot Keys. Refer to **Table 2-2.**

Table 2-2, Editing Keys

Label or Name	Key Face	Purpose
CLEAR		Clears the selected messages, values, commands, and program blocks.
ARROW		Allows you to move highlight bars and cursor around the screen.
ENTER		Selects blocks for editing, activates menu selections, activates number entry, or presets XYZ positions.

Manual Operation Keys

Manual Operation Keys allow you to control machine movements manually. These keys are located below the Editing Keys. Refer to **Table 2-3.**

Table 2-3, Manual Operation Keys

Label or Name	Key Face	Purpose
JOG		Cycles the CNC through manual movement modes (JOG: RAPID, JOG: FEED, JOG: 100, JOG: 10, JOG: 1).
U+		Manually moves machine in positive U direction.
U-		Manually moves machine in negative U direction.
Z+		Manually moves machine in positive Z direction.
Z-		Manually moves machine in negative Z direction.
Y+		Manually moves machine in positive Y direction.
Y-		Manually moves machine in negative Y direction.

(Continued...)

Table 2-3, Manual Operation Keys (Continued)

Label or Name	Key Face	Purpose
X+		Manually moves machine in positive X direction.
X-		Manually moves machine in negative X direction.
SERVO RESET		Activates servo motors.
SPINDLE FORWARD		Starts spindle in a clockwise direction (viewed from the top of the motor). Optional.
SPINDLE REVERSE		Starts spindle in a counterclockwise direction (viewed from the top of the motor). Optional.
SPINDLE OFF		Stops the spindle.

Operator Keys

Operator Keys allow you to control machine movements manually. These keys are located below the Manual Operation Keys and on the right side panel of the CNC console. Refer to **Table 2-4**.

Table 2-4, Operator Keys

Label or Name	Key Face	Purpose
FEEDRATE OVERRIDE		Overrides the feed and/or rapid rate of the axes in Manual, Auto, and Single Step modes. It is a 13-position rotary switch, which ranges from 0 to 120 percent. (Each increment adjusts the feedback override by 10%). NOTE: The override range for rapid rate is 100%. The CNC will not exceed the maximum rapid rate.
SPINDLE OVERRIDE		Typically on the right side panel of the CNC console. Overrides the programmed spindle RPM rate. It is a 13-position rotary switch that ranges from 40 to 160 percent. (Each increment adjusts the spindle override by 10%.) This feature can be used only on machines with programmable spindles.
E-STOP		The red emergency stop button disconnects the machine's servos, stopping the spindle and all machine movement.

(Continued...)

Table 2-4, Operator Keys (Continued)

Label or Name	Key Face	Purpose
START		The green START key initiates all machine moves except jog.
HOLD		The red HOLD key pauses any running program or programmed move. (Press START to resume.)

Soft Keys (F1) to (F10)

The soft keys (function keys) located just below the monitor are labeled (F1) through (F10). Soft key functions are not hard-wired but change in various modes. Active soft keys are labeled on screen. Inactive soft keys remain blank.

Off-line Keyboard (Optional)

The CNC supports most standard PC keyboards. Refer to “Section 13 - Off-line Software.” All keypad inputs except **E-STOP** and **SERVO RESET** are available on a keyboard.

Software Basics

The CNC’s screens change as different modes are activated. Basic procedures and features of the software remain the same, regardless of the CNC’s mode.

Pop-up Menus

Pop-up menus are temporary menus that allow you to make additional selections. Refer to **Figure 2-3**. Each pop-up menu contains a highlight bar. The **ARROWS** move the highlight bar up and down the menu. Press **ENTER** to activate the highlighted selection. Press the soft key that activated it or press **CLEAR** to close a pop-up menu.

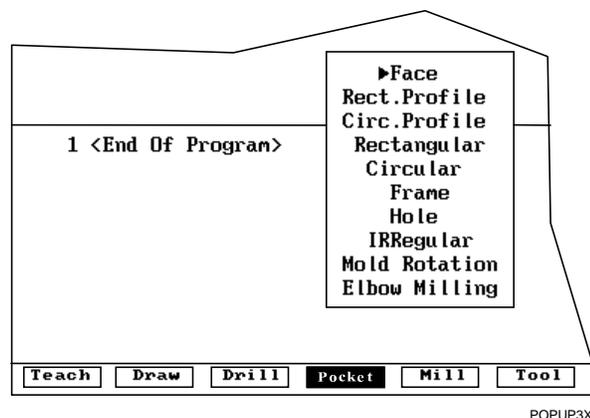


Figure 2-3, Pop-up Menu

Screen Saver

After a set period of inactivity, the CNC's screen dims to preserve the LCD. Press any key to restore the CNC to a ready status.

Switching Selections with the Toggle Key

Press the +/- key (toggle key) to toggle between options. (For example: Cw/Ccw, ToolComp.) This key also produces the negative symbol.

Clear Key

Press **CLEAR** to clear an entry in an entry field, a line from a program, or a message on the message line.

Operator Prompts

The CNC prompts when it requires specific information. When the CNC prompts for a text entry, use the ASCII Chart to enter ASCII characters from the keypad. Refer to "ASCII Chart" for more information. Enter numbers from the keypad.

ASCII Chart

When the CNC prompts for a text entry, the **ASCII (F2)** soft key is displayed. Press **ASCII (F2)** to toggle the ASCII Chart On and Off. The ASCII Chart allows you to enter text from the keypad. Refer to **Figure 2-4**.

A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
P	Q	R	S	T	U	V	W	X	Y	Z	~	!	@	#
\$	%	^	&	*	()	-	+	_	=	{	}	[]
	\	:	;	"	'	<	>	,	.	?	/	~	`	

ASCII

Figure 2-4, ASCII Chart Pop-up

Using the **ARROW** keys, select the character you want to enter. Press the **ENTER** key to display the character in the prompt area. Turn off the ASCII Chart, and press **ENTER** to complete the text entry.

Cursor and Highlight Functions

The CNC uses either a cursor or highlight to mark an item for selection or editing.

The highlight is displayed in the Edit Mode, Program Directory, Manual mode, and ASCII Chart. Use the **ARROWS** to move the highlight. The software highlights a selected item in a menu or window. Selected items can be activated or changed in some way. For instance, you highlight a program block in Edit Mode to edit it. You highlight an entry field label in a Graphic Menu to enter a value or switch between the available choices.

The cursor is displayed when the Tool Page activates. The cursor is a white underline that indicates where letters and numbers will be inserted.

Entering Text

Use the ASCII Chart or a keyboard to enter text.

To enter text using the **ASCII (F2)** chart:

1. Press **ASCII (F2)**. ASCII Chart activates.
2. Highlight desired character.
3. Press **ENTER**. The CNC displays the selected character at the cursor.
4. Select all required characters.
5. Press **ASCII (F2)** to close the ASCII Chart.

Typing Over and Inserting Letters and Numbers

The ASCII Chart has two text entry modes: Typeover (default) and Insert.

In the Typeover Mode, new characters replace characters marked by the cursor.

In the Insert Mode, new characters appear at the cursor and existing characters move to the right. When the Insert Mode is active, **Ins (F3)** highlights.

To put the CNC in the Insert Mode:

1. When the CNC prompts for a name, press **Ins (F3)**. **Ins (F3)** highlights.

Deleting Characters

To delete characters:

1. With the ASCII Chart active, move the cursor to underline the character being deleted.
2. Press **Del (F4)**. The selected character disappears.

NOTE: Press **CLEAR** to delete an entire word.

Messages/Error Messages

Messages generated by the CNC appear in the message area, present in all program-running modes. Refer to **Figure 2-5**. When the CNC generates more than one message, the message with the highest priority is displayed in the message area. Lower-priority messages remain in memory. The on-screen **MESSAGE** label highlights when pending messages remain in memory. There are two ways to review pending messages:

- Press **CLEAR**. The CNC clears the current message and displays the next message.
- From the Manual screen, press **MESSAGE (F1)**. Displayed messages appear in the Program Area of the screen.

Some messages are advisory; others stop CNC operation. For those that stop operation, you must put the CNC in the Manual Mode to correct the problem, and then clear the message.

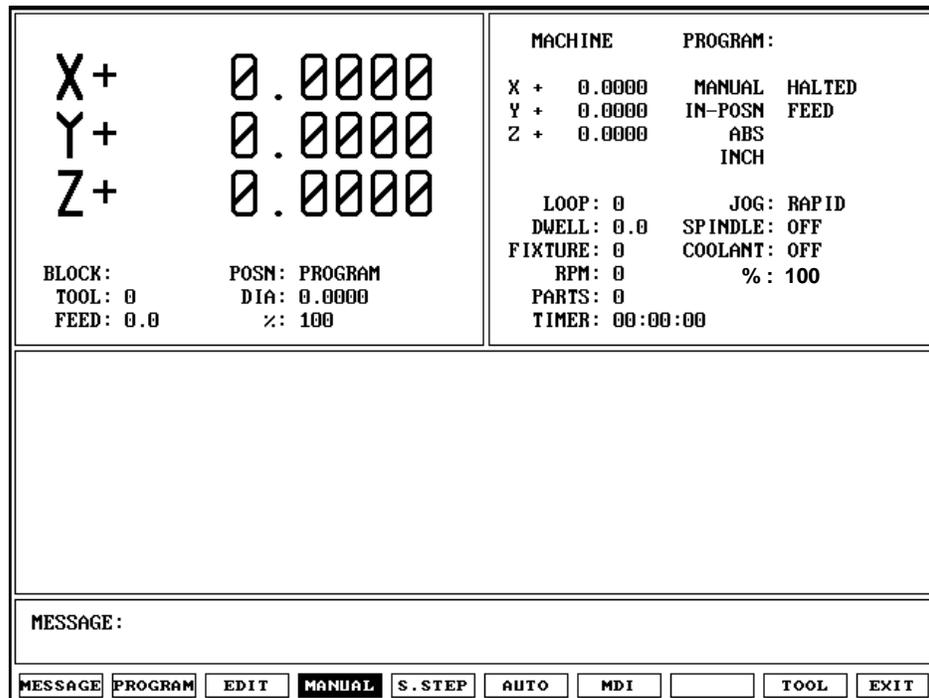


Figure 2-5, Messages Display

Section 3 - Manual Operation and Machine Setup

Powering On the CNC

To power on the CNC:

1. Press the power switch on the CNC cabinet. The CNC activates the startup screen and prompts you to **Press F10 to continue.**
2. Press **CONT. (F10)** to display the **Software Options** menu.
3. Highlight **CNC Control**, and press **ENTER** to activate Manual Mode.

Shutting Down the CNC

To shut down the CNC:

1. Press **E-STOP** to disengage the Servos and activate Manual Mode.
2. Press **EXIT (F10)** to display the **Software Options** menu.
3. Press the power switch on the CNC cabinet to power down the CNC.

Emergency Stop (E-STOP)

Use **E-STOP** to take all axes and spindle servos offline and halt machine movement.

To reset **E-STOP**:

1. Rotate the **E-STOP** switch in the direction of the arrows. The switch clicks when reset. The CNC does not automatically reactivate the servos after an **E-STOP** reset. You must reset the servos to move the machine or start the spindle.
2. Press **SERVO RESET** to reset the servos.

Performing an Emergency Stop

To perform an emergency stop:

1. Press **E-STOP** to disengage the servos and activate Manual Mode.

NOTE: You cannot activate E-STOP using an off-line keyboard.
--

Activating/Resetting the Servos

For safety reasons, the CNC powers up with the servo motors off. With the servos turned off, the CNC cannot move the machine and the spindle will not start. The CNC displays the **SERVO OFF!** message when the servos are off. Refer to “Section 2 - CNC Console and Software Basics” for reviewing messages.

During operation, press **E-STOP** to turn off the servos. Servos automatically turn off if the machine attempts to power past a limit switch.

To reset the servos:

1. After a limit switch has been tripped, the servos turn off. Manually reposition the machine inside its range of travel.
2. Press **E-STOP**.
3. Rotate the switch in the direction of the arrows to reset **E-STOP**. The switch clicks when reset.
4. Press **SERVO RESET** to reset the Servos.

NOTE: You cannot activate the servos using an off-line keyboard.

(Re-)Starting the Spindle

SPINDLE FORWARD and **SPINDLE REVERSE** do not work if **E-STOP** is active or if the servos go off-line.

To (re-)start the spindle:

1. Reset **E-STOP**.
2. Press **SERVO RESET**.
3. Press either **SPINDLE FORWARD** or **SPINDLE REVERSE**, as required.

Manual Mode Screen

The Manual Mode screen is the main CNC screen. All other operating screens activate from the Manual Mode screen. In Manual Mode, the **MANUAL (F4)** soft key label highlights. Refer to **Figure 3-1**.

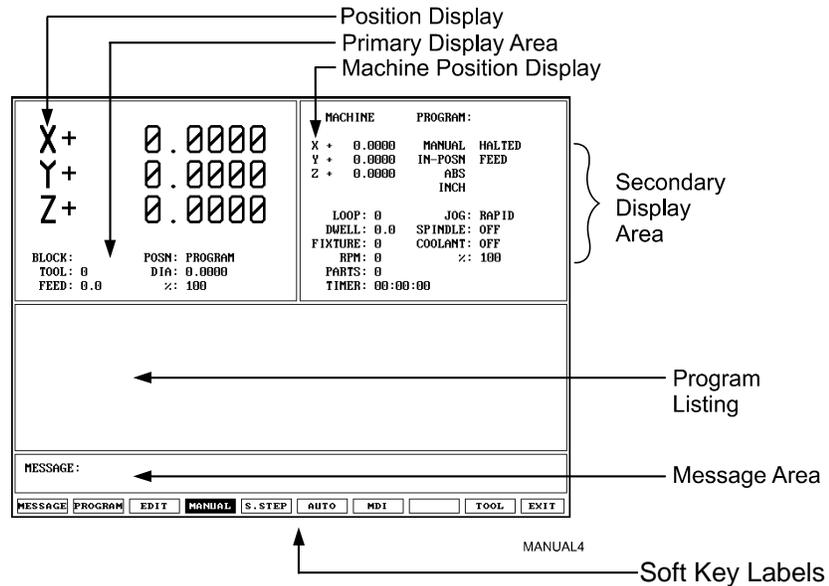


Figure 3-1, Manual Mode Screen

The Manual Mode screen features:

- Position Display** Displays X, Y, and Z position coordinates.
- Machine Position Display** Displays the axis positions in reference to Machine Home. (Not shown when Draw activates.)
- Secondary Display Area** Displays essential operating information.
- Primary Display Area** Displays additional operating information. (Not shown when Draw activates.)
- Message Area** Displays messages, prompts, and reminders.
- Soft Key Labels** Identify the function of the soft key directly underneath. Labels change from screen to screen. A highlighted label indicates an active mode.
- Program Listing** Displays program blocks as they run in Auto or S.Step Mode.

Primary Display Area Labels

BLOCK:	Current program block number.
TOOL:	Active tool.
FEED:	Current feedrate.
POSN:	Position Display Mode (Program or Distance to Go).
DIA:	Active tool diameter.
%:	Feedrate override setting (0% to 120% for Feed moves, 0% to 100% for Rapid moves).

Secondary Display Area Labels

PROGRAM	Name of selected program.
MANUAL/AUTO/S.STEP	Current operating mode.
IN-POSN	Indicates whether the machine has reached target (in-posn) or not.
ABS/INC	Current positioning mode.
INCH/MM	Current units mode.
HALTED/*HALTED/ RUNNING	HALTED (without asterisk) indicates machine is in a programmed hold, or has completed its program. *HALTED (with asterisk) indicates hold was activated by an event, or HOLD was pressed. RUNNING indicates normal program run.
FEED/RAPID/ARC:	Current move mode.
LOOP:	Number of loops remaining (when running a subprogram that has loops).
DWELL:	Seconds remaining in a dwell.
FIXTURE:	Active fixture offset number.
RPM:	Spindle RPM (Optional). May display programmed RPM or actual RPM. Refer to builder's documentation for details.
%:	Spindle override setting (40% to 160%) Optional.
JOG:	Current jog mode.
SPINDLE:	Spindle status (FWD/REV/OFF). Optional.
COOLANT:	Coolant status. Optional.

- PARTS:** Counts the number of successfully completed parts. (Increments by 1 every time the CNC encounters **EndMain** in a program run.) The counter resets to zero when you start a new program. (Refer to “Section 8 - Running Programs.”)
- TIMER:** Total program run time from **START** to **EndMain** execution. If the CNC holds, the counter pauses until the program restarts. The counter resets to **0** (zero) when you start a new program. (Refer to “Section 8 - Running Programs.”)

Position Display

Refer to **Figure 3-2**. The **POSN:** option sets the CNC to display machine position in one of two ways:

- Program** Position Display shows the programmed position.
- Distance To Go** Position Display shows the remaining distance to the commanded position. (**DISTTOGO**)

To switch the **POSN:** setting:

1. In Manual, S. Step, or Auto Mode, press **0** (zero) to switch the setting between **PROGRAM** and **DISTTOGO**.

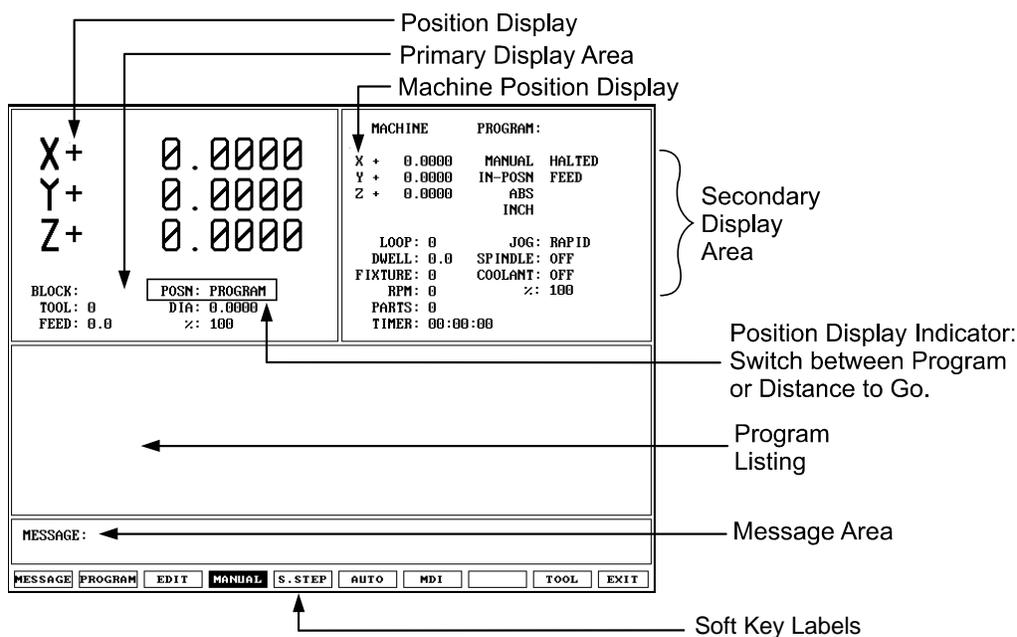


Figure 3-2, Position Display Options

Manual Machine Operation

Two types of manual operation are available:

- **Manual Mode** Digital Readout. Use handles to move the machine. Servos deactivate.
- **Auto Mode** Control machine using the keypad. Servos move the machine.

Manual Mode

In DRO Mode, the CNC acts as a digital readout and you use handles to move the machine.

To activate Manual Mode:

1. Press **E-STOP**.
2. Locate the **MANUAL/AUTO** switch, usually located on the back of the console.
3. Set the switch to **MANUAL**. Servos and spindles disconnect.
4. Reset **E-STOP**.
5. Press **SERVO RESET** to reactivate the spindle.

Auto Mode

To activate Auto Mode:

1. Press **E-STOP**.
2. Locate the **MANUAL/AUTO** switch and move the switch to **AUTO**.
3. Reset **E-STOP**.
4. Press **SERVO RESET**. The servos reactivate and the CNC now operates the machine.

Mode Settings

The operator controls every aspect of the CNC's operation. Settings that remain active for more than one operation or event are said to be modal. These settings remain active until changed or turned off.

Many of the CNC's features are modal. For example, move type (Rapid or Feed), feedrate (IPM), units (Inch or MM) or ABS/INCR.

The operator should make all of the mode adjustments required for the machine to move properly before making a manual move. Modes set from the Manual screen remain active until changed.

Refer to **Table 3-1** for the modes/settings from the Manual screen.

Table 3-1, Manual Mode Settings

Mode/Setting Name	Choices
Position Mode	Absolute/Incremental
Units Mode	Inch/Millimeter
Move Mode	Rapid/Feedrate
Plane	XY/YZ/XZ
Active Tool	Tool Diameter Compensation and Tool-Length Offsets

The operator can use the Manual screen to program the locations of the Absolute Zero Reference (X0, Y0) and the tool change position (Z0, Tool #0).

The following four types of moves are available in Manual Mode:

- Jog (Conventional)
- Jog (Continuous)
- One Shot Move
- Manual Data Input (MDI)

Activating Manual Mode Rapid or Feed

Press **JOG** to switch the Jog Mode. Two Jog Modes (Rapid and Feed) are CNC move modes. The CNC displays the active Rapid/Feed Mode in the Secondary Display Area.

Setting a Feedrate

You can change the feedrate at any time. When **FEEDRATE OVERRIDE** is set to 100%, the CNC executes Feed moves at the specified feedrate.

Adjust the active feedrate with **FEEDRATE OVERRIDE**. Each click of **FEEDRATE OVERRIDE** adjusts the feedrate by an increment of ten. **FEEDRATE OVERRIDE** has a range of 0% to 120%. Align the knob pointer with the white stripe to set the override to 100% (no override).

To change the active feedrate:

1. In Manual Mode, press the **4/FEED** hot key. The CNC prompts for a value.
2. Enter the desired feedrate and press **Save (F10)** or **ENTER** to return to the Manual screen.
3. Press **START** to activate the feedrate change.

– or –

Press **Manual (F4)** to cancel the feedrate change.

<p>CAUTION: If the CNC is shut down, then powered up again, the manufacturer's setup file will reload the CNC's default feedrate.</p>
--

Adjusting Rapid Move Speed

Use **FEEDRATE OVERRIDE** to adjust the Rapid move speed. Every click of **FEEDRATE OVERRIDE** adjusts the Rapid move speed by 10% of the default speed, with a range of 0% to 100%. Align the knob pointer with the white stripe on the console to set the override to 100% (no override).

Overriding the Programmed Spindle RPM

You can change the programmed spindle RPM at any time.

Adjust the programmed RPM with **SPINDLE OVERRIDE**. Refer to **Table 2-4, Operator Keys**. Each click of **SPINDLE OVERRIDE** adjusts the spindle speed by 10%. The **SPINDLE OVERRIDE** range is 40% to 160%. Align the knob pointer with the white stripe to set the override to 100% (no override).

Absolute/Incremental Modes

The CNC recognizes both absolute and incremental positions and can switch between the two modes at any time. In the Absolute Mode, the CNC measures all points from Absolute Zero. In the Incremental Mode, the CNC measures the next move from its present position. The active position mode is displayed in the Secondary Display Area.

- ABS** indicates Absolute Mode
- INC** indicates Incremental Mode

To activate the Absolute or Incremental Mode:

1. Put the CNC in the Manual Mode.
2. Press **ABS/INC** to toggle the setting.

Inch/MM Modes

The CNC uses two units of measurement, Inch and Millimeter. In Inch Mode, the CNC counts in inches. In Millimeter Mode, the CNC counts in millimeters.

Change the unit's mode from the Manual screen or in a program.

The CNC displays the active units mode in the Secondary Display Area.

To activate the Inch or Millimeter Mode:

1. With the Manual screen active, press **7/UNIT** to change the active Units Mode (Inch/mm). The Secondary Status Area displays the current mode.

Setting Absolute Zero

Absolute Zero is the point the CNC recognizes as X0, Y0 in the Absolute Mode. The CNC measures all absolute XY positions from this point.

The CNC uses a floating Absolute Zero. You can locate Absolute Zero anywhere within or beyond the machine's range of travel.

At power-up, the current machine location becomes Absolute Zero (X0, Y0, and Z0). At power down, Absolute Zero is lost. Setting Absolute Zero to a location on the part is called setting Part Zero.

NOTE: For machines without homing, it may be necessary to set machine home to make setting tool-length offsets easier. Either manually or by jogging take the Z-axis close to the top of travel. The servo must be turned on. Press **MDI (F7)**, press **Mill (F5)**, press **More (F7)**, curse down to **Home** and press **ENTER**, press **Z** the z will light up, press **Save**, press **Prev**, press **Exit**, and press **Start**. The Z-axis will change to zero.

NOTE: The active tool-length offset determines the Z-Axis Part Zero (Z0).

NOTE: Use a Home function to define a permanent Absolute Zero (Machine Home).

Defining Absolute Zero in X and Y Axes

To define Absolute Zero in X and Y:

1. Position the center of the spindle at the required part zero position.
2. In Manual Mode, press **X**. The CNC prompts for **X Axis Part Zero** position and displays **0.00**.
3. Press **Y**. The CNC prompts for **Y Axis Part Zero** position and displays **0.00**.
4. Press **ENTER** to accept the entered values (X0, Y0).

Presetting the X- or Y-Axis

To preset the X- or Y-axis to a predetermined position:

1. Adjust the machine's position to a known distance from the desired Part Zero.
2. Press **X**. The CNC prompts for the **X Axis Part Zero** position and displays **0.00**.
3. Enter the desired **X Axis** position (include sign) and press **Y**. The CNC prompts for the **Y Axis Part Zero** position and displays **0.00**.
4. Enter the desired **Y Axis** position (include sign), and press **ENTER** to set the new position coordinates.

CAUTION: To enter positions, press **ENTER** and not **START**. **START** will initiate a move to the entered coordinates.

Fixture Offsets (Work Coordinate System)

Sometimes it is convenient to be able to shift Part Zero with more than one coordinate system. (For example, if you must machine two or more workpieces in the same setup.) When you use multiple coordinate systems, you assign a different Part Zero for each workpiece. This simplifies programming because you can enter the moves required to machine a part from any selected reference point.

All Absolute moves are referenced from the active Fixture Offset location. The Fixture Offset location is referenced from Machine Home. You can define up to nine Fixture Offsets in the Fixture Offset Table. Refer to **Figure 3-3**.

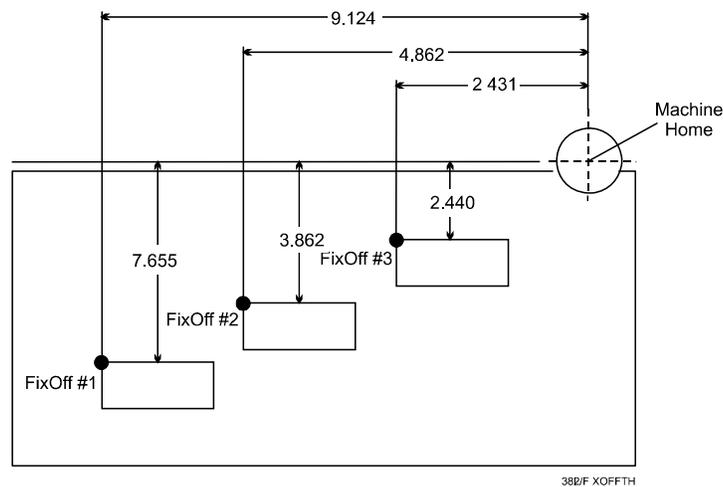


Figure 3-3, Locating Fixture Offsets

When you choose the location of Fixture Offsets, consider the following:

- Choose a reference point that corresponds to the information provided on the part print.
- Be consistent. To prevent confusion, use the same offset location for each part. (In the diagram, this is the upper-left corner.)
- The distance between the Fixture Offset and Machine Zero on each axis is the value entered in the Fixture Offset Table. Include the \pm direction of movement. For the diagram above:

Fixture Offset #	Coordinates Entered in Fixture Offset Table
FixOff #1	X-9.124, Y-7.655, Z (not shown)
FixOff #2	X-4.862, Y-3.862 Z (not shown)
FixOff #3	X-2.431, Y -2.440 Z (not shown)

Fixture Offset locations also can be used to preset the current location to entered coordinates or reset the current machine position to 0, for selected axes. (Refer to “Section 4 - Writing Programs” for information on how to program Fixture Offsets and use the Fixture Offset Table.)

Setting Tool Change Position

The tool change position is usually the fully retracted position of the quill, set to Tool #0, Z0 (no tool-length offsets, Part Zero). The Tool #0, Z0 position is often used for tool changes or as a safe place to make rapid XY moves.

At power up, the current location is Tool #0, Z0. This position is not stored in memory when you shut down the machine.

Set Tool #0, Z0 before setting tool-length offsets. Changing the location of Tool #0, Z0 alters all tool-length-offset settings.

Refer to “Section 1 - CNC Programming Concepts” and “Section 10 - Tool Management” for more information on tool-length offsets.

NOTE: For machines without homing, it may be necessary to set machine home to make setting tool-length offsets easier. Either manually or by jogging take the Z-axis close to the top of travel. The servo must be turned on. Press **MDI (F7)**, press **Mill (F5)**, press **More (F7)**, curse down to **Home** and press **ENTER**, press **Z** the z will light up, press **Save**, press **Prev**, press **Exit**, and press **Start**. The Z-axis will change to zero.

NOTE: Program a move to Z0 with XY rapid moves. This will ensure that the quill clears the surface of the work during the rapid move, thus avoiding possible tool or machine damage.

Locating Tool #0, Z0

NOTE: This procedure is necessary only if your machine does not have homing capability.

To locate Tool #0, Z0:

1. In Manual Mode, press the **5/TOOL** hot key to display the **TOOL MOUNT** Help Graphic.
2. Press **Save (F10)**. The CNC prompts you to **Press START or execute MANUAL to Cancel**.
3. Press **START** to activate tool #0 and deactivate length offset.
4. Adjust the machine to the desired tool change or tool #0, Z0 position.
5. Press **Z**. The CNC prompts for absolute **Z-Axis** position and displays **0.00**.
6. Press **ENTER**. The CNC displays **0.00** on **Z**.

Presetting the Z-Axis

To preset the Z-axis to a predetermined position:

1. In Manual Mode, go to Tool #0, Z0.
2. Press **Z**. The CNC prompts for the absolute **Z Axis** position and displays **0.00**.
3. Enter the Z-Axis position (including sign), and press **ENTER**. The Z Axis display changes to the entered position. This indicates that the CNC has preset Z to the entered position.

Activating a Tool

Activate a tool to allow applicable length offset and diameter listed in the Tool Page. Refer to "Section 4 - Writing Programs" to activate a tool in a program.

To manually activate a tool:

1. In Manual Mode, press **5/TOOL**. The CNC prompts for the tool number.
2. Enter a tool number, and press **Save (F10)** to return to Manual Mode.
3. Press **START**. The CNC displays the active tool number in the Primary Display Area.

– or –

Press **MANUAL (F4)** to cancel the selection.

Activating a Plane

Default plane: XY.

To change the active plane:

1. In Manual Mode, press **9/PLANE**. The CNC prompts for a plane selection.
2. Press the **+/-** key to switch to the desired plane selection.
3. Press **Save (F10)** to return to Manual Mode.
4. Press **START** to activate the selected plane.

– or –

Press **MANUAL (F4)** to cancel the selection.

Activating a Spindle RPM (Requires Programmable Spindle Option)

To activate a spindle RPM:

1. In Manual Mode, press the \square (Decimal/RPM) key. The CNC prompts for a RPM value.
2. Enter the desired RPM value, and press **Save (F10)** to return to Manual Mode.
3. Press **START** to activate the selected spindle RPM.
– or –
 Press **MANUAL (F4)** to cancel the selection.

Jog Moves

Use the Manual Operation Keys to make jog moves. There are two keys for each axis of motion, one for the positive direction and one for the negative direction. Refer to **Figure 3-4**.

Enable Jog moves when:

- The CNC is in the Manual Mode, Teach Mode, or Tool Page.
- The servos are on.

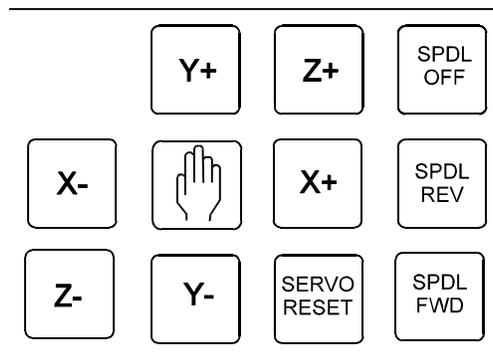


Figure 3-4, Manual Operation Keys

There are five available move modes. The machine builder determines the rate for each mode (Jog Rapid and Jog Feed) at machine setup. Press **JOG** to cycle through the available Jog settings. Refer to **Table 3-2**.

Table 3-2, Move Mode Selections

Mode	Description
Rapid	Default rapid speed for continuous jogs. Actual speed determined at machine setup.
Feed	Continuous jog at current feedrate.
Jog: 100	Conventional Jog Mode, increment set to 100 times resolution of machine.
Jog: 10	Conventional Jog Mode, increment set to 10 times resolution of machine.
Jog: 1	Conventional Jog Mode, increment set to actual resolution of machine.

In Manual Mode, the operator can change the Jog Mode anytime.

Changing the Jog Mode

To change the Jog Mode:

1. In Manual Mode, press **JOG** to change the mode selection. The CNC displays the selected mode in the Secondary Display Area.
2. Press **JOG** repeatedly to cycle through the available Jog settings.

NOTE: You can change the Jog Mode from the Manual Mode, Teach Mode, or Tool Page.

Jogging the Machine (Conventional)

To jog the machine one increment at a time:

1. In the Manual Mode, Teach Mode, or Tool Page, press **JOG** to cycle through the available settings: **JOG: 100**, **JOG: 10**, or **JOG: 1**. The CNC displays the selected setting in the Secondary Display Area.
2. Press the required axis key (**X+**, **X-**, **Y+**, **Y-**, **Z+**, or **Z**). Every time you press an axis key, the machine jogs along the selected axis at the indicated increment (100, 10, or 1 times machine resolution).

Jogging the Machine (Continuous)

To jog the machine along an axis continuously:

1. In Manual Mode, Teach Mode, or Tool Page, press **JOG** to cycle through the available settings. Set the machine to **JOG: RAPID** or **JOG: FEED**. The CNC displays the selected setting in the Secondary Display Area.
2. Press the required axis key (**X+**, **X-**, **Y+**, **Y-**, **Z+**, or **Z-**). The machine jogs in Rapid or Feed Mode along the selected axis until you release the key.

Operating the Handwheel (Optional)

The system supports an option that allows you to jog a selected axis using a remote handwheel.

NOTE: The handwheel operation described here assumes that the optional handwheel has been properly installed and configured.

The resolution of the handwheel depends on the selected Jog Mode. The CNC displays the selected Jog Mode on the screen.

Press the **JOG** key to cycle through the available Jog Modes. Refer to “Jog Moves” in this section for more information on selecting a Jog Mode. The handwheel works only in conventional Jog Modes (100, 10, and 1).

To operate the handwheel option:

1. From the Manual screen, press the **HANDWHL (F8)** soft key. The soft key highlights and the other soft keys are blank. The **Select HW Axis** popup menu activates. Refer to **Figure 3-5**.

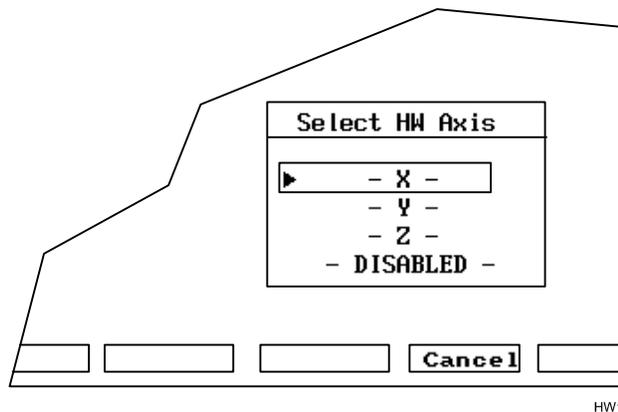


Figure 3-5, Handwheel Axis Selection Pop-Up

NOTE: The handwheel soft key will not display unless the Setup Utility has been configured for handwheel use.

2. From the Select HW Axis menu, highlight the axis that will be moved using the remote handwheel. Press **ENTER**. You can now move the selected axis using the remote handwheel.

3. Press the **JOG** key to select a Jog Mode: 100, 10, or 1. The axis will move 100, 10, or 1 times the machine resolution, respectively, per click of the handwheel.
4. Move the handwheel clockwise to move the selected axis in a positive direction, or counterclockwise to move the axis in a negative direction.

One-Shot Moves

In Manual Mode, you can position the machine using one-shot moves. One-shot moves can be in Absolute or Incremental Mode in Rapid or Feedrate.

To execute a one-shot move:

1. In Manual Mode, press **JOG** to select a move mode.

NOTE: All Jog modes, except Rapid Jog, run in Feed Mode.

2. Press **ABS/INC** to select a Positioning Mode (ABS/INC).
3. Enter the appropriate **X**, **Y**, and **Z** coordinates.
4. Press **START**.

NOTE: To pause one-shot moves, press **HOLD**. To resume, press **START**. To cancel one-shot moves, press **MANUAL (F4)**.

CAUTION: Press **START**, not **ENTER**, to initiate a one-shot move. If you inadvertently press **ENTER**, Absolute Zero moves to the entered position.

NOTE: The **1/RAPID** or **2/LINE** hot keys also can initiate one-shot moves. Activate the Absolute/Incremental Mode before you select the Rapid/Line Mode.

Manual Data Input

Manual Data Input (MDI) provides a quick way to program a single move, or a series of moves.

NOTE: From the Manual Data Input Mode, you can access all programming options and write a complete part program.

The CNC stores MDI blocks in a permanent program named **MDI.M**. Write MDI blocks the same way as program blocks. The MDI editor has the same features as the regular Program editor. Press **Draw** to test MDI moves.

The **MDI** screen resembles the **Edit** screen with **MDI.M** listed as the active program. Refer to **Figure 3-6**. The moves in MDI remain in the MDI.M program until cleared or edited. Run the MDI.M program at any

time. If you accidentally delete the MDI.M program, the CNC automatically creates a new one the next time the MDI Mode activates.

To save the MDI.M program, rename it. (Refer to “Section 9 - Program Management” for instructions.)

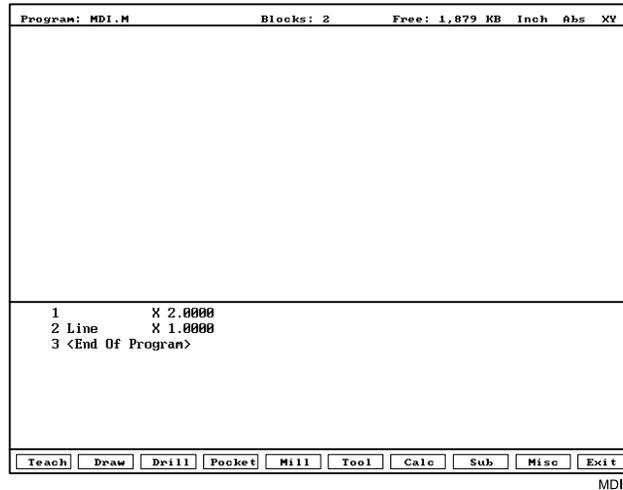


Figure 3-6, Manual Data Input (MDI) Screen

To execute an MDI command:

1. From the Manual screen, press **MDI (F7)** to display the Program Editor. The CNC automatically loads the MDI.M program and displays the program name on the bottom right part of the screen.
2. Enter the required program blocks.
3. If necessary, verify moves in Draw.
4. Press **Exit (F10)** to return to the Manual screen.
5. Press **START** to execute the MDI move.

– Or –

Press **MANUAL (F4)** to cancel the MDI move.

NOTE: Refer to “Section 4 - Writing Programs” for instructions on how to write program blocks.

Disengaging the Z-Axis Drive System

The 3000M provides the capability to switch between two-axis and three-axis operation. In Manual mode, just loosening the quick-release knob on the front of the Z-axis drive system can disengage the Z-axis. To re-engage the Z-axis drive system, lift the quill handle until you feel the drive assembly becomes seated, then tighten the quick release knob.

In Automatic mode, program a **ZMode TwoAxes** or a **ZMode ThreeAxes** to specify the particular area of the program that should be ran in two-axes or three-axes, respectively. When a **ZMode TwoAxes** is run in a program, the operator is prompted to disengage the Z-axis. When a **ZMode ThreeAxes** is run in a program, the operator is prompted to re-engage the Z-axis. Once the Z-axis re-engaged, the operator can jog the Z-axis to the homing mark on the front of the Z-axis drive system and perform a Homing sequence. Refer to **ZMode** block in “Section 5, Drilling Cycles” for more details.

When a program or a section of a program is ran in two-axes mode, the operator is prompted to manually position the Z-axis as needed.

Section 4 - Writing Programs

Program Basics

Each program consists of blocks of instructions that direct machine movements. Give each program a unique name.

Many settings remain active until changed or turned off. These are modal settings. For example, move type (Rapid/Feed), feedrate (IPM), units (Inch/MM), or ABS/INCR.

Write programs with combinations of moves, mode changes, and canned cycles. The CNC has a built-in library of canned cycles stored in its permanent memory.

Developing Part Programs

First, decide how to clamp the part and where to set Part Zero (Absolute Zero). Locate Part Zero at a point on the work positively positioned by the clamping fixture. This allows consistent machining of subsequent parts. Since Absolute positions are measured from Part Zero, locate Part Zero at a convenient location.

Determine the required tools and set the length offset for each tool. If necessary, adjust the position of the knee so that the longest tool clears the top of the part in a Rapid, Tool #0, Z0 move to Z0. Treat tool information like part of the program.

Refer to the blueprint to select a Part Zero. Note the moves, positions, and tools needed to cut the part. The CNC has a calculator that finds coordinates for complex geometric shapes (refer to "Section 12 - Calculators").

To develop a part program:

1. Go to the Program Directory (the **PROGRAM** screen) and create the program for the part (refer to "Section 9 - Program Management").
2. Go to the Program Editor (the **Edit** screen) to open the new program and begin to write blocks (refer to "Section 6 - Editing Programs").
3. The first block in a program is usually an Absolute Mode block. Put the CNC in the Absolute Mode at the start of a program to enable absolute positioning. (Use Incremental Mode only when specifically needed.)
4. Put the CNC in the appropriate Inch/MM Mode in the second block.
5. In the first move of the program, rapid to Tool #0, Z0 to retract the quill fully for the next move.
6. In the second move, rapid to a convenient part-change position.

- Execute moves toward a part in two steps: A Rapid X, Y move at a clear height, followed by a Z move to 0.1 inch (2mm) above the surface of the cut (standard starting height). If necessary, activate the first tool mount at this time.

NOTE: Program tool changes in separate blocks to make them easier to find.

- Subsequent blocks in the program are the moves, cycles, and tool changes required to machine the part.
- Make the last three blocks of the program as follows: a) a Rapid move to Tool #0, Z0, b) a Rapid XY move to the same part change position used at the start of the program, and c) an **EndMain** block.
- To verify and troubleshoot finished programs, run them in Draw Graphics Mode. Refer to “Section 7 - Viewing Programs with Draw.”
- Secure the work on the table with the appropriate work-holding device.
- Go to the Manual screen and set Part Zero at a convenient point on the part. Refer to “Section 3 - Manual Operation and Machine Setup.”
- Go to the Tool Page and organize the tooling. Assign each tool a number (in the order of use). Assign length offset and tool diameter as appropriate. Refer to “Section 10 - Tool Management.”
- If Fixture Offsets are used, define them in the Fixture Offsets Table. Refer to “Programming Fixture Offsets” in this section.
- Before you cut a part, perform a dry run. There are several ways to get a close look at the programmed moves. Run the program in Motion Mode to hold between each move or in Single-Step Mode to hold between each block. Run the program with no tool installed, or with a tool installed, but the knee lowered so the tool never contacts the work.
- After a successful dry run, the program is ready for production. Back up the program for safekeeping. Refer to “Section 9 - Program Management.”

Writing Program Blocks

You can program a block for a move type, mode, or cycle using one of the following: hot keys, soft keys, or pop-up menus.

To program a block, activate its graphic menu and fill in the appropriate values. To save a program block, press **Save (F10)** or press **ENTER** on the last entry field in the graphic menu. The CNC adds the new block to the Program Listing.

The **<End Of Program>** block is the last line of a program. The CNC automatically numbers new blocks and inserts them in front of the **<End Of Program>** block.

Using Graphic Menus

The Program Editor displays full screen graphic menus to write and edit program blocks. Refer to **Figure 4-1**.

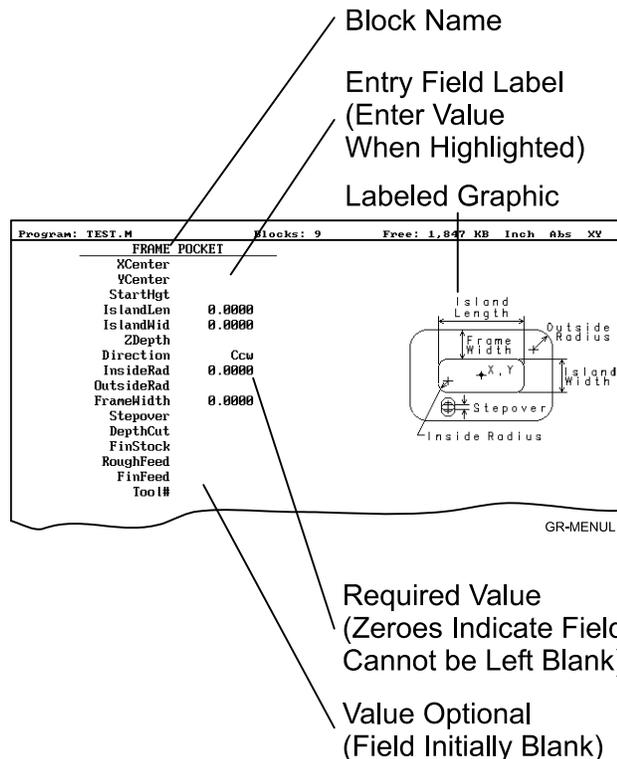


Figure 4-1, Sample Graphic Menu

Graphic menus activate with the first entry field highlighted. To enter values, highlight the appropriate entry field, and press **ENTER** to advance the highlight to the next entry field. With the last entry field highlighted, press **ENTER** to close the menu and add the block to the program.

Press **Save (F10)** from any entry field to close the graphic menu and add the block to the program. Move the highlight from field to field using the **ARROW** keys. Fill out entry fields in any order.

Press **CLEAR** to remove values in the highlighted field.

There are two types of entry fields in a graphic menu:

Optional entry fields Blank when the graphic menu activates.

Required entry fields Contain **0.000** when the graphic menu activates.

Required entry fields contain a 0.0000 default value. Change the value as required. Optional entry fields do not require a value. When left blank, the CNC usually assumes a default value or position. If the optional field is a position, the value defaults to the current position. If the optional field is a mode or tool change, the current mode and tool remain active. If the optional field is an angle, the value defaults to 0.0 degrees.

Type decimal points and negative signs where needed. Otherwise, the CNC assumes a positive whole number.

Press **+/-** to insert a negative sign or toggle selections in some entry fields (for example, **Cw/Ccw** fields).

No Move Blocks

No Move Blocks does not initiate machine moves. Use No Move Blocks to set modes (Incremental/Absolute, etc.), activate tools (**Tool#**), and set feedrates (**Feed**).

Programming an Absolute/Incremental Mode Change

A **Dim** (dimension) block sets the Absolute (**Abs.**) or Incremental (**Incr.**) Mode.

To program a **Dim** block:

1. In Edit Mode, press **ABS/INC**. The **SET ABS/INCR DIMENSION** graphic menu prompts you to select **Abs** or **Incr**.
2. Press **+/-** to toggle the mode.
3. Press **Save (F10)** or **ENTER** to add the block to the Program Listing.

Programming an Inch/MM Mode Change

A Unit block sets the Inch (**Inch**) or Millimeter (**MM**) Mode.

To program a Unit block:

1. In Edit Mode, press **7/UNIT**. The **SET INCH/MM UNIT** graphic menu prompts you to select **Inch/MM**.
2. Press **+/-** to toggle the selection.
3. Press **Save (F10)** or **ENTER** to add block to the Program Listing.

Programming a Tool Change

Identify tools with tool numbers. When you activate a tool, its tool length and diameter offsets activate. List these values on the corresponding row of the Tool Page. Refer to “Section 10 - Tool Management” for information on the Tool Page.

NOTE: The CNC also activates listed coolant (optional) and spindle operations (optional).

Tool-length offset remains in effect until a different tool activates. Always turn off tool-diameter compensation and ramp off before activating a new tool.

NOTE: You can program tool numbers with most moves; they do not require a separate block. Each time a tool activates, the CNC holds the program to permit installation of the new tool. Programming unnecessary tool changes slows down production.

Activate Tool #0 to set the tool-length offset and diameter to 0.0.

To change a tool:

NOTE: An absolute move to Tool #0, Z0 fully retracts the quill. An incremental command to Z0 maintains the current position.

1. In Absolute Mode, program a Rapid Z0, Tool #0 to cancel length offsets and retract the quill to a safe position.
2. Program a Rapid move to the tool change XY position (usually Machine Zero).
3. Program a block to activate the required tool (example: **Tool#1**). When the CNC encounters the **Tool#** command, it holds optional spindle and coolant functions. The operator can now change the tool.
4. Press **START** to resume operation. The CNC activates applicable tool compensation, spindle, and coolant activities.

NOTE: Program tool changes in separate blocks to make them easier to find.

Activating a Tool

To activate a tool:

1. In Edit Mode, press **5/TOOL**. The **TOOL MOUNT** graphic menu prompts for **Tool #**.
2. Type tool number and press **ENTER** to add the **Tool#** block to the Program Listing.

Activating Tool-Diameter Compensation

Refer to “Section 1 - CNC Programming Concepts” for basic information on tool-diameter compensation. You can compensate most move types. Compensation is modal and remains active until you cancel or change it.

Turn compensation on or off in (Rapid or Line) ramp moves. Ramp moves offset the tool on the programmed path by half the tool diameter. Tool compensation affects all subsequent moves until canceled.

The **ToolComp** command, available in Line or Rapid graphic menus, sets the required tool compensation. Settings include:

- **Left** (of the path)
- **Right** (of the path)
- **Off** (cancel compensation)

When the field is left blank, the current compensation, if any, remains in effect.

Many canned cycles include automatic tool compensation. Activate the correct tool diameter to ensure accuracy in these cycles. The required tool activates within the cycle.

Refer to **Table 4-1** for a list of move and cycle compensation requirements.

Table 4-1, Move and Cycle Compensation Requirements

Move or Cycle	Program a Rapid or Line move to activate tool comp before you program the move or cycle. Tool diameter must be active.	Activate/deactivate compensation automatically when you program the move or cycle. Tool diameter must be active.
Rapid	X	---
Line	X	---
Modal	X	---
Arc	X	---
Ellipse (Has special requirements. Refer to “Section 5 - Programming Canned Cycles, Ellipses, and Spirals” for more information.)	X	---
Spiral (No compensation available.)	---	---

(Continued...)

Table 4-1, Move and Cycle Compensation Requirements (Continued)

Move or Cycle	Program a Rapid or Line move to activate tool comp before you program the move or cycle. Tool diameter must be active.	Activate/deactivate compensation automatically when you program the move or cycle. Tool diameter must be active.
Face (Affects only step-over.)	---	X
Rectangular Profile Cycle	---	X
Circular Profile Cycle	---	X
Rectangular Pocket Cycle	---	X
Circular Pocket Cycle	---	X
Frame Pocket Cycle	---	X
Irregular Pocket Cycle	---	X
Mold Rotation Cycle (Has special requirements. Refer to "Section 5" for more information.)	---	X
Elbow Mold Cycle (No compensation available.)	---	---

Programming a Dwell

Dwell pauses a running program for a specified length of time, in seconds. Dwell resolution is 0.1 sec. When the operator types 0.0 seconds (infinite dwell), the CNC will hold indefinitely. Press **START** to restart the CNC after an infinite dwell.

To program a Dwell using hot keys:

1. In Edit Mode, press **8/DWELL**. The **DWELL** graphic menu prompts for length of time in seconds.
2. Type the time, and press **ENTER** to add Dwell block to the Program Listing.

To program a Dwell using soft keys:

1. In Edit Mode, press **Sub (F8)** to display the Secondary soft key functions.
2. Press **Dwell (F7)** to activate the **DWELL** graphic menu.
3. Type the time, and press **ENTER** to add Dwell block to the Program Listing.

Programming a Return to Machine Zero

NOTE: The CNC measures all typed coordinates in the Machine Home graphic menu from Machine Zero. The CNC homes axes one at a time, in the order indicated in the Setup Utility.

A **Home** block re-establishes a permanent reference position located on the machine. The position is called Machine Zero. Program a Home block using one of the two methods described in **Table 4-2**.

Table 4-2, Homing Methods

Homing Method	Required Action
1. Indicate axes	Activate Machine Home graphic menu. Press required X, Y, Z -axis keys. On each axis selected, machine feeds from the current position to the limit switch, reverses direction and travels to the first detected zero crossing, and sets Machine Zero at that point.
2. Type coordinates	Activate Machine Home graphic menu. For each required axis, highlight the axis entry field and type a coordinate (example: X0, Y-1, Z-4). The machine rapids to the typed coordinate, then feeds to the limit switch, reverses direction and travels to the first detected zero crossing, and sets Machine Zero at that point.

Use Homing Method 1 to execute a homing sequence in feed. Use Homing Method 2 to execute a homing sequence that rapids to the typed coordinate, then initiates the homing sequence.

To activate the Machine Home graphic menu:

1. In the Edit Mode, press **Mill (F5)** to display the Mill soft keys.
2. Press **More (F7)** to display the More pop-up menu.
3. Highlight **Home** and press **ENTER** to display the Machine Home graphic menu.

The method used to set Machine Zero depends on which options the builder installs. Check with the machine builder for detailed information.

Programming Fixture Offsets

Refer to **Figure 4-2**.

NOTE: Presets and **SetZero** will work with Fixture Offsets.

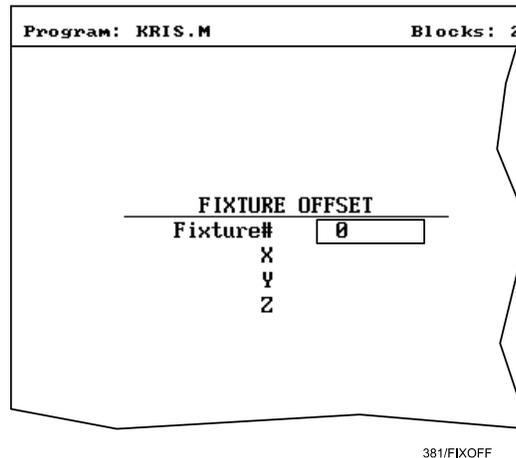


Figure 4-2, Fixture Offset Graphic Menu

To program a Fixture Offset:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft keys.
2. Press **More (F7)** to display the More pop-up menu.
3. Select **Offsets** and press **ENTER** to display the Fixture Offset graphic menu.
4. Fill in the labeled entry fields, as follows:

- | | |
|-----------------|--|
| Fixture# | The Fixture-Offset number. Indicates which set of values from the Fixture Offsets Table will be activated or changed. Type a number 1 through 9 , corresponding to the Fixture Offsets Table, to activate or change an offset. Type 0 to cancel fixture offsets. |
| X | X-offset coordinate. If you do not type a value, the CNC activates the offsets listed in the Fixture Offsets Table for the typed Fixture# . If you do type a value, the CNC applies the typed offset. When the program runs, the CNC updates the Fixture Offsets Table with the specified X offset and clears the previous value. Optional. |
| Y | Y-offset coordinate. If you do not type a value, the CNC activates the offsets listed in the Fixture Offsets Table for the typed Fixture# . If you do type a value, the CNC applies the typed offset. When the program runs, the CNC updates the Fixture Offsets Table with the specified Y offset and clears the previous value. Optional. |

- Z** Z-Offset coordinate. If you do not type a value, the CNC activates the offsets listed in the Fixture Offsets Table for the typed **Fixture#**. If you do type a value, the CNC applies the typed offset. When the program runs, the CNC updates the Fixture Offsets Table with the specified Z offset and clears the previous value. Optional.

To cancel Fixture Offsets:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft keys.
2. Press **More (F7)** to display the More pop-up menu.
3. Select **Offsets**, and press **ENTER** to display the **Fixture Offset** graphic menu.
4. Select **Fixture#**. In the highlighted entry field, type **0** to cancel Fixture Offsets. (Do not fill in the other entry fields.)

Fixture Offsets Table

The Fixture Offsets Table, accessed using the Tool Page, contains the typed values for Fixture Offsets 1 through 9. Refer to **Figure 4-3**.

NOTE: Handwheel and Jog features are available while the Fixture Offsets Table is active.

Fixture Offsets			
	X	Y	Z
1.▶	0.0000	1.0000	0.0000
2.	0.0000	0.0000	0.0000
3.	0.0000	0.0000	0.0000
4.	0.0000	0.0000	0.0000
5.	0.0000	0.0000	0.0000
6.	0.0000	0.0000	0.0000
7.	0.0000	0.0000	0.0000
8.	0.0000	0.0000	0.0000
9.	0.0000	0.0000	0.0000

381/FIXOFFTB

Figure 4-3, Fixture Offsets Table

Activating the Fixture Offsets Table

To activate the Fixture Offsets Table:

1. In the Tool Page, press **OFFSETS (F1)**.

Changing Fixture Offsets in the Table

There are three ways to change the values in the table: manually type a value, or calibrate the fixture offset table entry to the machine's current location (shown on the axis display), or adjust an existing entry.

To change a fixture offset to a manually typed coordinate:

1. Highlight a Fixture Offset (row 1 through 9).
2. Press an axis key (**X**, **Y**, or **Z**).
3. Type a value, and press **ENTER** to store the value in the table.

To calibrate the fixture offset table entry to the machine's current location:

1. Highlight a Fixture Offset (row 1 through 9).
2. Press **CalibX (F5)**, **CalibY (F6)**, or **CalibZ (F7)** to store the current machine position for the selected axis in the table.

To adjust a fixture offset table entry:

1. Highlight a Fixture Offset (row 1 through 9).
2. Press the **ABS/INC** key, or **ALT + E** on a keyboard, to display a pop-up message, "Enter axis and adjustment value."
3. Type the axis to adjust (**X**, **Y**, or **Z**) and the amount of the adjustment. The adjustment value can be positive or negative.
4. Press **ENTER** to adjust the value, and display the adjusted value in the table.

Resetting Absolute Zero (Part Zero)

Absolute Zero is the X0, Y0 position for absolute dimensions. Refer to "Section 1 - CNC Programming Concepts" for information on Absolute Positioning. A **SetZero** block sets the Absolute Zero Reference of one or more axes to a new position. Use **SetZero** in one of two ways: to reset X0 Y0 or to preset the current location to typed coordinates.

In axis presetting, non-zero XY values set the current machine position to the typed coordinates. In axis resetting, X0 and Y0 values set the current machine position as the new Absolute Zero Reference.

When the CNC executes the block, the X and Y values (zero or non-zero) in the graphic menu redefine Absolute Zero.

In **Figure 4-4, Executing a SetZero Block**, diagram A shows Part Zero and tool position prior to a **SetZero** block. In this example, the operator programs a **SetZero** block with the following coordinates: **X2, Y-1**.

Diagram B shows Part Zero and tool position following the **SetZero** block. The coordinates at the tool position become **X2, Y-1**. This, in effect, moves Part Zero, as indicated.

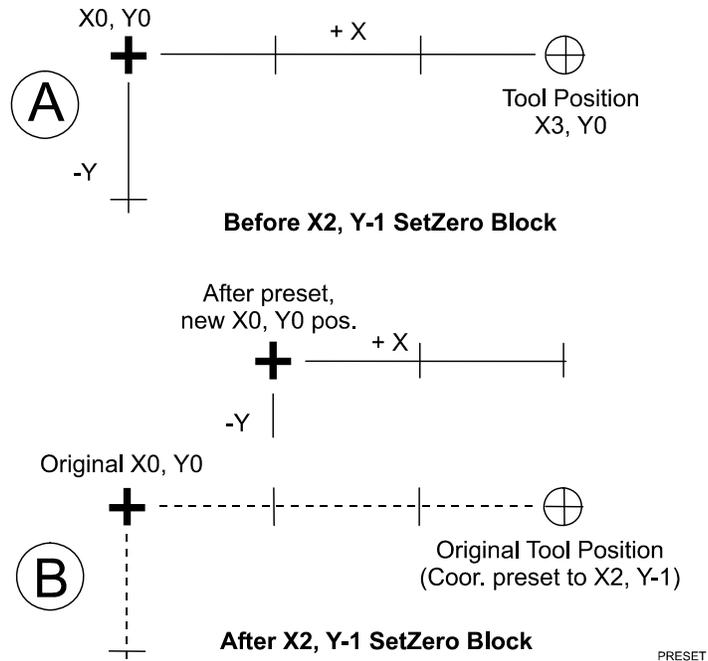
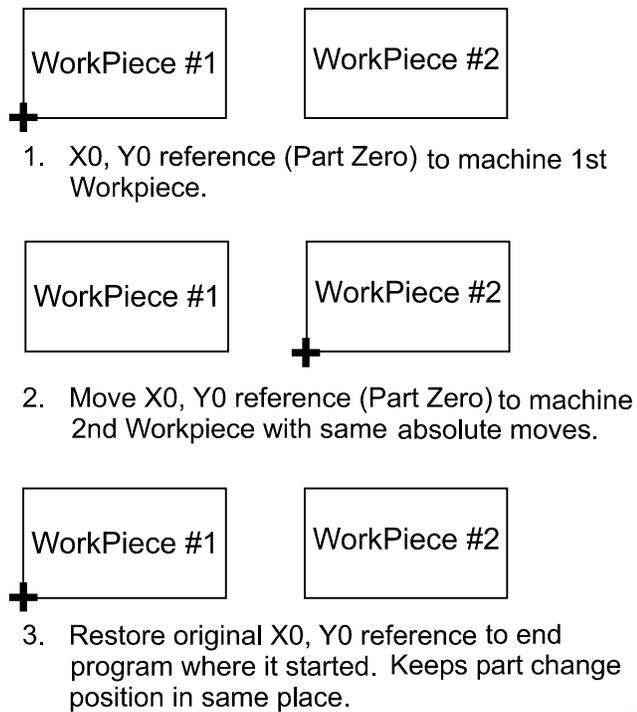


Figure 4-4, Executing a SetZero Block

Change Absolute Zero to cut more than one part with the same moves. Restore the location of the original X0, Y0 reference at the end of the program so that programmed part change positions do not move each time the program runs. Refer to **Figure 4-5**.



DUALPT

Figure 4-5, Using SetZero in a Program

When an axis entry field (**X**, **Y**, or **Z**) remains blank in a graphic menu, the CNC does not change the position of that axis. Refer to **Figure 4-6**.

NOTE: In most programs, the Z-axis position does not change. Changing the Z-axis position changes the Tool #0, Z0 position, which alters all existing tool-length offsets.

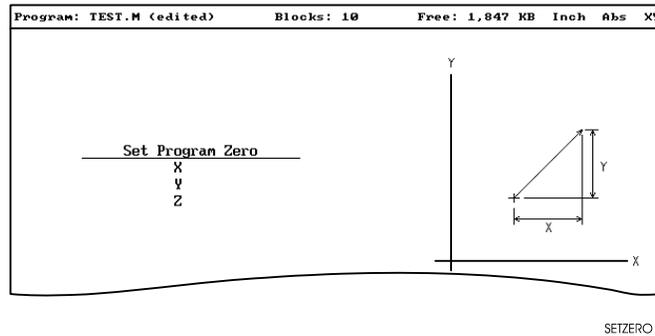


Figure 4-6, Set Zero Graphic Menu

To program a **SetZero** block:

1. In Edit Mode, press **Mill (F5)** to change the Soft Key labels.
2. Press **More (F7)** to display the pop-up menu.
3. Position the highlight to select **SetZero**, and then press **ENTER**. The **Set Program Zero Graphic** menu prompts for the absolute coordinates of the machine's current position.
4. Type the appropriate **X**, **Y**, and **Z** coordinates and press **ENTER** to add the block to the Program Listing.

Programming a Plane Change

The CNC executes Arc moves and compensates for tool diameters in three different planes (**XY**, **YZ**, and **XZ**). By default, the CNC operates in the XY plane. Program a Plane block to change the CNC's active plane. Following moves in the XZ or YZ plane, program a second Plane block to return to the XY plane. Refer to "Section 1 - CNC Programming Concepts" for basic information on planes.

A Plane block also changes the active plane of the Program Editor. The Program Editor customizes Arc graphic menus for the active plane.

When a plane block is deleted from the Program Listing, the active plane of the Editor does not change.

To program a Plane block using hot keys (the keypad):

1. In Edit Mode, press **9/PLANE**. The **SET PLANE** graphic menu prompts for plane selection.
2. Press the **+/-** key to change the selection to the desired plane. Press **ENTER** to add the block to the Program Listing.

To program a Plane block using soft keys:

1. In Edit Mode press **Mill (F5)** to display the Mill soft keys.
2. Press **More (F7)** to display the **More** pop-up menu.
3. Highlight **Plane**, and press **ENTER**. The **SET PLANE** graphic menu prompts for plane selection.
4. Press **+/-** to change to the desired plane. Press **ENTER** to add the block to the Program Listing.

Programming a Feedrate Change

A **Feed** block sets the feedrate for Line moves, arcs, and cycles that do not contain specifically programmed feedrates. Feed blocks also set the feedrate for modal moves. Add Feed blocks whenever necessary.

NOTE: A **Feed** block does not activate the Feed Mode.

To program a **Feed** block from the hot keys (the keypad):

1. In Edit Mode, press **4/FEED** to display the **FEEDRATE** graphic menu.
2. Type the required feedrate, and press **ENTER** to add the block to the Program Listing.

To program a **Feed** block from the soft keys:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft keys.
2. Press **More (F7)** to display the **More** pop-up menu.
3. Highlight **Feed**, and press **ENTER** to activate the **FEEDRATE** graphic menu.
4. Press **Save (F10)** or **ENTER** to add Feed block to the Program Listing.

If the machine has a 4th axis (typically, U) a feedrate can be programmed for the U-axis using the **FeedU** block. If the U-axis is a rotary axis, the units for the **FeedU** block are degrees per minute (deg/min); otherwise, the units are the same as in the Feed block.

To program a **FeedU** block from the soft keys:

1. In Edit, or MDI, press **Mill (F5)** to display the Mill keys.
2. Press **More (F7)** to display the More keys
3. Highlight **FeedU** and press **ENTER** to display the FeedU graphic menu.
4. Type the desired value, and press **Save (F10)** or **ENTER** to add the **FeedU** block to the program.

Programming a Spindle RPM

If your CNC has a programmable spindle RPM, you can set the RPM in one of the following ways.

- Each tool has an associated RPM entry in the tool page. When a tool is made active, its associated RPM is used.
- You can use the RPM block in the program. This is especially necessary when you use more than one spindle speed with a specific tool.

To program an RPM block from the hot keys (the keypad):

1. In Edit Mode, press the **▫** (Decimal/RPM) key to display the **Spindle RPM** graphic menu.
2. Type the required spindle RPM and press **ENTER** to add the block to the Program Listing.

To program an RPM block from the soft keys:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft keys.
2. Press **More (F7)** to display the **More** pop-up menu.
3. Highlight **RPM** and press **ENTER** to activate the **Spindle RPM** graphic menu.
4. Type the required RPM, and press **Save (F10)** or **ENTER** to add RPM block to the Program Listing.

Straight Moves

Programming a Rapid Move

Rapid moves run at the CNC's Rapid rate and save time when positioning for a cut or a canned cycle.

Use Rapid moves to activate/deactivate tool diameter compensation and cutter compensation. Refer to **Figure 4-7**.

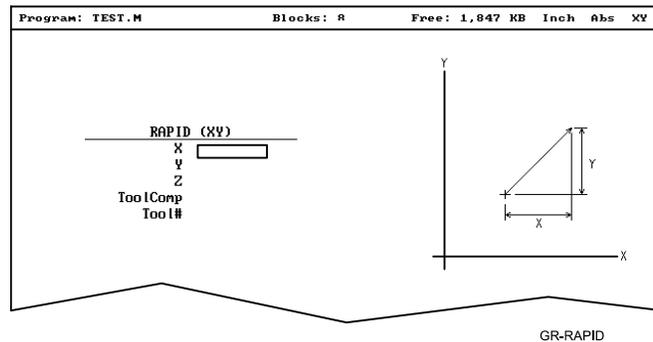


Figure 4-7, Rapid Move Graphic Menu

To program a Rapid move using hot keys (the keypad):

1. In Edit Mode, press **1/RAPID** to activate the **RAPID (XY)** graphic menu.
2. Type the **X**, **Y**, and **Z** coordinates in the appropriate entry fields.
3. Press **+/-** to set **ToolComp** (optional) and press **ENTER** to highlight **Tool#**.
4. Type a tool number (optional), and press **ENTER** to add the Rapid block to the Program Listing.

To program a Rapid move using soft keys:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft keys.
2. Press **Rapid (F2)** to activate the **RAPID (XY)** graphic menu.
3. Type the appropriate values and settings in the labeled entry fields.

Programming a Line Move

Straight line moves run in Feed. Refer to **Figure 4-8**.

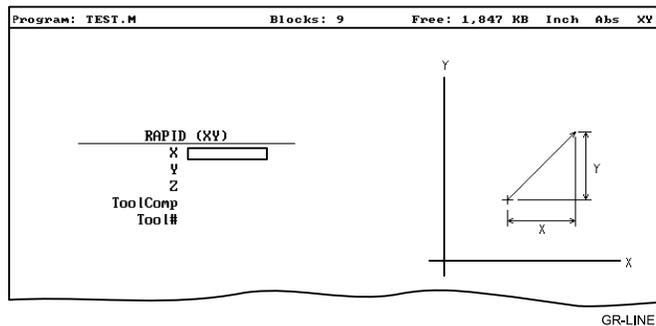


Figure 4-8, Line Move Graphic Menu

To program a Line move using hot keys (the keypad):

1. In Edit Mode, press **2/LINE** to activate the **LINE (XY)** graphic menu.
2. Type the appropriate values and settings in the labeled entry fields.

To program a Line move using soft keys:

1. In Edit Mode, press **Mill (F5)** to change the Soft Key labels.
2. Press **Line (F3)** to display the **LINE (XY)** graphic menu.
3. Type the appropriate values and settings in the labeled entry fields.
4. With the last entry field highlighted, press **ENTER** to add the block to the Program Listing.

Programming a Modal Move

A modal move is a straight move executed in the active Rapid or Feed Mode.

To program a Modal move:

1. In Edit Mode, press **X**, **Y**, or **Z**. The **MODAL MOVE** graphic menu prompts for the X, Y, and Z positions.
2. Type the required positions.
3. From the last field on the graphic menu, press **ENTER** or **Save (F10)** to add the modal move block (Xn Yn Zn) to the program.

NOTE: When using modal moves, be sure the CNC is in the required Rapid or Line Mode. The CNC executes Line Mode moves in Feed Mode.

Teach Mode (Programming from the Part)

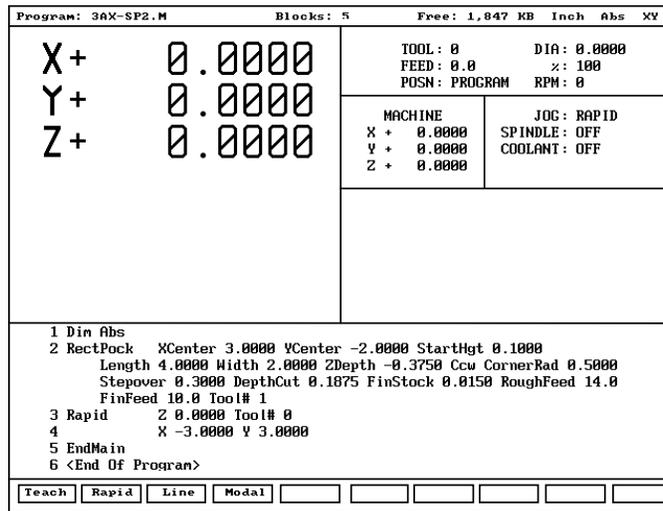
In Teach Mode, the CNC writes program blocks that duplicate manually executed moves. Teach Mode allows you to select endpoints from a part diagram to program a move without knowing the coordinates.

Teach Mode also operates in DRO Mode.

To activate the Teach Mode from the Edit screen:

1. Press **Teach (F1)**. The **Teach (F1)** soft key highlights.

Use Teach Mode to program a Line, Rapid, or Modal move. Refer to **Figure 4-9**.



381TEACH

Figure 4-9, Teach Mode Screen

To program a Teach move:

1. In Edit Mode, press **Teach (F1)** to activate Teach Mode. The display shows the current location.
2. Manually position the machine at the endpoint.
3. Press **Rapid (F2)**, **Line (F3)**, or **Modal (F4)** to indicate the required move type. The CNC inserts the appropriate block into the listing.
4. Program all required moves in Teach Mode.
5. Press **Teach (F1)** to deactivate Teach Mode.

Line or Rapid Moves

Using the X, Y, or XY endpoints, the CNC can write Line or Rapid moves. The CNC calculates the missing endpoint(s). Define the move as part of a right triangle with the components identified as in **Figure 4-10**.

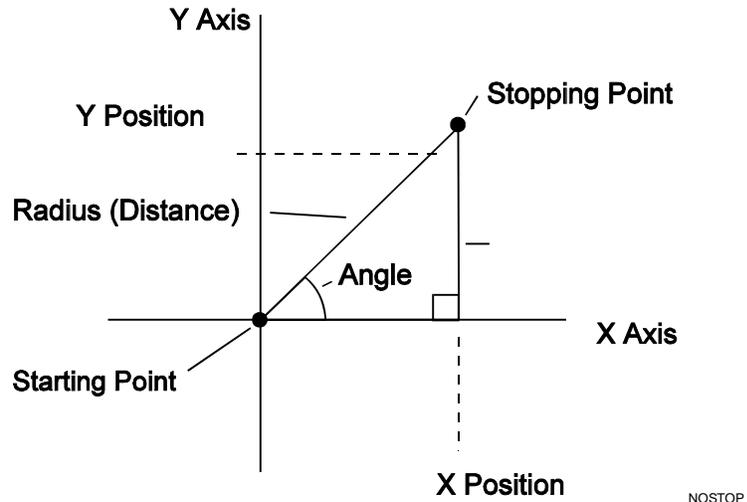


Figure 4-10, Move Orientation

The CNC can calculate move endpoints, given:

- Angle and radius
- X position and angle
- Y position and angle
- X position and radius
- Y position and radius

The **Rapid** and **Line** graphic menus are similar. However, the **Rapid** graphic menus do not contain **CornerRad** or **Feed** entry fields. Use either the Absolute or Incremental Mode.

Programming a Move Using XY Location, Radii, or Angles

To program a move using a Line or Rapid block:

1. In Edit Mode, press **Mill (F5)** and select either **Rapid (F2)** or **Line (F3)**.

– or –

In Edit Mode, press **1/RAPID** or **2/LINE** to display the **RAPID (XY)** or **LINE (XY)** graphic menu.

2. Press **More. . . (F4)** to display the Template pop-up menu. Refer to **Figure 4-11**.

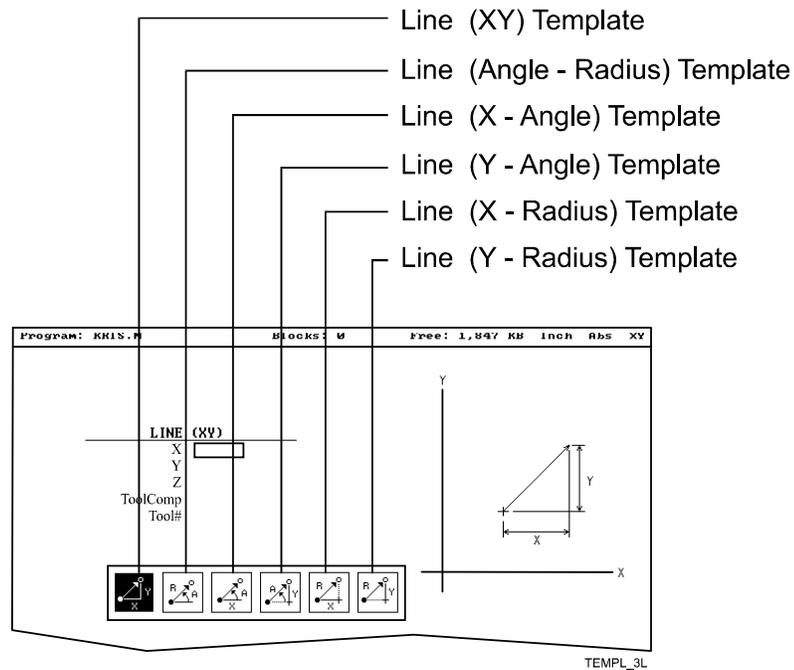


Figure 4-11, Rapid/Line Template Pop-up Menu

3. Highlight the appropriate template, and press **ENTER** to display the graphic menu.
4. Type the required values/settings in the entry fields.

Arcs

Selecting the Plane for an Arc

Refer to “Section 1 - CNC Programming Concepts” for information on planes and Arc directions. The CNC executes Arcs in the XY plane by default. For an Arc in the XZ or YZ plane, program the plane change before the Arc move. The plane change customizes the Arc graphic menus for the required plane.

The graphic menus for moves in the XY, XZ, and YZ planes contain the same entry fields. Entry fields for selected plane positions require a value.

After a move in the XZ or YZ plane, return the CNC to the XY plane.

NOTE: To activate a new plane in the Program Editor, program a plane change block.

Program Arc moves:

- Using the endpoint and radius
- Using the center and endpoint
- Using the center and angle

Programming an Arc Using an Endpoint and Radius

To define the **Endpoint - Radius Arc**, type the direction of the Arc, the endpoint, and the radius. The CNC cuts an Arc of the specified radius from the current position to the endpoint. You must correctly define the modal endpoint coordinates in the **Absolute** or **Incremental** Mode.

In the XY plane, if the Z-axis starting and end points differ, the arc is a helix.

Two Arcs can intersect any two points: an Arc with an included angle less than 180 degrees and an Arc with an included angle greater than 180 degrees. Refer to **Figure 4-12**.

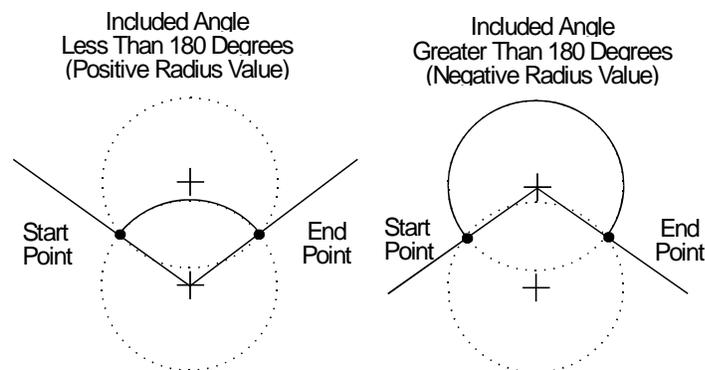


Figure 4-12, Endpoint Radius Arc Types

To program an Arc with an included angle less than 180 degrees, type a positive radius value. To program an Arc with an included angle greater than 180 degrees, type a negative radius value. The CNC selects which Arc center to use, based on the sign of the typed value.

To program an Arc using an endpoint and radius, using hot keys (the keypad):

1. In Edit Mode, press **3/ARC**. The **ARC (END POINT - RADIUS)** graphic menu prompts for labeled values.
2. Fill in the entry fields as labeled.

To program an Arc using an endpoint and radius, using soft keys:

1. In Edit Mode, press **Mill (F5)** to display the Mill secondary soft keys.
2. Press **Arc (F4)** to display the **ARC (END POINT - RADIUS)** graphic menu. Refer to **Figure 4-13**.

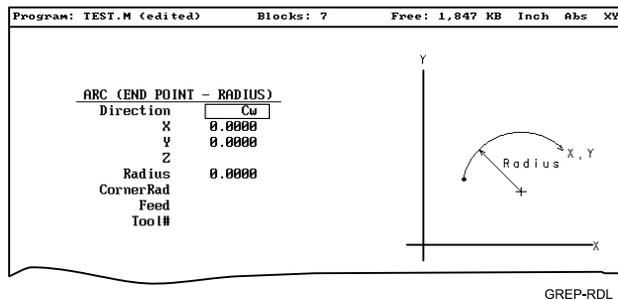


Figure 4-13, ARC (END POINT - RADIUS) Graphic Menu

3. Fill in the **ARC (END POINT - RADIUS)** entry fields:

Direction	Specifies a clockwise (Cw) or counterclockwise (Ccw) direction. Press +/- to toggle the setting. Selection required.
X	The X coordinate of the Arc endpoint. Value required.
Y	The Y coordinate of the Arc endpoint. Value required.
Z	The Z coordinate of the endpoint. Optional.
Radius	The radius of the Arc. Value required (positive or negative).
CornerRad	Corner radius setting. Optional. (Refer to "Section 1 - CNC Programming Concepts" for details.)
Feed	Feedrate. Optional.
Tool#	Active tool. Optional.

Programming an Arc Using the Center and Endpoint

NOTE: Use Center and Endpoint Arcs to cut helical threads.

To define the **Center - Endpoint Arc**, type the endpoint, arc center, and direction. The CNC cuts an Arc from the current position to the end point.

In Absolute Mode, the CNC measures the Arc center and endpoint from Absolute Zero. In Incremental Mode, the CNC measures the Arc center and end point from the starting position of the arc.

NOTE: Ensure that the required Absolute or Incremental Mode is active.

When the Z-axis start and end points differ in the XY plane, the Arc is a helix. The **Revs** value determines the number of rotations used to machine the helix.

The CNC calculates the radius from the specified starting point and Arc center. Therefore, the endpoint must lie along the Arc's path. If the endpoint does not lie along the Arc's path, the CNC will adjust the center or end point. Configure whether the CNC will adjust the center or endpoint in the Setup Utility. Endpoint is preferred. The machine builder sets the maximum arc circle center adjustment in the Setup Utility. If the error exceeds the setup tolerance, the CNC generates an error message.

To program a **Center - End Point Arc using hot keys** (the keypad):

1. In Edit Mode, press **3/ARC** to display the **ARC (CENTER – END POINT)** graphic menu. Refer to **Figure 4-14**.

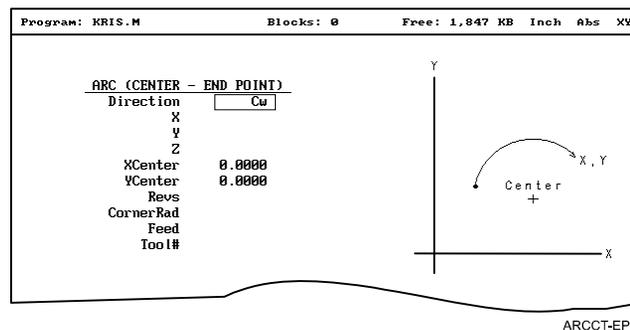


Figure 4-14, ARC (CENTER – END POINT) Graphic Menu

2. Press **More... (F4)** to display the Arc template pop-up menu. Refer to **Figure 4-15, Arc Template Pop-Up Menu**.
3. Highlight the **Center – End Point** template and press **ENTER** to display the **ARC (CENTER – END POINT)** graphic menu.
4. Type the required values or settings in the entry fields.

To program a **Center – End Point Arc** using soft keys:

1. In Edit Mode, press **Mill (F5)** to display the Mill secondary soft keys.
2. Press **Arc (F4)** to display the **ARC (END POINT - RADIUS)** graphic menu.
3. Press **More... (F4)** to display the Arc Template pop-up menu. (Refer to **Figure 4-15**).

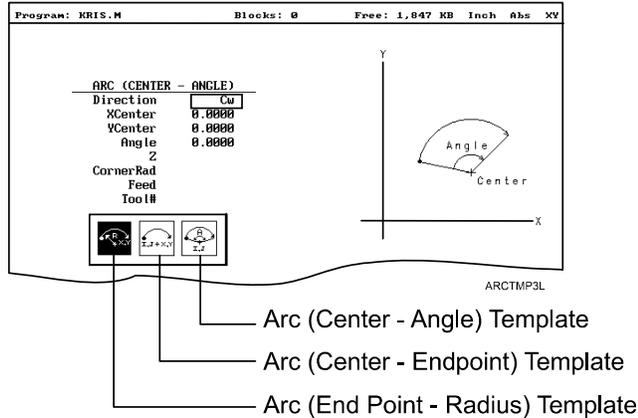


Figure 4-15, Arc Template Pop-Up Menu

4. Highlight the **Center – End Point** template and press **ENTER**. The **ARC (CENTER – END POINT)** graphic menu prompts for the labeled values.
5. Type the required values or setting in the following labeled entry fields:

Direction	Allows you to select a clockwise (Cw) or counterclockwise (Ccw) direction. Press +/- to toggle the setting. Selection required.
X	The X coordinate of the Arc end point. Optional.
Y	The Y coordinate of the Arc end point. Optional.
Z	The Z coordinate of the Arc end point. Optional.
XCenter	The X coordinate of the Arc center. Value required.
YCenter	The Y coordinate of the Arc center. Value required.
Revs	Number of revolutions from start position to endpoint. Optional.
CornerRad	Corner radius setting. Optional. (Refer “Section 1 - CNC Programming Concepts” for details.)
Feed	Feedrate. Optional.
Tool#	Active tool. Optional.

Programming an Arc Using the Center and the Included Angle

To define the **Center - Angle Arc**, type the arc center and the included angle. The CNC cuts the Arc from the present position until the Arc travels the specified number of degrees. The CNC calculates the radius, which is the distance between the start position and the center point.

Specify the appropriate Absolute or Incremental Mode for the angle and center point. Refer to **Figure 4-16** and **Figure 4-17**.

The direction (Cw/Ccw) of the Arc and the sign (+/-) of the angle control the path of the tool.

If the Z-axis starting and end points differ, the Arc will be a helix.

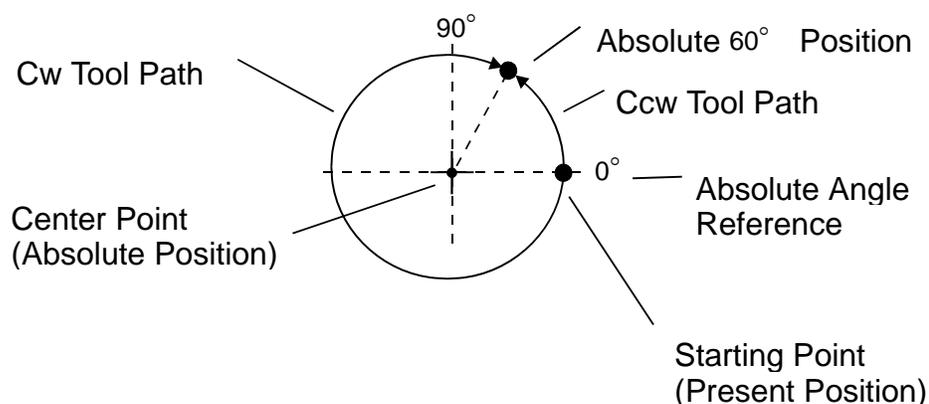


Figure 4-16, Absolute Mode, Center - Angle Arc

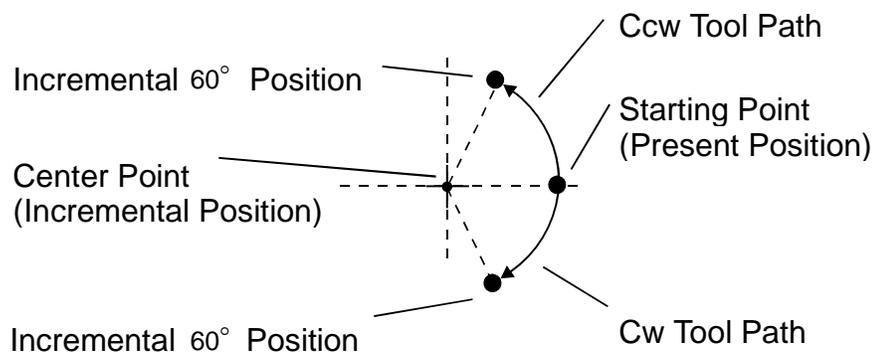


Figure 4-17, Incremental Mode, Center - Angle Arc

Refer to **Figure 4-18**.

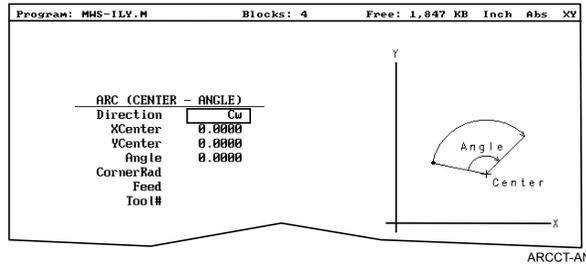


Figure 4-18, Arc (Center - Angle) Graphic Menu

To program an Arc using the center and the included angle using hot keys (the keypad):

1. In Edit Mode, press **3/ARC** to display the **ARC (ENDPOINT - RADIUS)** graphic menu.
2. Press **More... (F4)** to display the Arc Template pop-up menu. (Refer to **Figure 4-15, Arc Template Pop-Up Menu.**)
3. Highlight the **Center - Angle** template and press **ENTER** to display the **ARC (CENTER - ANGLE)** graphic menu.
4. Type the required values and settings in the entry fields.

To program an Arc using the center and the included angle using soft keys:

1. In Edit Mode, press **Mill (F5)** to activate the Mill secondary soft keys.
2. Press **Arc (F4)** to display the **ARC (END POINT - RADIUS)** graphic menu.
3. Press **More... (F4)** to display the Arc template pop-up menu. Refer to **Figure 4-15, Arc Template Pop-Up Menu.**
4. Highlight the **Center - Angle** template and press **ENTER** to display the **ARC (CENTER - ANGLE)** graphic menu.
5. Fill in the **ARC (CENTER - ANGLE)** entry fields:

Direction	Specifies a clockwise (Cw) or counterclockwise (Ccw) direction. Press +/- to toggle the setting. Selection required.
XCenter	The X coordinate of the Arc's center. Value required.
YCenter	The Y coordinate of the Arc's center. Value required.
Angle	Included angle of the Arc. Value required.
Z	The Z coordinate of the Arc endpoint. Optional.
CornerRad	Corner radius setting. Optional. (Refer to "Section 1 - CNC Programming Concepts" for details.)
Feed	Feedrate. Optional.
Tool#	Active tool. Optional.

Programming M-Code Blocks

The CNC supports M-Code functions. Enable available M-Codes at installation. Refer to the machine builder's technical data to determine which M-Codes are available.

Some programmed events initiate the same functions activated using M-Codes. Refer to **Table 4-3** for a list of the most commonly used M-Code functions.

Table 4-3, M-Code Functions

M-Code	Programmed Event	Standard Function Usage (ref. EIA Standard RS 274-D)
M02	EndMain block.	Stops spindle and coolant.
M03	Activation of a tool that lists a FWD spindle direction on the Tool Page.	Start CW spindle rotation.
M04	Activation of a tool that lists a REV spindle direction on the Tool Page.	Start CCW spindle rotation.
M05	Activation of a tool block (to allow a tool change). Activation of a tool that contains a Tool Page spindle direction OFF .	Stop spindle in normal manner.
M03, M05, M04	Execution of Tapping Cycle.	Start CW spindle rotation, stop spindle then start CCW spindle rotation.
M08	Activation of a tool with coolant ON listed on the Tool Page.	Turn coolant pump ON .
M09	Activation of a tool with coolant OFF listed on the Tool Page.	Turn coolant pump OFF .

To program an **MCode** block:

1. In Edit Mode, press **MCode (F8)**. The Graphic menu prompts for the **MCode** number and **X, Y, Z** values.
2. Type the values and press **Save (F10)** or **ENTER** to add **MCode** block to the program.

Dry Run M-Codes

In Dry Run Mode, the machine axes (X, Y, and Z) move through the program without cutting into the work. The CNC disables coolant operation and the work may or may not be placed on the table.

Activate Dry Run Mode with M-Codes 105 and 106. Deactivate it with M107. Refer to **Table 4-4**. Dry Run feedrates are set in the Setup Utility. They are often set at greater speeds than conventional feedrates. You can set them at any desired rate. Refer to the *3000M CNC Setup Utility Manual*, P/N 70000499, for details.

Table 4-4, Dry Run Mode M-Codes

M-Code	Function	Description
M105	Dry Run Mode ON , all axes	Enables machine Z Dry Run Mode. Program runs at dry run feedrates specified in the Setup Utility.
M106	Dry Run Mode ON , no Z-axis	Enables machine Dry Run Mode. Program runs at dry run feedrates specified in the Setup Utility, without moving the Z-axis.
M107	Cancel Dry Run Mode	Cancel active Dry Run Mode.

U-Axis Synchronization M-Codes

For systems that have a U-axis, the U-axis can be synchronized with XYZ or unsynchronized. The following M-codes are used to specify U-axis synchronization in Manual Data Input (MDI) or in a program. The builder sets the default configuration. Refer to **Table 4-5**.

Table 4-5, U-Axis Synchronization M-Codes

M-Code	Function	Description
M900	Activate 4 th (U) axis synchronization	Axes to be synchronized must also be programmed. Example: Mcode 9000 U will synchronize the U-axis with XYZ feedrate. All moves feedrates will be vectored.
M901	Deactivate 4 th (U) axis synchronization	Indicate axis to deactivate synchronization. Example: Mcode 9001 U will unsynchronize the U-axis with XYZ feedrate.

Section 5 - Programming Canned Cycles, Ellipses, and Spirals

Drilling Cycles

NOTE: Program all blocks by filling in the entry fields of a Graphic Menu.

Drill cycles simplify the programming required for repetitive drilling, boring, and tapping operations. Select specific drill cycles from the Program Editor's Drill (**F3**) pop-up menu:

- Basic (drill cycle)
- Pecking
- Boring
- Chip Break
- Tapping
- Drilling Off
- Pattern
- Bolt Hole
- Thread Mill
- ZMode

Drill cycles are modal. When the CNC encounters a block for any type of Drill cycle, it executes that cycle at the endpoint of each subsequent move until it encounters a **DrillOff** block. To change drill cycle parameters between moves, program a new drill block.

Program a **ZMode** block to switch the operation between two-axes and three-axes as needed in a program. To program a **ZMode** block, select **ZMode** from the **Drill (F3)** pop-up menu and then press **+/-** key to select: **TwoAxAuto**, **TwoAxStep**, or **ThreeAxes**.

If the CNC is in **ThreeAxes** mode and a **TwoAxStep** or **TwoAxAuto** block is ran, the operator is prompted to disengage the Z-axis and, subsequently, to position the Z-axis manually when needed. When in **TwoAxStep** block, the machine will stop on every move, as when drilling. When in a **TwoAxAuto** block, as when milling a contour, the motion is continuous until the CNC sees a **ZMove**.

If the CNC is in **TwoAxAuto** or **TwoAxStep** mode and a **ThreeAxes** block is ran, the operator is prompted to re-engage the Z-axis. To re-engage the Z-axis drive system, lift the quill handle until you feel the drive assembly become seated, then tighten the quick release knob. Once the Z-axis is re-engaged, the operator can jog the Z-axis to the homing mark on the front of the Z-axis drive system and (if needed) press **START** to perform a Homing sequence or **Manual (F4)** to cancel the Homing sequence. Another press of the **START** key allows the program to continue to run normally.

Basic Drill Cycle

The Basic Drill Cycle is a modal operation. When the CNC receives a **BasicDrill** command, it performs the drilling operation at the endpoint of every subsequent block until it receives a **Drilling Off** block. To change Basic Drilling dimensions cancel the current cycle and program a new cycle.

During the cycle, the tool rapids to the **StartHgt**, then Z feeds to **ZDepth**. To provide clearance for the next move, at the end of the cycle, the tool rapids to **ReturnHgt**.

Z-axis depth can be changed by placing a new Z depth on the same line as the X- and/or Y-axis location of the hole you want the new depth applied. A Z address on a line of its own will cause the control to drill the new depth at the current location.

Program a **DrillOff** block to deactivate the cycle. You can program any number of patterns and moves before turning off the cycle.

To program a **BasicDrill** block:

1. In Edit mode, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
2. Highlight **Basic**, and press **ENTER** to display the **BASIC DRILLING** Graphic Menu. Refer to **Figure 5-1**.

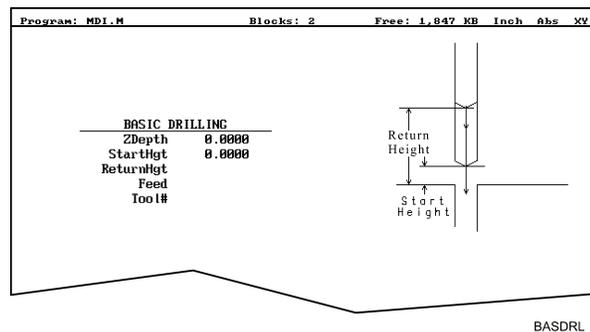


Figure 5-1, BASIC DRILLING Graphic Menu

3. Enter the required values and settings in the entry fields. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds the **BasicDrill** block to the program listing.
4. Program subsequent moves to position the tool at the required drilling location(s). The CNC will drill a hole at the endpoint of every move.
5. After programming the last drill move, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
6. Highlight **Drilling Off**, and press **ENTER** to cancel the Drilling Mode.

BASIC DRILLING entry fields:

ZDepth The absolute depth of the finished hole. (Required)

NOTE: **ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.

StartHgt The absolute Z position the CNC rapids to before feeding into the work. (Required)

ReturnHgt The absolute position to which the tool returns at the end of the cycle. (Optional)

Feed Feedrate. (Optional)

Tool# Active tool. (Optional)

Peck Drilling Cycle

Peck drilling is a modal operation. When the CNC receives a **PeckDrill** command, it peck-drills at the endpoint of every subsequent block until it receives a **Drilling Off** block. To change Peck Drilling dimensions, cancel the current **PeckDrill** cycle and program a new cycle.

The cycle starts when the CNC is in position. The tool rapids to the Z start height (**StartHgt**); feeds to the **Peck** depth; then rapids back to the **StartHgt**. This cycle repeats until the tool reaches **ZDepth**. At the end of the cycle, the tool rapids to the **ReturnHgt** to provide clearance for the next move.

To program a Peck Drilling cycle:

1. In Edit mode, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
2. Highlight **Pecking**, and press **ENTER** to display the **PECK DRILLING** Graphic Menu. Refer to **Figure 5-2**.

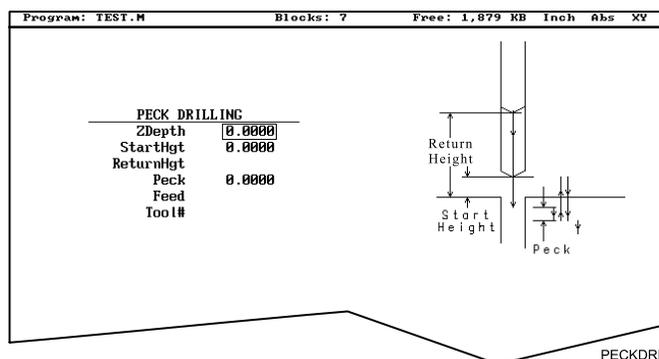


Figure 5-2, PECK DRILLING Graphic Menu

3. Enter the required values and settings in the entry fields. With the last entry field highlighted, press **ENTER**. The Display clears and the CNC adds the **PeckDrill** block to the program listing.
4. Program subsequent moves to position the tool at the required drilling location(s). The CNC will drill a hole at the endpoint of every move.

5. After programming the last drill move, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
6. Highlight **Drilling Off**, and press **ENTER** to cancel the Drilling Mode.

PECK DRILLING entry fields:

ZDepth Absolute depth of the finished hole. (Required)

NOTE: **ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.

StartHgt Absolute Z position the CNC rapids to before feeding into work. (Required)

ReturnHgt Absolute position the tool returns to at the end of the cycle. (Optional)

Peck Depth drilled in each peck. (Required)

Tool# Active tool. (Optional)

Boring Cycle

Boring is a modal operation. When the CNC encounters a **Boring** block, it executes a Boring Cycle at the endpoint of every subsequent move until it sees a **Drilling Off** block. To change Boring Cycle dimensions between moves, deactivate the cycle and program a new boring block.

The cycle starts when the CNC is in position. The tool rapids to the **StartHgt**, feeds to **ZDepth**, and then feeds back to **StartHgt**. At the end of the cycle, the tool moves to **ReturnHgt** to provide clearance for the next move.

When running a **Dwell** block, the CNC pauses at **ZDepth** for the indicated time period (in seconds). **Dwell** resolution is 0.1 sec. When you enter 0.0 sec., the CNC dwells until manually restarted.

To program a Boring cycle:

1. In Edit mode, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
2. Highlight **Boring**, and press **ENTER** to display the **BORING CYCLE** Graphic Menu. Refer to **Figure 5-3**.

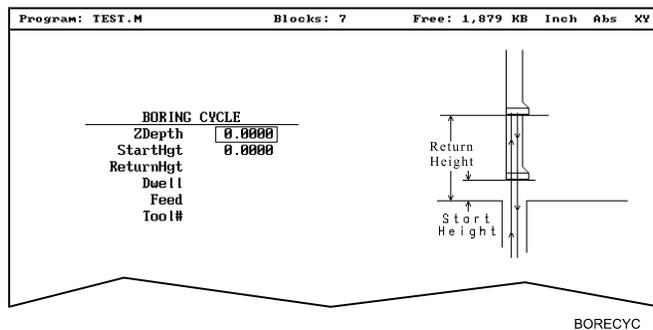


Figure 5-3, BORING CYCLE Graphic Menu

BORING CYCLE entry fields:

ZDepth Absolute depth of the finished hole. (Required)

NOTE: **ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 MM) above the work surface.

StartHgt Absolute Z position the CNC rapids to before feeding into work. (Required)

ReturnHgt Absolute position to which the tool returns at the end of the cycle. (Optional)

Dwell Length of time for pause at **ZDepth**. (Optional)

Feed Feedrate. (Optional)

Tool# Active tool. (Optional)

3. Enter the required values and settings in the entry fields. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds **Boring** block to the program listing.
4. Program subsequent moves to position the work at the required boring locations. The CNC executes the Boring Cycle at the endpoint of every move.
5. After programming the last boring position, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
6. Highlight **Drilling Off**, and press **ENTER** to cancel the cycle.

Chip Break Cycle

The Chip Break Cycle is modal. Once the CNC encounters a **ChipBreak** block, it executes the Chip Break Cycle at the endpoint of each block until it sees a **Drilling Off** block. To change Chip Break values between moves, deactivate the cycle and program a new one.

The cycle starts when the CNC is in position. The tool rapids to the **StartHgt**, feeds to the **FirstPeck**, retracts 0.02 inches [0.4 mm (default value)], then feeds to the next peck. Retract moves occur at the end of each peck in order to break the chip. This cycle repeats until the tool reaches **ZDepth**. At the end of the cycle, the tool moves to **ReturnHgt** to provide clearance for the next move.

Enter a **PeckDecr** value to decrement the depth of each peck by the specified amount. The **MinPeck** sets the minimum peck the cycle can decrement. A **ChipBrkInc** is the size of the retract move that breaks the chip.

Peck to the **RetractDep**, retract to the **StartHgt**, and then peck to the next **RetractDep** increment. The first full retract occurs one **RetractDep** increment after the first peck.

To program a Chip Break cycle:

1. In Edit Mode, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
2. Highlight **Chip Break**, and press **ENTER** to display the **CHIP BREAKING CYCLE** Graphic Menu. Refer to **Figure 5-4**.

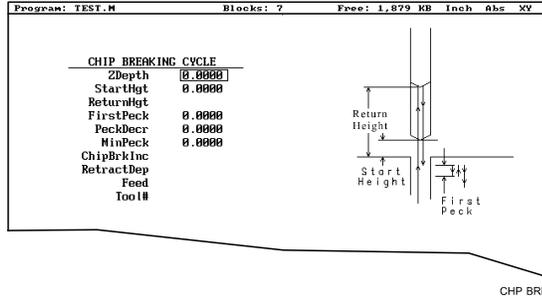


Figure 5-4, CHIP BREAKING CYCLE Graphic Menu

CHIP BREAKING CYCLE entry fields:

ZDepth Absolute depth of the finished hole. (Required)

NOTE: **ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.

StartHgt Absolute Z position to which the CNC rapids before feeding into work. (Required)

ReturnHgt Absolute position to which the tool returns at the end of the cycle. (Optional)

FirstPeck Absolute depth drilled in each peck. (Required)

PeckDecr Amount to subtract from previous peck (positive dimension). (Required)

MinPeck Smallest peck allowed. (Required)

ChipBrkInc Size of chip break retract. (Optional)

RetractDep Z increment before full retract. (Optional)

Tool# Active tool. (Optional)

3. Enter the required values and settings in the entry fields. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds the **Chip Break** block to the program listing.
4. Program the drilling location(s). The CNC will drill a hole at the endpoint of every move until it receives the **Drilling Off** command.
5. Program the **Drilling Off** command. After programming the last drill move, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
6. Highlight **Drilling Off**, and press **ENTER** to add the **DrillOff** block to the program Listing.

Tapping Cycle

The Tapping Cycle is available only on machines equipped with spindle RPM control and M-Codes (M3, M4, and M5).

In order for the cycle to operate, you must enter **Spindle RPM** value for the active tool on the Tool Page. During execution, the CNC uses the **Spindle RPM** value from the Tool Page and the programmed threads per inch (or pitch) value from the block to calculate the proper feedrate for tapping.

When the cycle runs the CNC rapids to the **StartHgt** and feeds to the **ZDepth**. The spindle stops and reverses direction to retract the tool from the hole. At **ReturnHgt**, the spindle stops and changes back to the original direction in preparation for the next programmed move.

Use the Tapping Cycle with any available pattern. A **Drilling Off** block cancels the cycle.

NOTE: The system supports spindle FWD, REV, OFF (M3, M4, M5), and spindle RPM control. At machine setup the machine builder determines which M-Codes to install.

To program a **Tapping** block:

1. In Edit mode, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
2. Highlight **Tapping**, and press **ENTER** to display the **TAPPING CYCLE** Graphic Menu. Refer to **Figure 5-5**.

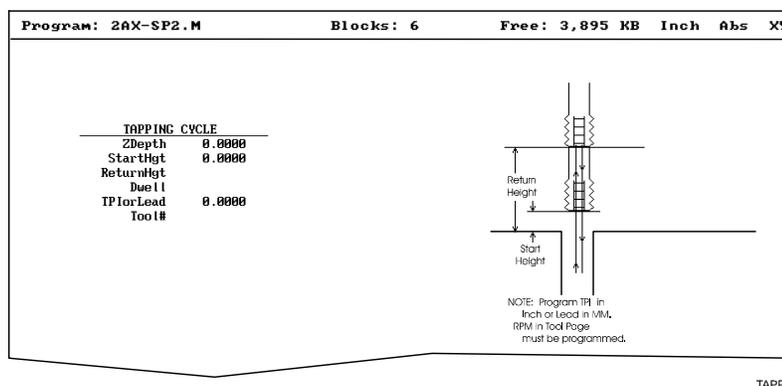


Figure 5-5, Tapping Cycle Graphic Menu

TAPPING CYCLE entry fields:

ZDepth Absolute depth of the tapped threads. (Required)

NOTE: **ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.

StartHgt Absolute Z position to which the CNC rapids before feeding into work. (Required)

ReturnHgt Absolute position to which the tool returns at the end of the cycle. (Optional)

TPIorLead	TPI in Inch or Lead in millimeters. (Required)
Tool#	Active tool. (Optional)
Dwell	Length of time for pause at top of hole and at ZDepth . (Optional)

3. Enter the required settings and values in the entry fields. With the last entry field highlighted, press **ENTER** to clear the display and add the new tapping cycle block to the program listing.
4. Program subsequent moves to position the location of the tapped holes. The CNC will tap a hole at the endpoint of every move until it receives a **Drilling Off** command.
5. To program a **Drilling Off** block, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
6. Highlight **Drilling Off**, and press **ENTER** to display the **Drill Off** block in the program listing.

Drill Pattern

The Pattern Cycle instructs the CNC to execute a pattern of regularly spaced moves. Locate a Pattern Cycle between a Drill Cycle and a **DrillOff** block. The CNC executes the Drill Cycle at every endpoint in the pattern.

In a Pattern Cycle, enter a size, location, spacing, Tool #, and rotation angle of the pattern.

To program a Drill Pattern cycle:

1. In Edit Mode, press **Drill (F3)** to display the **Drill** cycle pop-up menu.
2. Select **Pattern** from the **Drill** pop-up menu, and press **ENTER** to display the **PATTERN DRILL** Graphic Menu. Refer to **Figure 5-6**.

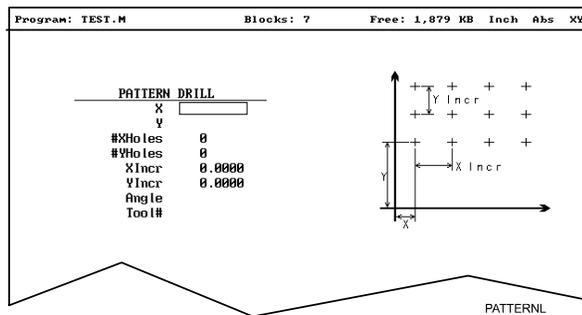


Figure 5-6, Pattern Drill Graphic Menu

PATTERN DRILL entry fields:

X	X coordinate of corner hole. If no entry made, CNC puts corner hole at present location. (Optional)
Y	Y coordinate of corner hole. If no entry made, CNC puts corner hole at present location. (Optional)

- #XHoles** Number of rows that lie along the X-axis. Must enter value greater than 0. (Required)
- #YHoles** Number of rows that lie along the Y-axis. Must enter value greater than 0. (Required)

NOTE: Enter **1** in either the **#XHoles** or the **#YHoles** field to drill a single row or column.

- Xincr** X-axis increment (spacing) of holes. (Required)
- Yincr** Y-axis increment (spacing) of holes. (Required)
- Angle** This value rotates the pattern. The XY corner hole is the pivot, and the rotation angle is the number of degrees counterclockwise from the X-axis or 3 o'clock position.

NOTE: Use a **Drilling Off** block to cancel cycle.

3. Enter the required values and settings in the entry fields.

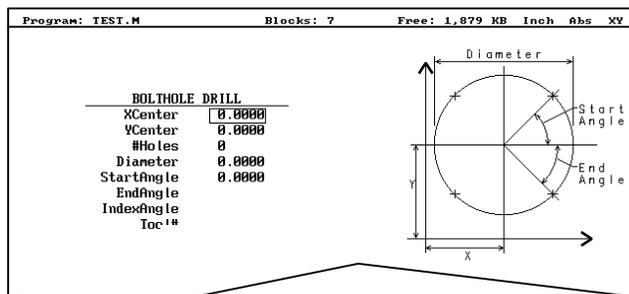
Bolt Hole Pattern

The Bolt Hole Cycle instructs the CNC to run a series of moves with endpoints that form a circular pattern. At each of these endpoints, you can run a previously programmed Drill Cycle.

You should first program a Drill Cycle to describe the hole being drilled. Then follow the Drill Cycle by one or more moves, patterns, or Bolt Hole cycles to position the CNC for the Drill Cycle. A **DrillOff** block cancels the cycle.

To program a Bolt Hole Cycle:

1. In Edit Mode, press **Drill (F3)** to display a pop-up menu.
2. Select **Bolt Hole** from the pop-up menu, and press **ENTER** to display the **BOLTHOLE DRILL** Graphic Menu. Refer to **Figure 5-7**.



BOLTHOLE

Figure 5-7, BOLTHOLE DRILL Graphic Menu

3. Enter the following required values and settings in the **BOLTHOLE DRILL** entry fields:

XCenter Absolute X center of the bolt hole pattern. If no entry is made, the CNC puts the center of the Bolt Hole pattern at X0. (Required)

YCenter Absolute Y center of the bolt hole pattern. If no entry is made, the CNC puts the center of the Bolt Hole pattern at Y0. (Required)

NOTE: Use absolute center point coordinates whenever possible.

#Holes Number of equally spaced holes in the circular pattern. (Required)

Diameter Diameter of the circular pattern. (Required)

StartAngle The number of degrees (from the 3 o'clock position) to the first hole. (Required)

EndAngle The number of degrees (from the 3 o'clock position) to the last hole. (Optional)

IndexAngle The number of degrees that the 3 o'clock reference position rotates around the center (rotates entire pattern). (Optional)

NOTE: Use a **Drilling Off** block to cancel the cycle.

Thread Milling Cycle

WARNING: The first move in this cycle is a rapid move to the center of the thread before moving the Z-axis. Make sure the tool is properly located before calling up this cycle.

Thread Milling Cycle simplifies the programming required to mill a thread. Use the thread milling for cutting inside or outside threads. It will cut either Inch or MM, left or right hand, and Z movement up or down. A single tooth or multi-toothed tool may be used. Start can be at the top or bottom of the hole or boss. Tool Length Offset is set the same as with any other tool or operation. A tool diameter also has to be set in the tool table, as cutter compensation is built into this cycle (cutter compensation is not allowed during the use of this cycle). The tool must be positioned at center of hole or boss, or the parameters for **XCenter** and **YCenter** must be specified in the cycle.

XCenter, **YCenter**, **ArcInRad**, **StockAmt**, **Passes**, **RoughFeed**, **FinFeed**, and **TaperAng** are all optional Input; all other parameters must be programmed. If the feed rates are not programmed, the CNC will use last feed rate used.

In a tapered thread the Major Diameter is always the major diameter of thread where the **ZStart** is set. Inside diameter is at finished depth and outside diameter is the diameter of boss.

To program a Thread Milling Cycle:

1. In Edit Mode, press **Drill (F3)** to display a pop-up menu.
2. Select **Thread Mill** from the pop-up menu, and press **ENTER** to display the **Thread Mill** Graphic Menu. Refer to **Figure 5-8**.

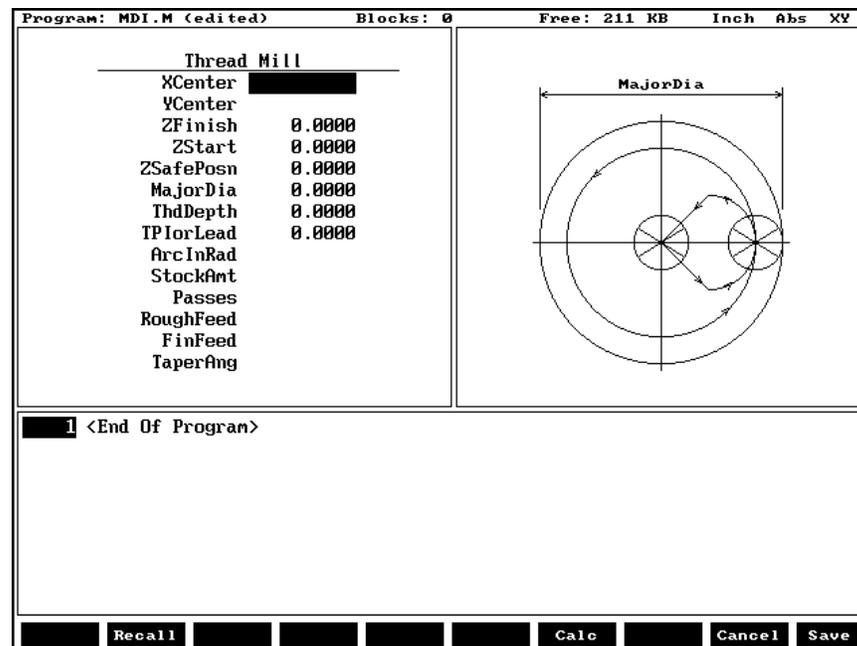


Figure 5-8, Thread Mill Graphic Menu

3. Enter the following required values and settings in the **Thread Mill** entry fields:

XCenter	Absolute X coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread as the current tool position. (Optional)
YCenter	Absolute Y coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of the thread as the current tool position. (Optional)
ZFinish	Absolute Z position where the thread cut will finish. This can be above or below the start position depending on the direction of the thread cut, up or down. (Required)
ZStart	Absolute Z position where the thread cut starts. This can be above or below the finish position depending on the direction of the thread cut, up or down. If not set, the cycle will use the current Z tool position. (Optional)
ZSafePosn	An Absolute safe Z position above the part for rapid moves in X and/or Y. (Required)

WARNING: ZSafePosn must be above the part to avoid a crash while positioning.

MajorDia	Major thread diameter. For a tapered thread, it will be the major diameter at the Z start position. If you have a tapered hole and you start at the top and cut down, you would have a different major diameter than if you started at the bottom and cut up. (Required)
	+ Value cuts in the CW direction
	- Value cuts in the CCW direction

ThdDepth	Depth of thread. The incremental depth of thread on one side: (Required)
	+ Value is inside thread
	- Value is outside thread

TPIorLead	Threads per inch (TPI) or lead of thread. (Required)
------------------	--

NOTE: The minimum number of threads per inch is "1"

ArclnRad Size of radius arcing into start of thread. (Optional)

NOTE: If **ArclnRad** is a positive value or not set and the thread is “inside,” the cycle will always return to the center between passes.

If **ArclnRad** is a negative value, the cutter will move to the start or end point that is closest to the center if inside thread, and farthest away from center if outside thread.

If **ArclnRad** is not specified and the thread is “outside,” the cutter will back away from the largest diameter by an amount equal to the thread depth.

StockAmt Amount to leave for a finish pass after the roughing passes. (Optional)

Passes Number of roughing cuts to be taken. (Optional)

NOTE: If **StockAmt** is not set or set to zero and **Passes** is 1 or 0, the cycle will make just one pass at the full depth..

If **StockAmt** is set to greater than zero and **Passes** is 1 or 0, the cycle will make just one pass at the stock depth and one pass at full thread depth.

If **Passes** is set to a negative number, all non-cutting positioning moves will be rapid.

RoughFeed Feedrate for roughing. (If not set, the cycle will use the current active feedrate.) (Optional)

FinFeed Feedrate for the finish pass. (If not set, the cycle will use the current active feedrate.) (Optional)

TaperAng Angle on one side of the thread (not the included angle). The angle is measured from the right side going counter clockwise with a positive number and clockwise with a negative number. A straight pipe tap with an inside cut would be -1.7833. If not set, then the thread is straight. (Optional)

If X and Y are not programmed, position tool center of the thread before calling the “Thread Mill” cycle:

- X and Y will rapid to the starting position of the thread.
- Z will rapid to the safe height specified in **ZSafePosn**.
- The Z axis will feed down to the start cut position **ZStart**. This could be above or below the Z position specified in the **ZFinish** position.
- Depending on what is in the **ArclnRad** parameter, the tool will arc into the first cut position.

- Spiral up or down, depending on the difference between **ZFinish** and **ZStart** and go counterclockwise or clockwise depending if **MajorDia** is plus or minus.
- Then arc-out and feed to the thread center for inside threads, or a safe distance away from the thread for outside threads depending on the value in **ArcInRad**.
- Then feed back to the **ZStart** height.
- Then feed X and Y to the next depth of cut. The depth of each roughing pass will be the thread depth specified in the **ThdDepth** parameter minus the stock amount specified in the **StockAmt** parameter, divided by the number of roughing passes specified in the **Passes** parameter.
- The cycle repeats this process until the final finish pass.
- It will then cut the thread at the full thread major diameter.

When cutting a tapered on an inside thread, care should be taken. An error will be generated if the diameter on the small end of the taper becomes too small for the tool to fit along with arc in and out moves. Not entering an arc-in value in the **ArcInRad** parameter will allow the cycle to move to the center of the hole for maximum clearance.

Sample Thread Program

This program will cut an 8 TPI thread starting at 0.1 above the hole. The major diameter of the thread is 1 inch. It will take 2 roughing cuts and 1 finish cut with a 0.625 cutter.

```
Dim Abs
Tool# 1
Rapid      X 1.0000 Y 1.0000
Rapid      Z 0.1000
ThreadMill  Finish -1.0  ZStart 0.1  ZSafePosn .5  MajorDia 1.0
            ThdDepth .0625  TPIorLead 8  ArcInRad 0.1
            StockAmt 0.002  Passes 2  RoughFeed 20.0
            FinFeed 5.0
Rapid      Z 5.0000
EndMain
```

The tool will spiral down the thread pitch of 8 threads per inch, finishing at a depth of -1. The starting height is 0.1, the safe rapid Z height is 0.5, the major thread diameter is 1 inch, and depth of thread is 0.0625. The arc-in radius is 0.1 and the stock amount for the finish pass is 0.002. The rough feedrate is 20.0 and the finish feedrate is 5.0.

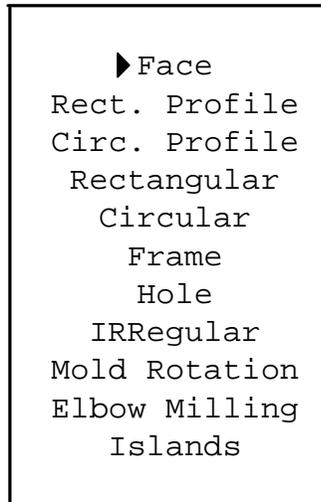
NOTE: If you would like all non-cutting positioning moves to be rapid, set **Passes** to a negative number. The idea is to initially set **Passes** as a positive number and after proving out the program, change it to a negative number for faster production. If you only need one pass to size and you want the positioning moves to be rapid, set **Passes** to -1.

Pocket Cycles

NOTE: Program all blocks by filling in the entry fields of a Graphic Menu.

Pocket canned cycles simplify the programming of repetitive moves required to mill out pockets. Select the pocket-canned cycles from the Program Editor **Pocket (F4)** pop-up menu. Refer to **Figure 5-9**.

NOTE: Programming a **Tool#** in a pocket cycle automatically activates the necessary tool compensation for that cycle.



POCKETPU

Figure 5-9, Pocket Pop-up Menu

Select specific pocket cycles from the Program Editor's Pocket (**F4**) pop-up menu:

- Facing Cycle
- Rectangular Profile Cycle
- Circular Profile Cycle
- Rectangular Pocket Cycle
- Circular Pocket Cycle
- Frame Pocket Cycle
- Hole – Mill Cycle
- Irregular Pocket Cycle
- Mold Rotation
- Elbow Milling Cycle
- Pockets with Islands

Facing Cycle

Facing cycles simplify the programming required to face the surface of a part.

Execution begins one tool radius from the start point. The selected step-over determines the approach axes.

NOTE: A **ZDepth** entry is not necessary if you program only one level plus finish stock.

Facing cycles can start in any corner of the surface and cut in any direction, depending on the sign (+/-) of the **Length** and **Width** values. Program a slightly oversize **Length** and **Width** to ensure complete facing of the surface.

At the end of the cycle, the tool rapids to **StartHgt**, then rapids back to the start position. Refer to **Figure 5-10**.

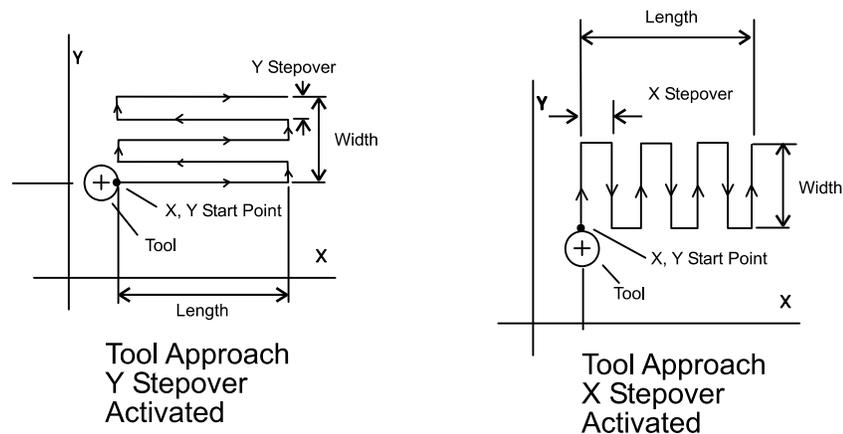


Figure 5-10, Face Cycle Tool Approach

To program a Facing cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Move the highlight to select **Face**, and press **ENTER** to display the **FACE POCKET** Graphic Menu. Refer to **Figure 5-11**.

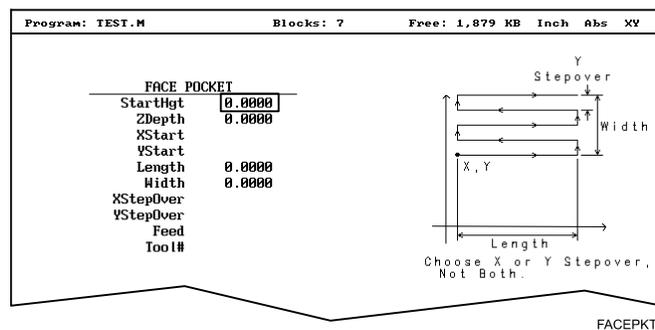


Figure 5-11, FACE POCKET Graphic Menu

NOTE: **ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.

FACE POCKET entry fields:

ZDepth	Absolute depth of the finished surface. (Required)
StartHgt	Absolute Z position the CNC rapids to before feeding into work. (Required)
XStart	X coordinate of the starting point. (Optional)
YStart	Y coordinate of the starting point. (Optional)

NOTE: Enter the required absolute **XStart** and **YStart** coordinates when possible.

Length	X-axis length to be faced. (Required)
Width	Y-axis length to be faced. (Required)
XStepover	Width of cut in the X-axis direction. When no value is entered, the CNC defaults to 70% of the active tool radius. Maximum step-over permitted is 70% of the active tool radius.
YStepover	Width of cut in the Y-axis direction. When no value is entered, the CNC defaults to 70% of the active tool radius. Maximum step-over permitted is 70% of the active tool radius.

NOTE: Enter either an X or Y step-over. Do not enter both.

NOTE: The Program Editor inadvertently will allow you to write a block containing a **Stepover** value greater than 70% of the active tool radius. To reveal this type of error, test the program in the Draw Graphics Mode.

Tool# Active tool. (Optional)

3. Enter the required values and settings in the entry fields.

Rectangular Profile Cycle

The Rectangular Profile Cycle cleans up the inside or outside profile of a rectangle. When this cycle runs, the CNC rapids to the Ramp #1 starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.

The machine feeds into the profile along Ramp #1, cuts the rectangle to the **Length** and **Width** specified, and then ramps away from the work along Ramp #2.

When cutting an inside profile, the **RECTANGULAR PROFILE** Graphic Menu displays ramp moves. Refer to **Figure 5-12**.

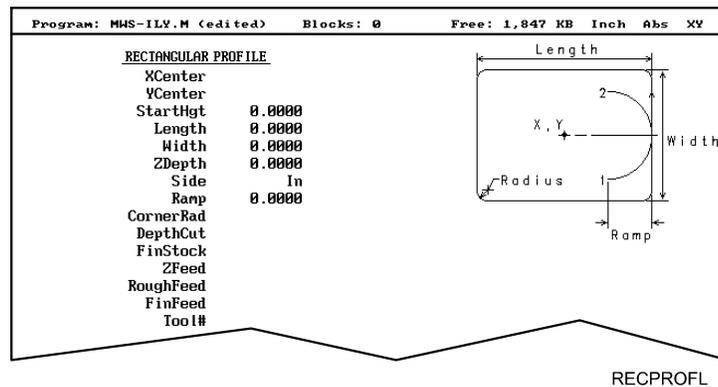


Figure 5-12, RECTANGULAR PROFILE Graphic Menu

When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2, as shown in **Figure 5-13**.

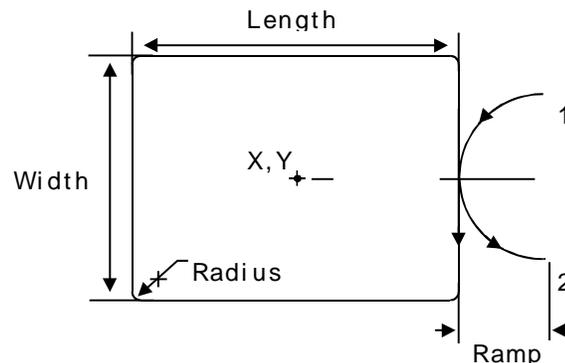


Figure 5-13, Outside Profile Ramp Moves

The Rectangular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **ProfRect** block.

When you enter a **DepthCut** value, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

When you enter a **FinStock** value, the CNC leaves the specified stock on the profile and depth for a finish pass. The CNC cuts the rectangle to the **Length**, **Width**, and **ZDepth** dimensions on the finish pass. Enter a negative **FinStock** to leave the finish stock without making a finish pass.

When you do not enter a **RoughFeed** or **FinFeed**, the CNC executes feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Rectangular Profile cycle:

In Edit Mode, press **Pocket (F4)** to display the **Pocket** pop-up menu.

1. Highlight **Rect.Profile**, and press **ENTER**. The **RECTANGULAR PROFILE** Graphic Menu prompts for labeled values.
2. Fill in the **RECTANGULAR PROFILE** entry fields labeled as follows:

XCenter	X coordinate of the center. If no coordinate is entered, the CNC centers the pocket at its present position. (Optional)
YCenter	Y coordinate of the center. If no coordinate is entered, the CNC centers the pocket at its present position. (Optional)
StartHgt	The absolute Z position to which the CNC rapids before feeding into the work. (Required)

NOTE: **StartHgt** is 0.100 inches (2.0 mm) above the work surface. **ZDepth** must be lower than **StartHgt**.

Length	Finished length of rectangle. (Required)
Width	Finished width of rectangle. (Required)
ZDepth	Absolute depth of the finished profile. (Required)
Side	Toggles cutting mode between the inside (In) or outside (Out) of the profile. Press +/- to toggle the selection. (Required)
Ramp	Radius of the ramping moves. (Required)
CornerRad	Corner radius setting. If you enter a negative value, both direction of cut and the starting and endpoints reverse. (Optional)
DepthCut	Z-axis increment used for each pass. (Optional)
FinStock	Amount of stock left by the machine before the finish pass. [Default: 0] If you enter a negative value, the CNC leaves the stock without making a finish pass. (Optional)
ZFeed	Z-axis feedrate. (Optional)
RoughFeed	Rough pass feedrate. (Optional)
FinFeed	Finish pass feedrate. (Optional)
Tool#	Active tool. (Optional)

Circular Profile Cycle

The Circular Profile Cycle cleans up the inside or outside profile of an existing circle. Refer to **Figure 5-14**.

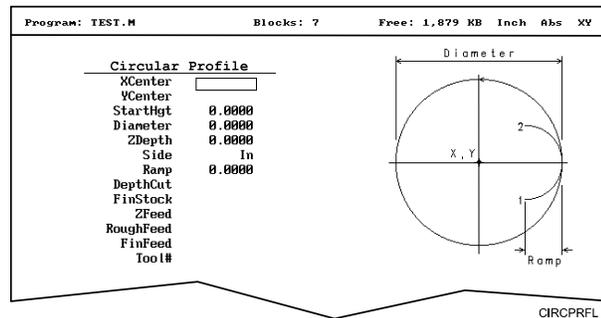


Figure 5-14, CIRCULAR PROFILE Graphic Menu

When executed, the CNC rapids to Ramp#1 starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.

The machine feeds into the profile along Ramp #1, cuts the circle to the **Diameter** specified, and then ramps away from the work along Ramp #2.

When cutting an outside profile, the tool ramps into the work along Ramp #1 and away from the work along Ramp #2 as shown in **Figure 5-15**.

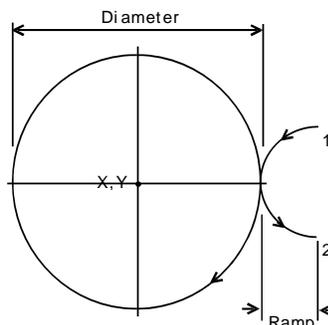


Figure 5-15, Ramp Position for Outside Profile

The Circular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **ProfCirc** block.

If you enter a **DepthCut**, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting to the **DepthCut** on each pass.

When you enter a **FinStock** value, the CNC leaves the specified stock on the profile and depth for a finish pass. The CNC finishes to the entered diameter on the finish pass. Enter a negative **FinStock** to leave the finish stock without making a finish pass.

If you do not enter a **RoughFeed** or **FinFeed**, the CNC executes feed moves at the current feedrate. **RoughFeed** controls feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Circular Profile cycle:

1. In Edit Mode, press **Pocket (F4)** to display the **Pocket** pop-up menu.
2. Highlight **Circ.Profile**, and press **ENTER** to display the **CIRCULAR PROFILE** Graphic Menu. Refer to **Figure 5-14, CIRCULAR PROFILE Graphic Menu**.

3. Fill in the **CIRCULAR PROFILE** entry fields:

XCenter	X coordinate of the center. [Default: present position] (Optional)
YCenter	Y coordinate of the center. [Default: present position] (Optional)
StartHgt	Absolute Z position to which the CNC rapids before feeding into work. (Required)

NOTE: StartHgt is 0.100 inches (2.0 mm) above the work surface.
ZDepth must be lower than **StartHgt**.

Diameter	Finished diameter of circle. If you enter a negative value, both the direction of cut and the starting and endpoints reverse. (Required)
ZDepth	Absolute depth of the finished profile. (Required)
Side	Selection for cutting on the inside of the profile (In) or the outside (Out). Press +/- to toggle the selection. (Required)
Ramp	Radius of the ramp into and away from the cut. Value required (0 value allowed).
DepthCut	Z-axis increment used for each pass. (Optional)
FinStock	Amount of stock left by the machine before the finish pass. [Default: 0] Enter a negative value to leave the stock without making a finish pass. (Optional)
ZFeed	Z-axis feedrate. (Optional)
RoughFeed	Rough pass feedrate. (Optional)
FinFeed	Finish pass feedrate. (Optional)
Tool#	Active tool. (Optional)

Rectangular Pocket Cycle

Rectangular Pocket cycles simplify the programming required to mill out rectangular pockets. When executed, the CNC rapids to the center of the lower left radius, rapids to the **StartHgt**, then ramps into the work toward the narrow center of the pocket. From the pocket center, the CNC mills increasingly larger rectangles until it reaches the specified **Length** and **Width**.

The Rectangular Pocket Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **RectPock** block.

If you enter **DepthCut**, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The CNC cuts the rectangle to the **Length**, **Width**, and **ZDepth** dimensions on the finish pass. Enter a negative **FinStock** to leave the finish stock without adding a finish pass.

If you do not enter a **RoughFeed** or **FinFeed** value, the CNC executes feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Rectangular Pocket cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Highlight **Rectangular**, and press **ENTER** to display the **RECTANGULAR POCKET** Graphic Menu. Refer to **Figure 5-16**.

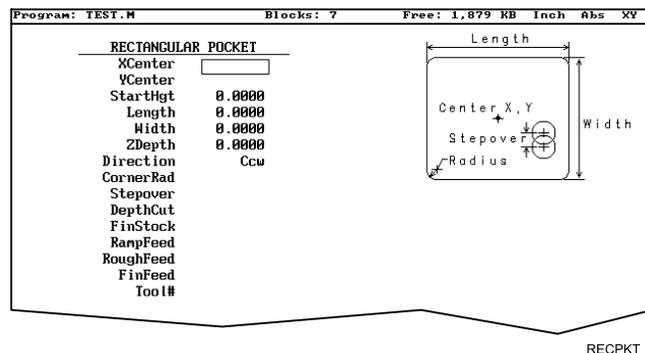


Figure 5-16, RECTANGULAR POCKET Graphic Menu

3. Fill in the **RECTANGULAR POCKET** entry fields:

XCenter X coordinate of the center. If you do not enter a coordinate, the pocket centers at the present position. (Optional)

YCenter Y coordinate of the center. If you do not enter a coordinate, the pocket centers at the present position. (Optional)

StartHgt Absolute Z position to which the CNC rapids to before feeding into the work. (Required)

NOTE: **StartHgt** is 0.100 inches (2.0 mm) above the work surface. **ZDepth** must be lower than **StartHgt**.

Length Inside Y length of the finished pocket. (Required)

Width Inside X length of the finished pocket. (Required)

ZDepth Absolute depth of the finished pocket. (Required)

Direction Allows you to select a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the selection. (Required)

Stepover Width of cut. If you do not enter a value, the CNC defaults to 70% of the active tool radius. The maximum step-over permitted is 70% of the active tool diameter. (Optional)

DepthCut Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock. (Optional)

FinStock Amount of stock left by the machine before the finish pass. [Default: **0**] Enter a negative value to leave the stock without making a finish pass. (Optional)

ZFeed Z-axis feedrate. (Optional)

RoughFeed Rough pass feedrate. (Optional)

FinFeed Finish pass feedrate. (Optional)

Tool# Active tool. (Optional)

Circular Pocket Cycle

Circular Pocket cycles simplify the programming of circular pockets. When executed, the CNC rapids to the center, rapids to the **StartHgt**, and then ramps into the work. The tool will circle outward from the center starting position, until it reaches the pocket **Diameter**. The tool circles back toward the center until the pass is complete.

The Circular Pocket Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **CircPock** block.

Use **DepthCut** to specify the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The cycle cuts the profile to the **Diameter** and **ZDepth** dimensions on the finish pass. Enter a negative **FinStock** to leave the finish stock without making a finish pass.

Leave **RoughFeed** and **FinFeed** blank to execute feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Circular Pocket cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Move the highlight to select **Circular**, and press **ENTER** to display the **CIRCULAR POCKET** Graphic Menu. Refer to **Figure 5-17**.

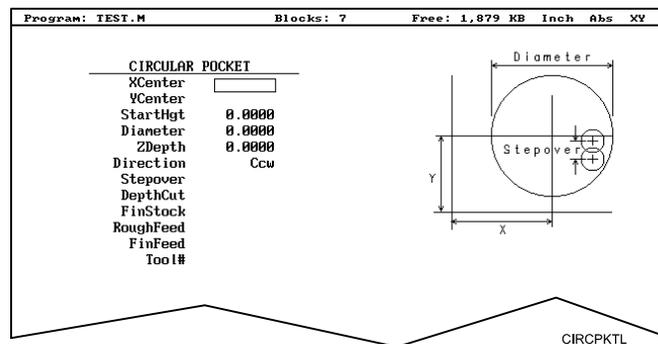


Figure 5-17, CIRCULAR POCKET Graphic Menu

3. Fill in the **CIRCULAR POCKET** entry fields:

XCenter	X coordinate of the center. [Default: present position] (Optional)
YCenter	Y coordinate of the center. Default: present position] (Optional)
StartHgt	Absolute Z position to which the CNC rapids before feeding into the work. (Required)

NOTE: **StartHgt** is 0.100 inches (2.0 mm) above the work surface.
ZDepth must be lower than **StartHgt**.

ZDepth	Absolute depth of the finished hole. (Required)
Direction	Allows you to select a clockwise (Cw) or counterclockwise (Ccw) direction. Press +/- to toggle the selection. (Required)
Stepover	Width of cut. If no value is entered, CNC defaults to 70% of the active tool radius. Maximum step-over permitted is 70% of the active tool diameter. (Optional)
DepthCut	Depth the machine takes in a single pass. Defaults to a single ZDepth cut minus the finish stock. (Optional)
FinStock	Amount of stock left by the machine before the finish pass. [Default: 0] Enter a negative value to leave the stock without making a finish pass. (Optional)
RoughFeed	Rough pass feedrate. (Optional)
FinFeed	Finish pass feedrate. (Optional)
Tool#	Active tool. (Optional)

Frame Pocket Cycle

A Frame Pocket Cycle simplifies the programming required to mill out a Frame. When executed, the CNC rapids to a starting position near the island, rapids to **StartHgt**, then ramps into the work while moving across the **Frame**. The CNC cuts from the outside edge to the island in rectangles of decreasing size to complete the pass.

The Frame Pocket Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **FramePock** block.

If you enter a **DepthCut** value, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The CNC cuts the frame to the **IslandLen**, **IslandWid**, and **FrameWidth** dimensions on the finish pass. Enter a negative **FinStock** value to leave the finish stock without adding a finish pass.

Leave **RoughFeed** and **FinFeed** blank to execute feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Frame Pocket cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Highlight **Frame**, and press **ENTER** to display the **FRAME POCKET** Graphic Menu. Refer to **Figure 5-18**.

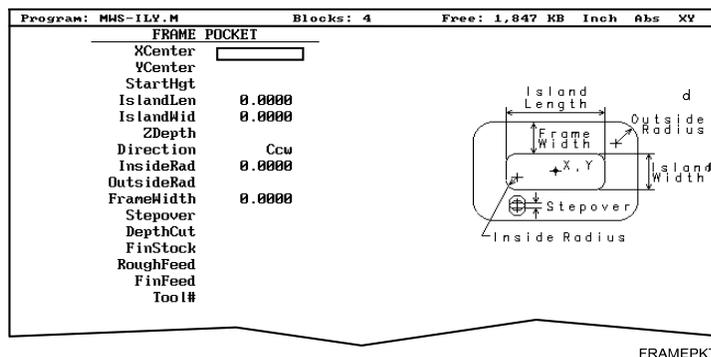


Figure 5-18, FRAME POCKET Graphic Menu

3. Fill in the **FRAME POCKET** entry fields:

XCenter	X coordinate of the center. [Default: pocket centers at present position] (Optional)
YCenter	Y coordinate of the center. [Default: pocket centers at present position] (Optional)
StartHgt	Absolute Z position to which the CNC rapids before feeding into the work. (Required)

NOTE: StartHgt is 0.100 inches (2.0 mm) above the work surface.
ZDepth must be lower than **StartHgt**.

IslandLen	Outside length (X-axis) of finished island. (Required)
IslandWid	Outside width (Y-axis) of finished island. (Required)
StartHgt	Absolute Z position to which the CNC rapids before feeding into the work. (Required)
Direction	Allows you to select conventional or climb milling for the pocket. The selections are clockwise (Cw) and counterclockwise (Ccw). Press +/- to toggle the selection. (Required)
InsideRad	Radius of the island corners. (Required)
OutsideRad	Outside radius of the frame corners. (Required)
FrameWidth	Width of the finished frame. (Required)
Stepover	Width of cut. If no value is entered, the CNC defaults to 70% of the active tool radius. Maximum step-over permitted is 70% of the active tool diameter. (Optional)
DepthCut	Z-axis increment used for each pass. (Optional)
FinStock	Amount of stock left by the machine before the finish pass. [Default: 0] Enter a negative value to leave the stock without making a finish pass. (Optional)
RampFeed	Z-axis feedrate. (Optional)
RoughFeed	Rough pass feedrate. (Optional)
FinFeed	Finish pass feedrate. (Optional)
Tool#	Active Tool. (Optional)

Hole - Mill Cycle

Use Hole Mill cycles to cut through holes, clean up the inside diameter of existing holes, or counter-bore existing holes. When executed the CNC rapids to the ramp, feeds into the circumference along the ramp, and cuts to the **Diameter**. After it completes the hole, the CNC ramps away from the circumference and rapids back to the center.

The Hole Mill Pocket Cycle automatically compensates tool diameter. Activate the correct tool diameter before or within the **HolePock** block.

Use **StartHgt** and **ZDepth** together, if at all. Enter a **DepthCut** to execute the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** amount on each pass.

Use **FinStock** to leave the specified amount on the profile and make an additional pass cutting to the **Diameter**. Enter a negative **FinStock** value to leave finish stock, without executing a finish pass.

Leave **RoughFeed** and **FinFeed** blank to execute feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Hole Mill Pocket cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Highlight **Hole**, and press **ENTER** to display the **HOLE-MILL POCKET** Graphic Menu. Refer to **Figure 5-19**.

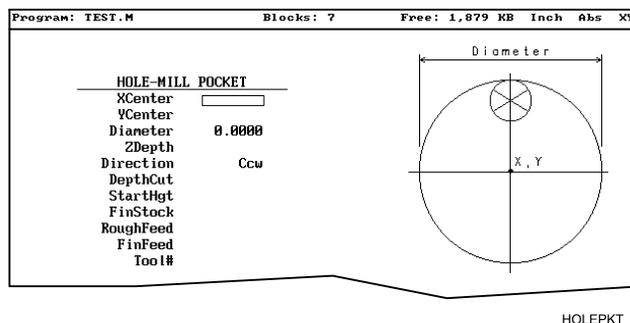


Figure 5-19, HOLE-MILL Pocket Graphic Menu

3. Fill in the **HOLE-MILL POCKET** entry fields:

XCenter	X coordinate of the center. [Default: pocket centers at present position] (Optional)
YCenter	Y coordinate of the center. If no coordinate is given, it defaults to present position. (Optional)
Diameter	Diameter of the pocket. (Required)
ZDepth	Absolute depth of the finished pocket. (Optional)

NOTE: Use **StartHgt** and **ZDepth** together, if at all. **ZDepth** must be lower than **StartHgt**.

Direction	Allows you to select a clockwise (Cw) or counterclockwise (Ccw) direction. Press +/- to toggle the selection. (Required)
DepthCut	Z-axis increment used for each pass. (Optional)
StartHgt	Absolute Z position to which the CNC rapids before feeding into the work. (Optional)
FinStock	Amount of stock left by the machine before the finish pass. [Default: 0] Enter negative value to leave a finish stock, without making a finish pass. (Optional)
RoughFeed	Rough pass feedrate. (Optional)
FinFeed	Finish pass feedrate. (Optional)
Tool#	Active tool. (Optional)

Irregular Pocket Cycle

An irregular pocket cycle simplifies the programming of an irregular pocket. An **IRRegular** pocket block must contain a subprogram. The main portion of the block holds the information required to mill the pocket. The moves in the subprogram define the pocket's profile.

Begin the subprogram with a Rapid move to an absolute starting position on the profile. Following blocks provide the moves that define the outline. After the profile is completed, return to the starting position.

NOTE: To test the subprogram that defines the outline, program a temporary subprogram call in the beginning of the main program and run the program in Simulation Draw Mode. Delete or comment out the subprogram call before actually running the program. Refer to "Subprograms" in this section for information on how to program a subprogram. Refer to "Section 6 - Editing Programs" for information on how to comment out a block.

The cycle mills out the pocket interior in progressive side-to-side moves that step over across the pocket. The machine reverses direction when it reaches any part of the pocket outline. You must orient the step-overs, starting point, and direction so that an irregularity in the outline does not prevent the entire pocket from being milled. Refer to **Figure 5-20**.

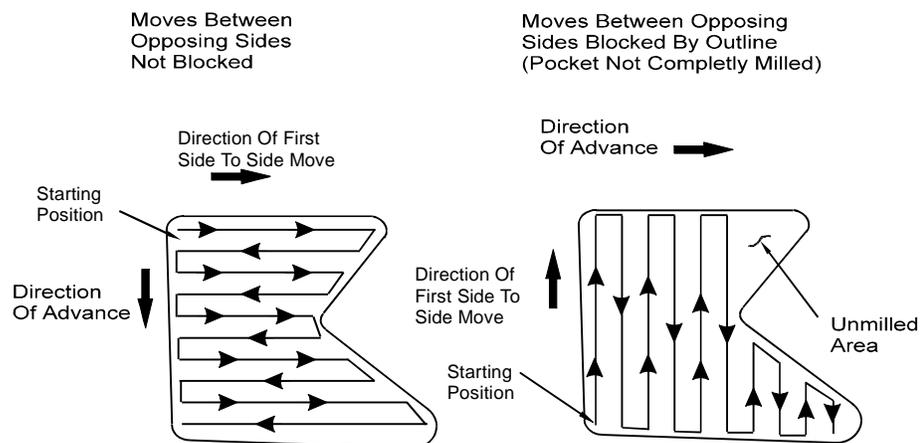


Figure 5-20, Stepover Move Orientation

Determining Move Direction

Two factors determine how moves step over across the pocket:

- The starting position of the first move.
- The direction of the first move.

The CNC uses the moves in the subprogram to generate the default starting position and the direction of the first move. Refer to **Figure 5-21**.

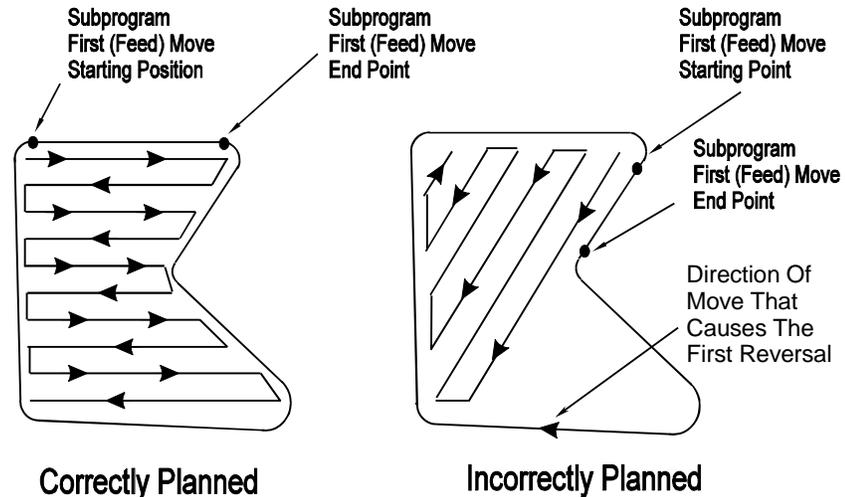


Figure 5-21, Orienting Steptover from the Subprogram

You should set the default starting position at the intersection of the first and last moves in the subprogram (compensated for tool diameter and **Finstock**). If this was done, do not enter values in the **Angle**, **X**, and **Y** fields. The default direction will be parallel to the first Line move in the subprogram.

When the **Angle**, **X**, and **Y** fields are left blank and the first move of the subprogram is an Arc, the CNC sets the default starting position at the intersection of the Arc and the last feed move in the subprogram (compensated for tool diameter and **Finstock**). The default starting direction will be parallel to a tangent drawn through the Arc's endpoint.

The CNC will step over perpendicular to the first move, following the outline of the subprogram's moves.

NOTE: Use a Line move for the first move in the subprogram whenever possible.

For most irregular pockets, leave the **Angle**, **Xstart**, and **YStart** fields blank.

Enter an **Angle** to force the direction of the first move to the entered absolute angle. The angle must point to a direction inside the pocket in order for the cycle to run. Refer to **Figure 5-22**.

NOTE: When an Arc in the subprogram defines the start position of the cycle and you specify an **Angle** value, ensure that the angle points inside the pocket. An angle tangent to the starting point of an Arc does not point to the inside of the Arc. If necessary, adjust the angle so that it points toward the inside of the pocket (± 1.00 degrees).

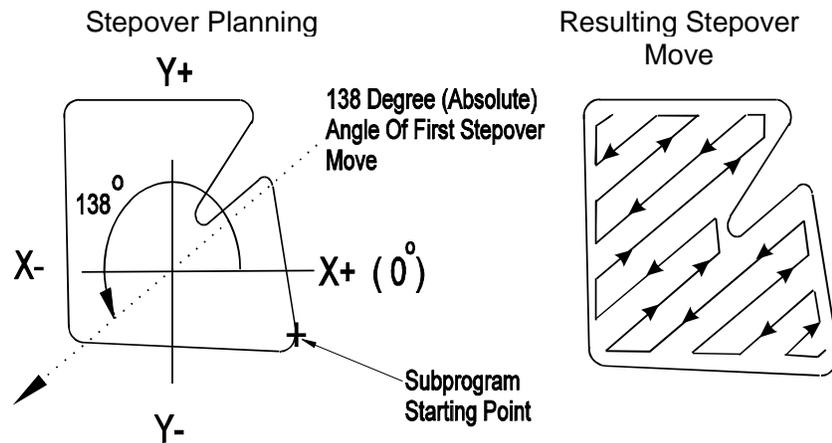


Figure 5-22, Specifying Move Angle

Enter **X** and **Y** values to force the starting position of the first move to the specified position, compensating for tool diameter and **FinStock**.

Irregular Pocket Execution

When executed, the CNC rapids to the starting position, rapids to **StartHgt**, then moves to the Z depth of the first pass.

The Irregular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **Pocket** block.

Use **XStart** and **YStart** to start the cycle with a ramping move from the **XStart**, **YStart** position to the step-over starting position.

The CNC mills out the pocket with side-to-side moves that step over across the pocket.

Use **DepthCut** to specify the number of passes required to get from the top of the part to **ZDepth**, cutting to the **DepthCut** on each pass.

Use **FinStock** to leave specified stock on the profile and depth for a finish pass. The CNC cuts the pocket to final dimensions on the finish pass. Enter a negative **FinStock** value to leave the finish stock without making a finish pass.

When you do not enter a **RoughFeed** or **FinFeed** value, the CNC executes feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program an Irregular Pocket cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Highlight **IRRegular**, and press **ENTER** to display the **IRREGULAR POCKET** Graphic Menu. Refer to **Figure 5-23**.

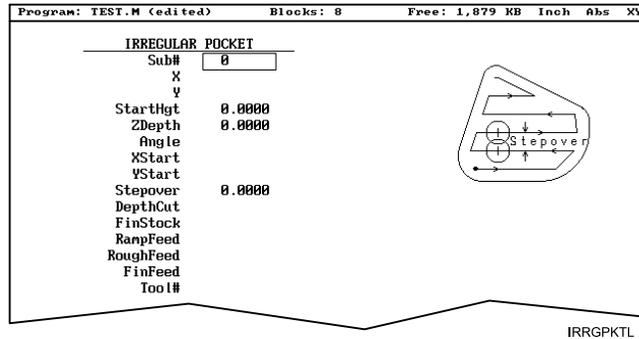


Figure 5-23, IRREGULAR POCKET Graphic Menu

3. Fill in the **IRREGULAR POCKET** entry fields:

Sub#	Subroutine that defines shape of the pocket. (Required)
X	X coordinate of the step-over starting position. If no value is entered, the CNC defaults to the starting position of the first move of the subprogram. (Optional)
Y	Y coordinate of the step-over starting position. If no value is entered, the CNC defaults to the starting position of the first move of the subprogram. (Optional)

NOTE: Use **X** and **Y** values together, if at all.

StartHgt	Absolute Z position to which the CNC rapids before feeding into work. (Required)
-----------------	--

NOTE: **StartHgt** is 0.100 inches (2.0 mm) above the work surface to be cut. **ZDepth** must be lower than **StartHgt**.

ZDepth	Absolute depth of the finished pocket. Required.
Angle	Absolute angle of the first move. If blank, the first move of the step-over is parallel to the first line move of the subprogram. (Optional)
XStart	X coordinate of the ramp move to the starting position. (Optional)

YStart Y coordinate of a ramp move to the starting position. (Optional)

NOTE: Use XStart and YStart values together, if at all.
--

Stepover Width of cut. (Required)

DepthCut Z-axis increment used for each pass. (Optional)

FinStock Amount of stock left by the machine before the finish pass. [Default: **0**] Use a negative value to leave stock without making a finish pass. (Optional)

RampFeed Z-axis feedrate. (Optional)

RoughFeed Rough pass feedrate. (Optional)

FinFeed Finish pass feedrate. (Optional)

Tool# Active tool. (Optional)

Using Irregular Pocket in Subprograms

The Irregular Pocket is typically used with the main program. In cases where the Irregular Pocket is being used in a subprogram, an M-code must be used to specify the tool number whose tool diameter must be used in the calculations of the Irregular Pocket. The M-code must be placed immediately before the Irregular Pocket. If Irregular Pocket is used in the main program, it is not necessary to use this M-code.

Format: MCode 9367 X (tool#)

Tool# is the number of the tool diameter whose diameter must be used in the calculations of the Irregular Pocket.

Example:

```
MCode 9367 X 3.0000
Pocket      Sub# 1 StartHgt 0.1000 (etc.)
```

This will use the diameter of Tool# 3 in the calculations of the Irregular Pocket.

Pockets with Islands

This cycle allows islands in irregular pockets. The main pocket must be the lowest subroutine number. Normally, this would be one (1). Pockets with Islands can be programmed using:

- DXF (see "Section 15, Using DXF for Pockets with Islands")
- Subroutines

More than one Island cycle can be programmed at a time. They may be strung together, but on separate lines. Islands can be programmed inside of islands. Five islands can be put on a line. The shape number subroutine number is used as inputs.

Using Subroutines for Pockets with Islands

This example using subroutines for Pockets with Islands uses the following illustration. See **Figure 5-24** and **Table 5-1 Pockets with Islands Subroutines Programming Example**.

The numbers are the subroutine numbers.

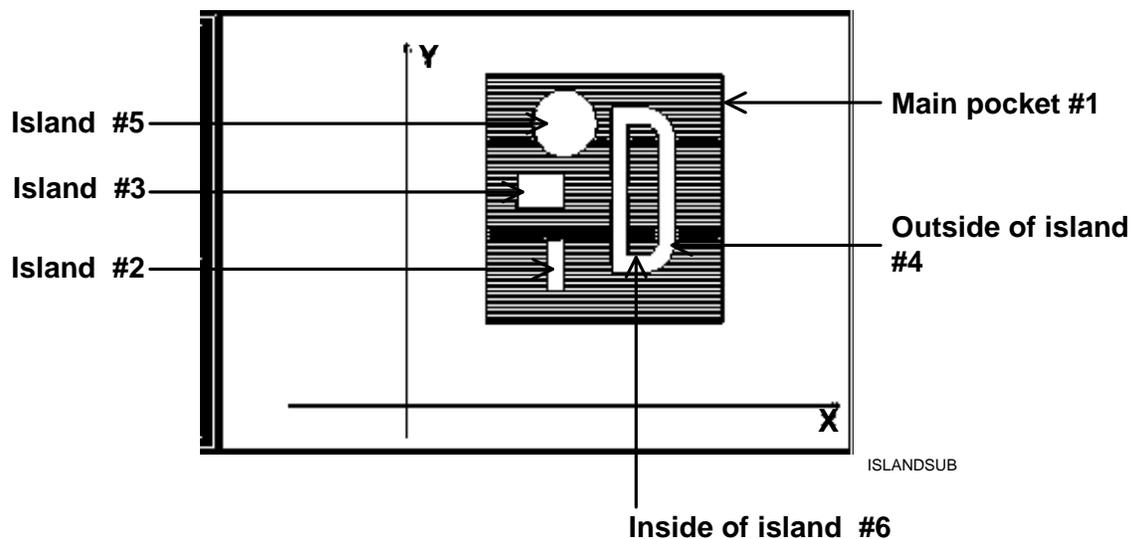


Figure 5-24, Subroutines Pockets with Islands Example Workpiece

In **Table 5-1** Island # 4 (FourthIsl) has a – (minus) in front of it, this is because the comp needs to be on the inside, as it is a pocket inside of an island.

Table 5-1 Pockets with Islands Subroutines Programming Example

```

Unit Inch
Dim Abs
Tool# 0
Rapid          X 0.0000 Y 0.0000 Z 0.0000 Feed 50.0000
Islands        FirstIsl 4.0000 SecondIsl 3.0000 ThirdIsl 2.0000
FourthIsl -6.0000 FifthIsl 5.0000
Pocket         Sub# 1.0000 StartHgt 0.2000 ZDepth -0.9000 Stepover
0.1900 DepthCut 0.2500 RoughFeed 50.0000 FinStock 0.0100
FinFeed 50.0000
RampFeed 50.0000 Tool# 1.0000
MCode 5 Z 5.0000
EndMain
Sub 1
Rapid          X 5.0000 Y 5.0000 Feed 50.0000
Line           X 20.0000
Line           Y 20.0000
Line           X 5.0000
Line           Y 5.0000
EndSub
Sub 2
Rapid          X 9.0000 Y 7.0000 Feed 50.0000
Line           X 10.0000
Line           Y 10.0000
Line           X 9.0000
Line           Y 7.0000
EndSub
Sub 3
Rapid          X 7.0000 Y 12.0000 Feed 50.0000
Line           X 10.0000
Line           Y 14.0000
Line           X 7.0000
Line           Y 12.0000
EndSub
Sub 4
Rapid          X 13.0000 Y 8.0000 Feed 50.0000
Line           X 15.0000
Arc Ccw        X 17.0000 Y 10.0000 Radius 2.0000
Line           Y 16.0000
Arc Ccw        X 15.0000 Y 18.0000 Radius 2.0000

```

Line	X 13.0000
Line	Y 8.0000
EndSub	
Sub 5	
Rapid	X 8.0000 Y 17.0000 Feed 50.0000
Arc Ccw	X 12.0000 Y 17.0000 Radius 2.0000
Arc Ccw	X 8.0000 Y 17.0000 Radius 2.0000
EndSub	
Sub 6	
Rapid	X 14.0000 Y 9.0000
Line	X 15.0000
Arc Ccw	X 16.0000 Y 10.0000 Radius 1.0000
Line	Y 16.0000
Arc Ccw	X 15.0000 Y 17.0000 Radius 1.0000
Line	X 14.0000
Line	Y 9.0000
EndSub	

Subprograms

Program repetitive operations in a subprogram called from the main program.

- Call (or nest) subprograms within other subprograms. The CNC supports up to ten levels of nesting.
- Repeat or loop subprograms moving along any axis in increments each time the loop runs.
- Rotate, scale, or mirror subprograms.

The following examples describe two situations where subprograms save time.

Situation: 1 (Repetitive Drilling Cycle)

When a workpiece must be center-drilled, drilled, then counterbored, each of the three tools must go to the same hole positions consecutively. Ten hole positions would require thirty programmed hole locations (ten for each tool). Program the ten hole locations in a subprogram called three times from the main program (once for each tool).

Situation: 2 (Rough and Finish Cycles)

Use subprograms for jobs that require both roughing and finishing cycles. Rough out the outside of a workpiece with a roughing mill, then finish it with a finishing mill. Program the profile in a subprogram. The main program calls the subprogram twice, once for each tool. You can set the tool diameter to .5300 inches for the .5-inch roughing mill and to .5000 inches for the .5-inch finishing mill. Tool #1 will leave 0.0150 inch excess stock per side. Tool #2 finishes the work to size.

Subprogram Structure

When using subprograms, define the end of the main program and the start and end of each subprogram.

Subprogram Example

1	Dim Abs		
2	Rapid	X 5.0000	Y -5.0000
3	Call 1		
4	Rapid	X 6.0000	Y -6.0000
5	Call 1		
6	Rapid	X 7.0000	Y -5.0000
7	Call 1		
8	EndMain		End of Main Program
9	Sub 1		Start of Subprogram 1
10		Z -0.0625	
11	Dim Incr		
12	Line	X 0.375	
13	Line	Y 0.375	
14	Line	X -0.375	
15	Line	Y -0.375	

```
16 Dim Abs
17           Z 0.1000
18 EndSub
19 <End Of Program>
```

End of Subprogram 1
End of Program

The main program begins at Block 1 and runs through Block 8. The subprogram begins at Block 9 and runs through Block 18.

When the main program reaches Block 3, the CNC jumps to Block 9, runs the subprogram through Block 18, then returns to the main program, Block 4.

Blocks 5 and 7 call the subprogram again.

Organizing Programs Containing Subprograms

To write a program that includes a subprogram:

1. Write the main program and include the subprogram **Call** blocks.
2. Insert an **EndMain** block at the end of the main program.
3. Insert a **Sub** block, followed by a unique subprogram call number (1 to 9999), on the first block of the subprogram. Example: Sub1.
4. Write the subprogram blocks.
5. Finish the subprogram with an **EndSub** block.
6. End the program with an **<End of Program>** block.

Calling Subprograms from the Main Program

To call a subprogram from the main program:

1. In Edit Mode, press **Sub (F8)** to display the Subprogram soft key labels.
2. Press **Call (F3)**. The Graphic Menu prompts for the subprogram number.
3. Enter a subprogram number, and press **ENTER** to add **Call** block to program.

Ending Main Programs

To program an **EndMain** block:

1. In Edit Mode, press **Sub (F8)** to display the Soft key subprogram labels.
2. Press **EndMain (F4)** to display the **EndMain** block in the program listing.

Starting Subprograms

Start subprograms with a **Sub** block.

To program a **Sub** block:

1. In Edit Mode, press **Sub (F8)** to display the **Sub** soft keys.
2. Press **Sub (F2)**. The CNC prompts for subprogram number.
3. Enter **Sub** number, and press **ENTER** to add a **Sub** block to the program. The **Sub** number must agree with the matching **Call** number.

Ending Subprograms

End subprograms with an **EndSub** block.

To program an **EndSub** block:

1. In Edit Mode, press **Sub (F8)** to display the **Sub** soft keys.
2. Press **EndSub (F2)** to add an **EndSub** block to the program.

Looping Subprograms

Looping subprograms repeat a set number of times before they return to the main program. The CNC tracks the number of loops.

NOTE: Only subprograms can loop.

To call a Loop subprogram:

1. In Edit Mode, press **Sub (F8)** to display the **Sub** soft keys.
2. Press **Loop (F5)** to activate the **LOOP SUB** Graphic Menu.
3. Fill in the **LOOP SUB** entry field:

Sub#	Subprogram identification number. (Required)
#Loops	Number of times loop repeats before it returns to the main program. (Required)
Xincr	Distance X-axis increments every cycle. (Optional)
Yincr	Distance Y-axis increments every cycle. (Optional)
Zincr	Distance Z-axis increments every cycle. Cannot be used with Xincr or Yincr . (Optional)
ZFeed	Feedrate used with Zincr . (Optional)
Tool#	Active tool. (Optional)

Rotating, Mirroring, and Scaling Subprograms (RMS)

Use **RMS** blocks to scale, rotate, and/or mirror subprograms. These functions turn off when the subprogram ends.

To call an RMS subprogram:

1. In Edit Mode, press **Sub (F8)**. Soft Key labels display subprogram functions.
2. Press **RMS (F6)**. The **ROTATE/MIRROR/SCALE SUB** Graphic Menu prompts for labeled values. Refer to **Figure 5-25**.

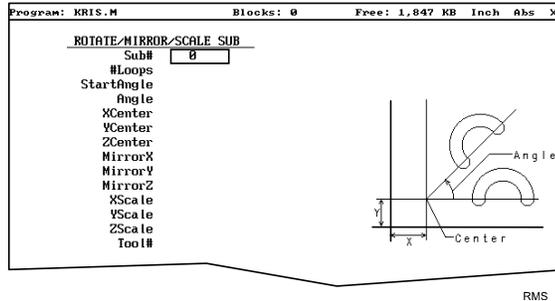


Figure 5-25, ROTATE/MIRROR/SCALE SUB Graphic Menu

3. Fill in the **ROTATE/MIRROR/SCALE SUB** entry fields defined below:

Sub# Subprogram number. (Required)
#Loops Number of times subprogram will loop before it returns to the main program. (Optional)

NOTE: RMS subprograms loop only when rotating.

StartAngle Number of degrees the pattern rotates for the first loop. (Optional)

NOTE: Sometimes, it is easier to program a part from the 3 o' clock position, then rotate it to desired angle.

Angle Number of degrees the pattern rotates per loop. (Optional)

XCenter Point of rotation X coordinate. (Optional)

YCenter Point of rotation Y coordinate. (Optional)

MirrorX Press **+/-** to toggle between **Yes** and **No**. If **Yes**, CNC mirrors the X-axis values. (Optional)

MirrorY Press **+/-** to toggle between **Yes** or **No**. If **Yes**, mirrors the Y-axis values. (Optional)

XScale X-axis scale factor. Multiplies all X-axis positions by the number entered. (Optional)

YScale Y-axis scale factor. Multiplies all Y-axis positions by the number entered. (Optional)

Tool# Active tool. (Optional)

Ellipses and Spirals

NOTE: It is possible for you to inadvertently write a block containing illogical entries. For safety, the CNC stops a program and generates a message when it detects an illogical block. Test programs in Draw to detect these blocks before machining a part.

Plane Selection

Refer to “Section 1 – CNC Programming Concepts” for basic information on planes. XY is the default plane for Ellipses and Spirals. For an XZ or YZ Ellipse or Spiral, program the plane change before the move.

The plane block allows the CNC to configure the Ellipse and Spiral Graphic Menus for the new plane.

The Graphic Menus for moves in the XY, XZ, and YZ planes contain the same entry fields. Entry fields for in-plane positions always require a value. Entry fields for off-plane positions do not.

After programming moves in other planes, always return to the XY plane.

Programming an Ellipse

The Ellipse Cycle simplifies the programming required to cut an ellipse. When the Ellipse Cycle runs, the CNC feeds from its current location, around the ellipse, to the endpoint. The start point and the endpoint must lie along the ellipse. Use incremental values to program an ellipse. The cutter must be in place before the ellipse is executed.

Standard tool compensation cannot be used with the Ellipse Cycle. Use the **CompSide** entry to activate tool compensation on an ellipse.

- | | |
|-------------|---|
| In | The CNC offsets the programmed path by the tool radius to the inside of the ellipse. |
| Out | The CNC offsets the programmed path by the tool radius to the outside of the ellipse. |
| None | (leave entry field blank). No compensation is activated during the cycle. |

Press the +/- key to toggle the selection.

NOTE: To clear the entry field, press CLEAR.

To program an Ellipse:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft key labels.
2. Press **More (F7)** to display the **More** pop-up menu.
3. Highlight **Ellipse**, and press **ENTER** to display the **ELLIPSE** Graphic Menu. Refer to **Figure 5-26**.

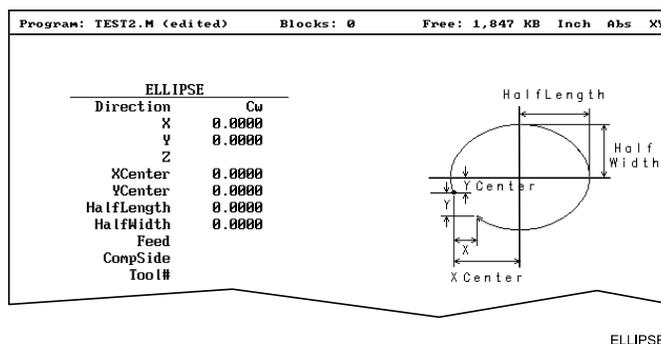


Figure 5-26, ELLIPSE Graphic Menu

4. Fill in the **ELLIPSE** entry fields:

- Direction** Allows you to select a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the selection. (Required)
- X** Incremental X coordinate of endpoint. (Required)
- Y** Incremental Y coordinate of endpoint. (Required)
- Z** Incremental Z coordinate of endpoint. (Required)
- XCenter** X coordinate of ellipse centerline. (Required)
- YCenter** Y coordinate of ellipse centerline. (Required)
- HalfLength** Half the overall X dimension of the ellipse. Incremental (Required)
- HalfWidth** Half the overall Y dimension of the ellipse. Incremental (Required)
- Feed** Feedrate. (Optional)
- CompSide** Offsets tool path to the inside (**In**) or outside (**Out**) of the ellipse by the tool radius. Refer to **Figure 5-27, Ellipse Tool Compensation**. If you leave the entry blank, no tool compensation is activated during the cycle. (Optional)
- Tool#** Active tool. (Optional)

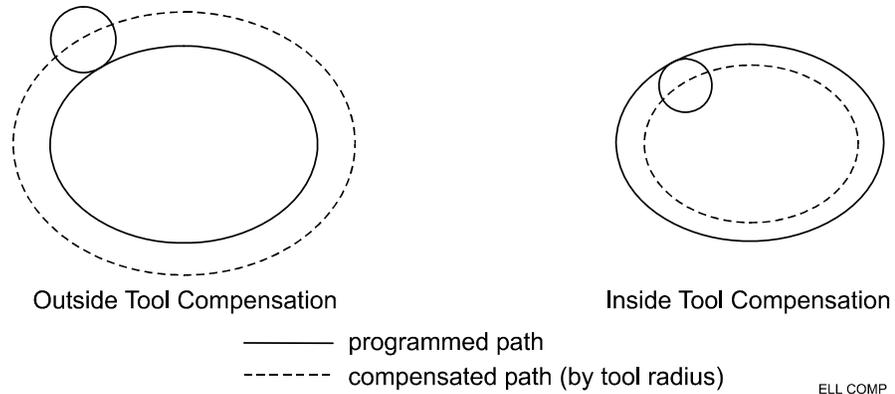


Figure 5-27, Ellipse Tool Compensation

Programming a Spiral

A spiral is an arc with a continuously changing radius. To program a spiral, enter the **Direction** of the cut, the **X** and **Y** coordinates of the endpoint, the coordinates of the center (**XCenter**, **YCenter**) and the number of revolutions (**Revs**). Program a spiral using incremental values.

When the cycle runs, the CNC cuts from its present position to the endpoint, spiraling around the center for the required number of revolutions. Each time the tool spirals past the starting point, it counts as one revolution. The machine automatically calculates and executes the required number of revolutions. There are two ways to set up spirals:

- Outward spirals** the present position is closer to the center than to the endpoint.
- Inward spirals** the endpoint is closer to the center than to the machine's current position.

To program a Spiral cycle:

1. In Edit Mode, press **Mill (F5)** to display the Mill soft key labels.
2. Press **More (F7)** to display the **More** pop-up menu.
3. Highlight **Spiral**, and press **ENTER**. The **SPIRAL** Graphic Menu prompts for labeled values. Refer to **Figure 5-28**.

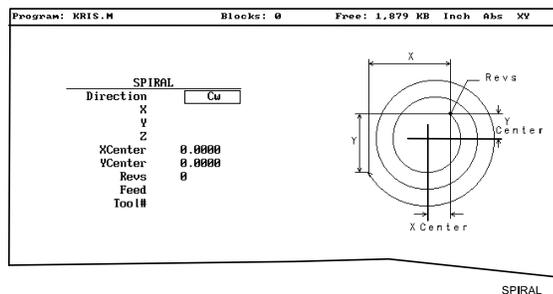


Figure 5-28, SPIRAL Graphic Menu

4. Enter the required values and settings in the entry fields:

Direction	Allows you to select a clockwise (Cw) or counterclockwise (Ccw) direction. Press +/- to toggle the selection. (Required)
X	Incremental X coordinate of endpoint. (Optional)
Y	Incremental Y coordinate of endpoint. (Optional)
Z	Incremental Z coordinate of endpoint. (Optional)
XCenter	Incremental X coordinate of center. (Required)
YCenter	Incremental Y coordinate of center. (Required)
Revs	Number of complete revolutions made in the cycle. (Required)
Feed	Feedrate. (Optional)
Tool#	Active tool. (Optional)

Mold Cycles

The following tasks are described in this topic:

- ❑ Programming a mold rotation
- ❑ Rotations around X- and Y-axes (small radius)
- ❑ Rotations around X- and Y-axes (large radius)
- ❑ Rotation around the Z-axis
- ❑ Programming an Elbow Milling Cycle

Programming a Mold Rotation

Use a Mold Rotation to mill cylindrically symmetrical cavities and cores. To define cylindrically symmetrical shapes, rotate a profile around an axis. Refer to **Figure 5-29**.

There are two categories of mold rotation:

- ❑ Rotation around X- or Y-axes
 - Small Radius
 - Large Radius
- ❑ Rotation around the Z-axis

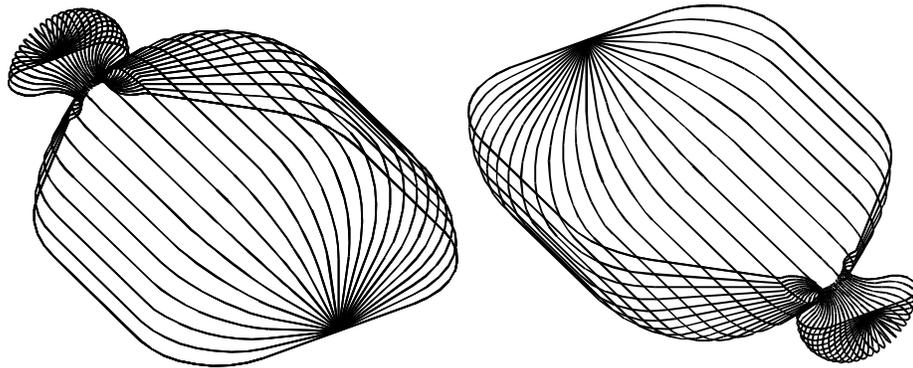


Figure 5-29, XY Symmetrical Axis Mold Rotations

Rotations Around X- and Y-Axes (Small Radius)

Each Mold Rotation block requires two subprograms: a forward subprogram (**FwdSub**) to define the profile moving away from the starting point and a reverse subprogram (**RevSub**) to define the profile moving back to the starting point. Refer to **Figure 5-30**.

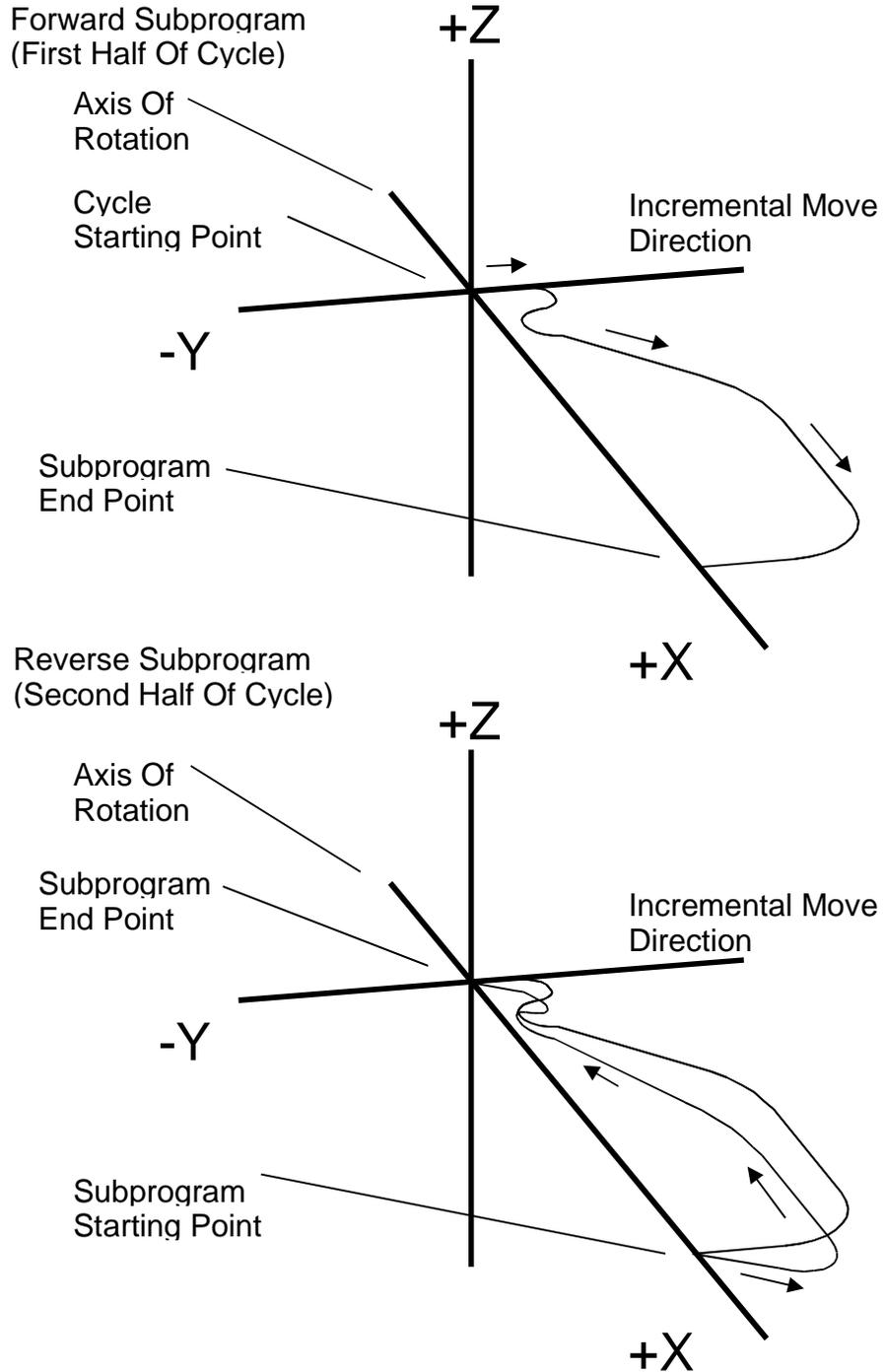


Figure 5-30, Subprogram Orientation

The CNC first executes the forward subprogram to the profile endpoint. It then executes the reverse subprogram back to the starting point. The CNC increments each cycle around the axis of rotation, from the **StartAngle** to the **EndAngle**. The number of **Cycles** and the size of the start and end angle values programmed into the block determine the amount of rotation for each increment.

The cycle starts to cut the first subprogram (profile) from the machine's current position.

In small radius rotations, subprogram start and end points can lie along the centerline of rotation. Make all subprogram moves incremental. The two subprograms (forward and reverse) must produce the same profile, but run in opposite directions. Refer to **Figure 5-31**.

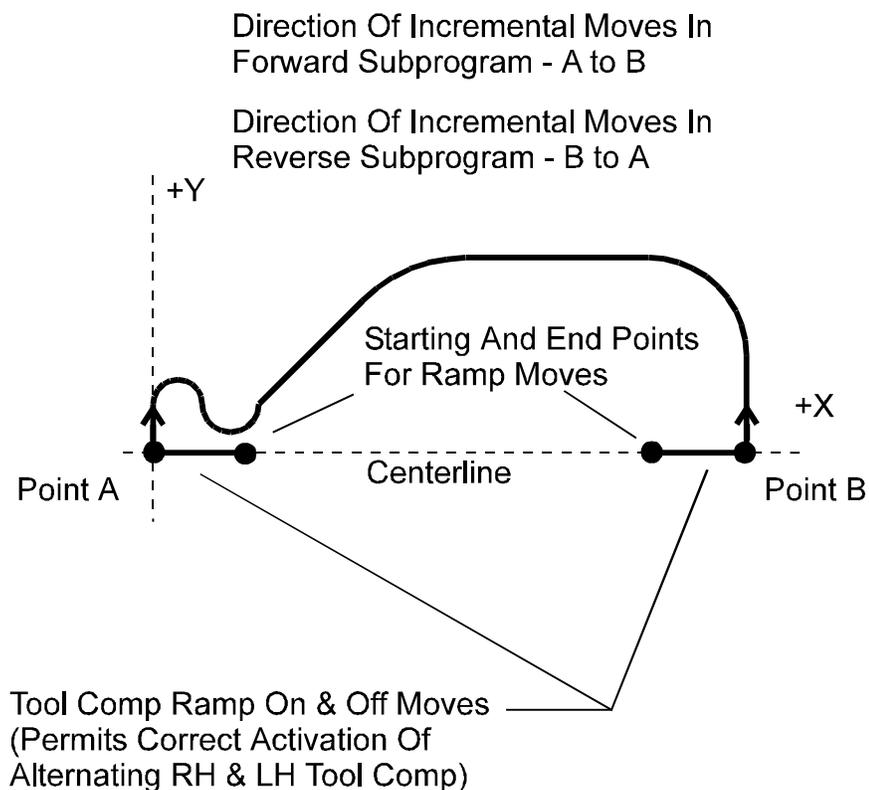


Figure 5-31, Subprogram Specifics

For rotation around the X-axis, include all subprogram moves in the +Y half of the XY plane. For rotation around the Y-axis, include all subprogram moves in the +X half of the XY plane.

When tool compensation is not used, adjust the path in the subprogram for the radius of the tool. If cutting a core, increase the path by one tool radius. If cutting a cavity, reduce the path by one tool radius.

Write compensated moves to the subprogram. Remember that tool compensation (left of path and right of path) will reverse in the two subprograms.

Program ramp moves in subprograms. Every time tool compensation activates, the CNC executes the programmed ramp on/off move for position adjustment.

Refer to "Section 1 - CNC Programming Concepts" for information on tool length offset and diameter compensation with ball-end mills.

The rotation occurs around the axis selected in the **AxisRot** field (centerline of rotation). Use **CAxisCL** and **BAxisCL** values to define the position of a centerline not located on the axis zero (X0, Y0, Z0). Refer to **Figure 5-32**.

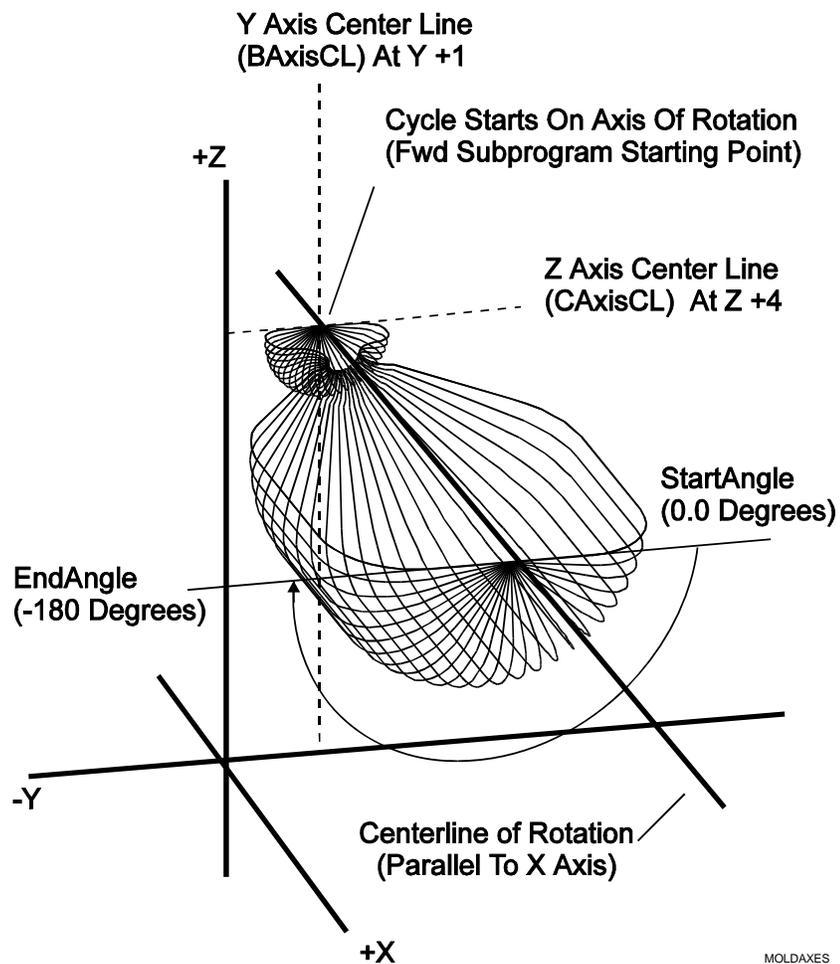


Figure 5-32, Axes of Rotation

When the shape rotates around the X-axis, the Y-axis position (in the **BAxisCL** field) and the Z-axis position (in the **CAxisCL** field) define the centerline.

When the shape rotates around the Y-axis, the X-axis position (in the **BAxisCL** field) and the Z-axis position (in the **CAxisCL** field) define the centerline.

The **StartAngle** and **EndAngle** are absolute angles that define the rotation. To program a cavity or core, first determine the orientation (positive or negative) of each angle in the subprogram.

To execute a Mold Rotation that requires more than one pass, write the subprograms for the finished shape and call the subprograms from more than one Mold Rotation block.

Vary the starting position and **CAxisCL** value of each block to bring each pass closer to the finished shape. To cut a core, reverse the sequence shown in the figure. Refer to **Figure 5-33**.

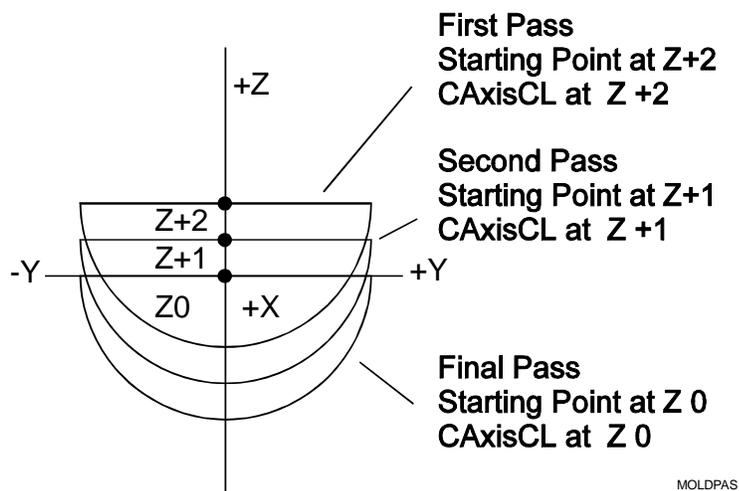


Figure 5-33, Cutting a Cavity Using More than One Pass

To rotate the XY shape around the Z-axis, enter a **ZAngle**. Refer to **Figure 5-34**.

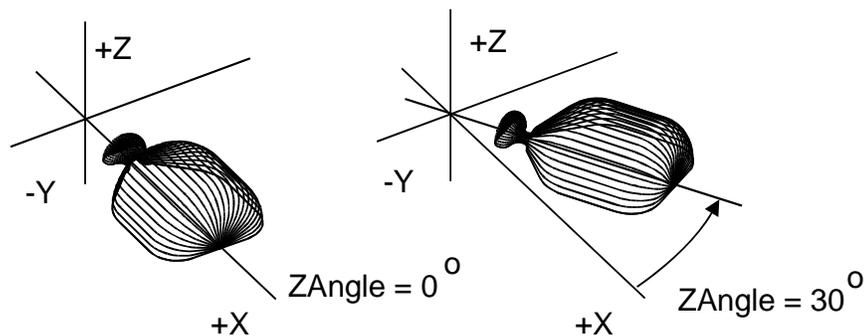
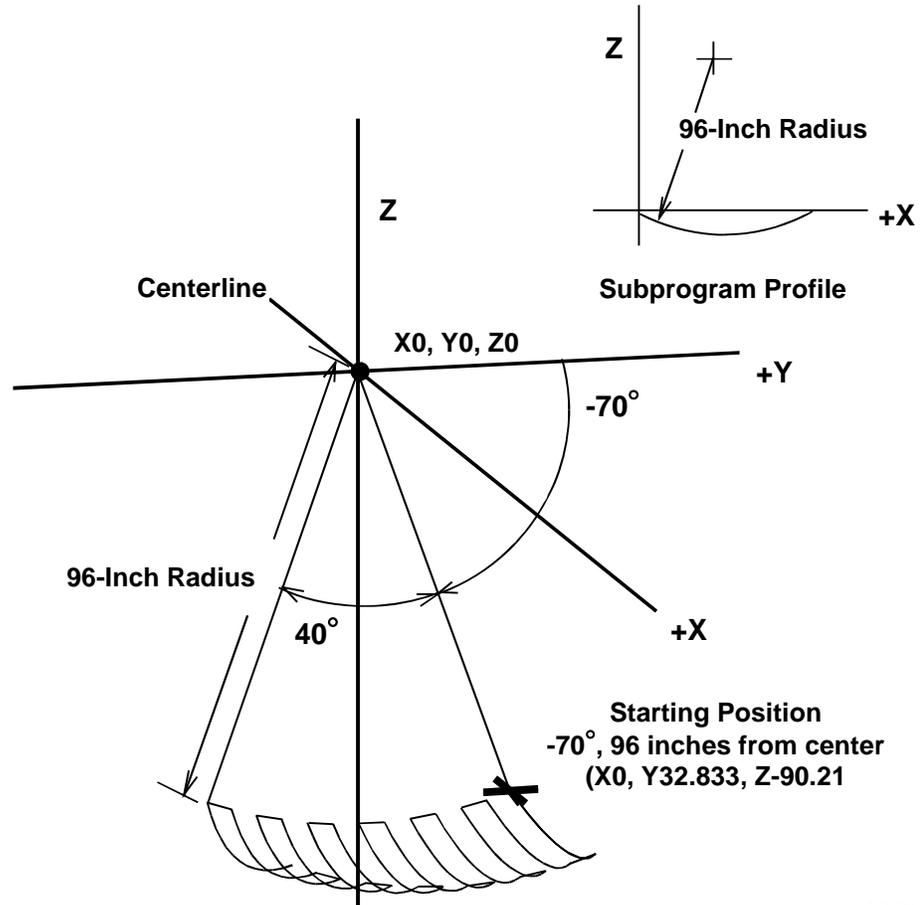


Figure 5-34, Rotating XY Mold Rotations Around Z

Rotations Around X- and Y-Axes (Large Radius)

The mold rotation cycle executes subprograms starting at the present position. To cut a large radius rotation, start the cycle at the required distance from the centerline. The CNC automatically adds the distance to the radius of the rotation. Refer to **Figure 5-35**.



LG-RAD2

Figure 5-35, Large Radius Mold Rotation

If the rotation is less than 180 degrees, a point on the start angle that is a specified distance from the centerline defines the starting position.

TIP: Use the Geometry Calculator to determine this position (refer to "Section 12 - Calculators").

Rotation Around the Z-Axis

The centerline of rotation is parallel to the Z-axis (**AxisRot**). The **BAxisCL** and **CAxisCL** values are the X and Y coordinates of the centerline. Enter the X coordinate in the **BAxisCL** field and the Y coordinate in the **CAxisCL** field. Refer to **Figure 5-36**.

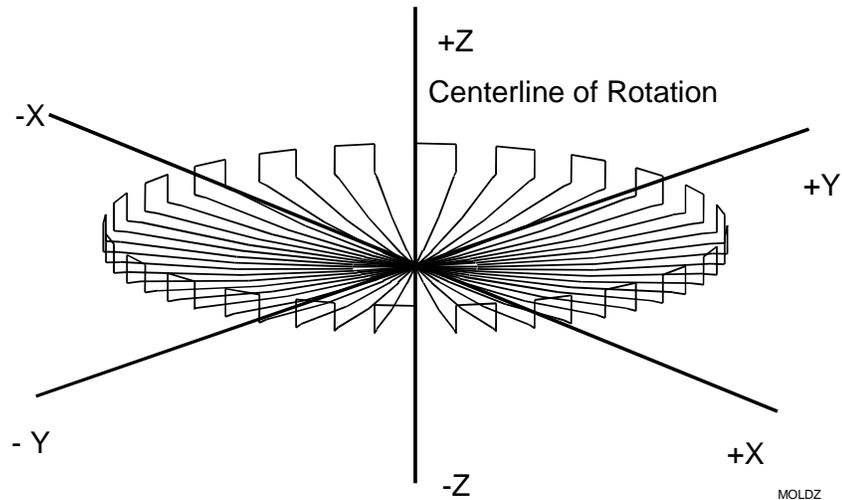


Figure 5-36, Z-Axis Mold Rotation

Position the machine at the center of rotation at cycle starts. The finished shape centers on the starting position. The Z-axis position of the starting point determines the Z-axis position of the finished shape.

Restrict subprogram moves for Z-axis rotations to the +X half of the XZ plane. Rules for using tool compensation are the same as those for X and Y-axis rotations. Refer to **Figure 5-37**.

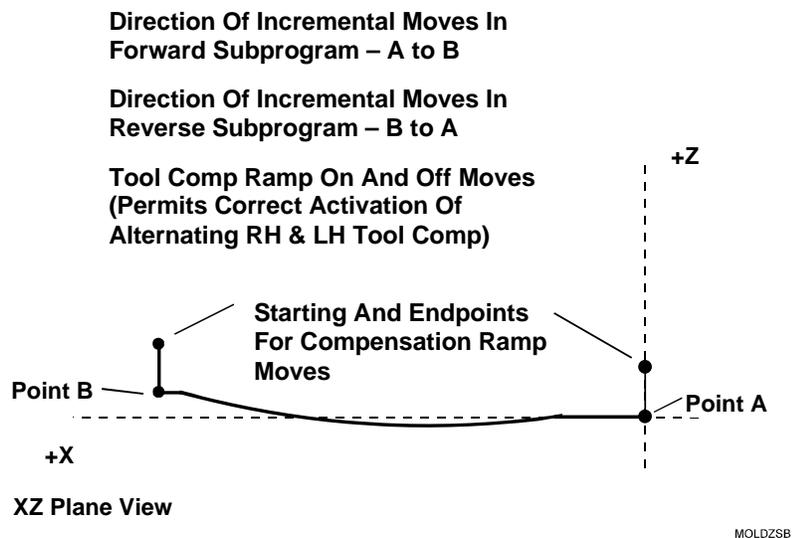


Figure 5-37, Z-Axis Rotation Subprogram Details

Figure 5-38 defines the Z-axis rotation start and end angles.

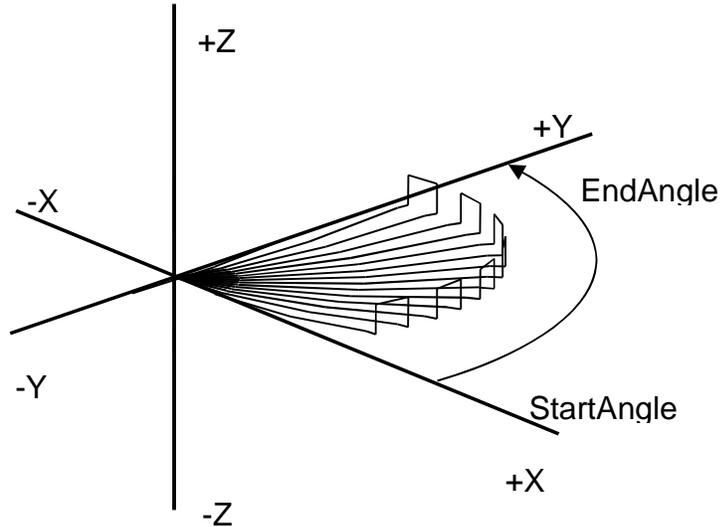


Figure 5-38, z-axis Rotation Start and End Angles

To program a **MoldRot** block:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Highlight **Mold Rotation**, and press **ENTER** to display the **MOLD ROTATION** Graphic Menu. Refer to **Figure 5-39**.

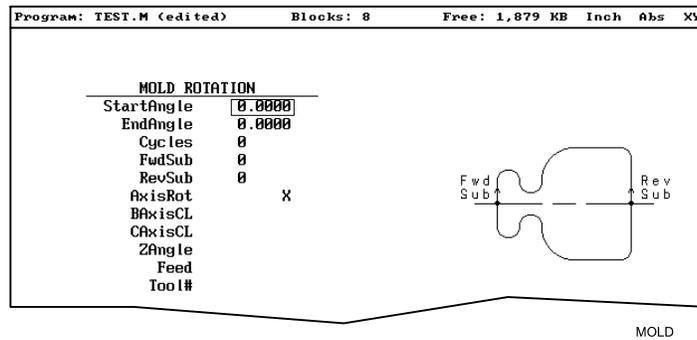


Figure 5-39, MOLD ROTATION Graphic Menu

3. Fill in the **MOLD ROTATION** entry field:

StartAngle	Absolute starting angle of the rotation. (Required)
EndAngle	Absolute final angle of the rotation. (Required)
Cycles	Number of subprogram cycles executed between start and stop angles. (Required)
FwdSub	Forward subprogram call. (Required)
RevSub	Reverse subprogram call. (Required)
AxisRot	Axis of rotation. Press +/- to toggle the selection. (Required)

BAxisCL	First position coordinate of the rotated axis. (Optional)
CAxisCL	Second position coordinate of the rotated axis. (Optional)
ZAngle	Rotated position of XY-axis mold. (Optional)
Feed	Feedrate. (Optional)
Tool#	Active tool. (Optional)

NOTE: After you program a mold rotation, activate the plane required for subsequent moves.

Programming an Elbow Milling Cycle

Elbow Milling cycles simplify the programming required to mill elbow-shaped cavities and cores. Finished elbows can have the same radius at each end (bent cylindrical shape) or a different radius at each end (bent conical shape). Refer to **Figure 5-40**.

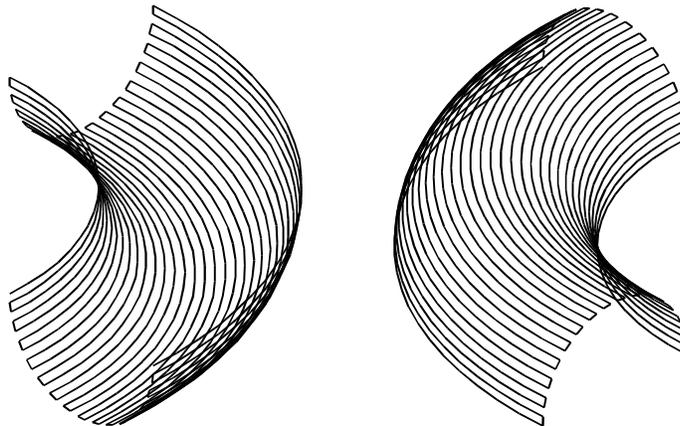
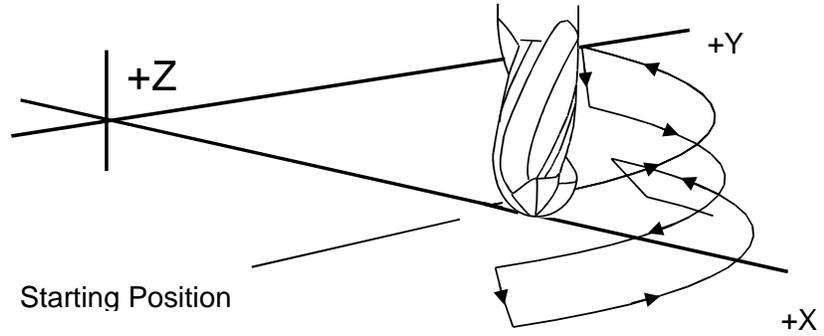


Figure 5-40, Elbow Cavity and Core

The Elbow Milling Cycle starts at the machine's present position. The CNC executes passes back and forth around the elbow's inner radius, from the start radius to the end radius. The tool advances to the cavity (or around the core) at the end of each pass until the elbow is completed. The cycle stops at the opposite side of the elbow, at the same Z-axis position from which it started. Refer to **Figure 5-41, Execution of Elbow Milling Cycle Moves**.



CW Execution Of Moves For Elbow Cavity

Figure 5-41, Execution of Elbow Milling Cycle Moves

Do not use tool compensation with the Elbow Milling Cycle. When cutting a particularly deep elbow, it may be necessary to cut the shape in several passes. It is easiest to program the elbow in a subprogram. The main program should consist of moves that position the machine at consecutively lower Z-axis starting positions. Call the subprogram at each successive starting position.

Consider the starting position of the machine when programming an Elbow Milling Cycle. The distance between the starting point and the **XCenter**, **YCenter** determines the elbow's inner radius. The line between the starting point and the **XCenter**, **YCenter** is the zero degree reference for the included angle. When **XCenter**, **YCenter** values are blank, the CNC uses X0, Y0 (default) values. Refer to **Figure 5-42**.

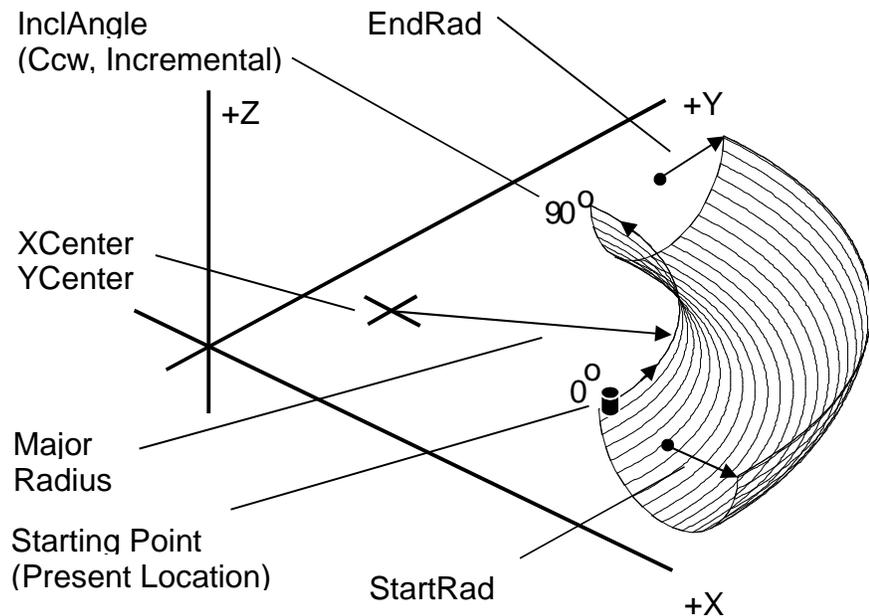


Figure 5-42, Elbow Milling Cycle Details

When the line between the starting point and the XCenter, YCenter does not lie along the X- or Y-axis, the orientation of the finished elbow will shift around the XY center accordingly. Refer to **Figure 5-43**.

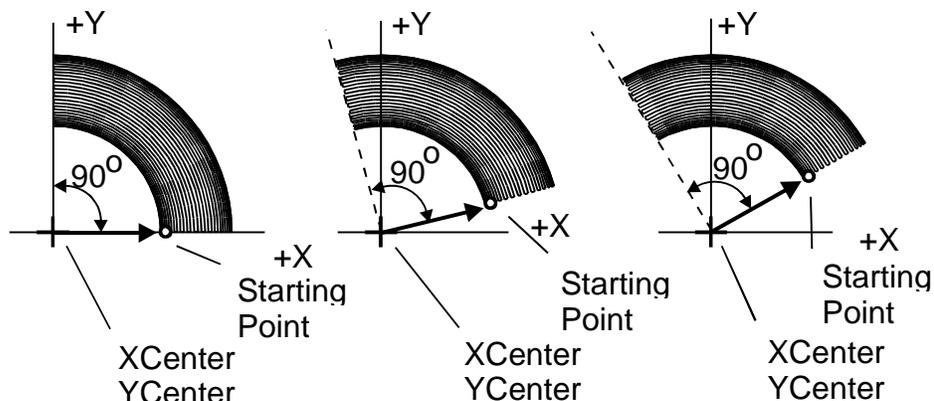


Figure 5-43, Starting Point and Effect on Orientation

The distance between the starting point and the XY center determines the elbow inner radius. Moving the starting point away from the XY center increases the overall size of the finished elbow, as shown. Refer to **Figure 5-44**.

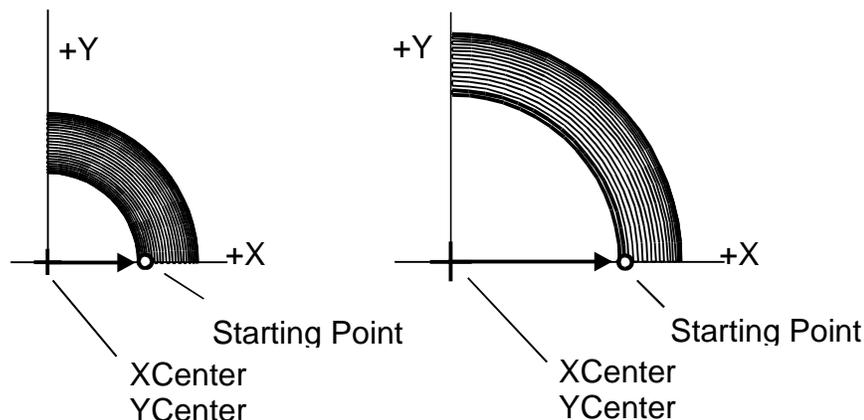


Figure 5-44, Starting Point and Effect on Size

The **Cycles** value determines the number of passes used to cut the elbow. Enter a negative cycle value to cut a core (above the XY plane). Use a positive cycle value to cut a cavity (below the XY plane). Refer to **Figure 5-45**.

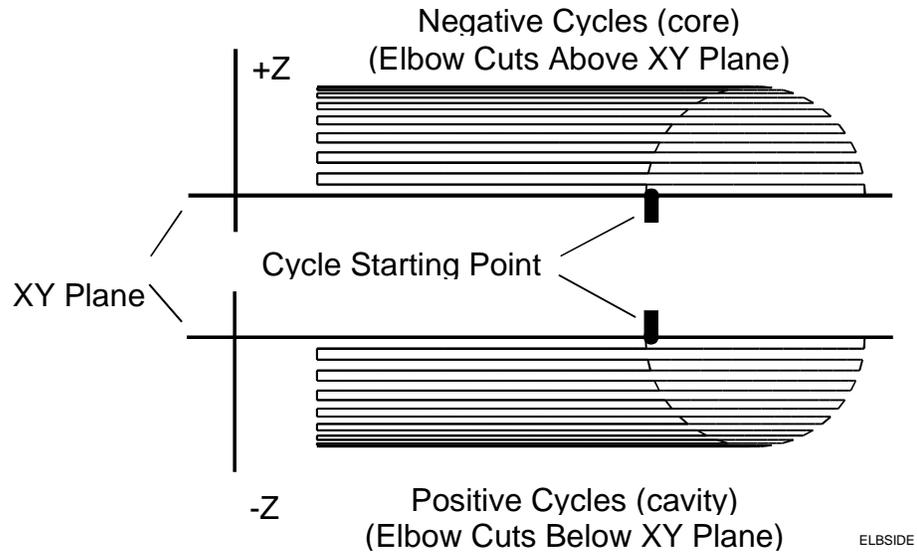


Figure 5-45, Core and Cavity Detail

Program an Elbow Milling Cycle with unequal **StartRad** and **EndRad** values to produce a conical elbow. Refer to **Figure 5-46**.

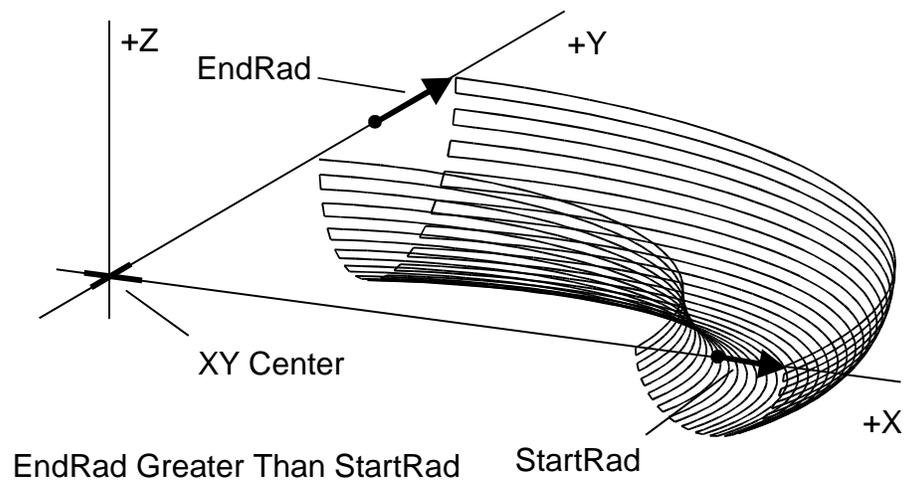


Figure 5-46, Conical Elbow Details

To program an Elbow Milling cycle:

1. In Edit Mode, press **Pocket (F4)** to display a pop-up menu.
2. Highlight **Elbow Milling**, and press **ENTER** to display the **ELBOW MILLING** Graphic Menu. Refer to **Figure 5-47**.

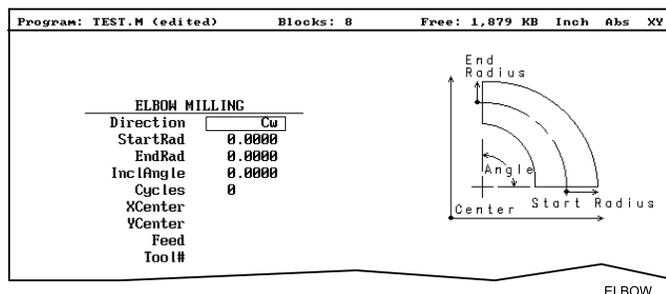


Figure 5-47, ELBOW MILLING Graphic Menu

3. Enter **ELBOW MILLING** entry fields:

Direction	Direction, clockwise (Cw) or counterclockwise (Ccw), of first pass from starting point. Press +/- to toggle the selection. (Required)
StartRad	Radius of the cavity/core at the starting end of the elbow. (Required)
EndRad	Radius of the cavity/core at the far end of the elbow. (Required)
InclAngle	Angle (centered at XY center) between the StartRad end and the EndRad end of the elbow. (Required)
Cycles	The number of passes used to cut the elbow. (Required)
XCenter	X coordinate of the elbow's inner radius center point. (Optional)
YCenter	Y coordinate of the elbow's inner radius center point. (Optional)
Feed	Feedrate. (Optional)
Tool#	Active tool. (Optional)

Engraving, Repeat, and Mill Cycles

This section describes operation of three new cycles:

- Engraving Cycle
- Repeat Cycle
- Mill Cycle

Engraving Cycle

The Engraving cycle provides a quick and easy way to engrave part numbers, legends, or any alpha/numeric inscription. The usual type of cutter is a sharp point or center-drill type tool. Options are given for engraving on an angle and mirror is supported for engraving molds. When executed, the CNC rapids to the start point, then to the StartHgt. It then feeds to the Zdepth specified and begins cutting the Text selected.

Programming the Engraving Cycle

To program the Engraving Cycle:

1. In Edit mode, press **Mill (F5)** and **More (F7)** to display the More pop-up menu, **Figure 5-48**. Highlight **Engrave** and press **ENTER** to display the Engraving Cycle screen, **Figure 5-49, Engraving Cycle Screen**.
2. Complete the entry fields (refer to **Table 5-2, Engraving Cycle Entry Fields**), and press **ENTER**.

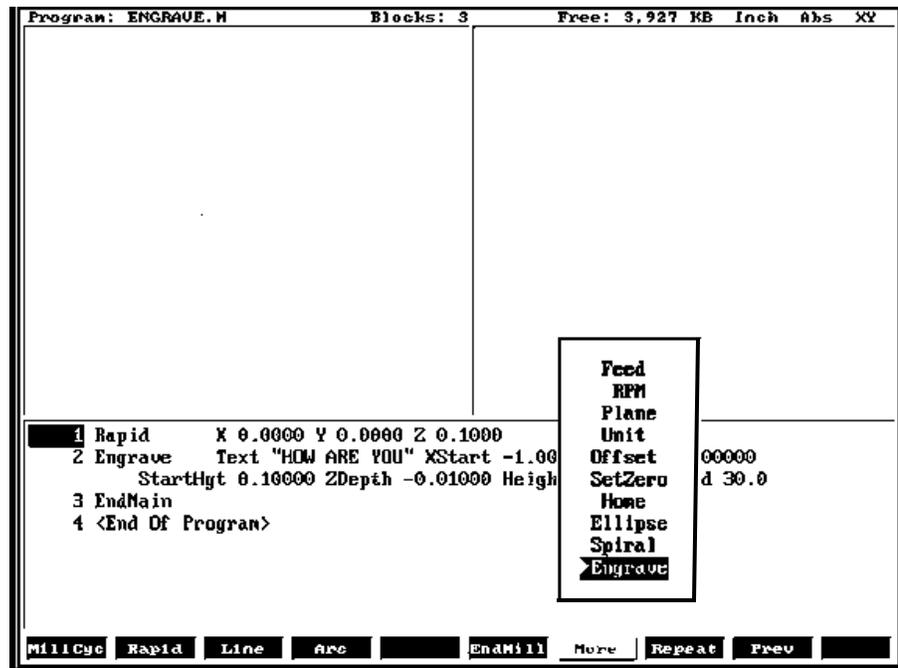


Figure 5-48, More Pop-up Menu

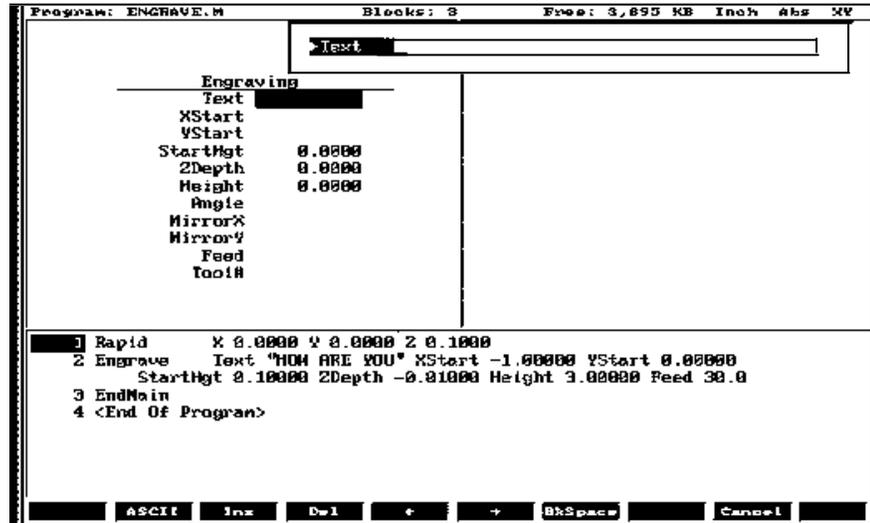


Figure 5-49, Engraving Cycle Screen

Table 5-2, Engraving Cycle Entry Fields

Entry Fields	Description
Text	When the cursor is on Text, it displays an entry field for the letters to be engraved. Letters A – Z, numbers 0 – 9, and: space, ampersand, plus, minus, comma, period, and slash right are supported. No lower case letters are allowed. If you have no external keyboard, use the ASCII Chart to enter text (see Figure 2-4, ASCII Chart Pop-up). Press ENTER to accept the text. (Required)
XStart	X coordinate for lower-left corner of the text. Defaults to current position if not given. (Optional)
YStart	Y coordinate for lower-left corner of the text. Defaults to current position if not given. (Optional)
StartHgt	Z absolute start height. Must be higher than Zdepth. (Required)
Zdepth	Z absolute depth of engraving. Must be below StartHgt. (Required)
Height	Letter height. Width will be proportional to height. Height is measured at the centerline of the cutter. (Required)
Angle	Angle in degrees. Default is 0 degrees. (Optional)
MirrorX	Mirrors all X moves. Set by using minus key (-) while in this field. (Optional)
MirrorY	Mirrors all Y moves. Set by using minus key (-) while in this field. (Optional)
Feed	Feedrate used while engraving. Default is current feedrate. (Optional)
Tool#	Active Tool (Optional) Used only on 3000M controls – not on 6000M

Sample Engraving Cycle Program

```

1   Dim Abs
2   Unit Inch
3   Rapid   X 0.00000 Y 0.00000
4   Tool# 1
5   Rapid   X 1.00000 Y 1.00000
6   Rapid   Z 0.10000
7   Engrave Text "ABCD" StartHgt 0.0100 ZDepth -0.0100 Height 0.5000
8   Rapid   Z 1.00000
9   Rapid   X 0.00000 Y 0.00000
10  EndMain
    
```

This program will rapid to X1.0 Y1.0. Z will rapid to 0.1 and the letters ABCD will be engraved 0.0100" deep and 0.500" high.

Repeat Cycle

The Repeat cycle allows a series of previously programmed blocks to be repeated. Some examples are going over the same contour while lowering the Z-axis, or drilling over a series of holes with a different drill cycle, or moving an operation to a different location using fixture offsets. Wherever it is used, the repeated blocks will be processed, just as if they were written in the program at that point.

Programming the Repeat Cycle

To program the Repeat Cycle:

1. In Edit mode, press **Mill (F5)** and **Repeat (F8)** to display the Repeat Cycle screen, **Figure 5-50**.

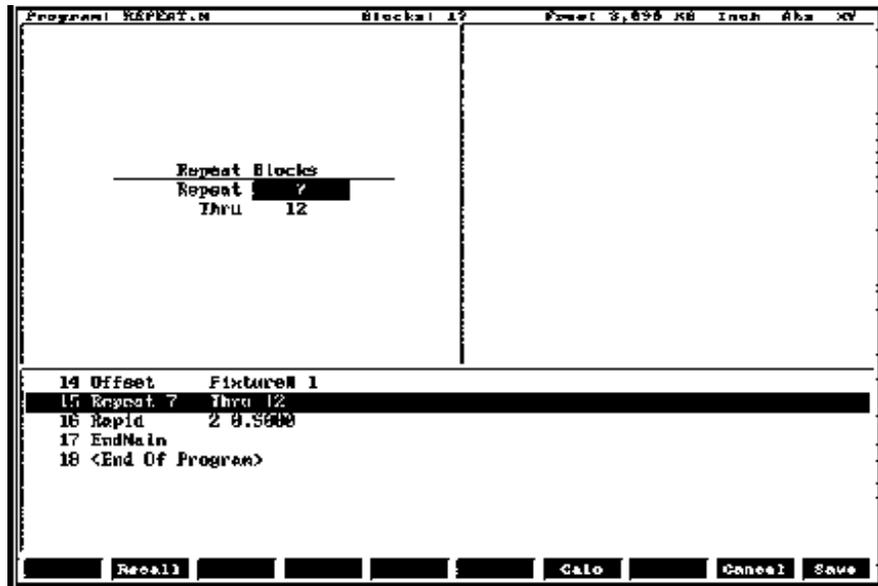


Figure 5-50, Repeat Cycle Screen

2. Complete the entry fields (refer to **Table 5-3**), and press **ENTER**.

Table 5-3, Repeat Cycle Entry Fields

Entry Field	Description
Repeat	Type the block number you want to begin repeating. (Required)
Thru	Type the block number you want to end the repeat. (Required)

3. When using a Modal Drilling Cycle with the Repeat feature, a **DrillOff** or non-move command must be included as the final block. For example, see “Sample Repeat Cycle Program” block 7–12 and block 15.

Sample Repeat Cycle Program

```

1   Dim Abs
2   Unit Inch
3   Offset  Fixture# 0
4   Rapid   X 0.0000 Y 0.0000
5   Tool# 1
6   Rapid   Z 0.1000
7   BasicDrill ZDepth -0.50000 StartHgt 0.10000 Feed 15.0
8   Rapid   X 1.00000
9           Y 1.0000
10          X 0.0000
11          Y 0.0000
12  DrillOff
13  Offset  Fixture# 1 X 3.0000 Y 0.0000
14  Offset  Fixture# 1
15  Repeat 7 Thru 12
16  Rapid   Z 0.5000
17  EndMain

```

This program will drill four holes. A Fixture Offset is used to relocate X Y zero. When the Repeat Cycle is encountered, it will drill four more holes at the offset location.

Mill Cycle

The Mill cycle is intended for contour milling operations. Cutter compensation, Z pecking, Z finish stock, RoughFeed, and FinishFeed are supported. The cycle will rapid to the XY start point (compensated, if comp is on) rapid to the start height and then feed to the Zdepth or DepthCut using the Zfeed. Subsequent milling blocks are then executed using the ToolComp parameter and Feed specified. The feedrate can be changed in the blocks that are being milled, but any change of feedrate must accompany an axis move and overrides the feedrate specified in the canned cycle (RoughFeed or FinFeed) from the point of the new feedrate forward in the cycle. Cutter Compensation cannot be changed from within the cycle. The cycle is terminated with the EndMill block; at which point, it rapids up to the StartHgt and returns to the un-comped XStart YStart location.

Programming the Mill Cycle

To program the Mill Cycle:

1. In Edit mode, press **Mill (F5)** and **MillCyc (F1)** to display the Mill Cycle screen, **Figure 5-51**.
2. Complete the entry fields (refer to **Table 5-4, Mill Cycle Entry Fields**), and press ENTER.

Programming the EndMill Block

To program the EndMill Block:

1. In Edit mode, press **EndMill (F6)** to end the cycle.

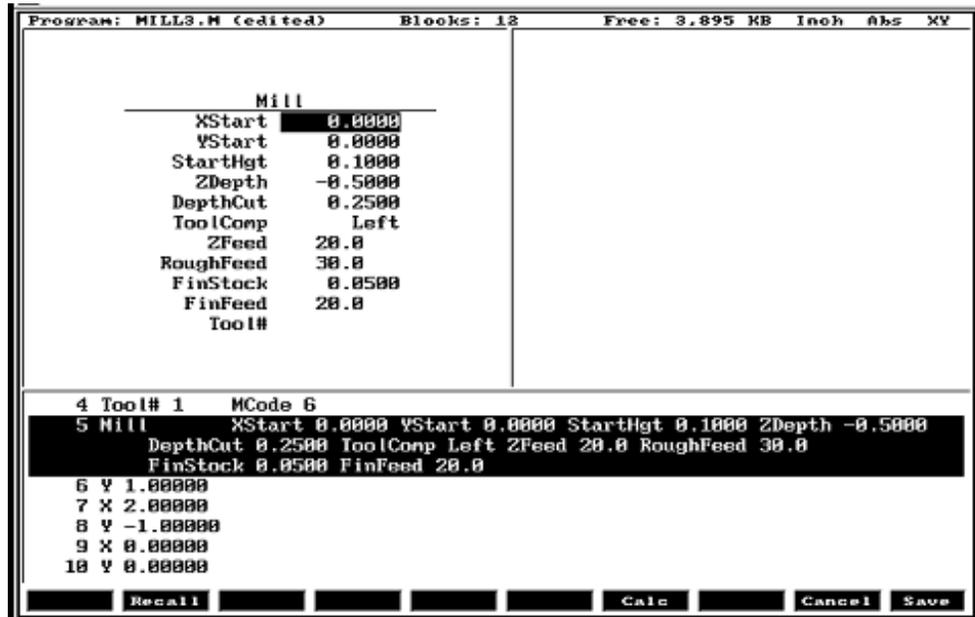


Figure 5-51, Mill Cycle Screen

Table 5-4, Mill Cycle Entry Fields

Entry Field	Description
XStart	X coordinate for start of Mill cycle. Defaults to current position if not given. (Required)
YStart	Y coordinate for start of Mill cycle. Defaults to current position if not given. (Required)
StartHgt	Z absolute start height. Must be 0.100" above work surface (0.2mm). (Required)
ZDepth	Absolute depth of finished contour. (Required)
DepthCut	Depth of cut taken in a single pass. Cuts will be adjusted so that all are equal pecks. (Optional)
ToolComp	Tool radius compensation Left or Right of programmed path. Set by using minus key (-) while in this field. (Optional)
ZFeed	Feedrate for Z-axis. Defaults to current feedrate. (Optional)
RoughFeed	Feedrate for X and Y-axis. Defaults to current feedrate. (Optional)
FinStock	Amount of stock to take for last Z peck. (Optional)
FinFeed	Feedrate used for FinStock. (Optional)
Tool #	Active Tool. (Optional) Used only on 3000M controls – not on 6000M

Sample Mill Cycle Program

```

1   Dim Abs
2   Unit Inch
3   Rapid   X 0.00000 Y 0.00000
4   Tool# 1   MCode 6
5   Mill    XStart 0.00000 YStart 0.00000 StartHgt 0.10000
        ZDepth -0.50000 DepthCut 0.25000 ToolComp Left ZFeed 20.0
        Feed 30.0
6   Y 1.00000
7   X 2.00000
8   Y -1.00000
9   X 0.00000
10  Y 0.00000
11  EndMill
12  EndMain

```

This program will contour a square, in two Z pecks of 0.250" each. The blocks 6 thru 10 are the contour moves that will be comped to the left of tool path direction. Block 11, EndMill is required to show the end of the contour. The cutter will be returned to the start point, X0 Y0 at the start height of 0.100".

Probing Cycles

Probing cycles have the following features:

- Tool probe cycles
- Spindle probe cycles

This section describes operation and an overview of the tool and spindle probe canned cycles in conversational format. Probing is an option in 3000M CNC system. The cycles provided perform the most common tool and spindle probing functions. If Probing has been added post-sale, besides Setup Utility changes, there may be Integral Programmable Intelligence (IPI) program modifications required.

The tool probe cycles are only supported on machines with automatic spindle forward/reverse and spindle speed, and homing with a permanent X, Y, and Z machine position. The method described assumes the use of negative tool-length offsets. In this method, the Tool-Length Offset (TLO) in the length column for each tool represents the distance from the tool tip at machine home to top of work piece and is a negative number. This method does not require the use of any Z work coordinate offset to be active. This procedure will find the effective tool diameter by turning the spindle on in reverse and touching two sides of the probe stylus, then storing the tool's diameter in the tool's diameter offset table.

The spindle probing cycles are designed to assist in part setup. Using these cycles, one or more features (edges) of a part can be measured. Using the data obtained with these measurements, calculations are made that can be used to set a given fixture offset. It is also possible to find the orientation angle of a part so as to not always have to align the part exactly.

Tool and spindle probing do not allow rotation, scaling, and mirroring. Plane will be set to XY when these cycles are complete.

Tool Probe Cycles

Before using your tool probe and tool probe cycles, you must setup the probe following the probe manufacturer's specifications.

The tool probe will update the tool registers only. If you are going to use the tool being measured after the probing cycle, you must recall that tool for the new offsets to be active.

This section covers the following topics:

- Tool probe cycle designations
- Description of tool probe cycles

For probing system parameter setup information, refer to *3000M CNC Setup Utility Manual*, P/N 70000499, "Section 2 – Builder Setup," "Probing."

Tool Probe Cycle Designations

The following summarizes the cycles available:

Probe Calibration (CalibTIPrb)

Tool Probe Calibration Cycle

This is used to set the Z datum for length preset, the effective probe stylus diameter for setting tool diameter registers, and establishes the center of the probe stylus.

NOTE: Calibration must be done at least once before using the tool probe. Once the probe has been calibrated, calibration does not need to be done again unless the probe is moved or a new part is being setup. The cycle must always know the relationship between the top of the part and the top of the probe to set the TLO.

Length and Diameter (LenDiamMea)

Tool Length and Diameter Offset Preset

Updates length and diameter tool registers.

NOTE: If the tool has a hole on the bottom so that the probe would fall between the tool teeth, do not use this cycle. Damage to the probe could result. In this case, use **Length Special** for manual length preset or **Diameter Special** for manual diameter preset.

Length Special (LenSpecMea)

Manual Tool-Length Offset Preset

Updates tool-length register. To be used for large face mill style tools or shell mill tools that have a hole in the center of the bottom of the tool.

Diameter Special (DiaSpecMea)

Manual Tool Diameter Preset

Updates tool diameter register for irregular shaped tools or tools with a hole in the center of the bottom.

Break and Wear (BrkWearDet)

Tool Breakage, Length and Diameter Wear Detection Breakage

Checks the tool and gives an alarm if not within tolerance.

Length and Diameter Wear – Check the Length and/or Diameter and updates the Length and/or Diameter wear registers up to a user-defined limit. Once the user-defined limit has been reached, the cycle will give an alarm and stop the program.

Description of Tool Probe Cycles

- For tool probing or tool length presetting, Tool-Length Offset (TLO) is the distance from machine home to top of work piece or wherever you set your part “**Z**” zero.
- Before starting to set your tools, you must calibrate the probe. Once the probe has been calibrated, calibration does not have to be done again unless you remove the probe or replace the stylus.
- Recalibration may also be required if the Z location of the top of the part changes, and is not compensated for by a Z work offset.
- From the program editor or MDI mode, press **Mill (F5)**, then **Probe (F10)** and **ToolPro (F1)** or **SpinPro (F3)**.

The following sections contain detailed descriptions of the tool probe cycles:

- Tool Probe Calibration Cycle (**CalibTIPrb**)
- Tool Length and Diameter Offset Preset (**LenDiamMea**)
- Manual Tool Length Measure for Special Tools (**LenSpecMea**)
- Manual Tool Diameter Measure for Special Tools (**DiaSpecMea**)
- Tool Breakage, Length, and Diameter Wear Detection (**BrkWearDet**)

Tool Probe Calibration Cycle (CalibTIPrb)

Format: CalibTIPrb DiamOfStd(n) DistDown(n)

This cycle is used to calibrate the probe. This sets the Z datum for length preset to the top of the part, establishing the center of the probe stylus, and the effective probe stylus diameter for setting tool diameter registers. Refer to **Table 5-5**.

Table 5-5, CalibTIPrb Entry Fields

Entry Fields	Description
DiamOfStd	The diameter of the part of the calibration standard that comes in contact with the probe stylus during calibration. This should be an exact measurement. (Optional override for Diameter of tool probe gauge)
DistDown	The distance to go down along the side of the probe stylus with the probe calibration standard when touching the side of the stylus for diameter calibration. The maximum DistDown value is 0.55" (13.97 mm). Without any DistDown value, the cycle will bring the calibration standard down past the top of the probe stylus the default 0.1" (2.54mm). If you put a number higher than 0.55" (13.97 mm), the control displays an error. (Optional) [Default: 0.1"]

To calibrate the tool probe:

1. Jog the calibration standard (the calibration standard should be in the spindle) to the top of your work piece, and set its tool-length offset to the top of the work piece or to wherever you would like your Z zero to be. To calibrate the tool number:
 - a) Jog the tip of the calibration standard to the proper spot
 - b) Press the **Calib Z** function key.
2. Manually jog the calibration standard over the probe stylus center and less than 0.1" (2.54 mm) above the probe stylus. It should be no more than 0.1" (2.54 mm) from the center of the stylus.
3. From the MDI mode, pick **F5 (Mill) > F10 (Probe) > F1 (ToolPro) > Probe Calibration**
For example: **CalibTIPrb** exit by pressing **F9** twice and **F10** to exit.
4. The Z-axis will initially go down and touch the top of the probe stylus at the feedrate specified in **Z first pick, MEDIUM feedrate** machine setup parameter. Then retouch at the slow feedrate, **Z final pick, SLOW feedrate** machine setup parameter, establishing the zero probe stylus top.
5. Then incrementally rapid up whatever value that is in **Z retract distance** machine setup parameter.

6. The spindle will come on at the RPM specified at the **RPM for calibration and tool measurement** machine setup parameter and then the calibration standard will move over an incremental amount that is equal to (Half the value entered in the **DiamOfStd** cycle parameter (or machine setup parameter Diameter of probe gauge) + Half the value entered in **Nominal probe stylus diameter** machine setup parameter + The value in the **XY retract distance** machine setup parameter). The direction the probe will move over depends on what is placed in the **Probe orientation** machine setup parameter:
 - 1 Go first to the left
 - 1 Go first to the right
 - 2 Go first to the front
 - 2 Go first to the back
7. The Z-axis will then do a guarded Z move down 0.1" (2.54 mm) or whatever amount was placed in the **DistDown** cycle parameter and then move over toward the probe stylus 0.3" (7.62 mm) or until it touches the probe stylus. If contact is not made with the probe or if contact is made during a guarded move, then an alarm will be generated and the canned cycle will terminate.
8. After the probe stylus is touched on the first side, the machine will then rapid up and over the stylus then down on the opposite side then over to the other two sides until it has touched the probe stylus on all four quadrants. This will establish the center of the probe stylus.
9. The spindle will then turn off and the machine will touch off on two sides of the probe with the spindle off finding the effective probe stylus diameter. Then, will rapid up above the probe stylus and over to the center.
10. Remove the calibration standard. You are now ready to start running the **Length and Diameter** cycle (or one of the other cycles for setting or checking length and diameter of the tool) to set your tool-length offsets or tool diameter registers.

Tool Length and Diameter Offset Preset (LenDiamMea)

Format: LenDiamMea Tool#(tool#) EstDiam(n) MeasType (Length, Diameter, or Both) DistDown(n) OvrFstFeed(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

Each tool must have the length set once before trying to set the diameter. Call this cycle up the first time using **Both** because it will automatically set the length first then the diameter.

- Calibrate the tool probe at least once before trying to automatically preset a tool. This is done initially, but if the stylus is ever changed or the probe is moved, then you must again calibrate the tool probe.
- This tool preset (**LenDiamMea**) can be run from within a program or from the MDI mode. Refer to **Table 5-6**.

Table 5-6, LenDiamMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) With only the Tool# cycle parameter present, the canned cycle will not step over half the tool's diameter but come straight down measuring the tool length and storing it in the tool register.
EstDiam	This is the rough diameter of the tool. This should be within 0.04" (1.0 mm). (Optional) If the EstDiam cycle parameter is present, the tool will step over half of its diameter, the spindle will turn on in reverse and then the canned cycle will measure the tool's length. A negative EstDiam value is for a left-handed tool and will cause the spindle to come on forward instead of reverse. For on center length measurement, do not give a EstDiam cycle parameter.
MeasType	The option specifies to measure length, diameter, or both and the appropriate tool registers are updated. (Optional) [Default: Length] Diameter Measure the diameter only Length Measure the length only Both Measure both length and diameter If Length is not set, the cycle will measure the tool length only. If Diameter or Both are programmed, you must also have a EstDiam cycle parameter or the control will display an error message.

(Continued...)

Table 5-6, LenDiamMea Entry Fields (Continued)

Entry Fields	Description
DistDown	<p>The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum DistDown value is 0.55" (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55" (13.97 mm), the control will issue an error message. If DistDown is not set, the cycle will use a default value of 0.1" (2.54 mm). (Optional) [Default: 0.1"]</p> <p>Ball nose cutters and special cutters that require a move down more than 0.55" (13.97 mm) are not supported.</p>
OvrFstFeed	<p>This is the override for the fast Z feedrate that was set in the machine setup parameter Z first pick, FAST feedrate. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)</p>
OvrMedFeed	<p>This is the override for the medium feedrate that was set in the machine setup parameter Z first pick, MEDIUM feedrate. This is used for the same reason as the OvrFstFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)</p>
OvrSlwFeed	<p>This is the override for the slow feedrate that was set in the machine setup parameter Z final pick, SLOW feedrate. This is used for the same reason as the OvrFstFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)</p>
OvrRPM	<p>This is the override for the RPM that was set in the machine setup parameter RPM for calibration and tool measurement. This is used for the same reason as the OvrFstFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)</p>

To use the tool preset probing cycle:

1. Install all the tools you wish to set, in the tool changer.
2. Input:
LenDiamMea Tool#(tool#) EstDiam(tool rough diameter) MeasType

If run from the inside of a program, this line needs to be repeated for every tool that you want to set.

3. Execute that line if you are in MDI mode, or run the program if you have set all the tools up in a program.
4. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, all your tools are measured and ready for use.

Shell mill style tools that have a hole in the center of the bottom will not work with this canned cycle; in this case, you must use the manual canned cycles **LenSpecMea (Length Special Manual Tool Length Measure for Special Tools)** for length and **DiaSpecMea (Diameter Special Manual Tool Diameter Measure for Special Tools)** for diameter. See **Table 5-6, LenDiamMea Entry Fields**. This cycle is only good for drills, taps, reamers, ball nosed endmills, and standard endmills with a flat bottom, the cycle updates length and diameter tool registers clearing anything in the wear registers.

Format: LenDiamMea Tool#(tool#)

With **Tool#** parameter only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **Z rapid to start position from home** machine setup parameter, then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **Z first pick, FAST feedrate** machine setup parameter, then will back up and retouch the probe at the feedrate that is in the **Z final pick, SLOW feedrate** machine setup parameter.
3. The tool-length register for that tool is now updated, and that tool's length-wear register is set to zero.
4. Then the Z-axis will rapid up to home position.
5. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 4 until all the tools have been measured.

Format: LenDiaMea Tool#(tool#) EstDiam(tool rough diameter)

With **Tool#** and **EstDiam** parameters only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **Z rapid to start position from home** machine setup parameter then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **Z first pick, FAST feedrate** machine setup parameter then will back up.
3. The machine will rapid over half the diameter of the cutter from the probe stylus center in the direction related to the **Probe orientation** machine setup parameter.
4. The spindle will then come on in reverse at the RPM specified in the **RPM for calibration and tool measurement** machine setup parameter and retouch the probe at the feedrate that is in the **Z final pick, SLOW feedrate** machine setup parameter.
5. The tool-length register for that tool is now updated, and that tool's length-wear register is set to zero.
6. Then the Z-axis will rapid up to the home position.
7. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 6 until all the tools have been measured.

Format: LenDiaMea Tool#(tool#) EstDiam(tool rough diameter)
MeasType (Both)

With **Tool#**, **EstDiam**, and **MeasType** parameters set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **Z rapid to start position from home** machine setup parameter then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **Z first pick, FAST feedrate** machine setup parameter then will back up.
3. The machine will rapid over half the diameter of the cutter from the probe stylus center in the direction related to the **Probe orientation** machine setup parameter.

4. The spindle will then come on counter clockwise at the RPM specified in the **RPM for calibration and tool measurement** machine setup parameter and retouch the probe at the feedrate that is in the **Z final pick, SLOW feedrate** machine setup parameter.
5. The tool-length register for that tool is now updated, and any value in the length wear register will be reset to zero.
6. Then the Z-axis will rapid up above the probe stylus the distance specified in the **Z retract distance** machine setup parameter and then rapid the X & Y axes over the center of the probe and turn the spindle on in reverse.
7. The machine will move the tool's edge off to one side of the probe stylus in the direction indicated in the **Probe orientation** machine setup parameter before making a guarded move down 0.1" (2.54 mm) or whatever value has been placed in the **DistDown** cycle parameter.
8. The machine will then touch the tool to the probe stylus on two opposite sides at the feedrate specified in the **Z first pick, MEDIUM feedrate** machine setup parameter with the spindle running at the RPM specified in the **RPM for calibration and tool measurement** machine setup parameter, backing up 0.02" (0.508 mm) after each first touch then retouching and the feedrate specified in the **Z final pick, SLOW feedrate** machine setup parameter, calculating the diameter of the tool and placing the calculated diameter value in the diameter register for the tool being preset and any value in the diameter wear register will be reset to zero.
9. Then the Z-axis will rapid up to the home position.
10. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 9 until all the tools have been measured.

Manual Tool Length Measure for Special Tools (LenSpecMea)

Format: LenSpecMea Tool#(tool#) DiamOfStd(n) OvrMedFeed(n)
OvrSlwFeed(n) OvrRPM(n)

This cycle is used to measure the length of large face mill style tools that have a hole in the center of the bottom of the tool. Refer to **Table 5-7**.

Table 5-7, LenSpecMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) With only the Tool# cycle parameter present, the spindle will turn on in reverse and the canned cycle will come straight down measuring the tool length and storing it in the tool-length register. The Tool# cycle parameter must be the same as the current tool in the spindle.
EstDiam	This is the rough diameter of the tool and is only used in this cycle to determine if the spindle should be turned on in reverse or forward. If you have a left-handed tool you would give a negative value to the diameter. If this parameter is left off the control will always turn on in reverse by default. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter Z first pick, MEDIUM feedrate . Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter Z final pick, SLOW feedrate . This is used for the same reason as the OvrMedFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the machine setup parameter RPM for calibration and tool measurement . This is used for the same reason as the OvrMedFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for the specific tool being checked or set.

You must have the tool positioned over the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less than 0.200" (5.08 mm).

NOTE: If the spindle is locked, you may have to unlock it to manually orient the tool tooth over the probe stylus.

To measure the tool length:

1. Jog the tool to the top of the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus.
2. From the MDI mode, input:

LenSpecMea Tool#(tool#) EstDiam(n), Exit and press the **START** button. Where **Tool#** is the tool number and **EstDiam** is roughly the diameter of the special tool.
For example: LenSpecMea Tool# 3 EstDiam 3.5
3. The spindle will turn on in reverse and the Z-axis should go down and touch the top of the probe stylus keeping the X and Y position the same then rapid up 0.02" (0.508 mm) and then retouch using the slow feedrate programmed in the machine variables. The cycle will then update the tool-length offset register clearing any value in the length wear register, turn the spindle off and return the tool to the Z height where it started.
4. The Tool Length has been set and you can now change to another tool, and repeat steps 1 through 3.

Manual Tool Diameter Measure for Special Tools (DiaSpecMea)

Format: DiaSpecMea Tool#(tool#) EstDiam(n) DistDown(n)
OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

This cycle is used to measure the diameter of irregularly shaped tools or tools with a hole in the center of the bottom. Refer to **Table 5-8**.

Table 5-8, DiaSpecMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) The Tool# cycle parameter must be the same as the current tool in the spindle.
EstDiam	This is the rough diameter of the tool. (Required) The diameter specified in this parameter should be larger than the actual diameter of the tool being measured but no more than 0.04" (1.0 mm) over. If you have a left-handed tool, you would give a negative value to the diameter so the spindle will turn on in forward direction.
DistDown	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum DistDown value is 0.55" (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55" (13.97 mm), the control will issue an error message. If DistDown is not set, the cycle will use a default value of 0.1" (2.54 mm). (Optional) [Default: 0.1"] Ball nose cutters and special cutters that require a move down more than 0.55" (13.97 mm) are not supported.
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter Z first pick, MEDIUM feedrate . Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter Z final pick, SLOW feedrate . This is used for the same reason as the OvrMedFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)

(Continued...)

Table 5-8, DiaSpecMea Entry Fields (Continued)

Entry Fields	Description
OvrRPM	This is the override for the RPM that was set in the machine setup parameter RPM for calibration and tool measurement . This is used for the same reason as the OvrMedFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for the specific tool being checked or set.

You must:

- Load the tool in the spindle and call up that tools offset.
- Know the distance from the top of the probe stylus down you will have to move so that the largest part of the tool diameter is even with the side of the probe stylus for diameter measurement. That value will be placed in **DistDown** if different then the default 0.1" (2.54 mm).
- Position the tool over the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less then 0.200 (5.08 mm).

To measure the tool diameter:

1. Jog the tool to the top of the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus.
2. From the MDI mode and the spindle off, input:

DiaSpecMea Tool#(n) EstDiam(n) DistDown(n), exit and press the **START** button. Where **Tool#(n)** is the tool number, **EstDiam(n)** is roughly the diameter of the special tool (this should be larger but not more then 0.100" (2.54 mm) larger), and **DistDown(n)** is the Z-axis move down needed if different then the default 0.100" (2.54mm) so that the largest part of the tool diameter comes in contact with the edge of the probe stylus.

For example:

DiaSpecMea Tool# 3 EstDiam 3.5 DistDown .25
exit and press the **START** button.

3. The Z-axis will feed down with the spindle on, touching the top of the probe stylus. Once the top of the probe is found, the Z-axis will rapid back up above the probe and move over to one side of the probe stylus. The tool will then move down the distance in **DistDown** or 0.1" (2.54 mm) if **DistDown** is not programmed. Then, with the spindle turning in reverse, the canned cycle will touch the side of the tool to the probe stylus twice on opposite sides establishing the tool's diameter. The new diameter will then be stored in that tool's diameter register and clear any value in the diameter wear register. The Z-axis will rapid up to machine home.
4. The Tool Diameter has now been set and you can change to another tool and repeat steps 1 through 3.

Tool Breakage, Length, and Diameter Wear Detection (BrkWearDet)

Format: BrkWearDet Tool#(tool#) EstDiam(n) MaxLenAdj(n)
 MaxDiaAdj(n) DistDown(n) Update(n) OvrMedFeed(n) OvrSlwFeed(n)
 OvrRPM(n)

Refer to **Table 5-9**.

Table 5-9, BrkWearDet Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) The Tool# cycle parameter will be the tool number you want checked.
EstDiam	This is the rough diameter on the bottom of the tool. (Optional) The diameter specified in this parameter should be roughly the diameter on the bottom of the tool that you want to be over the center of the probe stylus. If you have a left-handed tool, you would give a negative value to this diameter so the spindle will turn on forward verses reverse. When stepping over for checking the diameter of the tool, this cycle will use the diameter in the tool table for the tool being checked.
MaxLenAdj	The maximum length wear value limit. The cycle will check to see if the cutter length has changed by more then this amount and will alarm, stopping the program if exceeded. If not set, the cycle will not check the tool length. (Optional) NOTE: At least one, MaxLenAdj or MaxDiaAdj must be set or the cycle will alarm.
MaxDiaAdj	The maximum diameter wear value limit. The cycle will check to see if the cutter diameter has changed by more then this amount and will alarm, stopping the program if exceeded. If not set, the cycle will not check the tool diameter. (Optional) NOTE: At least one, MaxLenAdj or MaxDiaAdj must be set or the cycle will alarm.
DistDown	The distance to go down along the side of the probe stylus when doing a diameter check. The maximum DistDown value is 0.55" (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55" (13.97 mm), the control will issue an error message. If DistDown is not set, the cycle will use a default value of 0.1" (2.54 mm). (Optional) [Default: 0.1"] Ball nose cutters and special cutters that require a move down more than 0.55" (13.97 mm) are not supported.

(Continued...)

Table 5-9, BrkWearDet Entry Fields (Continued)

Entry Fields	Description
Update	If this is undefined or set to No , the Break and Wear cycle will not update the diameter or length wear register each time it checks a tool. If set to Yes , the cycle will update the wear registers. In both cases the control will alarm when the maximum limit set in MaxLenAdj or MaxDiaAdj has been exceeded. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter Z first pick, MEDIUM feedrate . Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter Z final pick, SLOW feedrate . This is used for the same reason as the OvrMedFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the machine setup parameter RPM for calibration and tool measurement . This is used for the same reason as the OvrMedFeed cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for the specific tool being checked or set.

Warning: Running this cycle without first initially setting the length and diameter offset could result in damage to the probe and/or the machine tool. **CalTIPrb (Probe Calibration)** and **LenSpecMea (Length and Diameter) Automatic Tool Length and Diameter set**, or **LenSpecMea (Length Special Manual Tool Length Measure for Special Tools)** and **DiaSpecMea (Diameter Special Manual Tool Diameter Measure for Special Tools)** must be run first before using the **BrkWearDet (Break and Wear)** cycle.

The **Break and Wear** cycle loads the tool, checks, and updates length and diameter wear registers, if specified, until a maximum value is exceeded, then it will alarm out stopping the program.

This cycle can be used in place of calling up a tool before running it.

You must know the distance from the top of the probe stylus down that you will have to move so that the largest part of the tool diameter is even with the side of the probe stylus for diameter measurement. That value will be placed in **DistDown** if different than the default 0.1" (2.54 mm).

To check the tool length and/or tool diameter for wear or breakage:

In place of the usual "**Tool#(tool#) MCode6**" command, use:

```
BrkWearDet Tool#(tool#) EstDiam(n) MJaxLenAdy(n) MaxDiaAdj(n)
DistDown(n) Update(n)
```

at a tool change according to the instructions above and the control will check the tool prior to using it. To activate the new offset wear values you must call that tool with "**Tool#(Tool#) MCode6**" after this cycle has been run.

Spindle Probe Cycles

This section describes operation and an overview of the conversational programming spindle probing cycles available in 3000M Probe option (included with Probe option).

These cycles are designed to assist in part setup only.

Before using your spindle probe for part setup, you must set the probe up according to the probe manufacturer's specification so it is set to turn on with a signal (if cordless) from the optical module sending unit and to automatically time out after approximately 120 seconds.

Rotation, Mirroring, and Scaling with (RMS) is not allowed while running these cycles. If any of these cycles are in a subprogram, you cannot call them using RMS. Plane will be set to XY when these cycles are complete.

This section contains the following topics:

- ❑ Spindle probe cycle designations
- ❑ Canned cycle parameter settings
- ❑ Description of spindle probe cycles

Spindle Probe Cycle Designations

The following summarizes the cycles available:

CalibPtPrb **Spindle Probe Calibration Cycle**
This is used to set the effective probe stylus diameter and set the compensation factor for any run-out of the probe stylus.

You will also need to calibrate the probe using the **CalibPtPrb** cycle.

NOTE: On machines that do not have spindle orientation or if you are using a corded probe or cordless UD probe and cannot orient the spindle 180 degrees during calibration, the spindle probe stylus needs to be indicated true to the spindle centerline. In this case, the accuracy of the spindle probe is only as good as the stylus concentricity to the spindle. Calibration must be done at least once before using the spindle probe. Once calibrated, calibration does not have to be done again unless you replace the probe stylus.

EdgeFind **Single Surface Measure/Edge Find**
This cycle will find a single surface and store that surface in a work or fixture offset register if programmed.

CornerOut **Outside Part Corner Find**
This cycle will find the X & Y surface on an outside corner of a part and store that location in a work or fixture offset register if programmed.

CornerIn **Inside Part Corner Find**
This cycle will find the X & Y surface in an inside corner of a part and store that location in a work or fixture offset register if programmed.

InOutBoss **Inside or Outside Hole or Boss Center Find**
This cycle will find the X & Y center of an inside hole or outside standing boss on a part and store that location in a work or fixture offset register if programmed.

InOutWeb **Inside or Outside Web or Slot Center Find**
This cycle will find the X or Y center of an inside or outside web or slot on a part and store that location in a work or fixture offset register if programmed. The slot or standing web must be parallel to either the X or Y axes.

- ProbeMove** **Protected Positioning Move**
This cycle allows for safe positioning of the probe around the part and will generate an alarm if an obstruction is encountered.
- SkewComp** **Skew Error or Angle Find**
This cycle will make two touches on a surface in the X- or Y-axes and stores the angle relative to the 3 O'clock position. This cycle can also activate **SkewComp** at the same time as it is measured or in a subsequent call at another place in the program without measuring again.

Canned Cycle Parameter Settings

Before you set the cycle parameters for the Spindle probe you must:

- Know what the exact diameter of the Ring Gauge (calibration standard).
- Know that the Ring Gauge is a standard that is specifically designed for calibrating the probe. The **GaugeDiam** cycle parameter is the diameter of hole that comes in contact with the probe stylus during calibration and should be an exact measurement.

Description of Spindle Probe Cycles

This section contains detailed descriptions of the spindle probe cycles:

- Spindle Probe Calibration (CalibPtPrb)**
- Edge Finding (EdgeFind)**
- Outside Corner Finding (CornerOut)**
- Inside Corner Finding (CornerIn)**
- Inside/Outside Boss/Hole Finding (InOutBoss)**
- Inside/Outside Web Finding (InOutWeb)**
- Protected Probe Positioning (ProbeMove)**
- Skew Error Find (SkewComp)**

Spindle Probe Calibration (CalibPtPrb)

Format: CalibPtPrb Boss Top(n) DistDown(n) DistBack(n)
GaugeDiam(n) DistInX(n) DistInY(n)

Refer to **Table 5-10**.

Table 5-10, CalibPtPrb Entry Fields

Entry Fields	Description
Boss	Set Boss to Yes if you are calibrating to a boss verses a ring gauge; otherwise, do not set or set to No . Default is No . (Optional)
Top	If set to Yes , the cycle will find the top of the part before calibrating the probe. If Boss cycle parameter is set to Yes , Top is forced to Yes as well; otherwise, the default is No . (Optional)
DistDown	The distance to go down from the top of the ring gauge or standing boss for calibration. This is only used if the Boss cycle parameter is set to Yes . Without any DistDown value, the cycle will bring the probe down past the top of the ring gauge after finding the top, 0.1" (2.54 mm). Note: If the stylus ball is greater than .2" (5.08 mm), DistDown must be set to at least half the ball diameter. (Optional)
DistBack	The DistBack parameter specifies the distance to back away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm) if not set. (Optional)
GaugeDiam	The diameter of the ring gauge hole the probe stylus will come in contact with. This is only to override the value in the machine setup parameter Diameter of spindle probe Gauge if needed and <u>should be an exact measurement</u> . (Optional)
DistInX	The distance from the starting point to move in the X-axis to find the top of the gauge. The default, if Boss is not set or set to No , is 0.1" (2.54 mm) beyond the edge of the ring gauge hole. If Boss is set to Yes , the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the gauge. The default is the current probe position. (Optional)

You must have:

1. The probe in the spindle.
2. The Ring Gauge mounted on the machine table.

To calibrate the probe:

1. Jog the probe to the approximate center of the ring gauge by eye and into the hole of the ring gauge at the depth that you wish the probe stylus to come in contact with the inside of the ring gauge hole.
2. From the MDI mode, go to the **F5 (Mill)**, then **F10 (Probe)**, then **F3 (SpinPro)**, then select **Probe Calibration**. Type the appropriate information, press **exit**, and then press **START**.
3. The probe will touch four sides of the inside of the hole. The spindle will rotate (if the machine has spindle orientation) 180 degrees and touch the same four sides again establishing the center of the ring gauge. The spindle will then orient and touch four sides one more time calibrating the probe.
4. Remove the ring gauge from the machine and you are now ready to start spindle probing.

NOTE: On machines that allow the spindle probe to be installed in the spindle with more than one orientation, the probe stylus must be indicated true to the spindle centerline or the probe will not be accurate once removed and replaced into the spindle again.

Edge Finding (EdgeFind)

Format: EdgeFind SearchDir(XPlus, XMinus, YPlus, YMinus, ZPlus, or ZMinus) Offset(0–9)

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The EdgeFind Edge Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-11**.

Table 5-11, EdgeFind Entry Fields

Entry Fields	Description
SearchDir	Axis and direction to find edge. XPlus, XMinus, YPlus, YMinus, ZPlus, or ZMinus (Required)
Offset	Work Coordinate to update with edge location in X- or Y-axes. If set, work coordinate will be updated if XPlus, XMinus, YPlus, and YMinus are specified for SearchDir , or tool-length offset will be set for the current tool number if SearchDir is set to ZPlus, or ZMinus . NOTE: Before any tool-length offset is active, you must recall that tool. Work coordinate register or Tool-length register will not be updated if Offset is not set and a warning message will tell the operator no update has taken place. [Default: 0] Range (0–9) (Optional)

To use the Edge Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus less than 0.1" (2.54 mm) away from the surface to be found.
3. Input EdgeFind SearchDir(?) Offset(n). If this is run from inside a program, this line needs to be repeated for every surface you wish to find.

Caution: When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see "Protected Probe Positioning (ProbeMove)").

4. Execute that line in MDI by exiting and pressing **START**.

Outside Corner Finding (CornerOut)

Format: CornerOut SearchQuad(XPlusYPlus, XMinusYPlus, XMinusYMinus, XPlusYMinus) Top(Yes/NO) DistDown(n) DistSide(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset (0–9))

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset must be active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The CornerOut Outside Corner Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-12**.

Table 5-12, CornerOut Entry Fields

Entry Fields	Description
SearchQuad	Quadrant of corner to find. XPlusYPlus = +,+ (upper right) XMinusYPlus = -,+ (upper left) XMinusYMinus = -,- (lower left) XPlusYMinus = +,- (lower right) (Required)
Top	If set to Yes , the cycle will find the top of the part before finding the X & Y corner coordinate. Default is No . If Top is not set or is set to No , the Z-axis must be at the picking depth. If Top = Yes , then the Z-axis must be within 0.1" (2.54 mm) above the part. The probe stylus must be positioned within 0.1" (2.54 mm) from the outside of the corner in X & Y. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if Top parameter is set to Yes . Without any DistDown value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistSide	The distance over from the corner to find X & Y edge. This will allow for a part corner that has a large chamfer or radius where you cannot pick the edge close to the theoretical corner or has an obstruction interfering with the default move. Default is 0.4" (10.16 mm). (Optional)

(Continued...)

Table 5-12, CornerOut Entry Fields (Continued)

Entry Fields	Description
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. (Optional)
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the corner. (Optional)
Y	Same as X only for the Y-axis. (Optional)
Z	Same as X only for the Z-axis. (Optional)
Offset	Work Coordinate to update with edge location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if Offset is not set. [Default: 0] Range (0–9) (Optional)

To use the Outside Corner Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus less than 0.1" (2.54 mm) away from the outside of the corner you wish to find in X & Y. If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.
3. Input CornerOut SearchQuad(XPlusYPlus) Offset(n)
If this is run from inside a program, this line needs to be repeated for every corner you wish to find or whose position you want to reestablish.

Caution: When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see "Protected Probe Positioning (ProbeMove)") or use the **X, Y, or Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside Corner Finding (CornerIn)

Format: CornerIn SearchQuad(XPlusYPlus, XMinusYPlus, XMinusYMinus, XPlusYMinus) Top(Yes/No) DistDown(n) DistSide(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0-9)

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The CornerIn Inside Corner Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-13**.

Table 5-13, CornerIn Entry Fields

Entry Fields	Description
SearchQuad	Quadrant of corner to find. XPlusYPlus = +,+ (upper right) XMinusYPlus = -,+ (upper left) XMinusYMinus = -,- (lower left) XPlusYMinus = +,- (lower right) (Required)
Top	If set to Yes , the cycle will find the top of the part before finding the X & Y corner coordinate. Default is No . If Top is not set or is set to No , the Z-axis must be at the picking depth. If Top = Yes , then the Z-axis must be within 0.1" (2.54 mm) above the part. The probe stylus must be positioned within 0.1" (2.54 mm) from the outside of the corner in X & Y. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if Top parameter is set to Yes . Without any DistDown value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistSide	The distance over from the corner to find X & Y edge. This will allow for a part corner that has a large chamfer or radius where you cannot pick the edge close to the theoretical corner or has an obstruction interfering with the default move. Default is 0.4" (10.16 mm). (Optional)

(Continued...)

Table 5-13, CornerIn Entry Fields (Continued)

Entry Fields	Description
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm) if not set. (Optional)
DistInX	The distance from the starting point to move in the "X" axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
DistInY	The distance from the starting point to move in the "Y" axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the corner. (Optional)
Y	Same as X only for the Y-axis. (Optional)
Z	Same as X only for the Z-axis. (Optional)
Offset	Work Coordinate to update with edge location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if Offset is not set. [Default: 0] Range (0–9) (Optional)

To use the Inside Corner Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus less than 0.1" (2.54 mm) away from the outside of the corner you wish to find in X & Y. If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part; otherwise, the Z-axis should be at the side picking depth.
3. Input CornerIn SearchQuad(XPlusYPlus) Offset(0–9)
If this is run from inside a program, this line needs to be repeated for every corner you wish to find or whose position you want to reestablish.

Caution: When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see "Protected Probe Positioning (ProbeMove)") or use the **X, Y, or Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside/Outside Boss/Hole Finding (InOutBoss)

Format: InOutBoss Side(In/Out) Length(n) Width(n) Top(Yes/No)
 DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0–9)
 RepeatMeas(Yes/No)

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The InOutBoss Inside or Outside Boss/Hole Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-14**.

Table 5-14, InOutBoss Entry Fields

Entry Fields	Description
Side	Inside or Outside. In = Inside Hole Out = Outside Boss (Required)
Length	Estimated length in the X-axis of boss/hole if rectangular or the Diameter if round. (Required)
Width	Estimated width in the Y-axis of boss/hole. Width is only specified if boss or hole is rectangular in shape. (Optional)
Top	If set to Yes , the cycle will find the top of the part before finding center of hole or boss. If Side parameter is set to Out , Top is forced to Yes as well; otherwise, the default is No . (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the center. This is only used if Top parameter is set to Yes . Without any DistDown value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm) if not set. (Optional)

(Continued...)

Table 5-14, InOutBoss Entry Fields (Continued)

Entry Fields	Description
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default, if Side is not set or set to In , is 0.1" beyond the edge of the boss/hole. If Side is set to Out , the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is the current probe position. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the boss/hole center. (Optional)
Y	Same as X only for the Y-axis. (Optional)
Z	Same as X only for the Z-axis. (Optional)
Offset	Work Coordinate to update with the center location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if Offset is not set. [Default: 0] Range (0–9) (Optional)
RepeatMeas	If set to Yes , the cycle will do a preliminary measure in the "X" axis to get on center before measuring the Y-axis, making a total of 6 touches. If set to No , the cycle will only measure "X" once for a total of 4 touches. Default is No . (Optional)

To use the Inside/Outside Boss/Hole Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus the approximate center in X & Y within 0.1" (2.54 mm). If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.
3. Input InOutBoss Side(In/Out) Length(n) Width(n) Offset(0–9)
If this is run from inside a program, this line needs to be repeated for every boss/hole you wish to find or whose position you want to reestablish.

Caution: When positioning the probe from within the program, you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see "Protected Probe Positioning (ProbeMove)") or use the **X, Y, or Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside/Outside Web Finding (InOutWeb)

Format: InOutWeb Side(In/Out) Length(n) Width(n) Top(Yes/No)
DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0–9)

- An inside Web is a slot. An outside Web is a standing rib.
- Webs can only be measured in the X- or Y-axis.
- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The InOutWeb Inside or Outside Web Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-15**.

Table 5-15, InOutWeb Entry Fields

Entry Fields	Description
Side	Inside or Outside. In = Inside Hole Out = Outside Boss (Required)
Length	Estimated X width of Web if measuring in the X-axis. Length or Width must be specified but only one, not both.
Width	Estimated Y width of Web if measuring in the Y-axis. Length or Width must be specified but only one, not both.
Top	If set to Yes , the cycle will find the top of the part before finding center of Web. If Side parameter is set to Out , Top is forced to Yes as well; otherwise, the default is No . (Optional)
DistDown	The distance to go down from the top of part to find X or Y coordinate of the center. This is only used if Top parameter is set to Yes . Without any DistDown value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm), if not set. (Optional)

(Continued...)

Table 5-15 ,InOutWeb Entry Fields (Continued)

Entry Fields	Description
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default, if Side is not set or set to In , is 0.1" beyond the edge of the web. If Side is set to Out , the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is the current probe position. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the web center. (Optional)
Y	Same as X only for the Y-axis. (Optional)
Z	Same as X only for the Z-axis. (Optional)
Offset	Work Coordinate to update with the center location in X- or Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if Offset is not set. [Default: 0] Range (0–9) (Optional)

To use the Inside/Outside Web Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus the approximate center in X or Y within 0.1" (2.54 mm). If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.
3. Input InOutWeb Side(In/Out) Length(n) Offset(0–9)
If this is run from inside a program, this line needs to be repeated for every web you wish to find or whose position you want to reestablish.

Caution: When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see "Protected Probe Positioning (ProbeMove)") or use the **X, Y, or Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Protected Probe Positioning (ProbeMove)

Format: ProbeMove X(n) Y(n) Z(n) Feed(n)

- When an X, Y, and/or Z move is programmed using the **ProbeMove** (Protected Positioning Cycle), the control will stop and alarm if the probe stylus is triggered before reaching the target set in the X, Y, and/or Z parameters.
- This cycle is intended to offer some degree of safety when moving the probe around the part; however, it is not fool proof and will not protect against gross bad programming where the probe body would encounter an obstruction before the probe stylus is triggered. Extreme care should be taken to avoid this condition as probe damage may result.
- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.
- The ProbeMove Protected Probe Positioning Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-16**.

Table 5-16, ProbeMove Entry Fields

Entry Fields	Description
X	X Target position relative to current active work coordinate.
Y	Y Target position relative to current active work coordinate.
Z	Z Target position relative to current active work coordinate combined with the current active tool-length offset.
Feed	Feedrate at which to travel to target. Feed is only active for the current move so it must be restated every time or the default will take precedence. The default is set in the machine setup parameter Positioning feedrate normally . (Optional)

To use the ProbeMove Protected Probe Positioning Cycle:

1. Place the probe in the spindle and make sure that its tool and work coordinate offsets are active.
2. Input
ProbeMove X(n) Y(n) Z(n) Feed(n)
If this is run from inside a program, this line needs to be repeated for every move you wish to make.
3. Execute that line in MDI by exiting and pressing **START**.

Skew Error Find (SkewComp)

Format: SkewComp Action(Find/FindActive/Activate) EstAngle(n)
DistPicks(n) Top(Yes/No) DistDown(n) DistBack(n) DistInX(n) DistInY(n)
X(n) Y(n) Z(n)

- RMS cannot be used with **SkewComp**, skew error find.
- Skew error is only supported for along the side edge of a part relative to the X,Y plane.
- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The probe must be pre-positioned to the proper spot in relation to the part in accordance with the specified **EstAngle** cycle parameter as described below or an X, Y, and/or Z should be included for pre-positioning.
- The SkewComp Skew Error Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 5-17, SkewComp Entry Fields**.

Table 5-17, SkewComp Entry Fields

Entry Fields	Description
Action	<p>Find Finds the skew angle, but does not activate skew compensation.</p> <p>FindActive Finds the skew angle, and activates skew compensation.</p> <p>Activate Activates skew compensation with the current skew value, but will not rerun the cycle on the part.</p> <p>[Default: Find]</p> <p>NOTE: If Activate is used, all other SkewComp parameters are ignored.</p> <p>NOTE: Before using SkewComp Activate, you must have called SkewComp at least once with Find or FindActive, or the error message "Skew error has not been found!" is displayed.</p> <p>Skew compensation will be activated around the current active work coordinate and will only work from within the program being run. Skew compensation cannot be activated directly or indirectly using SkewComp from the MDI mode.</p> <p>The operator can run the SkewComp from MDI but must place SkewComp Activate inside the program for skew compensation to take effect.</p> <p>An Offset work coordinate call will deactivate skew compensation, necessitating a re-issuance of SkewComp Activate to activate skew compensation.</p> <p>Using FindActive or Activate will default the control to Absolute. If you are in Incremental, you will need to switch back after the cycle has been run.</p> <p>(Optional)</p>
EstAngle	<p>Estimated amount of angle from 3 O'clock. Default is "0" which will cause the cycle to find the angle of the back edge of the part starting its first pick in the upper-left corner and making the second pick to the left of that, as you are facing the surface being picked. Examples:</p> <p>EstAngle=90 Would start in the lower-left side, picking in the X positive direction, finding the skew of the left side of the part.</p> <p>EstAngle=-90 Would start in the upper-right side, picking in the X negative direction, finding the skew of the right side of the part.</p> <p>EstAngle=180 Would start in the lower-right front edge of the part, picking in the Y positive direction, finding the skew of the front edge of the part.</p> <p>[Default: 0] (Optional)</p>
DistPicks	<p>The distance from the first pick to the second pick. Default is 2.0" (50.8 mm)</p> <p>(Optional)</p>
Top	<p>If set to Yes, the cycle will find the top of the part before finding part skew angle. Default is No. If Top is set to Yes, the probe stylus should be pre-positioned within 0.1" (2.54 mm) above the part. If Top is set to No, the probe stylus should be positioned at the Z-axis depth from which you want to make side picks. (Optional)</p>

(Continued...)

Table 5-17, SkewComp Entry Fields (Continued)

Entry Fields	Description
DistDown	The distance to go down from the top of part to find part skew angle. This is only used if Top parameter is set to Yes . Without any DistSide value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	<p>Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1" (2.54 mm) if not set. This would be used to make sure that the cycle is picking from far enough away from the edge so that it will not trigger the probe prematurely when stepping over to make the second pick.</p> <p>Hint: If the EstAngle parameter is relatively accurate, this parameter will not be needed because the default will be good enough.</p> <p>(Optional)</p>
DistInX	The distance from the starting point to move in the "X" axis to find the top of the part. The default is 1.0" (25.4 mm) toward the part at the angle specified in the EstAngle parameter. (Optional)
DistInY	The distance from the starting point to move in the "Y" axis to find the top of the part. The default is 1.0" (25.4 mm) toward the part at the angle specified in the EstAngle parameter. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the skew angle. (Optional)
Y	Same as X only for the Y-axis. (Optional)
Z	Same as X only for the Z-axis. (Optional)

To use the Skew Error Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus to the appropriate start position relative to the part as specified by the **EstAngle** cycle parameter in **Table 5-17**. X or Y should be within 0.1" (2.54 mm) of the part edge. If **Top = Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part; otherwise, the Z-axis should be at the side picking depth. If run from within a program, probe must be pre-positioned.
3. Input SkewComp Action(Find/FindActive/Activate) EstAngle(n)
If this is run from inside a program, this line needs to be repeated every time you wish to find a skew angle.

Caution: When positioning the probe from within the program you should always use the **ProbeMove (Protected Probe Positioning)** cycle (see “Protected Probe Positioning (ProbeMove)”) or use the **X, Y, or Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Section 6 - Editing Programs

Write and edit program blocks using the CNC's Program Editor (the Edit screen). Activate the Program Editor to put the CNC in the Edit Mode.

Activating the Program Editor

You can activate the Program Editor screen either from the Program Directory or from the Manual screen. When you activate the Program Editor from the Program Directory, the highlighted program opens for editing. When you activate the Program Editor from the Manual screen, the loaded program opens for editing.

To activate the Program Editor from the Program Directory:

1. In the Program Directory, highlight a program.
2. Press **Edit (F4)**. The Program Editor opens the selected program for editing.

To activate the Program Editor from the Manual screen:

1. In the Manual screen, press **EDIT (F4)**. The Editor opens the loaded program. If no program is loaded, the CNC displays the message **No Program Selected !**.
2. Press **PROGRAM (F2)** to activate the Program Directory.
3. Highlight a program.
4. Press **Edit (F4)**. The Program Editor opens the selected program for editing.

The Program Editor Screen

The Program Editor monitors mode changes written to a program. The mode indicators displayed in the Program Editor indicate the CNC's active modes. Refer to **Figure 6-1**.

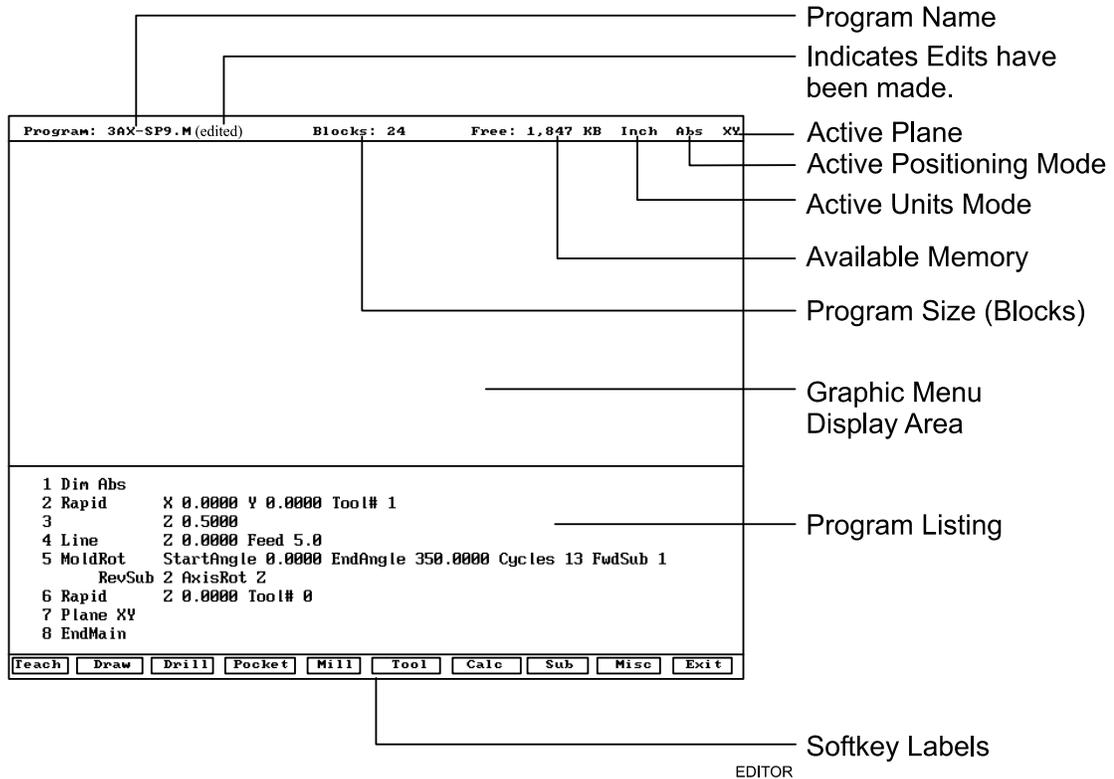


Figure 6-1, Program Editor

Program Name	Name of the program opened for editing.
(edited) Marker	Indicates that you have edited the program, but the edits have not been saved.
Active Plane	Current operating plane (XY , XZ , YZ).
Active Positioning Mode	Current measurement mode (Abs/Inc).
Active Units Mode	Current units mode (Inch/MM).
Available Memory	Amount of room remaining in the Editor's memory.
Program Size	Number of blocks in the program.
Graphic Menu Area	Area for displaying the Graphic Menus.
Program Listing	Current listing of the blocks in the open program.

Highlight	Selects a block for editing and acts as an insertion marker for adding new blocks. The CNC tracks program mode changes up to this point in the Program Listing.
Soft Key Labels	These labels define the soft key functions. The following sets of soft keys are available. <ul style="list-style-type: none">❑ Default set, normally visible❑ Misc soft keys, activated by pressing Misc (F9)❑ Sub soft keys, activated by pressing Sub (F8)

Saving Edits

The Program Listing displays text entered by the programmer. The CNC does not save edits until you exit the Editor. If the **(edited)** marker is visible at the top of the Program Editor, the open program contains unsaved edits.

To save edits:

1. In Edit Mode, press **Exit (F10)**. The CNC saves all edits and returns to the Program Directory.

Canceling Unsaved Edits

To cancel unsaved edits:

1. In Edit Mode, press **Misc (F9)**. Misc soft key labels are displayed.
2. Press **Quit (F8)**. The CNC displays the message **WARNING: Program has been edited. Sure you want to QUIT?**, and changes the soft key labels.
3. Press **Yes (F1)**. The CNC returns to the Program Directory without saving the edits. Press **No (F2)** to cancel.

Deleting a Program Block

To delete a program block:

1. In Edit Mode, highlight a block.
2. Press **CLEAR**.

Inserting a Program Block

To insert a program block:

1. In Edit Mode, highlight the block that will follow the inserted block.
2. Program the new block from the appropriate Graphic Menu. When you save the new block, it is displayed in front of the highlighted block. Blocks are automatically renumbered.

Editing a Program Block

To edit a program block:

1. In Edit Mode, highlight a block.
2. Press **ENTER** if the existing block is a move or cycle. The appropriate Graphic Menu opens.
3. Highlight the entry fields that require changes. Press **CLEAR** to erase the existing values.
4. Make the appropriate changes. Press **Save (F10)** to close the block.

<p>NOTE: When the program block's Graphic Menu offers two modes (for example, Cw/Ccw), highlight the block and press +/- to change the selection.</p>

Searching Blocks for Words or Numbers

Use **Search** to find a block number or word. **Search** looks only from the cursor position forward. To search an entire program, place the cursor at the beginning of the program, and then activate **Search**.

To search for a block number or word:

1. In Edit Mode, Press **Misc (F9)**. The soft key menu changes.
2. Press **Search (F3)**. The CNC prompts for the block number or word.
3. Use the ASCII chart to enter the block number or word. (You can also use the keypad to enter numbers.)
4. Press **ENTER**. The CNC searches and highlights the next block that contains the specified word or block number.

Scrolling the Program Listing

In Edit Mode, use the up and down **ARROWS** to scroll through the Program Listing.

Paging through the Program Listing

To scroll through the Program Listing one page at a time:

1. In Edit Mode, Press **Misc (F9)**. The CNC displays the Soft Key secondary functions.
2. Press the **PgUp (F4)** or **PgDn (F5)** keys to page forward or backward.
3. Press **Prev (F9)**. The CNC redisplay the Program Editor default soft keys.

Jumping to First or Last Block in the Program

To jump to the first or last block of the Program Listing:

1. In Edit Mode, press **Misc. (F9)**. The CNC displays the Soft Key secondary functions.
2. Press **Begin (F6)**. The Program Listing displays the first block of the program.

– or –

Press **End (F7)**. The Program Listing displays last block of the program.

Using Comments

The CNC will ignore comment blocks. You can add a new comment block to a program or convert an existing block into a comment. Comment blocks typically contain program setup or tool information, or are used to comment out existing blocks.

Writing a Comment Block

To write a comment block:

1. In Edit Mode, press **Misc. (F9)**. The CNC displays the Soft Key secondary functions.
2. Press **Comment (F2)**. The CNC prompts for a comment and the **ASCII** soft keys activate.
3. Use the ASCII Chart to enter comments.

Commenting Out Existing Blocks

To comment out an existing block:

1. In Edit Mode, highlight the block being commented out.
2. Press **0** on the keypad. The CNC displays an asterisk after the block number.

<p>NOTE: Off-line keyboard users, use the 0 key (not the asterisk key) to produce a comment block.</p>
--

Canceling a Comment

To cancel a comment:

1. In Edit Mode, highlight the comment block to be canceled.
2. Press **0**. The CNC deletes the asterisk and will no longer ignore the block during program execution.

NOTE: Off-line keyboard users, use the 0 key to switch the asterisk on or off.

Using Block Operations to Edit a Program

In conversational editor, use the **Misc (F9) – More (F1)** soft key to display the More pop-up menu. See **Table 6-1** for a description of the features. See **Figure 6-2, More Pop-up Menu**. To display the More pop-up menu:

1. In the conversational editor, select **Misc (F9)**.
2. Press **F1 (More)** to display the More pop-up menu.

Table 6-1, More Pop-up Menu

Feature	Hot Key	Description
Mark	'M'	Allows program blocks to be marked for a subsequent Cut, Copy, or Delete operation. Selecting Mark again turns off marking.
Copy	'C'	Copies marked blocks into scrap buffer for a subsequent Paste operation. Marking is turned off. Selecting Copy with no blocks marked copies the current block into the scrap buffer.
Paste	'P'	Paste contents of scrap buffer in current location (i.e., above current block).
Cut	'V'	Copies marked blocks into scrap buffer and deletes them. Marking is turned off. Selecting Cut with no blocks marked cuts the current block into the scrap buffer.
Delete	CLEAR	Deletes marked blocks.
Open Program	'O'	Allows the user to open another program for editing without leaving the editor. The scrap buffer is preserved which allows blocks to be copied or moved from one program to another program.

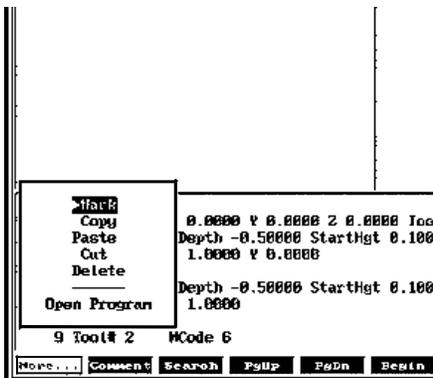


Figure 6-2, More Pop-up Menu

Section 7 - Viewing Programs with Draw

Draw Modes

The CNC has two Draw Modes, Draw Simulation Mode and Real-Time Draw Mode.

Draw Simulation Mode The CNC runs programs to simulate machine movements in the viewing area. The machine does not move.

Real-Time Draw Mode The CNC displays machine movements in the viewing area as it makes them.

In this manual, **Draw** refers to the CNC's Draw Simulation Mode. **DRAW** refers to the CNC's Real-Time Draw Mode. This reflects the corresponding labels on the soft keys.

NOTE: Although both Draw screens have viewing areas in the upper right corner of the screen, the operation of each varies. Real-Time DRAW looks like a programmed operation screen (Manual, Auto, or Single Step) with a viewing area in the upper right corner. The **DRAW (F10)** soft key highlights and the soft keys change. Draw Simulation looks like an **Edit** screen with a viewing area in the upper right corner, with the Draw soft keys active.

This section describes how to use the Draw Simulation Mode to view programs. It also describes how to change the viewing settings to allow you to inspect the programmed moves in detail.

Use the Draw Simulation Mode soft keys to change Draw view settings. Refer to "Section 8 - Running Programs" for instructions on how to run DRAW while cutting a part.

Draw allows you to verify a program before cutting any material. In Draw, you can change planar views, zoom in or out, or scale the drawing to better inspect the program.

NOTE: You can make display-setting changes at any time in the Draw Simulation Mode, but not in the Real-Time Draw Mode.

Activate **Draw Simulation Mode** from the Edit Mode or Manual Data Input (MDI) Mode. Activate **Real-Time Draw Mode** from Manual, Single Step, or Auto Mode.

Starting Draw

Start **Draw Simulation Mode** from the Edit Mode or Manual Data Input (MDI) Mode. The **DISPLAY (F5)** and **Parms (F9)** settings determine how Draw looks and runs. Adjust view settings before you start the simulation. Make setting changes using the soft keys.

In Draw Simulation Mode, the CNC does not halt the operation of the program for Dwells and tool mounts.

To activate the Draw Simulation Mode:

1. In Edit Mode and with a program loaded, press **Draw (F2)**. The CNC displays the viewing area in the upper right corner of the screen and activates the Draw soft keys.
2. Press **DISPLAY (F5)**. The CNC displays a pop-up menu, with **Fit** highlighted.
3. Press **ENTER**. **Fit** scales the image to fit in the viewing area.
4. Press **Run (F3)** to run the loaded program. The CNC displays an image of the tool path in the viewing area and the machine remains idle.

NOTE: The Draw display remains on the Edit screen until cleared.

NOTE: If you set Draw to display both compensated and uncompensated moves, Draw runs the program twice: once with compensated moves and once without compensated moves. The CNC displays the tool paths of both versions simultaneously on the screen, for comparison.

Draw Screen Description

The Draw screen looks like the Edit screen with the addition of a viewing area in the upper right corner. The CNC activates the Draw soft keys and highlights the **Draw (F2)** soft key. Refer to **Figure 7-1**.

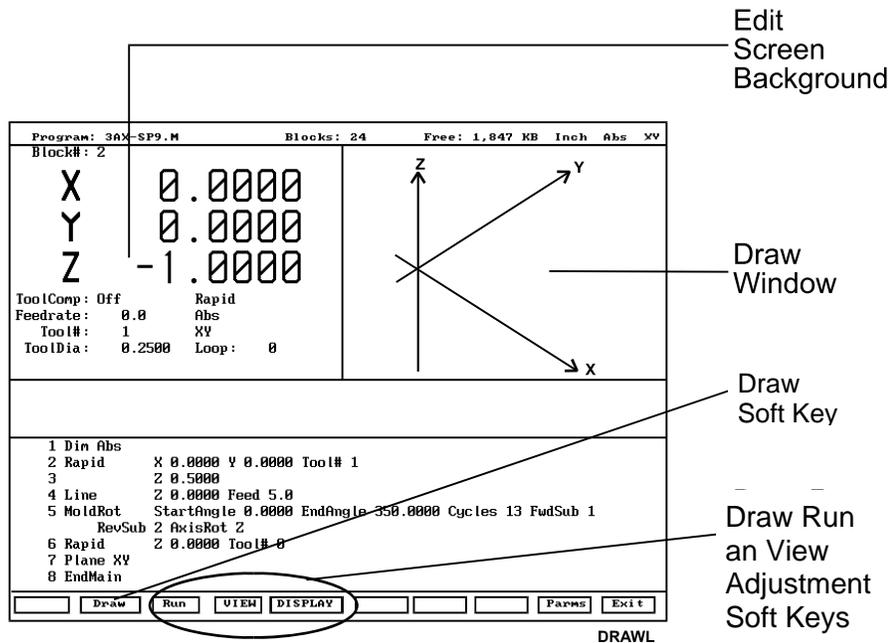


Figure 7-1, Draw Simulation Mode

Putting Draw in Hold

Press **Hold (F8)** or **HOLD** to pause a program running in Draw. Press **Start (F7)** or **START** to resume.

Canceling Draw

Press **Cancel (F9)** to stop program execution in Draw Simulation Mode.

Draw Parameters

In Draw Mode, the CNC displays rapid moves as dotted lines; feed moves as solid lines; and tools and drilled holes as cylinders.

Some view parameters deactivate during program simulation. To set viewing parameters before you run the program, activate the **Parms (F9)** pop-up menu. Refer to **Figure 7-2**. To set viewing parameters while you run a program, press the appropriate soft keys.

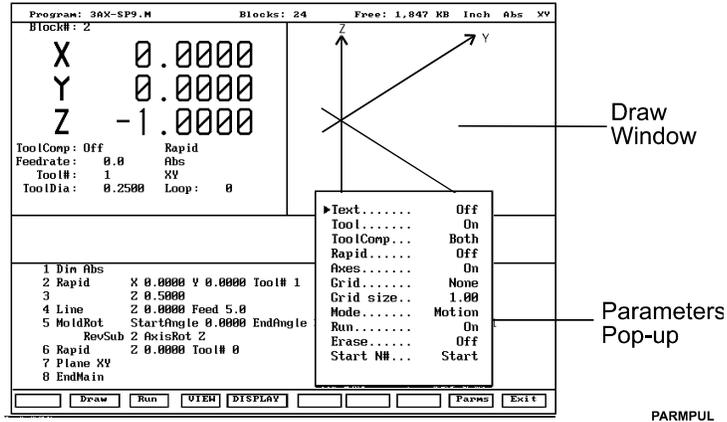


Figure 7-2, Draw Parameters Pop-up Menu

Text On or Off

Turn the **Text** parameter **On** to display program blocks as they run. This can help you pinpoint a problem. Draw runs slightly faster with **Text** turned **Off**. [Default: **On**]

To toggle the **Text** parameter setting **On/Off**:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Text**, and press **ENTER**. Text toggles between **On** and **Off**.
3. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

NOTE: In Draw Mode, press **Text (F4)** to change the **Text** parameter.

Tool On or Off

Turn **Tool On** to display a drawing of the tool as it moves through the part. Draw displays only the active tool. The tool must have a diameter on the Tool Page or the CNC will not display it in Draw. Draw scales the displayed tool (cylinder) to the corresponding diameter.

[Default: **On**]

To toggle the **Tool** parameter setting **On/Off**:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Tool**, and press **ENTER**. Tool toggles between **On** and **Off**.

NOTE: Press **Tool (F5)** to toggle the Tool parameter **On/Off**.

3. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Drawing Compensated Moves

Use **ToolComp** to view compensated moves in Draw.

ToolComp allows you to:

- Ignore** a program's compensated moves.
- Use (display)** a program's compensated moves.
- Both** run the program twice. (First without compensation and then with compensation.) Draw displays both tool paths simultaneously for comparison.

[Default: **Both**]

To set the compensation parameter:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **ToolComp**, and press **ENTER**. The CNC displays a second pop-up menu.
3. Highlight the desired selection, and press **ENTER**. The CNC closes the smaller pop-up menu.
4. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Showing Rapid Moves

Draw displays Rapid moves as dotted lines. Toggle the parameter **Off** to eliminate screen clutter. This parameter does not affect how the program runs, only how it is displayed. [Default: **On**]

To toggle the **Rapid** parameter **On** and **Off**:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Rapid**, and press **ENTER**. **Rapid** toggles **On** and **Off**.
3. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

SHORTCUT: Press **Rapid (F6)** to toggle the rapid move dotted lines **On** and **Off**.

Setting Grid Line Type

Draw displays a two-dimensional grid with dotted or solid lines. The CNC does not display the Grid lines in **Iso** view. [Default: **None**]

To set the **Grid** parameter:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Grid**, and press **ENTER**. The CNC displays the Grid pop-up menu.
3. Highlight the desired selection, and press **ENTER**. The CNC closes the Grid pop-up menu.
4. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Setting Grid Size

The CNC measures the adjustable grid size in the active Inch/MM Mode. [Default: **1.00**]

To set the **Grid size**:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Grid size**, and press **ENTER**. The CNC displays the number entry field.
3. Type the desired size, and press **ENTER**. The CNC displays the new value on the pop-up menu.
4. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Putting Draw in Motion, Single-Step, or Auto Mode

The Draw Simulation Mode executes programs in one of the following ways:

- Automatic Mode Fully automatic (**Auto**)
- Single-Step Mode One program block at a time (**S.Step**)
- Motion Mode One move at a time (**Motion**)

NOTE: To switch the CNC running mode: press **Auto (F1)** for the Automatic Mode; press **S.Step (F2)** for the Single-Step Mode; and press **Motion (F3)** for the Motion Mode.

In Automatic Mode (**Auto**), the CNC executes blocks sequentially until one of the following situations occurs: it reaches the end of the program, the CNC holds, or an error stops the program.

In Single-Step Mode (**S.Step**), the CNC executes one block each time you press **START**. This allows you to run the program one block at a time.

In Motion Mode (**Motion**), the CNC executes one motion at a time. This means that the CNC does not halt for blocks with no motion commands (for example, **Dim** blocks), as in Single-Step Mode. This decreases program run time.

Press **START** to execute each motion.

NOTE: Off-line keyboard users can use the spacebar to continue the program after each hold in the Single-Step or Motion Mode.

To select the default operating mode:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Mode**, and press **ENTER**. The CNC displays the Mode pop-up menu.
3. Highlight the desired mode, and press **ENTER**. The CNC closes the Mode pop-up menu.
4. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Automatic Draw Restart

The **Run** parameter determines whether Draw automatically restarts after a **DISPLAY** or **VIEW** setting changes. This allows you to make more than one setting change before restarting Draw.

With **Run** turned **Off**, Draw starts and restarts when you press **Run (F3)**. With **Run** turned **On**, Draw automatically starts after each **DISPLAY** or **VIEW** change. [Default: **On**]

To set the **Run** parameter:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Run**, and press **ENTER**. **Run** toggles **On** and **Off**.
3. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Erasing Display

The **Erase** parameter allows you to clear the display each time a new program runs in Draw (**Erase** turned **On**) or to display the new program tool path without clearing the previous graphic (**Erase** turned **Off**). This allows you to compare two or more program paths. [Default: **On**]

To set the **Erase** parameter:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Erase**, and press **ENTER**. **Erase** toggles **On** and **Off**.
3. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Running Draw for Selected Blocks

Use the **Start N#** and **End N#** commands to run part of a program or subprogram in Draw. Specify the **Start N#** and **End N#** block settings on the Parameters menu. The default settings are the **Start** (first) and **End** (last) blocks.

To run a subprogram, select the starting and ending blocks from the main program. Include the **Call**, **Loop**, or **RMS** block.

If a starting block is in a subprogram and the ending block is in the main program, the CNC will stop and generate an error message at the **EndSub** block.

Starting Draw at a Specific Block

To start Draw at a specific block:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **Start N#**, and press **ENTER**. The CNC displays the **Start N#** pop-up menu.
3. Highlight the desired selection, and press **ENTER**. If you selected **Start Of Program**, Draw will start at block 1. If you selected **Other Block**, type the block number, and press **ENTER**.
4. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Ending Draw at a Specific Block

To end Draw at a specific block:

1. In Draw Mode, press **Parms (F9)**. The CNC displays the Parameters pop-up menu.
2. Highlight **End N#**, and press **ENTER**. The CNC displays the **Start N#** pop-up menu.
3. Highlight the desired selection, and press **ENTER**. If you selected **End Of Program**, Draw will stop at the last program block. If you selected **Other Block**, type the block number, and press **ENTER**.
4. Press **Parms (F9)**. The CNC closes the Parameters pop-up menu.

Adjusting Draw Display

Draw has several display settings for the moves shown in the viewing window. Activate these settings from the **DISPLAY (F5)** pop-up menu. Refer to **Figure 7-3**.

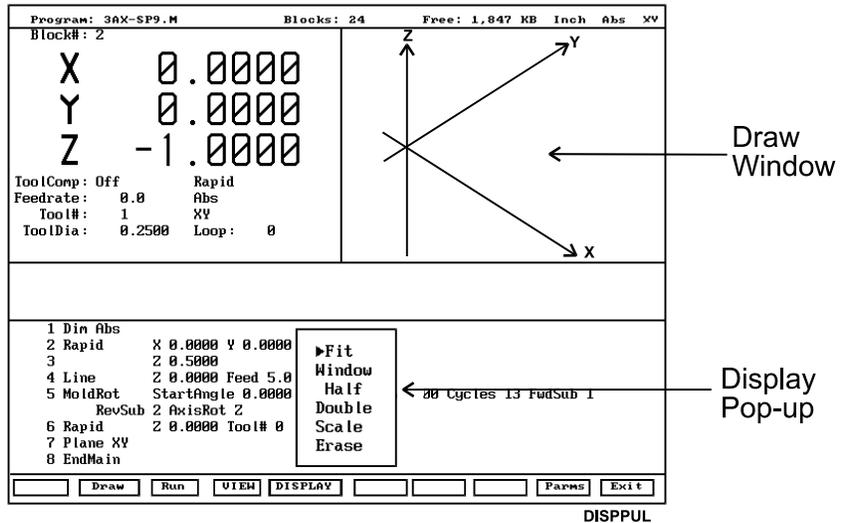


Figure 7-3, Display Pop-up Menu

Fitting the Display to the Viewing Window

To fit the display in the viewing area:

1. In Draw Mode, press **Display (F5)**. The CNC displays a pop-up menu.
2. Highlight **Fit**, and press **ENTER**. The CNC closes the pop-up menu and adjusts the display to fit in the viewing window.

Halving Display Size

To reduce the display size by half:

1. In Draw Mode, press **Display (F5)**. The CNC displays a pop-up menu.
2. Highlight **Half**, and press **ENTER**. The CNC scales the Display size to half its original size.

Doubling Display Size

To double the size of the display:

1. In Draw Mode, press **Display (F5)**. The CNC displays a pop-up menu.
2. Highlight **Double**, and press **ENTER**. The CNC closes the pop-up menu.

Scaling the Display by a Factor

To scale the Draw display by a factor:

1. In Draw Mode, press **Display (F5)**. The CNC displays a pop-up menu.
2. Highlight **Scale**, and press **ENTER**. The CNC closes the pop-up menu and prompts for the scale factor.

NOTE: Enter a value preceded by a decimal point to reduce the size or a whole number to enlarge the size.
--

3. Type the required factor, and press **ENTER**. The CNC scales the display by the specified factor.

Zooming In

To zoom in on a portion of the display:

1. In Draw mode, press **Display (F5)**. The CNC displays a pop-up menu.
2. Highlight **Window**, and press **ENTER**. The CNC displays a white window inside the viewing window.
3. Use the **ARROWS** to center the window over the area of interest.
4. Press **Expand (F5)** or **Compres (F6)** to expand or compress the window size.

NOTE: Use Reset (F7) to restore the window to its original size.
--

5. With the window sized and positioned, press **Enter (F10)**. The CNC closes the window. The next time **Draw** runs, the CNC will zoom in on the selected part of the display.

Erasing Display

To erase the display:

1. In Draw Mode, press **Display (F5)**. The CNC displays the Display pop-up menu.
2. Highlight **Erase**, and press **ENTER**. The CNC clears the display.

Changing Draw Views

Activate different view orientations from the **VIEW (F4)** pop-up menu. Refer to **Figure 7-4**.

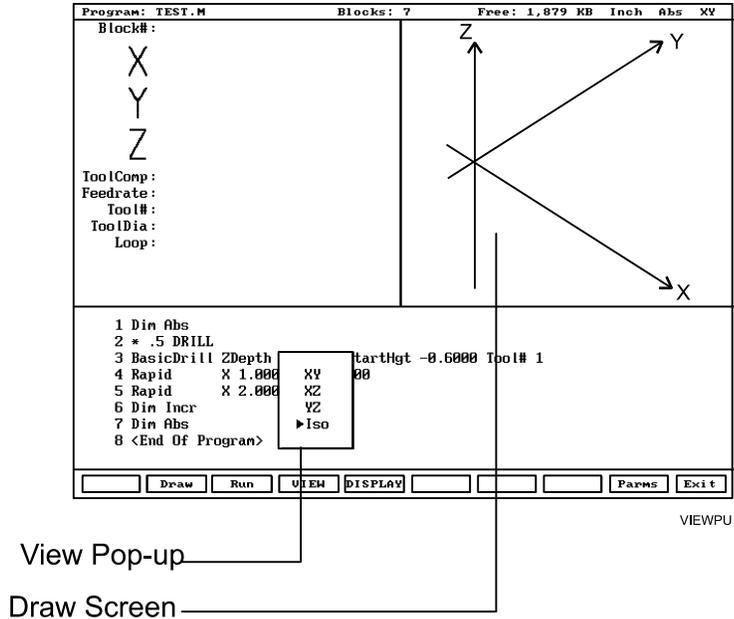


Figure 7-4, View Pop-up Menu

Selecting the View

To view Draw Mode from a different plane:

1. In Draw Mode, press **VIEW (F4)**. The CNC displays the VIEW pop-up menu.
2. Highlight a plane (**XY**, **YZ**, **XZ**, or **Iso**), and press **ENTER**. Draw Mode orients the display to the selected plane: **XY** (top view); **YZ** (side or end view); **XZ** (front view) or **Iso** (3-D or isometric view). The Iso view does not show grid lines.

Section 8 - Running Programs

There are three modes of programmed operation:

- Single-Step Mode** Run a program one block at a time.
- Motion Mode** Run a program from motion to motion. Available only in Single-Step Mode.
- Auto Mode** Run a program automatically.

The Auto and Single-Step screens are based on the Manual Mode screen. Use the soft key labels to distinguish between modes. The CNC highlights the soft key for the active mode.

The CNC runs only the currently selected program. Select programs from the Program Directory. (Refer to “Section 9 - Program Management.”)

In the Manual Data Input Mode (MDI), you do not need to select MDI code blocks.

The MDI Mode allows you to program a few quick moves without creating and saving a program. MDI is available only in the Manual Mode. (Refer to “Section 3 - Manual Operation and Machine Setup.”)

All programming tools, moves, and cycles are available in the MDI Mode.

Selecting Programs for Running

Before you run a program, select it into the CNC’s memory from the Program Directory. Refer to “Section 9 - Program Management” for instructions on selecting a program.

Running a Program One Step at a Time

The Single-Step screen provides access to the Single-Step Mode (**S.STEP**) and the Motion Mode (**MOTION**) screens. Either mode allows you to step through the program and verify the moves before production.

The Single-Step screen differs from the Manual screen as follows:

- ❑ **S.STEP** screen has fewer active soft keys
- ❑ **S.STEP (F5)** soft key highlights
- ❑ **S.STEP** indicator is displayed in status box (upper-right corner of screen)

To run a program in Single-Step Mode:

1. Select the required program and return to the Manual screen.
2. Press **S.STEP (F5)**. The CNC activates the Single-Step Mode.
3. Press **START**. The CNC executes a single block or motion.

NOTE: In Auto Mode, press **S.STEP (F5)** to activate Single-Step Mode.

Single-Step Mode vs. Motion Mode

In Single-Step Mode, the CNC holds after each block, even if a block does not include a move command. Press **START** to execute the following block.

In Motion Mode, the CNC holds after each machine move. It will not pause on blocks that do not command a motion. Press **START** to execute each machine move.

To switch the CNC between Single-Step (**S.STEP**) and Motion (**MOTION**) Modes:

1. Press **MOTION (F7)** to highlight the Active soft keys.

Holding or Canceling a Single-Step Run

Press **HOLD** to pause program execution. To restart the program, press **START**. To cancel a program that is on hold, press **MANUAL (F4)**. This cancels active canned cycles and tool compensation. All other modal settings remain active.

Single-Step Execution of Selected Program Blocks

To select a starting block using **ARROWS**:

1. Select the desired program and return to the Manual screen.
2. Press **S.STEP (F5)** to activate Single-Step Mode.

NOTE: Press **MOTION (F7)** to activate the Motion Mode.

3. Highlight the desired starting block.
4. Press **START** to execute the next block or motion.

SEARCH directs the CNC to search the program for a block number, a block containing a number, or a block containing specific text. Searching forward from the cursor location, the CNC highlights and displays the first occurrence of the entered item.

To select a starting block with the **SEARCH** feature:

1. Select the desired program and return to the Manual screen.
2. Press **S.STEP (F5)** to activate Single-Step Mode.
3. Press **SEARCH (F3)**. The CNC prompts for search text or number.
4. Type the search text or number, and press **ENTER**. The CNC runs the search and highlights the first occurrence of the search criteria.
5. Press **START** to start the program from the highlighted block.

Switching from Single-Step to Auto

To switch the CNC from Single-Step Mode to Auto Mode:

1. In the Single-Step Mode, press **AUTO (F6)**. The CNC completes the current move then holds.
2. Press **START** to restart and run the rest of the program in Auto Mode.

Auto Program Execution

Auto Mode is the CNC's production mode. Execute all or part of a program in Auto Mode. Activate Auto Mode from the Manual or Single-Step screens.

The AUTO screen differs from the Manual screen as follows:

- **AUTO** screen has fewer active soft keys
- **AUTO (F6)** highlights
- **AUTO** indicator is displayed in the status box (upper-right corner of the screen)

To run a program in Auto Mode:

1. Select the required program and return to the Manual screen.
2. Press **AUTO (F6)** to activate Auto Mode.
3. Press **START** to run the entire program for production.

Holding or Canceling an Auto Run

Press **HOLD** to pause program execution. To restart a program after a hold, press **START**.

To cancel program execution when a program is on hold, press **MANUAL (F4)**. This also cancels any active tool compensation and canned cycles. All other modal settings remain active.

Starting at a Specific Block

CAUTION: Select the specified starting block carefully. Modes and compensations enabled in the program before the specified block may no longer be effective.

To select a starting block before you run the program, using the **ARROWS**:

1. Select the required program and return to the Manual Mode.
2. Highlight the required starting block.
3. Press **START** to automatically run the program starting from the selected block.

SEARCH directs the CNC to search the program for a block number, a block containing a number, or a block containing specific text. The CNC highlights and displays the first block found that contains the specified information. **SEARCH** seeks only forward in the program.

To select the starting block before a program run, using the **SEARCH** feature:

1. Select the required program and return to the Manual screen.
2. Press **AUTO (F6)** to activate the Auto Mode.
3. Press **SEARCH (F3)**. The CNC prompts for search criteria.
4. Type the search number or text, and press **ENTER**. The CNC runs the search and then highlights and displays the first occurrence of the search criteria.
5. Press **START** to run the program forward from the highlighted block.

Clearing a Halted Program

When the CNC encounters a program block that generates an error, it displays a Warning message and stops the program. Press **MANUAL (F4)** to reactivate the keypad.

A program error could generate more than one message. Refer to "Section 2 - CNC Console and Software Basics" for instructions on reviewing undisplayed error messages.

After you have corrected the program, restart it from the appropriate block.

Using Draw while Running Programs

In Real-Time Draw, the CNC displays the moves being executed. When Draw activates, the secondary display area changes into a DRAW viewing area and **DRAW (F10)** highlights.

Real-Time Draw Mode and Draw Simulation Mode contain the same settings. Make setting adjustments from the Draw Simulation Mode. (Refer to “Section 7 - Viewing Programs With Draw.”) Press **CLEAR** at any time to clear the Draw display.

NOTE: Do not confuse the two types of Draw screens. The Draw Real-Time Mode screen looks like a Manual screen with a viewing window in the upper right corner. In Draw Real-Time, **DRAW (F10)** highlights and the program execution soft keys activate. Draw Simulation Mode resembles an Edit screen that has a window in the upper right corner, with the Draw soft keys active.

To activate Draw while running a program:

1. Select the required program and put the CNC in the required execution mode (**S.STEP** or **AUTO**).
2. Press **DRAW (F10)** to change the upper-right status box to a window.
3. Press **START** to start the program. The CNC displays moves as they are executed.

Parts Counter and Program Timer

The CNC keeps track of program run-time (**TIMER**) and the number of completed parts (**PARTS**). The CNC displays Run-time in hours, minutes, and seconds. These two features are available in the Manual, Auto, and S.Step Modes. Refer to **Figure 8-1**.

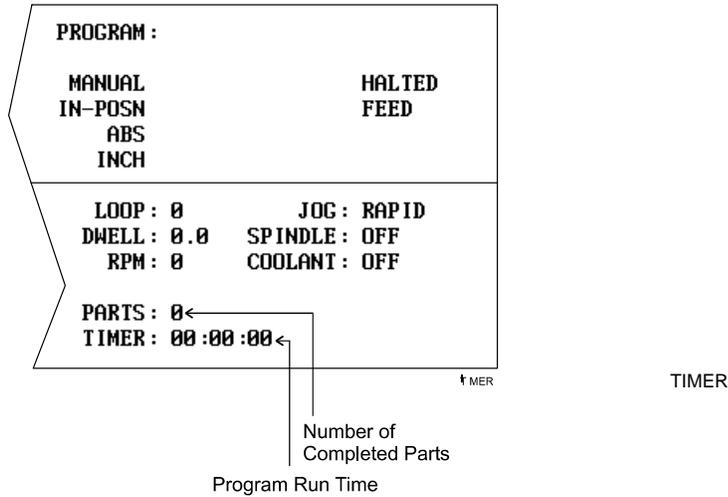


Figure 8-1, Program Timer and Parts Counter

The Timer begins timing the program run when you press **START** and stops when it encounters an **EndMain** block. Therefore, ensure that an **EndMain** block has been included at the end of the program.

The timer pauses if the CNC holds. The timer stops if you switch to Manual Mode. The Timer value remains the same until you switch to Auto or S.Step Mode again. Then, the timer resets to zero.

The Parts counter starts at zero and increments by one every time the CNC runs an **EndMain** block. Therefore, ensure that an **EndMain** block has been included at the end of the program. The CNC continues to count parts when you re-run the program in Auto or Single-Step. The parts counter value is maintained when you switch to Manual Mode, but will reset to 0 when you switch back to Auto or Single-Step Mode. The Parts Counter value can be modified via M-Codes. Refer to **Table 8-1, M-Codes Used with Parts Counter and Program Timer**.

Table 8-1, M-Codes Used with Parts Counter and Program Timer

M-Code	Function
M9355 X0	Prevents the parts counter from resetting to zero.
M9356 X0	Disables the Timer and Counter.
M9376 Xx	Presets any value into the parts counter register. For example, program M9376 X5 to preset 5 in the parts counter register.
M9377 Xx	Adds entered number to the parts counter. For example, if the current parts counter value is 4 and you then program M9377 X6, the new parts counter value will be 10.

Background Mode

In the Background Mode, you can create, edit, delete, copy, or print other programs while you run the selected program.

Most program management utilities are available in the Background Mode. You can also send or receive (but not execute) programs through RS-232 communications. The CNC does not display the soft keys for prohibited functions in Background Mode. The following functions are not available:

- Draw Graphics
- DNC execution
- Selecting another program to run
- G-Code to conversational converter
- Editing the currently running (selected) program

To activate the Background Mode:

1. While the selected program is running in the Auto or Single-Step Mode, press **Program (F2)**. The CNC activates the Program Directory.

In Background Mode, the CNC displays messages for the running program in a message box. To clear the messages, press **CLEAR**.

For information on the Edit Mode, refer to “Section 6 - Editing Programs.” For information on RS-232 communications, refer to “Section 11 - Communications and DNC.” For information on the program management utilities, refer to “Section 9 - Program Management.”

Section 9 - Program Management

Program Directory

The Program Directory provides access to all program management and disk functions. These include creating, selecting, deleting, undeleting, and copying programs. The Program Directory also provides access to the floppy disk drives and the communication configuration screens.

To activate the Program Directory from the Manual screen, press the **PROGRAM (F2)** soft key.

The Program Directory **USER** list displays programs stored in the CNC. The CNC program names are followed by a **.M** extension. Change the display mode to view programs with other formats. Refer to **Figure 9-1**.

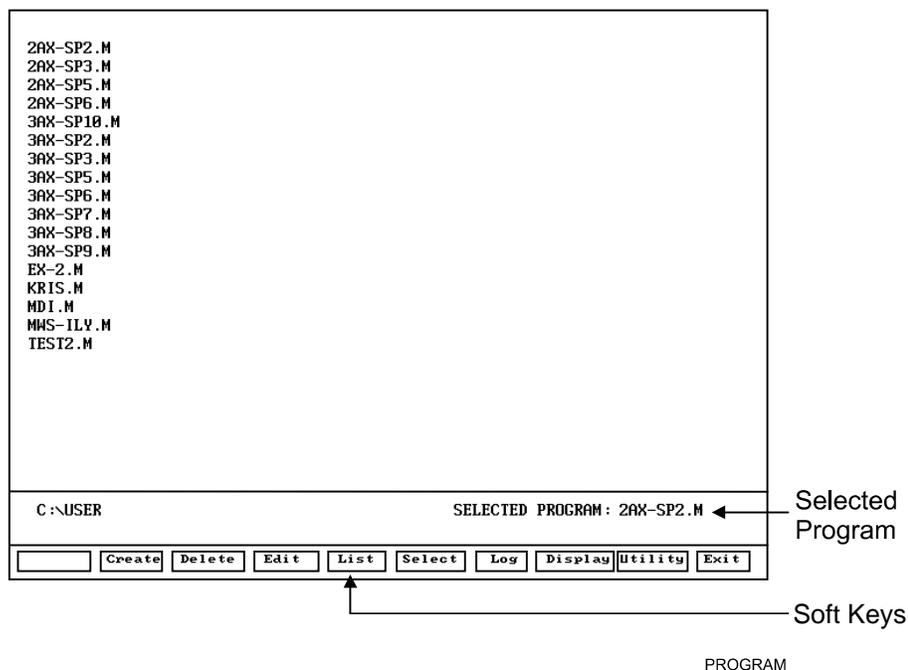


Figure 9-1, Program Directory

Changing the Program Directory Display

The Program Directory has four display modes:

- Display only CNC programs (names with **.M** extensions).
- Display CNC programs (names with **.M** extensions) with size, date, and time of last edit.
- Display all programs in all formats.
- Display all programs in all formats with size, date, and time of last edit.

Press **Display (F8)** to cycle the Program Directory through different display modes. Usually, only part programs, with a **.M** extension, are shown.

CAUTION: The Program Directory can provide access to internal CNC programs. Tampering with internal programs can cause a machine malfunction.

Creating a New Program

To create new programs in the Program Directory:

1. In the Program Directory, press **Create (F2)** to display the prompt: **NEW PROGRAM:** _ on the Message line.
2. Press **ASCII (F2)** to activate the ASCII Chart.
3. Use the ASCII Chart and the number keypad to enter the new program's name.
4. Press **ASCII (F2)** to deactivate the ASCII Chart.
5. Press **ENTER** to place the new program in the **USER** list in alphabetical order.

Choosing Program Names

Program names can contain a maximum of eight characters. The CNC displays program names in uppercase characters. Use letters, numbers, and dashes to name programs. Give each program a unique name. The CNC automatically assigns the **.M** extension.

NOTE: You can use blank spaces in the program name.

Loading a Program for Running

You must load a program before you can run it. Only one program can be loaded at a time.

To Load a program:

1. In the Program Directory, use **ARROWS** to highlight a program.
2. Press **Select (F6)**. The CNC loads the program. The name of the currently loaded program displays next to the **SELECTED PROGRAM** label at the bottom of the screen.

Selecting a Program for Editing and Utilities

Delete (F3), **List (F5)**, and most other utilities carry out their functions on the highlighted program.

NOTE: The highlighted program is selected for cutting parts.

To select a program for editing:

1. From the Program Directory, press **Display (F8)** to switch the display setting until the CNC displays only the CNC programs (*.M).
2. Highlight the name of the required program.
3. Press **Edit (F4)** to activate the Program Editor and display the highlighted program on the Edit screen.

Maximizing Program Storage Space

The CNC has a fixed amount of space available for programs. Use the System Information screen to check available space. Refer to **Figure 9-2, System Information Screen**.

Total Space Available for the System	Available program storage space
Total Free User Space	Remaining new programs storage space

During program execution, the CNC generates a second program with the same name followed by an **.S** extension. The **.S** programs contain information required by the CNC. Normally, **.S** programs are slightly larger than the part programs that generated them. Deleting a part program also deletes the associated **.S** file.

The CNC is not the optimal place for long-term storage of part programs. Back up part programs to floppy disks (or other computers).

If too many old programs accumulate, the CNC could run out of memory. To access more memory, temporarily delete inactive **.S** files. The CNC will automatically replace the **.S** files the next time you run the programs.

By default, the Program Directory displays only part programs (programs followed by a **.M**). To list **.S** files on screen, press **Display (F8)** until the CNC displays the **.S** programs in the Program Directory.

Program File Utilities

You can access some utilities directly from the soft keys, others through the **Utility (F9)** pop-up menu. The Utility pop-up menu lists 14 utilities on two pages. Highlight **MORE ...**, and press **ENTER** to activate the second page of the menu.

Displaying Program Blocks (Listing a Program)

List (F6) displays the blocks in a program. This allows you to look over a program without making inadvertent changes. The **List** feature displays only CNC programs (programs with a **.M** extension on their names).

The soft keys available on the List screen include: **Draw (F2)**, **Search (F3)**, **PgUp (F4)**, **PgDn (F5)**, **Begin (F6)**, **End (F7)**, and **Exit (F10)**. They function the same way as in the Program Editor (refer to “Section 4 - Writing Programs”).

To list a program:

1. Highlight the required program.
2. Press **List (F5)**. The CNC displays the program blocks. The CNC will not make editing changes in the List Mode.

Deleting a Program

To delete a program:

1. From the Program Directory, highlight a program.
2. Press **Delete (F3)**. The CNC prompts you to confirm the deletion. The soft keys change for your response.
3. Press **Yes (F1)**. To delete the selected program.

– or –

Press **No (F2)** to cancel the Deletion.

NOTE: Deleting a program automatically deletes its associated .S file.
--

Reading Disks in Floppy Drives (Logging to Other Drives)

Most utilities apply to programs in the **USER** list by default. The **Log** feature allows you to use the utilities on floppy disks.

To log the CNC to a floppy disk drive:

1. From the Program Directory, press **Log (F7)**. The CNC prompts you to select a drive.
2. Highlight a drive, and press **ENTER**. The Program Directory lists files stored on the selected drive.

Marking and Unmarking Programs

Most utilities can operate on more than one program at a time. The Program Directory allows you to select one, some, or all of the programs in the **USER** list.

Marking Programs

To mark a program:

1. From the Program Directory, highlight the program.
2. Press **ENTER**. The marked program highlights and the highlight bar advances to the next name.
3. Press **ENTER** to mark the next program.

– or –

Highlight another program in the list, and press **ENTER**.

4. Repeat these steps to mark all required programs.

Unmarking Marked Programs

To unmark a previously marked program:

1. Highlight the marked program, and press **ENTER**. The highlight clears.

Marking all Programs

To mark all programs:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **Mark All**, and press **ENTER** to mark all programs for pending operation.

Unmarking all Marked Programs

To unmark all marked programs:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **Unmark All**, and press **ENTER**. The CNC unmarks all programs.

Deleting Groups of Programs

To delete a group of programs:

1. From the Program Directory, mark the required programs and press **Delete (F3)**. The CNC prompts you to confirm the deletion. The soft keys change for your response.
2. Press **Yes (F1)**. The CNC deletes the selected programs. Press **No (F2)** to cancel the command.

Restoring Programs

The CNC allows you to recover most deleted programs. Sometimes, only part of a deleted program can be recovered. Restore (undelete) a program as soon as possible after it is deleted.

To restore a program:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **Restore**, and press **ENTER**. If the CNC finds programs to restore, it displays a pop-up menu listing recoverable programs. If the CNC cannot recover any deleted programs, it displays the message: **No programs available for restore**.

NOTE: When you delete a program, the CNC inserts a ? in place of the first character of the program.

3. Highlight the desired program, and press **ENTER**. The CNC prompts for the first character of the deleted program's name. The ASCII soft keys activate.
4. Use the ASCII Chart to enter the first character, and press **ENTER**. The CNC displays an information message.
5. Press **Cont (F10)**. The CNC transfers the program to the **USER** list.

NOTE: Restored programs may not contain all of the original information. Review the entire program to verify.
--

Copying Programs to Floppy Disks

To copy a program to a floppy disk:

1. From the Program Directory, mark the required programs.
2. Press **Utility (F9)**. The CNC displays the Utility pop-up menu, with the **Copy** selection highlighted.
3. Press **ENTER**. The CNC displays the **Copy to:** pop-up menu.
4. Highlight the target drive. (usually **A:** for floppy drives) Press **ENTER**. The CNC copies the marked programs to the target drive.

Renaming Programs

To rename a program:

1. From the Program Directory, highlight a program.
2. Press **Utility (F9)**. The CNC displays the Utility pop-up menu.
3. Highlight **Rename**, and press **ENTER**. The CNC prompts for a new program name and the ASCII chart soft keys display.
4. Use the ASCII chart and the number keypad to enter the new name. Press **ENTER**. The CNC renames the program.

Printing Programs

Print paper copies of part programs to any standard IBM PC-compatible printer. The CNC sends all marked programs to the printer.

To print one or more programs:

1. From the Program Directory, highlight or mark the desired program(s).
2. Press **Utility (F9)**. The CNC displays the Utility pop-up menu.
3. Highlight **Print**, and press **ENTER**. The CNC prompts you to confirm the command. The soft keys change for operator response.
4. Press **Yes (F1)** to send programs to the printer. Press **No (F2)** to cancel.

Formatting Floppy Disks

Transfer programs to formatted floppy disks. Most disks are pre-formatted. The CNC can also format disks.

To format a floppy disk:

1. From the Program Directory, press **Utility (F9)**. A Utility pop-up is displayed.
2. Highlight **Floppy Format**, and press **ENTER**. The CNC displays a warning message and prompts you to confirm the command. The soft keys change based on your response.

CAUTION: Formatting a floppy disk erases all data on the disk.

3. Press **Yes (F1)**. CNC prompts for disk insertion. Press **No (F2)** to cancel.
4. Insert the floppy disk in the CNC's disk drive and press **Cont (F10)**. The CNC displays disk statistics as it formats.

Converting G-Code Programs to CNC Conversational Format

The CNC runs programs written only in the ANILAM Conversational Language Format. The G-Code conversion utility translates G-Code programs to the ANILAM Conversational Language Format. Thus, it is unnecessary for you to rewrite part programs written for other machines.

By default, the CNC names converted programs the same as the original program, but with an **.M** extension added. You can assign any valid name to a converted program. The conversion utility does not alter the original G-Code program in any way. The CNC will not convert **.M** or **.S** programs.

By default, the conversion utility writes the translated program to the CNC's current drive, usually the **USER** listing. You can force the location of the new program to any valid location by selecting **No (F2)** when the CNC prompts you to confirm the new program name.

Ensure that the CNC has the available space to store the converted program, which will be slightly larger than the original. The CNC can convert only one program at a time.

The G-Code utility supports an enhanced version of the RS-274D specification for G-Codes. Some G-Code blocks might not convert directly. The CNC inserts unconverted G-Code blocks as comment blocks, marked by an asterisk (*).

Review and test all converted programs with Draw before running them. Refer to "Section 7 - Viewing Programs With Draw" for details.

NOTE: The Program Directory's default display setting displays only program names that end with the **.M** extension. Press **Display (F8)** to switch the display setting and view programs with other formats.

To convert a G-Code program to conversational format:

1. To transfer the program to the CNC, copy it from a floppy disk or download it via the RS-232 link. Refer to "Section 11 - Communication and DNC" for downloading instructions.
2. From the Program Directory, mark the program being converted.
3. Press **Utility (F9)**. The CNC displays the Utility pop-up menu.
4. Highlight **G-Code Converter**, and press **ENTER**. The CNC prompts you to confirm the conversion. The soft keys change for operator response.
5. Press **Yes (F1)**. The CNC prompts you to confirm the new program name. The soft keys change for operator response. Press **No (F2)** to cancel.

6. To use the same name and location, press **Yes (F1)**. The CNC converts the program and displays the conversion statistics.

– or –

To enter a different name (or location), press **No (F2)**. The CNC prompts for the new name and activates the ASCII soft keys.

7. Type the desired name (or path), close the ASCII pop-up menu, and press **ENTER**. The CNC writes the conversational program. The Status box tracks the progress.

Refer to **Table 9-1** and **Table 9-2** for G-Code compatibility specifications.

Table 9-1, G-Code Conversion Number Formats

	Maximum Number of Digits Right/Left of Decimal Place		Example of Properly Formatted Value	
	Mode		Mode	
	Inches	MM	Inches	MM
Axes	4.4	5.3	X9999.9999	X99999.999
Feed	3.1	4.1	F999.9	F9999.9

Table 9-2, G-Code Equivalents

G-Code Format	Conversational Equivalent
X	X
Y	Y
Z	Z
F	Feed
M	MCode
S	RPM
T	Tool#
G00	Rapid
G01	Line
G02 Xn Yn Zn Rn In Jn Kn Cn	Arc Clockwise
G03 Xn Yn Zn Rn In Jn Kn Cn	Arc Counterclockwise
X	X
Y	Y
Z	Z
R	Radius
I	XCenter
J	YCenter
K	ZCenter
C	Angle

Table 9-2, G-Code Equivalents (Continued)

G-Code Format	Conversational Equivalent
G04 Tn	Dwell
G05 Xn Yn In Jn An Bn Ln	Ellipse
X	X
Y	Y
I	XCenter
J	YCenter
A	HalfLength
B	HalfWidth
L	Cw, Ccw
G06 Xn Yn Zn In Jn Ln	Spiral
X	X
Y	Y
Z	Z
I	XCenter
J	YCenter
L	Revs
G17	Plane XY
G18	Plane XZ
G19	Plane YZ
G40	ToolComp Off
G41	ToolComp Left
G42	ToolComp Right
G53 On Xn Yn Zn	Fixture Offsets
O	Offset Fixture#
X	X
Y	Y
Z	Z
G75 Xn Yn Mn Wn Hn Zn An Bn In Jn Un Vn Cn Sn Kn Pn	Frame Pocket
X	XCenter
Y	YCenter
M	IslandLen
W	IslandWid
H	StartHgt
Z	ZDepth
A	Stepover
B	DepthCut
I	RampFeed

Table 9-2, G-Code Equivalents (Continued)

G-Code Format	Conversational Equivalent
J	RoughFeed
U	InsideRad
V	OutsideRad
C	FrameWidth
S	FinStock
K	FinFeed
P	ReturnHgt
G76 Dn Jn Sn Kn	Hole Pocket
D	Diameter
J	RoughFeed
S	FinStock
K	FinFeed
G77 Xn Yn Hn Zn Dn An Bn In Sn Kn	Circular Pocket
X	XCenter
Y	YCenter
H	StartHgt
Z	ZDepth
D	Diameter
A	Stepover
B	DepthCut
I	RoughFeed
S	FinStock
K	FinFeed
G78 Xn Yn Mn Wn Hn Zn Un An Bn In Jn Sn Kn	Rectangular Pocket
X	XCenter
Y	YCenter
M	Length
W	Width
H	StartHgt
Z	ZDepth
U	CornerRad
A	Stepover
B	DepthCut
I	RampFeed
J	RoughFeed
S	FinStock
K	FinFeed

(Continued...)

Table 9-2, G-Code Equivalents (Continued)

G-Code Format	Conversational Equivalent
G79 Xn Yn Cn An Bn Hn Dn	Bolt Hole Drill
X	XCenter
Y	YCenter
C	IndexAngle
A	StartAngle
B	EndAngle
H	#Holes
D	Diameter
G80	Drilling Off
G81 Zn Rn Fn Pn	Basic Drill
Z	ZDepth
R	StartHgt
F	Feed
P	ReturnHgt
G83 Zn Rn Fn In Pn	Peck Drill
Z	ZDepth
R	StartHgt
F	Feed
I	Peck
P	ReturnHgt
G85 Zn Rn Fn Pn	Boring
Z	ZDepth
R	StartHgt
F	Feed
P	ReturnHgt
G87 Zn Rn Fn In Jn Kn Wn Un Pn	Chip Break
Z	ZDepth
R	StartHgt
F	Feed
I	FirstPeck
J	PeckDecr
K	MinPeck
W	ChipBrkInc
U	RetractDep
P	ReturnHgt
G90	Dim Abs
G91	Dim Incr

Table 9-2, G-Code Equivalents (Continued)

G-Code Format	Conversational Equivalent
G170 Hn Zn Dn En Xn Yn An Bn Fn Pn	Facing Cycle
H	StartHgt
Z	ZDepth
D	XStart
E	YStart
X	Length
Y	Width
A	XStepover
B	YStepover
F	Feed
G171 Xn Yn Hn Dn Zn An Rn Bn Sn In Jn Kn Pn	Circular Profile Cycle
X	XCenter
Y	YCenter
H	StartHgt
D	Diameter
Z	ZDepth
A	Side (In/Out)
R	Ramp
B	DepthCut
S	FinStock
I	ZFeed
J	RoughFeed
K	FinFeed

(Continued...)

Table 9-2, G-Code Equivalents (Continued)

G-Code Format	Conversational Equivalent
G172 Xn Yn Hn Mn Wn Zn An Rn Un Bn Sn In Jn Kn	Rectangular Profile Cycle
X	XCenter
Y	YCenter
H	StartHgt
M	Length
W	Width
Z	ZDepth
A	Side (In/Out)
R	Ramp
U	CornerRad
B	DepthCut
S	FinStock
I	ZFeed
J	RoughFeed
K	FinFeed

Checking Disks for Lost Data

Sometimes, computer disks contain lost program fragments. This might happen if a computer loses power while saving a program. Lost program fragments do not display in the **USER** listing, but continue to take up disk space.

To check a disk for lost program fragments:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **Check Disk**, and press **ENTER**. The CNC prompts you to select the required drive.
4. Highlight the drive, and press **ENTER**. The CNC checks the disk. If it finds lost clusters, it prompts for recovery instructions and the soft keys change for operator response.
5. Press **Yes (F1)**. The CNC attempts to recover lost clusters on the disk. Press **No (F2)** to cancel.

Displaying System Information

The System Information screen displays specific details about the CNC and its software. The information comes in handy during machine setup or troubleshooting. Refer to **Figure 9-2**.

System Information
<p>Product: 3000M Machine Serial ID: 0000000000 PC software version: 1.02G DSP firmware version: 0.33A DOS version: 6.22 Processor detected: 486 Total memory (RAM): 2,048 KB Free memory (RAM): 2,048 KB (100%) Free base memory (RAM): 288 KB Total space drive C: 1,547 MB Free space drive C: 1,488 (96.19%)</p>
<p>Press F10 to continue.</p>

SYSINFO

Figure 9-2, System Information Screen

To display the System Information screen:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **System Info**, and press **ENTER**. The CNC displays the System Information screen.

SHORTCUT: On an offline system or via PC keyboard, press **ALT + I** to display the System Information screen.

Copying Programs from/to Unspecified Locations

To copy programs from floppy disk drives to the **USER** listing:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **Copy ?**, and press **ENTER**. The CNC prompts for program source and activates the ASCII soft keys.
4. Use the ASCII Chart to enter the complete location (path) of the required program, and press **ENTER**. The pop-up menu prompts for the destination drive or **Other . . .**
5. Highlight **Other**, and press **ENTER**. The CNC prompts for the destination and activates the ASCII soft keys.
6. Use the ASCII Chart to enter **C:\USER\[program name].M**, and press **ENTER**. The CNC copies the program to the **USER** listing.

<p>NOTE: As an alternative, log on to the floppy disk drive containing the program, mark the program, and use the Copy to: utility.</p>

Renaming Programs from/to Unspecified Locations

To rename a program located on any drive:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **Rename ?**, and press **ENTER**. The CNC prompts for the location of the source program and activates the ASCII soft keys.
4. Use the ASCII Chart to enter the location (complete path) of the required program, and press **ENTER**. The CNC prompts for the program destination and activates the ASCII soft keys.
5. Use the ASCII Chart to enter the location and new name (complete path) of the program, and press **ENTER**. The CNC renames the program.

<p>TIP: Use the Rename ? utility to move a program to a different drive. Log on to the drive containing the program and activate the Rename ? utility. Enter the desired drive as part of the new program name. The CNC saves the program to the specified drive.</p>
--

Printing from Floppy Drives

The CNC can print to any standard IBM PC compatible printer.

To print programs from floppy drives:

1. From the Program Directory, press **Utility (F9)**. The CNC displays the Utility pop-up menu.
2. Highlight **More . . .**, and press **ENTER**. The CNC displays the second page of the pop-up menu.
3. Highlight **Print ?**, and press **ENTER**. The CNC prompts for the source program on the message line and activates the ASCII soft keys.
4. Use the ASCII Chart to enter the location (complete path) of the required program, and press **ENTER**. The CNC prompts you to confirm the instruction. The soft keys change for operator response.
5. Press **Yes (F1)**. The CNC sends the programs to the printer.

– or –

Press **No (F2)** to cancel the print command.

NOTE: As an alternative, log on to the desired drive, mark the desired program, and use the Print utility.
--

Section 10 - Tool Management

Tool Page

The Tool Page contains the tool-length offsets and tool diameter values for each tool. When a tool activates, the CNC automatically activates its length offset and diameter compensation values.

The active diameter value affects compensated moves and cycles with built-in tool compensation.

Enter tool-length offset and tool diameter values on the numbered rows of the Tool Page. The numbered lines on the Tool Page identify the tool number (**Tool #**) that activates those values.

Program a **Tool #** command as a separate block or with another block.

The Tool Page can hold information for a maximum of 99 tools.

The Tool Page stores spindle speed, spindle direction, and coolant information for machines with these functions.

On machines with collet-type tool holders, it is impractical to store tool-length offsets on the Tool Page. Set the tool-length offsets at tool change. Compensated moves and cycles with built-in compensation still require a tool diameter value in the Tool Page.

Entering the Tool Page

To enter the Tool Page from the Manual Mode:

1. In Manual Mode, press **TOOL (F9)**. The CNC activates the Tool Page.

To enter the Tool Page from the Edit Mode:

1. In Edit Mode, press **Tool (F6)**. The CNC activates the Tool Page.

Tool Page Description

To edit tool information, highlight the required row (**Tool #**) and enter the appropriate values. The CNC displays the highlighted row at the bottom of the screen. The cursor indicates where entered information will appear on the screen. Refer to **Figure 10-1**.

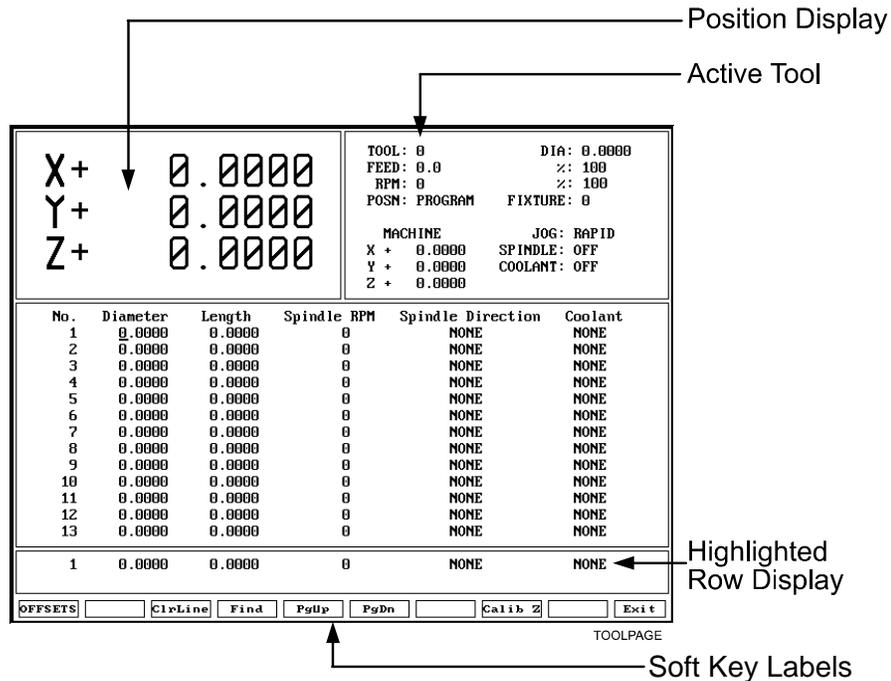


Figure 10-1, Tool Page

Row numbers correspond to tool numbers. When the CNC executes a program block that activates a tool number, the CNC applies the values on the corresponding row of the Tool Page.

The Tool# 0 command cancels active diameter and length offsets. Set the Tool# 0, Z0 position at the fully retracted position of the quill.

All Jog features operate in the Tool Page.

No. Row numbers link the values on a row of the Tool Page to a tool number. A program block that activates a tool number activates the values and settings on that row of the Tool Page.

Diameter Indicates the appropriate tool diameter.

Length Z-axis length offsets. Length offsets adjust the Z-axis position for the length of the active tool.

Spindle RPM The required spindle RPM on machines set up for spindle RPM control. On machines not set up for spindle functions, the feature is used only with the tapping cycle.

Spindle Direction	Option. On machines equipped for spindle direction control (machines built with M-functions), these settings control spindle direction.
Coolant	On machines equipped for coolant control (machines built with M-functions), these settings control the coolant pumps.

Using the Tool Page

Use the up and down ARROWS to highlight and select tool numbers (row numbers). Use the left and right ARROWS to move from column to column. The Tool Page automatically converts values to the active inch/mm equivalents.

NOTE: The Tool Page is the only place the CNC converts values from Inch to MM Mode, and vice versa.

Press **PgUp (F5)** or **PgDn (F6)** to scroll up and down the Tool Page.

Finding Tools by Number

To locate tools by number:

1. From the Tool Page, press **Find (F4)**. The CNC displays the prompt: **Enter Tool Number: _**.
2. Type a tool number, and press **ENTER** to display the appropriate tool information.

Changing Tool Page Values

CAUTION: Set the CNC for the required Inch/MM Mode. The CNC displays the active Inch/MM Mode in the top left corner of the Tool Page.

To change Tool Page values:

1. Highlight the desired row. Position the cursor in the desired column.
2. Type the new value.
3. Press **ENTER** or any **ARROW** to accept the value.

Clearing a Tool (Whole Row)

To clear a Tool Page row:

1. Go to the Tool Page and highlight the required row.
2. Press **ClrLine (F3)** to reset all values in the highlighted row to zero.

Clearing a Single Value

To clear a single value:

1. Go to the Tool Page and highlight the required row.
2. Position the cursor at the required column.
3. Press **CLEAR** to change the value to zero.

Adjusting a Single Value

To adjust a single value:

1. Go to the Tool Page and highlight the required row.
2. Position the cursor at the required column.
3. Press the **ABS/INC** key, or **ALT + E** on a keyboard, to display the message, "Enter adjustment value."
4. Type the amount of the adjustment. The adjustment value may be positive or negative.
5. Press **ENTER** to adjust the value, and display the adjusted value in the table.

Setting Tool-Length Offset

Setting tool-length offsets actually produces a Z-axis Part Zero position for each tool used in a program. All tool-length offsets are measured from Machine Zero, a permanent, physical reference on the machine. Determine the exact location of Tool #0, Z0 before you enter any tool-length offsets. Refer to "Section 3 - Manual Operation and Machine Setup" for instructions on setting the location of Tool# 0, Z0.

NOTE: For machines without homing, it may be necessary to set machine home to make setting tool-length offsets easier. Either manually or by jogging take the Z-axis close to the top of travel. The servo must be turned on. Press **MDI (F7)**, press **Mill (F5)**, press **More (F7)**, curse down to **Home** and press **ENTER**, press **Z** the z will light up, press **Save**, press **Prev**, press **Exit**, and press **Start**. The Z-axis will change to zero.

NOTE: All Jog features operate in the Tool Page.

Automatically Setting Tool-Length Offsets from the Tool Page

To automatically set tool-length offsets from the Tool Page:

1. Activate the Tool Page and highlight a tool number.
2. Load the tool and manually position its tip at the part Z0 position.
3. Press **Calib Z (F8)** to insert Z position in the length offset column.

Manually Setting Tool-Length Offsets from the Tool Page

To manually set tool-length offsets from the Tool Page:

1. Retract the Z-axis to the Tool# 0, Z0 position.
2. Load the tool and manually position its tip at the part Z0 position.
3. Manually type the $\pm Z$ position, as it is displayed in the position display, and press **ENTER**. The CNC enters the Z position in the offset column.

NOTE: Most tool-length offsets are negative numbers.

Setting Tool-Length Offset for Ball-End Mills

When using a ball-end mill to cut contoured surfaces, use tool diameter and length offsets. Set Z0 half the tool's diameter from the tool tip.

To set the offset for a ball-end mill:

1. Use the procedure outlined for a standard end mill to set the tool-length offset.
2. Add half the tool's diameter to the value listed in the **Length** column.
3. Highlight the **Length** column and enter the calculated value.

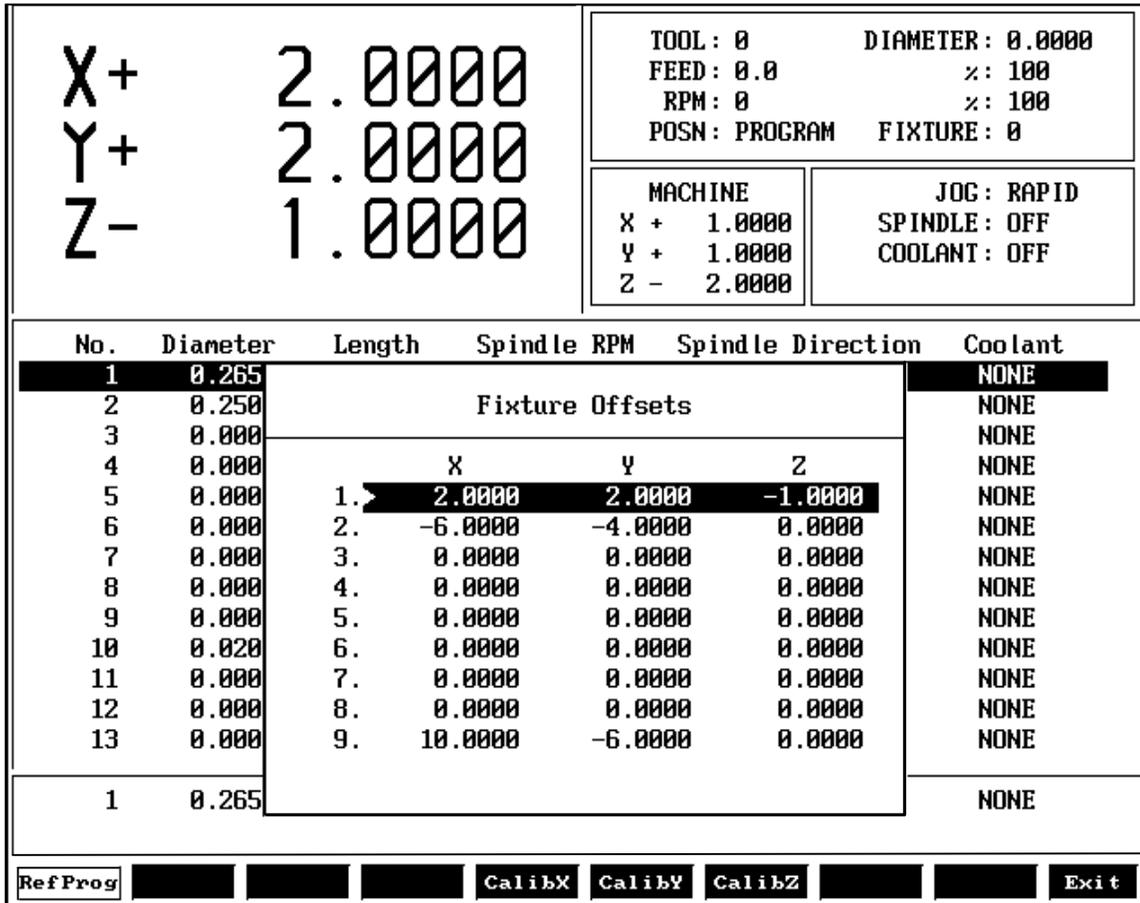
Fixture Offsets

Press **Offsets (F1)** to activate the Fixture Offsets Table. You can define a maximum of nine fixture offsets in the table.

Refer to "Section 3 - Manual Operation and Machine Setup" for a description and definition of fixture offsets. Refer to "Section 4 - Writing Programs" for information on how to program fixture offsets and enter values into the Fixture Offsets Table.

Setting RefProg Offset

Activate the **RefProg** key by pressing **Tool (F9)**, **Offsets (F1)**, then **RefProg (F1)**. Refer to **Figure 10-2**.



3000MREPROG

Figure 10-2, RefProg “Fixture Offsets” Pop-up

When **RefProg** is highlighted, use the calibration keys: **CalibX (F5)**, **CalibY (F6)**, or **CalibZ (F7)** to calibrate an axis work coordinate offset, the calibration value is in reference to the current active offset versus the machine home. With soft key **RefProg** not highlighted, the value placed in the axis register offset table is the value you see displayed under the label “Machine” that is in reference to the machine home. Refer to **Figure 10-2** for the “Machine” label in the middle of the Tool Page.

Three-axis example:

1. Set up all your tools with their offsets coming from machine zero in the Z-axis to the top of the part.
2. To machine the same parts on the reverse side (or an edge); flip the part over onto another vise. At this point all of your tool length offsets have changed, so you have two choices:
 - Reset all the tools
 - Shift the Z in a work offset

If you shift the Z-axis in the work offset, you do not need to reset all your tools. To shift the Z-axis in the work offset:

1. Call up any tool and jog it to the top of the now flipped over part.
2. Go to the work coordinate table, press **RefProg** key and then **CaibZ** to set the Z shift register for the difference between the two setups:
 - When you want to do machining on the first set up, you call up one work offset and use all the tools as normal.
 - Then when you want to machine on the other side and the other fixture, you call up the other work offset and all the tools will now be in reference to the top of the part without having to change every tool offset.

Section 11 - Communication and DNC

Communication

The CNC can exchange data with any other RS-232 compatible devices. The baud rate, parity, data bits, stop bits, and software parameters of the CNC and the other machine or computer must agree.

The machine builder sets up the default communication parameters at installation. The operator can temporarily adjust these parameters from the Communication screen. Every time the CNC starts, it reloads the default settings.

CAUTION: Only qualified personnel should change the settings in the CNC Setup Utility. Incorrect settings could disable the machine.

NOTE: The CNC loads the updated setup file when you change setup parameters or when you turn on the CNC.

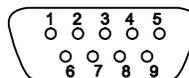
Installing the RS-232 Cable

RS-232 communication requires a cable connection between the sending and receiving machines or computers. Connect one end of the cable to the RS-232, 9-pin connector on the back of the console. Connect the other end to the other machine or computer.

NOTE: The machine builder determines the exact location of the RS-232 connector.

Use a cable designed for RS-232 communication. An RS-232 cable has internally crossed wires between Pins 2 and 3. Data sent from Pin 3 (transmit) of one machine must go to Pin 2 (receive) of the other. Refer to **Figure 11-1**.

RS-232 Connector
(Male 9-Pin)



Pin 2 - Recieve Data
Pin 3 - Transmit Data

DB-9

Figure 11-1, RS-232 Communication Connector

NOTE: The same requirements apply for Pins 2 and 3 when one of the connectors is a 25-pin serial connector.

Do not use computer extension cords for RS-232 communication (Pins 2 and 3 are not crossed).

Setting Communication Parameters

This manual does not describe the merits of the different parameter settings. Refer to an appropriate computer communication reference for this information.

To change communication parameter settings:

1. From the Communication screen, press **Param (F9)** to display the Parameter Settings pop-up menu. Refer to **Figure 11-3**.

Parameter Settings	
1. ▶Port.....	COM1
2. Baud rate..	9600
3. Parity.....	EVEN
4. Data bits..	7
5. Stop bits..	1
6. Software...	On
7. Data Type..	ASCII

Figure 11-3, Parameter Settings Pop-Up Menu

NOTE: Both the sending and receiving devices must have the same baud, parity, data bits, stop bits, and software parameter settings.

2. Highlight the setting to be changed, and press **ENTER**.
3. When you select the **Port**, **Baud**, or **Parity** parameter, the CNC activates an additional pop-up menu. In the pop-up menu, select the appropriate setting, and press **ENTER** to store the setting.
4. For the **Data Bits**, **Stop Bits**, **Software**, or **Data Type** parameter, press **ENTER** to cycle through the available selections.

Selecting the Communication Port

Most CNC installations have at least one RS-232 connector, which is usually labeled and mounted on the back of the console. The CNC sends signals, using its **COM1** port, to the RS-232 connector. This configuration cannot be changed. Personal computers that run the offline software can use **COM2** if they have the necessary hardware.

Setting the Baud Rate

The CNC supports the following baud rates: 110, 150, 300, 600, 1200, 2400, 4800, 9600, or 19200. [Default: **9600**].

Setting Parity

The CNC supports the following parity settings: Odd, Even, or None. [Default: **Even**].

Setting Data Bits

The CNC supports the following data-bit settings: 5-bit, 6-bit, 7-bit, or 8-bit. [Default: **7-bit**].

Setting Stop Bits

The CNC supports the following stop bit settings: 0-bit, 1-bit, or 2-bit. [Default: **1-bit**].

Software Settings

The CNC supports the following protocol settings: ON or OFF. Software protocol is frequently referred to as **Xon** or **Xoff**, or in commercial communication packages, **handshaking**. [Default: **ON**].

Setting Data Type

The CNC supports the following data display types: ASCII or BINARY. This setting does not affect the data exchanged, only how the CNC displays it on the screen during transmission. [Default: **ASCII**].

Testing the Data Link

The CNC has a utility to test the data link. With the CNC connected to another machine, set the parameters on both machines. If the other machine has a manual test screen, activate it.

Activating the Test Link Screen

With the Communication screen active, press **TestLnk (F7)** to display the Test Link screen. Refer to **Figure 11-4**.

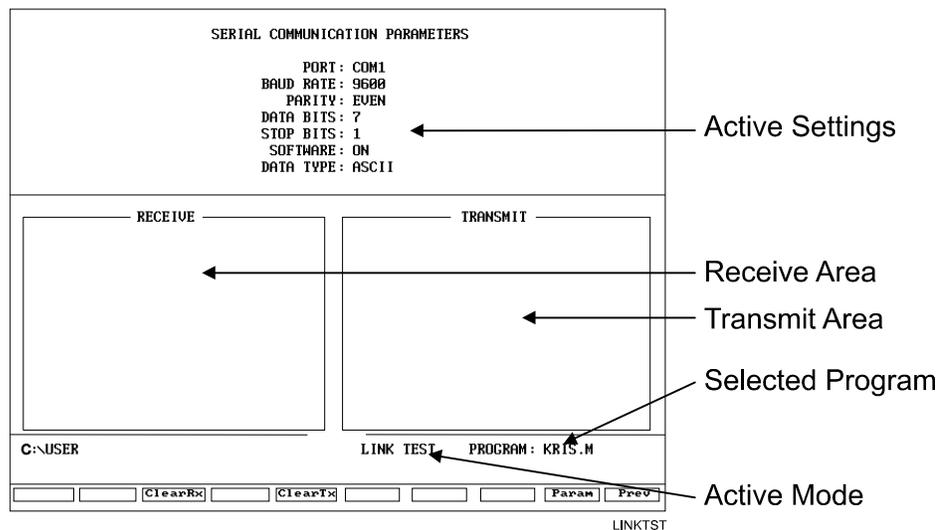


Figure 11-4, Test Link Screen

Setting Test Link Display Modes

To test the link, visually verify that the test data sent matches the test data received. The data type setting determines how the CNC displays characters on the screen.

If the data type is set to ASCII, the CNC displays letters and numbers. If the data type is set to binary, the CNC displays hexadecimal equivalents. Set both machines to use the same data type display.

NOTE: The CNC displays Hexadecimal characters as pairs of numbers or numbers and letters.

To change the Test Link screen data display:

From the Test Link screen, press **Data (F1)** to switch between ASCII and binary display formats. The CNC displays the current mode in the Active Settings area of the Test Link screen.

Testing the Link

To test the Link:

1. Set up an RS-232 connection with another machine (or computer).
2. Set the other machine to receive.
3. With the Test Link screen active, use the CNC keypad to enter any series of numbers. The CNC displays these characters in the transmit area.
4. Verify that the other machine (or computer) correctly received the entire transmission.
5. Set the other machine (or computer) to transmit.
6. Manually transmit a series of characters from the other machine (or computer).
7. Verify that the CNC correctly received the entire transmission.

Clearing the Receive Area

Press **ClearRx** to clear the receive area.

Sending a Program

Before you activate the Communication screen, highlight the appropriate program in the Program Directory. The CNC displays the active program name next to the **PROGRAM** label on the screen. After the CNC sends the selected program, it prompts you to select another program to transmit during the sending process.

To send a program:

1. With the Communication screen active, the parameters set, and the link tested, press **Send (F3)**. The CNC prompts you to send the selected program or to change the program being sent.
2. Press **Select (F1)** to send the program.

– or –

Press **Change (F3)** to display the ASCII soft keys and enter the name of the desired program.

Receiving a Program

Before transmission, enter a name for the received program.

To receive a program:

1. With the Communication screen active, the parameters set, and the link tested, press **Receive (F1)**. The CNC prompts you to enter a name for the incoming program and displays the ASCII soft keys.
2. Type the desired name, and press **ENTER**. The CNC displays the message **Ready to Receive...**
3. Begin transmission from the sending machine.

Setting the Transmission and Receiving Display

When transmitting or receiving with Text Mode on, the CNC displays the transmitted program on the screen. When Text Mode is off, the display area remains blank. When Text Mode is on, **Text (F4)** highlights.

When data is exchanged, the CNC displays the **Transfer in Progress** symbol just above the soft key line.

When transmitting or receiving, press **Text (F4)** to toggle in and out of Text Mode. Transmission or receiving is faster with Text Mode Off.

Holding Transmission/Receiving Operations

Press **Hold (F1)** to pause the program being sent or received. Press **Resume (F2)** to continue the exchange.

<p>CAUTION: Set the software parameter (Xon/Xoff) ON for both machines. If this parameter is OFF, a hold command could overload the CNC's buffer, resulting in data loss.</p>
--

Running in DNC

Under Direct Numeric Control (DNC), the CNC runs the program received over the RS-232 link. The CNC can run incoming programs in Single Step Mode or Automatic Mode. Real-Time Draw Mode is available under DNC.

CAUTION: The CNC runs programs written only in its conversational language format. Do not attempt to run other types of programs.

In the DNC Mode, the CNC cannot look ahead to verify completion of a subprogram. Therefore, do not run programs in the DNC Mode that contain subprogram calls.

The DNC screen is similar to other operating screens, but contains communication information. Refer to **Figure 11-5**.

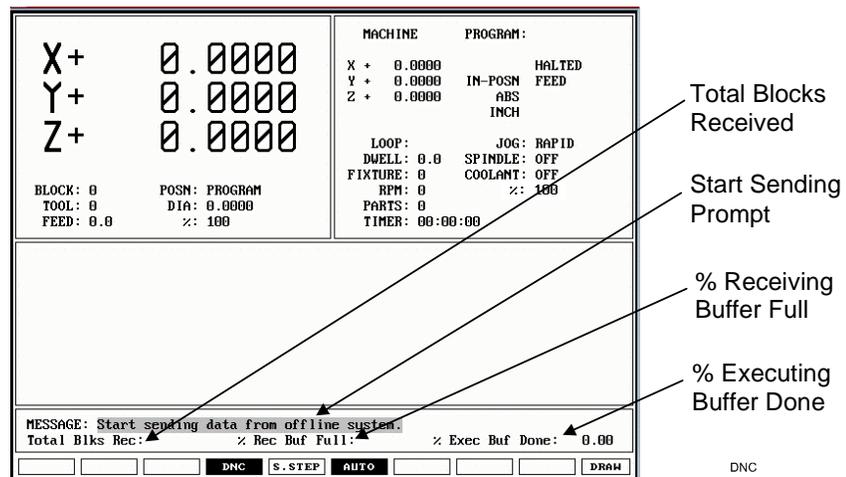


Figure 11-5, DNC Screen

- Total Blks Rec:** The total number of program blocks the CNC has received.
- % Rec Buf Full:** The percentage of receiving buffer used.
- % Exec Buf Done:** The percentage of executing buffer used.

Because an RS-232 link transmits program blocks faster than the receiving machine can run them the CNC must manage the memory and timing between the two machines.

The CNC uses two buffers: One to store incoming program blocks and one to store blocks being run. When the CNC has run all of the blocks in the execution buffer, it moves the blocks in the receiving buffer to the executing buffer. The receiving buffer continues to store the blocks being sent over the RS-232 cable.

If necessary, to avoid data loss when using the software protocol (Xon/Xoff), the CNC can signal the sending machine to pause.

NOTE: Most machines default to the Buffer Mode for DNC operations. Some machines can be set to default to the Drip Feed Mode for DNC. In Drip Feed Mode, the transmitting machine sends one block at a time and the receiving machine executes each block as it receives it, without buffering the data.

To put the CNC in Direct Numeric Control Mode:

1. With the Communication screen active, parameters set, and the link tested, press **DNC (F4)** to highlight the DNC soft key.
2. Press **Receive (F1)** to activate the DNC screen. The CNC prompts you to start transferring from the other machine.
3. Press **START** to run the incoming program.

Using Data Control (DC) Codes

Data Control (DC) codes are sometimes required to automate the operation of a paper tape reader or punch. Refer to **Table 11-1** for the available Data Control Codes.

Table 11-1, Data Control Codes

ASCII	Function	Hex Code	CNC Key
DC1	Reader Start	0x11	1
DC2	Punch Start	0x12	2
DC3	Reader Stop	0x13	3
DC4	Punch Stop	0x14	4

The **ASCII** column lists the codes required to perform the corresponding Function. The **Hex Code** column lists the hexadecimal equivalent of the ASCII code. The **CNC Key** column lists the key you press on the CNC keypad or a PC keyboard to transmit the required DC code.

A reader or punch will turn ON (start) or OFF (stop) in response to these codes. To test reader or punch communication, activate the Communication screen and press any of these keys. The paper tape reader or punch should respond appropriately.

Using DC Codes In Receive Mode

Usually a receive operation involves the paper tape reader. You must start the reader, thereby initiating the reading of the paper tape. Do this after the CNC has been set up to receive the file.

1. Set up to the CNC to receive a program.
2. Press **1** to send the DC1 code to the reader. The reader starts.
3. After the CNC has read the tape, if necessary, press **3** to stop the reader. The CNC sends the DC3 code to the reader and the reader stops.

Using DC Codes In Send Mode

Usually, a send operation involves the paper tape punch. Set up and start the punch before you start the send operation.

1. Select the program that will be sent to the tape punch.
2. Press **2** to send the **DC2** code to the punch. The punch starts.
3. Press **Send (F3)** to start sending the program. If necessary, change the name of the program.
4. When the program has been completely transferred to the punched tape, if necessary, press **4** to stop the punch. The CNC sends the **DC4** code to the punch and the punch stops.

Section 12 - Calculators

CNC Calculator Package

The CNC features a powerful calculator package that contains three separate calculators:

- Mathematics Calculator
- Right Triangle Calculator
- Geometry Calculator

You can recall calculator solutions directly into the labeled fields of a Graphic Menu. All three calculators have separate memory space for storing and recalling solutions.

Math Calculator

The math calculator performs all of the operations commonly offered in a scientific calculator. These include basic math, trigonometry, unit conversion, logs, exponential operations, angle/radian conversions, and finding the inverse.

To activate the Math Calculator:

1. Press the **CALC** programming hot key on the keypad.

– or –

In Edit or MDI Mode, press **Calc (F7)** to display the **Select Type of Calculator:** pop-up menu. Refer to **Figure 12-1**.

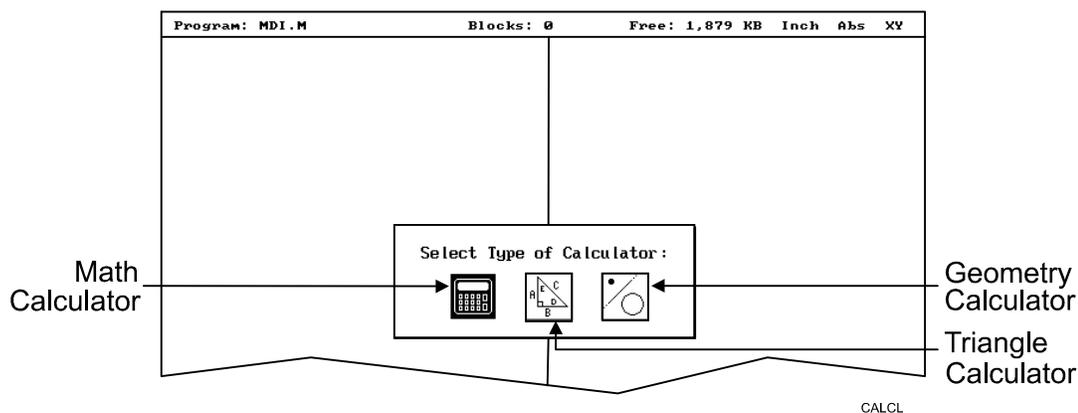


Figure 12-1, Calculator Selection Menu

- Highlight the **Math Calculator** template, and press **ENTER** to display the math calculator pop-up window. Refer to **Figure 12-2**.

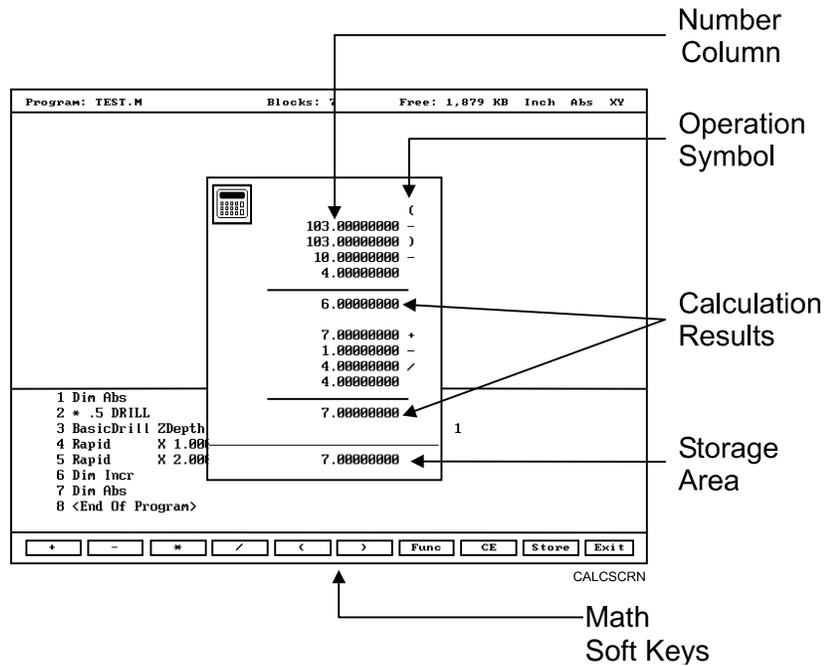


Figure 12-2, Math Calculator and Soft Keys

Math Calculator Basics

The CNC displays numbers in the storage area, as entered. Select the math operation using the appropriate soft key. Refer to **Table 12-1, Calculator Selection Menu**.

Type the first number of the calculation, and press an operation soft key. The CNC displays the number in the column. Next, select the desired operation to perform with the next number. Type the second number, and so forth.

After typing the last number of a calculation press **ENTER** to place the final number in the column and calculate the answer. The CNC displays the result in the storage area. Press **Store (F9)** to copy the result into the calculator's memory.

Press **CE (F8)** to clear previous calculations.

If you enter illogical combinations of operations and numbers, the CNC displays an error message or a row of asterisks.

Table 12-1, Math Operation Soft Keys

Operation	Soft Key Label	Soft Key Number
Addition	+	(F1)
Subtraction	-	(F2)
Multiplication	*	(F3)
Division	/	(F4)
Left Hand Parenthesis	((F5)
Right Hand Parenthesis)	(F6)
Misc. Function Pop-up Menu	Func	(F7)
Clear	CE	(F8)
Store Number For Recall	Store	(F9)

NOTE: Off-line keyboard users can use the keypad operators.

Operations Involving Two Numbers

To add, subtract, multiply, or divide two numbers:

1. With the math calculator active, type the first number, followed by the appropriate function hot key: **+** (F1), **-** (F2), ***** (F3), or **/** (F4). The CNC adds the number and operation symbol to the column.
2. Type the second number, and press **ENTER** to add the second number to the column and display the sum in the storage area.

Math with a Column of Numbers

The CNC handles a column of numbers requiring multiple operations, such as a continuous equation, in standard mathematical order (multiplication, division, addition, and subtraction). Press **ENTER** to generate the result.

Using Parentheses

Use parentheses to indicate arithmetical operations in a non-standard order. (Example: finding a sum before using it as a multiplication factor.) The CNC performs operations within parentheses first.

When the CNC encounters parentheses, it solves the operation within the parentheses immediately, and substitutes that value for the parenthetical expression in the column.

Use parentheses in pairs. The parenthetical expression must contain a left parenthesis, **(**, at the beginning of the expression, and a right parenthesis, **)**, at the end of the expression. Otherwise, the CNC will not calculate the result.

The CNC performs operations within parentheses top to bottom, as they appear in the column, with innermost expressions solved first.

For example, the following expression:

$$(7 + 4 + ((6 * 9) - 1)) / 8$$

generates a result of 8.

Order of calculation:

$$6 * 9 = 54$$

$$54 - 1 = 53$$

$$53 + 7 = 60$$

$$60 + 4 = 64$$

$$64 / 8 = 8$$

Using Additional Functions

The **Func (F7)** key activates a pop-up menu that provides access to additional math functions. These functions perform their listed operations on a single number. Refer to **Table 12-2**.

Table 12-2, Function Selection Pop-up Listing

Pop-up Menu Label	Function
Sine	Sine Function
Cosine	Cosine Function
Tangent	Tangent Function
ASine	Arcsine Function
ACosine	Arccosine Function
ATangent	Arctangent Function
SQRT	Square Root Function
SQR	Squaring Function
LN	Natural Log Function
Log	Log Function Base 10
Exp	Exponential Function
ToMetric	Inch to Metric Conversion
ToInch	Metric to Inch Conversion
ToDegs	Radian to Degree Conversion
ToRads	Degree to Radian Conversion
Inverse	Inverse Function

To use an additional function:

1. With the math calculator active, type the number and press **Func (F7)** to display the Function pop-up menu to the right of the calculator.
2. Highlight a function, and press **ENTER** to display the Result in the storage area.

Storing Numbers from the Math Calculator

Press **Store (F9)** to copy the number from the storage area to the calculator's memory. The math calculator's memory stores 64 numbers. You can recall numbers stored in memory directly into a program.

Right Triangle Calculator

The Right Triangle Calculator solves the angles and sides of a right triangle given: any two sides, any two angles, or an angle and a side. Store any or all of the values in the calculator's memory.

Activating the Triangle Calculator

To activate the Right Triangle Calculator:

1. Press the **CALC** programming hot key on the keypad.

– or –

With the CNC in Edit or MDI Mode, press **Calc (F7)** to display the Calculator selection menu.

2. Highlight the **Triangle Calculator** template, and press **ENTER** to activate the Triangle Calculator. Refer to **Figure 12-3**.

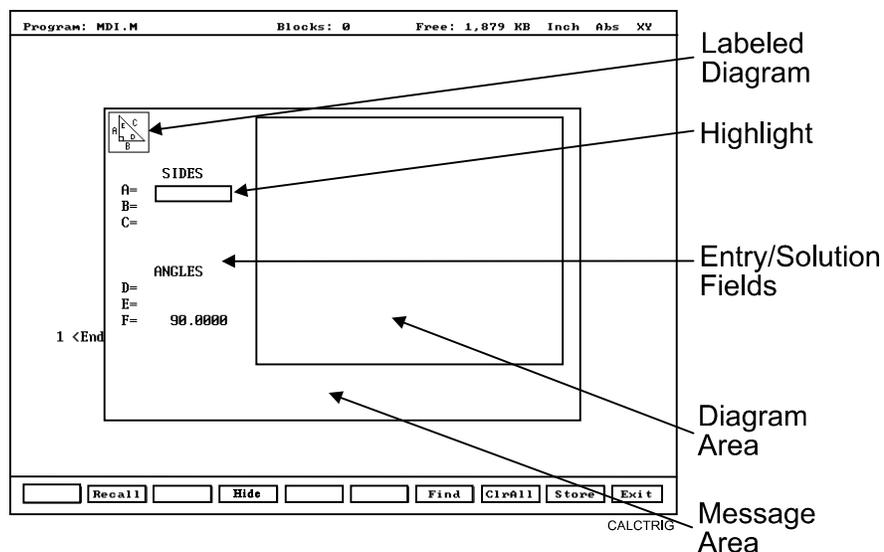


Figure 12-3, Right Triangle Calculator Screen

Using the Triangle Calculator

The Right Triangle Calculator solves only right triangle problems. The Right Triangle Calculator's pop-up screen contains three main areas: the entry/solution area, the diagram area, and the message area.

The labeled entry fields in the entry/solution area correspond to the sides and angles shown in the diagram.

To use the Right Triangle Calculator, enter known element values: any two angles, any two sides, or one angle and one side.

After you enter two known elements, the CNC calculates and displays values for the remaining sides and/or angles. The CNC places an asterisk after solved element values and displays a scaled drawing of the solved triangle in the diagram area.

To clear a single incorrect value, highlight the value and press **CLEAR**. To clear all displayed values, press **CirAll (F8)**.

An illogical entry generates an error message.

Recall Right Triangle Calculator solutions directly into a program (for positive (+) incremental moves only). To adjust these solutions to produce absolute coordinates, recall the values into the Math Calculator and add an offset value.

To solve for the sides and angles of a right triangle:

1. With the Right Triangle Calculator active, highlight the field for the first side or angle and enter the value.
2. Highlight a second side or angle and type the value.
3. Press **Find (F7)** to display a scaled drawing of the triangle is displayed in the solution box. The CNC displays calculated sides and angles in unused fields.

Storing Right Triangle Calculator Results

The CNC will store any side or angle value. Maximum: 64 values.

To store a value:

1. Solve the required triangle.
2. Highlight the value to be stored.
3. Press **ENTER** to store the selected value in memory.

Hiding the Right Triangle Calculator Screen

To temporarily hide the Right Triangle Calculator Screen:

1. Press **Hide (F4)**. The CNC temporarily hides the Right Triangle Calculator Screen.
2. Press **Find (F7)** to display the screen again.

Geometry Calculator

The CNC uses Cartesian coordinates (X, Y, Z-axis values) to define most positions. However, you must sometimes determine position coordinates based on the known construction of other elements on the print, including lines, circles, and angles.

The Geometry Calculator provides an assortment of line, circle, angle, and point templates. Use these templates to sketch a geometry construction that identifies the unknown position. The Geometry Calculator inserts a point at the required position. The calculator automatically solves the coordinates of all points. Recall stored coordinates as necessary in a program.

Activating the Geometry Calculator

To activate the Geometry Calculator:

1. Press the **CALC** programming hot key on the keypad.
– or –

In Edit or MDI Mode, press **Calc (F7)** to display the Calculator selection menu.

2. Highlight **Geometry Calculator** template, and press **ENTER** to display the Geometry Calculator.

Geometry Calculator Screen

The Geometry Calculator consists of a pop-up screen with a display area, a message area, and 18 geometry templates. The CNC displays the geometry construction in the display area. Refer to **Figure 12-4**.

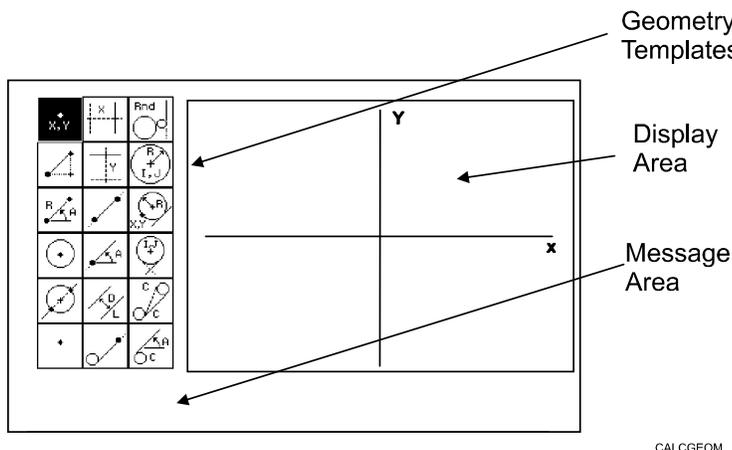


Figure 12-4, Geometry Calculator

Use the **DISPLAY (F5)** soft key selections to alter the view of the display area. The **DISPLAY (F5)** options work the same as they do in the Draw Mode display.

The calculator prompts for required values and selections. Enter all the appropriate values and selections for the prompts.

Using the Geometry Calculator

Use the **ARROWS** to select a template. Press **ENTER** to activate the selected tool.

Points, lines, and circles are the basic elements of all sketches. Use the 18 geometry templates to define these elements. Each geometry tool defines an element differently. More than one tool will probably be needed to define the required geometry construction. Templates that require pre-existing points, prompt you to activate one of the point identification templates.

Display-area grid lines appear as solid lines; constructed lines and circles appear as dotted lines; and points are marked with an **x** (lowercase x).

The CNC assigns a number to each element in a sketch. Sketches can contain a maximum of 50 elements.

All elements in the sketch also appear on the Geometry List. The element numbers on the Geometry List correspond to the element numbers in the sketch.

The Geometry List contains the following details about each element:

Circles Absolute position of center and radius.

Lines Orientation (angle) and absolute position at which it crosses X- and/or Y-axis.

Points Absolute positions.

The calculator automatically numbers each element and stores the following details in memory:

Points Absolute position.

Circles Absolute position of center.

NOTE: The Geometry Calculator does not automatically save the coordinates of an intersection between two elements. Insert a point at the intersection to save its coordinates.

When you delete an element from the sketch, the CNC deletes all stored information regarding that element.

There are three categories of Geometry templates:

Point templates Refer to **Table 12-3, Point Templates**.

Line templates Refer to **Table 12-4, Line Templates**.

Circle templates Refer to **Table 12-5, Circle Templates**.

Review the tables to see the requirements for each template. Many templates require feature(s) of an existing element as a reference. Experiment to become familiar with how each template operates.

NOTE: After a series of deletions and additions, the display may appear incomplete. Press **Display (F5)** and select **Redraw** to refresh the screen.

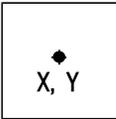
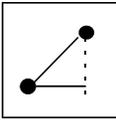
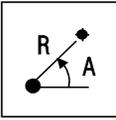
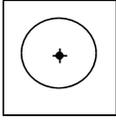
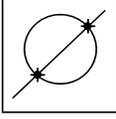
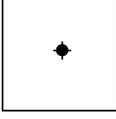
Point Templates

Some point templates insert points at operator-defined positions. Other point templates use other elements as references. Refer to **Table 12-3**.

Many line and circle templates display a **Select point definition . . .**, message when you activate them. This indicates that the selected tool requires you to select (or create) a reference point.

Templates that insert points at circle centers and element intersections prompt for the required element number(s). The CNC displays all possible intersections; select one.

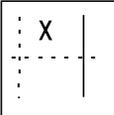
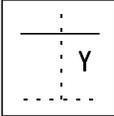
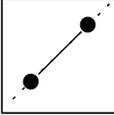
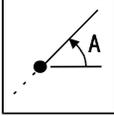
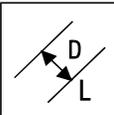
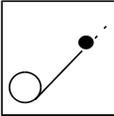
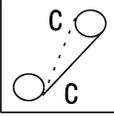
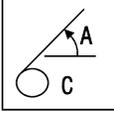
Table 12-3, Point Templates

Template	Purpose	Requirements
The following line templates are in the first column of the Calculator screen.		
	Defines a point in a sketch.	Must know the absolute X and Y position of the point.
	Defines a point at an X and Y increment from the existing point.	Must know the incremental X and Y distances from the existing point.
	Defines a point at the radius and angle from the existing point.	Must know the distance from the existing point. Must know the number of degrees and direction (+/-) from the 3 o'clock position.
	Defines a point at the center of an existing circle.	The circle already must be an element of the sketch.
	Defines the point of intersection between two existing elements. Prompts to select which intersection to use when more than one intersection exists.	Sketch must contain two intersecting (or tangential) elements.
	Identifies an existing point. Usually used when the construction of a new element requires a reference point.	Point already must be an element of the sketch.

Line Templates

Line templates use other elements or axis positions as references. Templates that draw lines tangent to circles, display all possible tangent lines and prompt you to select one. Refer to **Table 12-4**.

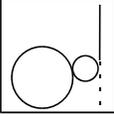
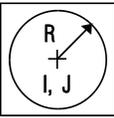
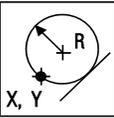
Table 12-4, Line Templates

Template	Purpose	Requirements
The following line templates are in the second column of the Calculator screen.		
	Constructs a line parallel to the Y-axis.	Must know the absolute X-axis position of the line.
	Constructs a line parallel to the X-axis.	Must know the absolute Y-axis position of the line.
	Constructs a line between any two points. Prompts you to select a point tool to define each endpoint.	Use any point to locate the two endpoints.
	Constructs a line through a point rotated a specified number of degrees from the 3 o'clock position. Prompts you to select any point tool to define the point of rotation.	Must know the number of degrees and direction (+/-) from the 3 o'clock position. Use any point to locate the point of rotation.
	Constructs a line parallel to the existing line at the specified (positive or negative) distance.	Existing line already must be an element of the sketch.
	Constructs a line through a selected point and tangent to an existing circle. Prompts you to select any point tool for point. Displays tangent lines on both sides of the circle. Select the required tangent.	Circle already must be an element of the sketch. Use any point to locate the point.
The following line templates are in the third column of the Calculator screen.		
	Constructs a line tangent to any two circles. Displays the four possible tangent lines. Select the required tangent.	Two circles already must be elements of the sketch.
	Constructs a line rotated a specified number of degrees from the 3 o'clock position and tangent to the existing circle.	Circle already must be an element of the sketch. Must know the number of degrees of rotation from the 3 o'clock position.

Circle Templates

Circle templates use other elements as a positioning reference. Templates that draw circles tangent to other circles, lines, or points display all possible tangent circles and prompt you to select one. Refer to **Table 12-5**.

Table 12-5, Circle Templates

Template	Purpose	Requirements
The following line templates are in the third column of the Calculator screen.		
	Constructs a circle of the specified radius tangent to any two existing elements. All possible tangential circles are displayed. Select the required tangent.	Sketch must contain at least two elements.
	Constructs a circle of the specified radius centered on a position. Prompts you to select a point tool to define the center point.	Use any point to locate the center. Must know the radius.
	Constructs a circle of the specified radius tangent to an existing line and through an existing point. Prompts you to select a point tool to define the point.	Line already must be an element of the sketch. Use any point to locate the point.
	Constructs a circle centered on a point and tangent to an existing line. Prompts you to select any point tool to define the center point.	Line already must be an element of the sketch. Use any point to locate the center.

Deleting Selected Elements

To delete an element from the sketch:

1. With the Geometry Calculator active, press **GEOMETR (F7)** to display a pop-up menu.
2. Highlight **Delete Item**, and press **ENTER**. The CNC prompts for the element number being deleted.
3. Type the element number, and press **ENTER** to delete the element.

Deleting All Elements

To delete all elements from the display area:

1. With the Geometry Calculator active, press **GEOMETR (F7)** to display a pop-up menu.
2. Highlight **Delete All**, and press **ENTER**. The CNC prompts to confirm the deletion.
3. Press **Yes (F1)** to delete all elements from the screen. Press **No (F2)** to cancel the deletion.

Listing All Geometry Elements

The CNC stores information on all points, circles, and lines created in the Geometry Calculator in the **Geometry List**. For a point, the CNC lists the X,Y coordinates; for a circle, the circle center and radius; for a line, the endpoint coordinates and sometimes the angle in reference to the 3 o'clock, 0 degree position.

To display the Geometry List:

1. With the Geometry Calculator open, press **GEOMETR (F7)** to display a pop-up menu.
2. Highlight **Geometry List**, and press **ENTER** to display the Geometry List.

Calculating the Distance between Two Elements

Use **Calc. Distance** to calculate the shortest distance between any two elements drawn with the Geometry Calculator.

NOTE: For circles, the CNC calculates the shortest distance from the circle center to the other element.

To calculate the distance between two elements in the Geometry Calculator:

1. With the Geometry Calculator open, press **GEOMETR (F7)** to display a pop-up menu.
2. Highlight **Calc. Distance**, and press **ENTER**. The CNC prompts, **Enter number of first element:**
3. Enter the first element number. The CNC prompts, **Enter number of second element:**
4. Enter the second element number to calculate and display the shortest distance between the two elements.

Last Position Recall

Whenever the calculator prompts for an X position, Y position, or circle, you can recall the last X position, Y position, or circle used by the calculator.

To recall a position:

1. When the CNC prompts for an X position, Y position, or circle number, press the up and down **ARROWS** to recall the last position, Y position, or circle number used.

Recalling Values into a Program

The Program Editor always displays **Recall (F2)** when a Graphic Menu activates. Recall calculator solutions stored in memory directly to the entry fields of a Graphic Menu. You can recall stored values only from the same calculator in which they were stored. Use the menu to select the type of recall. Refer to **Figure 12-5**.

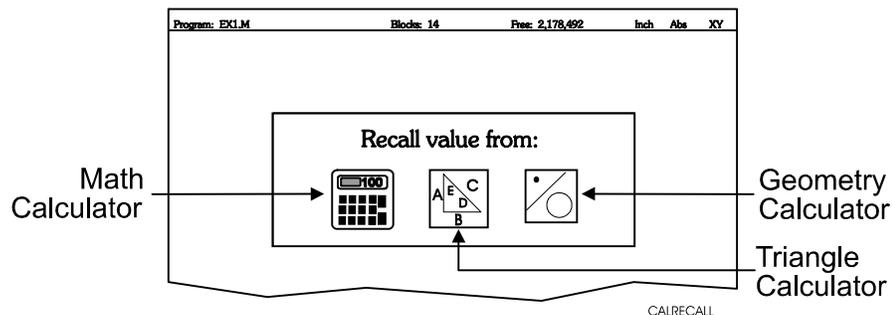


Figure 12-5, Recall Selection Pop-up Menu

Recalling Values from the Math Calculator

To recall a value from the Math Calculator:

1. From the Graphic Menu for the block being edited, highlight the field and press **Recall (F2)** to display the **Recall value from:** pop-up menu.
2. Highlight the **Math Calculator** template, and press **ENTER** to display the **Select value:** pop-up menu, with the most recently saved value displayed at the top of the list. Refer to **Figure 12-6**.

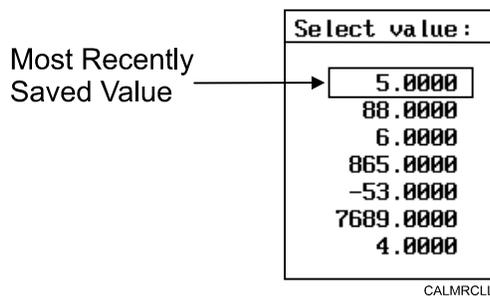


Figure 12-6, Math Calculator Select Value Pop-up Menu

3. Highlight the required value, and press **ENTER** to copy the stored value into the Graphic Menu.

Recalling Values from the Right Triangle Calculator

To recall values from the Right Triangle Calculator:

1. From the Graphic Menu for the block being edited, highlight the field receiving the recalled value.
2. Press **Recall (F2)** to display the **Recall value from:** pop-up menu. Refer to **Figure 12-7**.

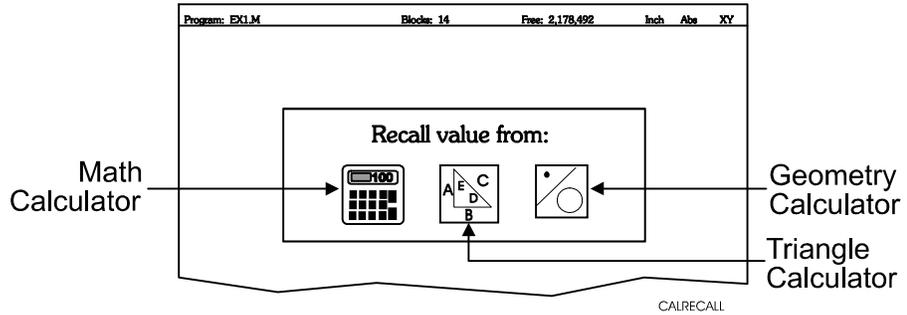


Figure 12-7, Recall value from: Pop-up Menu

3. Highlight the **Triangle Calculator** template, and press **ENTER** to display the Triangle Calculator **Select value:** pop-up menu. Refer to **Figure 12-8**.

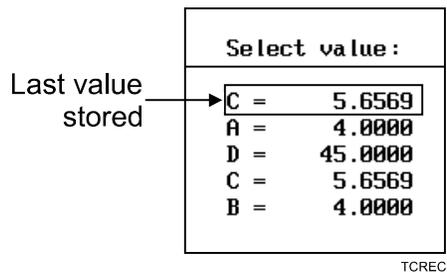


Figure 12-8, Triangle Calculator Select value: Pop-up Menu

4. Highlight the required value, and press **ENTER** to copy the stored value to the Graphic Menu.

Recalling Values from the Geometry Calculator

Recall Geometry Calculator values from the calculator's **Select point:** pop-up menu. The CNC displays this menu next to a copy of the sketch that generated the points. The recall list corresponds to the numbered elements on the sketch.

Positions on the recall list followed by a **[C]** are circle centers.

Positions on the recall list followed by a **[P]** are points.

Either one or both of the position coordinates can be recalled.

To recall a value from the Geometry Calculator:

1. From the Graphic Menu for the block being edited, highlight the field receiving the recalled value.
2. Press **Recall (F2)** to display the **Select point:** menu. Refer to **Figure 12-5, Recall Selection Pop-up Menu.**
3. Highlight the **Geometry Calculator** template, and press **ENTER** to display the **Select point:** pop-up menu. Refer to **Figure 12-9.**

[C] - Indicates Position is a Circle Center

[P] - Indicates Position is a Point

Select point :			
▶ 1. X	0.0000 Y	0.0000	[P]
2. X	1.0000 Y	1.0000	[C]
4. X	5.0000 Y	6.0000	[P]
6. X	6.0000 Y	6.0000	[P]
7. X	3.0000 Y	3.0000	[P]

CALGRCL

Figure 12-9, Geometry Calculator, Select point: Pop-up Menu

4. Highlight the required values, and press **ENTER**. The **Select term:** pop-up menu prompts for **Both X and Y** values, **X only**, or **Y only**.
5. Select the required terms, and press **ENTER** to copy the selected values to the Graphic Menu.

NOTE: Position coordinates can be recalled only to Graphic Menu entry fields that require X and/or Y coordinate values.

Recalling Values from One Calculator into Another

Both the Math Calculator and the Triangle Calculator recall values from other calculators.

In the **Triangle Calculator**, press **Recall (F2)** to activate recall. In the **Math Calculator**, press an **ARROW** to activate recall.

Section 13 - Off-line Software

The off-line version of the CNC software requires a computer with an **Intel® **Pentium® processor. The hard disk drive must have a minimum of 4MB of available space.

The off-line software is compatible with **Microsoft® **Windows® 98 operating system or later:

Passwords

Access to some parameters is restricted. Passwords protect four access levels. Operators are assigned limited access passwords, which allows them to set parameters used in normal machine operations. Service passwords allow a higher level of access. The factory password is not used outside the manufacturer's plant. When required, the CNC prompts you to type a password. Refer to **Table 13-1**.

Table 13-1, Off-line Passwords

Restriction Level	Password
Limited access	Q1
Service access	Q2
IPI access	Q3

To access protected parameters:

1. Type the service access password, and press **ENTER**. The CNC allows you to change the protected settings.

Exiting the Software

Press **ESC** to exit the software from the Main Menu selection screen.

Windows Installation (Use Windows Installation Disk)

1. Insert the installation disk in the floppy disk drive.
2. Go to the task bar and click on the **Start** button. Select **Run**. The CNC activates the Run.
3. In the **Open** entry field, type **A:setup** and click on **OK**. The CNC begins the installation procedure.
4. Answer each on-screen prompt as it is displayed on the screen.

NOTE: Substitute **B:setup** for **A:setup** if your 3.5-inch floppy disk is in the B: drive.

NOTE: During software installation, the CNC prompts you to install sample programs. The 3-axis sample programs begin with **3AX-**.

** Intel® and Pentium® are registered trademarks of Intel Corporation.

** Microsoft®, Windows® are registered trademarks of Microsoft Corporation in the United States and/or other countries.

Running from Windows

1. If you selected Desktop Icon (recommended) during the installation, click on the 3000M icon on your desktop.
2. If you selected Program Group or Start Menu, start from there.

Setting up the Icon

NOTE: By default, the computer creates the **C:\P3M** directory during software installation. If you use any other directory for the software, substitute the appropriate directory name in the instructions.

To set up the icon to run the software:

1. In windows program manager, select a folder (for example: **Applications**).
2. In the **File** pull-down menu, select **New** to activate the New program Object window.
3. The **Program Item** radial button is already selected. Click on **OK** to activate the Program Item Properties window.
4. Type the following in the indicated entry fields:

Description	3000M
Command	C:\ P3M\RUNCNC.BAT
Working Directory	P3M
5. Click on the **Change Icon** button. The CNC displays a prompt stating there are no icons available for the specified file. Click **OK** to activate the Change Icon window.
6. In the File Name entry field, type **C:\P3M\P3M.ICO**. Click **OK** to close the Change Icon window. Click **OK** to close the Program Item Properties window. The CNC displays the icon in the selected folder.

To run the off-line software:

1. Click on the icon created in the previous procedure.

System Settings

Maximum Memory Allocated

In the Setup Utility, you can adjust the amount of memory allocated to the CNC software. Set the Maximum Memory Allocated parameter between 2 MB and 18 MB. This limits the amount of memory available to the software and prevents the CNC program from tapping into Windows' large virtual memory supply. If you allocate too much memory to the CNC software you will dramatically increase startup time and make the software run slower.

To change the Maximum Memory Allocated:

1. In the CNC startup screen, select **Setup Utility**, and press **ENTER** to display the Setup Options menu.
2. Select **Operator Setup**, and press **ENTER** to display the General Software menu.
3. Select **Control Software**, and press **ENTER** to display the Control Software menu.
4. Select **Maximum Memory Allocated**, and press **ENTER**. Type the appropriate value, and press **ENTER**.

The only time it might be necessary to increase this parameter is when you edit a program that is larger than this value. In this case the CNC generates an error message indicating that there is not enough memory to edit the program. To correct this, change this parameter to the size of program plus 1MB.

Disabled Features

The following software features, found in the Program Directory's **Utility (F9)** pop-up menu, are not available under any Windows operating system:

- Check Disk
- Disk Optimization
- Program Restore

If you select any of these features, the CNC displays a message to inform you that the feature is disabled.

Using Soft Keys from a Keyboard

The (**F1** to **F10**) keyboard keys correspond to the CNC (**F1** to **F10**) soft keys, located on the console below the LCD. The **SHIFT** key activates secondary soft key functions, when applicable. To access secondary soft key labels on an off-line keyboard, simultaneously press the **SHIFT** key and the appropriate function key (**F1** to **F10**).

To make jog moves using the keyboard, use the **ALT + F1** to **ALT + F5** keys to select the increment. Press the **AXIS/DIRECTION** key to start the jog. (Example: Press **ALT + X** to initiate a positive X jog move.)

To make continuous jog moves, press **ALT + F1** (Rapid) or **ALT + F2** (Feed). Press the **AXIS/DIRECTION** key to start the jog. Press any key to stop a continuous jog.

Keypad Equivalent Keyboard Keys

Refer to **Table 13-2** for keyboard keystroke equivalents to the keypad keys, **Table 13-3, Off-line Keyboard, Tool Page Utilities** for off-line keyboard and tool page utilities, and **Table 13-4, Off-line Keyboard, Program File Directory Utilities** for off-line keyboard, program file utilities.

Table 13-2, Keyboard Keystroke Equivalents

Key Name	Key Face	Keyboard Keystroke Equivalent
ABS/INCR		ALT + E
(+/-) key		+ OR -
CLEAR		ALT + C – or – DELETE
ARROWS		ARROWS
ENTER		ENTER
RAPID		ALT + F1
FEED		ALT + F2
JOG: 100		ALT + F3
JOG: 10		ALT + F4
JOG: 1		ALT + F5
STOP CONTINUOUS JOG	none	ALT + T

Key Name	Key Face	Keyboard Keystroke Equivalent
X+		ALT + X
X-		ALT + K
Y+		ALT + Y
Y-		ALT + L
Z+		ALT + Z
Z-		ALT + N
START		ALT + S
HOLD		ALT + H
FEEDRATE OVERRIDE 0%		ALT + 0
FEEDRATE OVERRIDE 10%		ALT + 1
FEEDRATE OVERRIDE 20%		ALT + 2
FEEDRATE OVERRIDE 30%		ALT + 3
FEEDRATE OVERRIDE 40%		ALT + 4
FEEDRATE OVERRIDE 50%		ALT + 5
FEEDRATE OVERRIDE 60%		ALT + 6
FEEDRATE OVERRIDE 70%		ALT + 7
FEEDRATE OVERRIDE 80%		ALT + 8
FEEDRATE OVERRIDE 90%		ALT + 9
FEEDRATE OVERRIDE 100%		ALT + A

Key Name	Key Face	Keyboard Keystroke Equivalent
FEEDRATE OVERRIDE 110%		ALT + B
FEEDRATE OVERRIDE 120%		ALT + D
SPINDLE FORWARD		ALT + F
SPINDLE OFF		ALT + O
SPINDLE REV		ALT + G

Table 13-3, Off-line Keyboard, Tool Page Utilities

Function	Keystroke	Operation
Clear all entries on the Tool Page.	SHIFT + F3	Clears all entries from the Tool Page; prompts for confirmation before it clears the page.
Exit Tool Page without saving changes.	SHIFT + F10	Exits the Tool Page; unsaved edits are lost.

Table 13-4, Off-line Keyboard, Program File Directory Utilities

Function	Keystroke	Operation
Create a subdirectory.	SHIFT + F2	Prompt for NEW DIR: __ is displayed above the soft key labels.
Delete any program in the specified path.	SHIFT + F3	Prompt for PROGRAM TO DELETE: _ is displayed above the soft key labels.
Display System Information screen.	ALT + I	Displays System Information screen.

Editing with a Text Editor

You can also edit programs with the ASCII text editor. Refer to **Figure 13-1**.

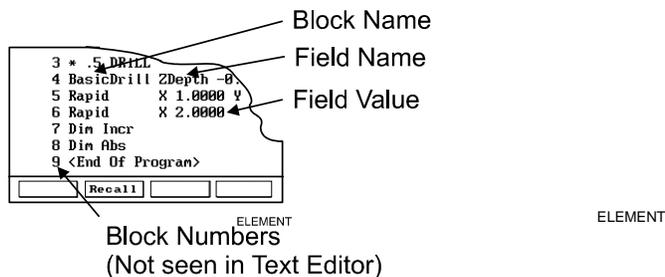


Figure 13-1, Program Block Elements

The following rules apply when you create or edit CNC blocks with a text editor:

- Start every program block with the block name. The name identifies the move, cycle, or mode change.
- All required entry field names and values must appear after the block name. Separate all block names, field names, and field values by at least one blank space.
- Do not enter block numbers when using ASCII text editors. The CNC automatically numbers the blocks when it loads the program.
- The Program Editor uses all uppercase letters to make the Program Listing easier to read, but not all text editors are case sensitive.
- Spell block names as they appear in the Program Listing.
- A decimal point is not necessary after a whole number.
- Include one Carriage Return/Line Feed combination (CR/LF) at the end of every block. Press **ENTER** to insert a CR/LF. The CR/LF will not appear on most ASCII text editor screens. Do not use more than one consecutive CR/LF.
- Use an asterisk (*) as the first character of a block to comment out blocks in a text editor.
- Rapid, Line, and Arc moves involve one, two, or three axes.
- List a **Feed** (feedrate) value at least once in the first feed move of the program.
- Program **ToolComp** (Left, Right, or Off) blocks with a Line or Rapid move of a length equal to or greater than the cutter radius.
- Arc center points are modal (defined in the active mode).

NOTE: You can set up machines with non-modal Arc centers.

- Arc direction (Cw/Ccw) is modal and remains in effect until you cancel or change it.
- Numbered values must conform to the specification listed for the G-code converter. (Refer to “Section 9 - Program Management.”)

Section 14 - Four-Axis Programming

Axis Types

The machine builder sets up the fourth-axis as linear, rotary or readout axis. The three basic axes are X, Y, and Z. The additional axis is designated as U (four-axis systems). This section will discuss the rotary axis option in detail.

Below are the programming formats for linear or rotary additional axes:

Linear: Program as Line or Rapid moves. Only rapid and linear line moves can be programmed. You must set an individual feedrate if a move is non-synchronous (Format: FeedU 20.0). U can be programmed along with X, Y, and Z-axis in rapid, linear, and arc moves. You can make U synchronous or non-synchronous to the XYZ moves. Program Sync codes:

MCode 900 U (for synchronous)

MCode 901 U (for non-synchronous)

on a block alone.

Rotary: Program rotary moves in degrees. The typical resolution is 0.001 degrees (set by builder). Minutes and seconds cannot be programmed. Therefore, you must convert minutes and seconds to a decimal value.

Conversion formula for minutes, seconds to decimal degrees:

Minutes to decimal: $\text{min}/60 = \text{decimal degrees}$.

Example: $15 \text{ min}/60 = 0.25 \text{ degrees}$

Seconds to decimal: $\text{sec}/3600 = \text{decimal degrees}$

Example: $30 \text{ sec}/3600 = 0.008 \text{ degrees}$

Example:

$$\begin{aligned} &5 \text{ deg. } 30 \text{ min. } 15 \text{ sec. } = \\ &5 + (30/60) + (15/3600) = \\ &5 + 0.5 + 0.004 = \\ &5.504 \text{ degrees} \end{aligned}$$

You must program feedrates for non-synchronous moves. Program the feedrate in degrees per minute (deg/min).

Format: FeedU 500.0 = 500 deg/min for the U-axis.

Rotary Axis Programming Conventions

A rotary axis (typically U-axis) will program differently based on the setting of the **Reset Rotary at 360** parameter, which is determined by the builder. The default for this parameter is **No**; in which case, the U-axis behaves like a linear axis. If set to **Yes**, the behavior of the rotary axis (U) is described below.

If programming the U-axis in Absolute:

The rotary axis will never rotate more than 180 degrees in one move. So, if a move of greater than 180 degrees is programmed, the control will resolve the number to a positive value less than 360 degrees and move to that target, taking the shortest distance (always less than 180 degrees). A move of exactly 180 degrees will always move positive and a move of exactly 360 degrees will not move at all.

If programming the U-axis in Incremental:

The rotary axis will move the exact amount of degrees programmed and in the direction indicated with the plus or minus sign. The display will reset to zero every time 360 degrees is crossed so that the highest value in the U-axis display will be 359.999 degrees depending on the displayed resolution.

Non-Synchronous or Synchronous Auxiliary Axis

You must decide whether the fourth-axis move will be synchronous or non-synchronous. A synchronous move is defined as a move in which all axes programmed (XYZU) reach target simultaneously. Use synchronous moves to thread-mill along a rotary axis or to machine an impeller part. Usually, the programmer uses an M-Code to set synchronization ON (Sync-On) or OFF (Sync-Off) for the designated axes.

[Default: **Sync-Off**]

Format: MCode 900 U

Example: MCode 900 U sets Sync-On for U-axis. If a U dimension is programmed on the same block as any rapid or linear feed move, the U-axis will reach target simultaneously with the other axes. The feedrate (Feed n) programmed for the XYZ move (X, Y, Z, XY, XZ, YZ, or XYZ) will be applied along the vector of the cut. If you program 10 in/min (Feed 10), the CNC adjusts the feedrates along each axis so that tool movement equals 10 in/min (vector speed). The feedrate displayed on the CNC screen is the vector feedrate.

Synchronous moves are used only when necessary. For example, you want to thread mill a part, with the centerline of the rotary axis parallel to the X-axis. Synchronize X and U. Turn sync OFF when not in use.

Format: MCode 901 U

Example: MCode 901 U will set Sync-Off for U-axis. If a U dimension is programmed on the same block as any rapid or linear feed move, the U-axis will not reach target simultaneously with the other axes. The feedrate (Feed n) programmed for the XYZ move (X, Y, Z, XY, XZ, YZ, or XYZ) will be applied along the vector of the cut. A separate feedrate is programmed for U (FeedU n). The feedrate programmed for XYZ (Feed n) will be applied to those axes.

Generally, Nonsynchronous moves are used for indexing work such as milling a side of a part or drilling at various rotary locations.

Nonsync fourth-axis moves require a separate feedrate for the additional axes (FeedU n). (If you do not program a separate feedrate, the U feedrate will be the Default for U, set by parameter or the last U feedrate programmed.)

Feedrate display is either XYZ (Sync-Off) or vectored (Sync-On).

In a 4-axis setup, the rotary table is set so the centerline of the rotary axis is parallel to the X-axis.

Programming Examples

All programming examples are for 4-axis machining with the rotary table mounted on the left end of the mill table, with the centerline of the rotary axis parallel to the X-axis. The face of the rotary table faces X+.

The examples contain both milling and drilling applications. Modal cycles BasicDrill to ChipBreak can be executed at rotary locations as in XYZ locations. Non-modal canned cycles can be executed at rotary locations. Use sync-OFF and position the rotary axis before you execute the cycle.

Example 1: Drill (Sync-Off)

Mount the fourth-axis as described above. Mount a part 6-inches wide and 8-inches long on the face of the rotary table. **Reset Rotary at 360** is set to **No**.

Table 14-1, Four-Axis Example 1 shows a drilling example with No Sync. You must drill ten 0.375-inch holes 36-degrees apart, 1-inch deep, 0.75-inches in from the end of the cylinder. Then, starting at X-2 U0, drill a spiral series of holes 36-degrees and X-.500 inches apart each. Set X0 at the right end, Y0 at the cylinder's centerline, U0 at a pre-milled keyway on the cylinder. Measure tool offsets from the top of the cylinder, with Y-axis at 0.

Table 14-1, Four-Axis Example 1

```
* 4-AX-DRL
* SET RESET ROTARY AT 360 TO "NO"
Dim Abs
Unit Inch
Rapid   Z 0.0000 U 0.0000 Tool# 0
MCode 5
Rapid   X 0.0000 Y 0.0000
Tool# 1
* #3 CENTERDRILL
RPM 2400
MCode 3
BasicDrill ZDepth -0.2200 StartHgt 0.1000 Feed 12.0000
Call 1
Tool# 2
* 3/8" DRILL
RPM 1850
MCode 3
ChipBreak ZDepth -1.0000 StartHgt 0.1000 FirstPeck 0.1800
PeckDecr 0.0120 MinPeck 0.1000 RetractDep 0.3334 Feed 14.0000
Call 1
EndMain
Sub 1
* ROTARY HOLE LOCATIONS
Dim Abs
Rapid   X -0.7500 Y 0.0000 U 0.0000
Loop    Sub# 2 #Loops 9
Dim Abs
Rapid   X -2.0000 U 0.0000
Loop    Sub# 3 #Loops 9
DrillOff
MCode 5
Dim Abs
Rapid   U 0.0000
Tool# 0
Rapid   Z 0.0000
Rapid   X 0.0000 Y 0.0000
EndSub
Sub 2
* U PLUS INCR.
Dim Incr
Rapid   U 36.0000
EndSub
Sub 3
* U MINUS INCR.
Dim Incr
Rapid   X -0.5000 U -36.0000
EndSub
```

Example 2: Mill (Sync-On)

Mount the fourth-axis as described above. Mount a part 3 inches in diameter and 5 inches long on the face of the rotary table. **Reset Rotary at 360** is set to **No**.

Table 14-2 shows a milling example only. Assume that a series of six 0.25-inch wide grooves must be milled 60-degrees apart, 0.25-inch deep at the start, tapering up to 0.125-inch deep and rotating 15 degrees at the far end. Set X0 at the right end, Y0 at the cylinder centerline, U0 at a pre-milled keyway on the cylinder. Set the tool offset so that the centerline of the 0.25-inch ball-end mill is at the centerline of the 3-inch diameter part (with Y-axis at 0).

Table 14-2, Four-Axis Example 2

```
* 4-AX-MILL
* SET RESET ROTARY AT 360 TO "NO"
Dim Abs
Unit Inch
Tool# 0
Rapid      Z 0.0000 U 0.0000
MCode 5
Rapid      X 0.0000 Y 0.0000
Tool# 1
*.25 BALL-END-MILL
MCode 3
RPM 2400
Loop      Sub# 1 #Loops 6
Dim Abs
Tool# 0
Rapid      Z 0.0000
MCode 5
Rapid      X 0.0000 Y 0.0000 U 0.0000
EndMain
Sub 1
* GROOVE
Dim Abs
Rapid      X 0.2250
Rapid      Z 2.8750
Line       X 0.0000 Feed 5.0000
MCode 900 U
Dim Incr
Line       X -3.3750 Z 0.1250 U -15.0000
MCode 901 U
Dim Abs
Rapid      Z 3.2250
Rapid      X 0.2250
Dim Incr
Rapid      U -45.0000
EndSub
```

Example 3: Mill (Sync-On)

Mount a fourth-axis as described above. Mount a part 4-inches in diameter and 5-inches long on the face of the rotary table. Support the part on the X+ end by a live center. The part has 0.25-inch, 45-degree chamfer on one end. **Reset Rotary at 360** is set to **Yes**. This will prevent the need to unwind the U-axis, saving operation time.

Table 14-3 shows a thread-milling example. Assume that a 4-8 UN 2A thread must be milled from the right end, 3-inches long. The tool is tapered to conform to the thread. Set X0 at the right end, Y0 at the cylinder's centerline, U0 at a pre-milled keyway on the cylinder. Measure the tool offset from the top of the part (with Y-axis at 0).

The X start position will be one pitch (0.125 in.) to the right of X0, so that the tool enters the work smoothly.

Table 14-3, Four-Axis Example 3

```
* 4-AX-THD
* SET RESET ROTARY AT 360 TO "NO"
Dim Abs
Unit Inch
Tool# 0
Rapid      Z 0.0000 U 0.0000
MCode 5
Rapid      X 0.0000 Y 0.0000
Tool# 1
* SPECIAL THD-TOOL
RPM 3500
MCode 3
Rapid      X 0.1250 Y 0.0000 U 0.0000
Rapid      Z 0.1000
Line       Z -0.0750 Feed 80.0
MCode 900 U
* SET RET ROTARY AT 360 TO "YES"
* THIS IS TO AVOID HAVING TO UNWIND U
* U AXIS MOVE IS
* (360 X 8 PITCH X 3" LONG)
* + 360 FOR 1 TURN X.125 LEAD-IN
* U MOVE WILL BE 9,000.00 DEGREES
* OR 25 TURNS
Dim Incr
Line       X -3.1250 U 90000.0000
MCode 901 U
Dim Abs
Tool# 0
Rapid      Z 0.0000
MCode 5
Rapid      X 0.0000 Y 0.0000 U 0.0000
EndSub
```

Section 15 - DXF Converter Feature

The DXF Converter feature allows information in a Drawing Exchange File (.DXF extension) to be used to create a CNC conversational (.M extension) file.

Contours and drill hole locations in the DXF file can be put in the CNC file in the form of subroutines, using a mouse and “point and click” approach.

The DXF Converter feature creates a CNC program that must be edited to be usable, but most of the program creation is already done.

Requirements

Off-line Software

The Personal Computer (PC) must have a mouse installed. The ANILAM Off-line Software is required. The ANILAM Off-line Software will run in a Windows environment. (See “Section 13 - Off-line Software.”)

Machine Software

A mouse or other pointing device (for example, track ball) must be installed to properly operate the DXF converter on the machine. As part of installing a mouse, it is necessary to install a driver. A suitable driver should be included with the mouse when purchased. Follow the manufacturer's instruction to install a **MS-DOS® compatible driver. Typically, this will require editing AUTOEXEC.BAT and adding a line to load the mouse driver before the CNC software starts running. See sample AUTOEXEC.BAT below. The mouse can be a serial mouse or a PS/2 style mouse (PS/2 style mouse only supported on systems manufactured after January 1, 2003). The serial mouse must be connected to the RS-232 port. If using a PS/2 style mouse, it must be connected to the keyboard connector on the console through a PS/2 Y-splitter cable. The mouse will only be operational while in the DXF Converter. Depending on the mouse and mouse driver, it may be necessary to have the mouse connected before turning the CNC on.

```
ECHO OFF
PROMPT $P$G
PATH C:\;C:\DOS
ECHO Loading CNC software
SET P5MSYS=C:\P5M
C:
CD \P5M
\MSMOUSE\MOUSE      <<<< Line added to load mouse driver
P5M
```

 ** MS-DOS® is a registered trademark of Microsoft Corporation in the United States and/or other countries.

Entry to the DXF Converter

To open the DXF Converter:

1. Open the ANILAM Off-line Software
2. Gain access to the Program page and highlight the DXF file you wish to convert. For details on how to work with the Program page see "Section 9 - Program Management."
3. Select the **Utility** soft key to display a pop-up menu
4. Select DXF Converter on the pop-up menu to bring the drawing into the DXF Converter.

The drawing display screen shows the file name in the upper-left border, the current X Y position of the mouse pointer at the upper-right corner, a message and information area along the bottom border, and several soft keys along the bottom edge.

Creating Shapes

The part drawing is used to create shapes. Shapes are then output to CNC programs as subroutines. There are many features to make the drawing screen easier to use. Layers may be turned on or off. Any area of the screen can be zoomed in or out. Refer to **Table 15-1, Mouse Operations** and **Table 15-2, DXF Hot Keys**.

Each shape is given a sequential number. The number is displayed on the screen at the beginning of the shape. Press **F1** to toggle Select mode on. With Select mode on, entities can be selected to be part of a shape. Left click with the mouse to make selections. When an entity is selected, it will change color.

When selecting entities, direction is determined by where the mouse pointer is positioned. If nearer the left side of a line, direction will be left to right.

There are two types of shapes:

- One used for contouring or feed motion
- One for drilling a series of holes

Contours

Pick an entity where the shape will begin. Pick the last entity in the shape. All entities that are connected will be chained together and change color to verify this. Some shapes have to be selected one entity at a time. This is determined by the way the part was drawn in the DXF file.

If an entity is selected that is not connected to the previous one, a message is displayed, "Entity not connected, connect anyway (Y/N)?" Answering Yes will join and allow the shape to be continued. This is sometimes needed when the drawing was not properly made. The same message is displayed when a shape is finished and a new shape is started. Answering No produces a second message, "Create new shape (Y/N)?" If this is answered Yes, a new shape number is displayed.

Entities in a shape can be un-selected by clicking them again. This un-selects everything previously selected to that point. To delete a shape, click on the first entity.

Drilling

When circle entities are selected, they are assumed to be drilling hole locations. As circles are selected, a dotted line shows the rapid path between holes. Selecting anything other than a circle, ends the drilling shape and produces the message, "Create new shape (Y/N)?"

If you wish to create a second drilling shape, select the first hole of the next shape by right clicking the mouse. The message, "Create new shape (Y/N)?" is displayed.

CNC Code

Each shape that is created is made into a subroutine. If the option was selected as a setup parameter, for each subroutine, there is a call in the main program. (Refer to **Table 15-4, Output Menu Descriptions.**) Running the CNC program in Draw mode allows the tool paths to be seen.

The file must be edited to add tool numbers, feed rates, cutter comp on or off, and so forth.

The tool paths are only as accurate as the DXF drawing file used.

The Output Parameter pop-up menu contains a Shift X and Shift Y entry. The purpose of these entries is to shift the X Y coordinate system of the CNC program. This is usually done, since the coordinates in the drawing file are based from some corner of the drawing and it is preferable to use some feature of the part as the X0 Y0 position. Refer to "Shift X, Shift Y Descriptions" later in this section.

Mouse Operations

Refer to **Table 15-1** and **Table 15-2, DXF Hot Keys**.

Table 15-1, Mouse Operations

Button	Event	Function
Left	Press–Drag–Release	Zoom Window
Right	Press–Drag–Release	Pan
Left	Double click	Zoom Fit
Left	Click Top Edge	View XY Plane
Left	Click Right Edge	View YZ Plane
Left	Click Bottom Edge	View XZ Plane
Left	Click Left Edge	View ISO
Left	SHIFT + Click	Zoom In
Right	SHIFT + Click	Zoom Out
Left	Click (Select mode on)	Select Entity
Right	Click (Select mode on)	Link Unconnected Entities or Create New Shape
Left	CTRL + Click	Basic Entity Information is displayed on the bottom of the screen.

DXF Hot Keys

Refer to **Table 15-2**.

Table 15-2, DXF Hot Keys

Hot Key	Event	Hot Key	Event
ALT + A	Zoom Fit	ALT + N	All Layers On
ALT + B	Redo View Change	ALT + O	View Plane ISO
ALT + C	Erase Text	ALT + P	Toggle Axes
ALT + D	Zoom In	ALT + Q	Quit / Exit
ALT + E	Set Chain Delta	ALT + R	Redraw
ALT + F	Toggle Entity Endpoints	ALT + S	Save Shape to Output Filename
ALT + G	Toggle Dotted Grid	ALT + T	Paste Entity Info to Shift Field
ALT + H	Zoom Out	ALT + U	Undo View Change
ALT + K	Cycle Tracking Resolution	ALT + X	View Plane XY
ALT + L	Toggle Layers	ALT + Y	View Plane YZ
ALT + M	Invert Layers	ALT + Z	View Plane XZ

Toggle Entity Endpoints (ALT + F)

This hot key places a small X at the end of every entity. If a shape is not chaining as expected, using this feature helps determine the problem. Look for X's in unexpected positions. Two lines may be overlapped, for example.

Using the mouse (**CTRL + left click**) with endpoints on will 'flash' each entity and also provide more information. Zoom the problem area for the best view.

DXF Soft Keys

Refer to **Table 15-3**.

Table 15-3, Soft Key Descriptions

Soft Key	Function	Description
F1	Toggle Select Mode	Select mode must be on when chaining shapes.
F3	Layers Menu	<p>Pop-up menu has:</p> <ul style="list-style-type: none"> ▪ All Layers on ▪ Invert Layers ▪ Toggle Layers <p>Layers can be turned on or off as desired.</p> <p>NOTE: The mouse (CTRL + left click), that is hold down the CTRL key while left clicking an entity, provides the layer name of the clicked entity at the end of the basic entity information displayed at the bottom of the screen. This is helpful to determine which layers to turn on and/or off.</p>
F4	View Menu	Pop-up menu has: XY, XZ, YZ, and ISO selections.
F5	Display Menu	Pop-up menu has: Fit, Window, Redraw, Half, and Double. Select the desired display.
F6	Misc.	Pop-up menu has: Entity Info, Set Shift, Toggle Endpoints, and Link or New Shape. Select the desired option. See "Miscellaneous DXF Soft Key, F6" for details.
F8	Save	Creates CNC code. The message, "Successfully created (filename) (.M)." is displayed when Save is activated. If no shapes are defined, a warning message is displayed.
F9	Setup	Pop-up menu provides access to Output or Display parameters. Refer to Table 15-4, Output Menu Descriptions and to Table 15-5, Display Menu Descriptions .

(Continued...)

Table 15-3, Soft Key Descriptions (Continued)

Soft Key	Function	Description
F10	Exit	<p>F10 exits the Setup menus, exits the DXF Converter, and returns to the Program page.</p> <p>Be sure to Save (F8) any work done before exiting. Anything not saved will be lost.</p> <p>If shapes have been created, a message, “Exit (Y/N)?” is displayed. This is a reminder to be sure you have saved your work.</p>

Miscellaneous DXF Soft Key, F6

Press **Misc. (F6)** to display the miscellaneous DXF pop-up menu with the following options:

- | | |
|--------------------------|--|
| Entity Info | <p>Displays entity information at the bottom of the screen. (See Table 15-1, Mouse Operations. This option is equivalent to pressing (CTRL + left click).</p> <p>On the off-line or using an auxiliary keyboard, the Entity Info is displayed by holding the CTRL key when you pick the entity.</p> |
| Set Shift | <p>Pastes the Entity Info onto the Shift field. (See Table 15-2, DXF Hot Keys. This option is equivalent to pressing (ALT + T).</p> |
| Toggle Endpoints | <p>Toggles entity endpoints. (See Table 15-2, DXF Hot Keys. This option is equivalent to pressing (ALT + F).</p> |
| Link or New Shape | <p>When creating a shape, select this option to terminate the current shape or to add new entities to the current shape. When a new entity is selected, the “Connect to existing shape (Y/N)?” message is displayed. To continue, press Yes (F1) or No (F2).</p> |

Output Menu Options

Refer to **Table 15-4**.

Table 15-4, Output Menu Descriptions

Parameter	Default	Input Definition
Output program name	DXF filename	Different filename. No extension required.
Shift X	0.000	X position offset value (See "Shift X, Shift Y Descriptions.")
Shift Y	0.000	Y position offset value (See "Shift X, Shift Y Descriptions.")
Output dim.	Absolute	Incremental or Absolute.
Create mode	Smart	Overwrite or Smart. Overwrite replaces existing program file. Smart replaces duplicate subs only.
Starting sub number	1	Beginning sub number. To keep previously created subs, use next higher number.
Output warnings	Yes	Yes or No. A setting of Yes outputs warnings in the program.
Re-calc Intersections	Yes	Yes or No. A setting of Yes re-calculates the intersections in the DXF file.
Output resolution	5	Number of decimal digits (3 to 6)
Create main in new	Yes	Yes or No No will not create main to call subs in new programs.
Convert values	None	None or (to)Metric or (to)Inch
Convert polyline	None	None or To Arc or Prompt (See "Convert Polyline Descriptions.")

Shift X, Shift Y Descriptions

The end of an entity or center of a circle can be used to automatically make this point the X0 Y0 of the program. To select the desired point:

1. Use the mouse (**CTRL + left click**) to display Basic entity information.
2. Press (**ALT + T**) to paste the coordinates into the Shift XY fields.

A line or arc entity uses the end closest to the mouse pointer.

Convert Polyline Description

Some DXF files have arcs as polylines. Set the parameter **Convert to Arc** to **Yes** to have an arc output in the CNC program.

Polylines that are not converted to arcs are executed as a series of short line moves.

Circles that have been converted to polylines by the DXF file creation software cannot be used for hole drilling or circle center information. Hence, given the option, it is the best choice that arcs and circles not be converted to polylines when creating the DXF file.

Display Menu Options

Refer to **Table 15-5**.

Table 15-5, Display Menu Descriptions

Parameter	Default	Input Definition
Mouse position resolution	4	Number of places to display
Axes	Off	Toggle display of axes
Grid	None	None, Solid, Dotted
Grid size	1	In current units
Chaining accuracy	.0001	Maximum separation between chainable entities
Picking accuracy	3	Pixel distance from pointer to selected entity
Shape color	Green	Choose color
Entity flash color	Yellow	Choose color

DXF Entities Supported

See **Table 15-6** for the DXF entities supported.

Table 15-6, DXF Entities Supported

Entities	Drawing	Transformation	Chaining	Information
Line	X	X	X	X
Point	X	X		X
Circle	X	X	X	X
Arc	X	X	X	X
Trace	X	X		X
Solid	X	X		X
Text	X	X		
Shape	X	X		
Insert	X	X		
Attdef	X	X		
Attribute	X	X		
Vertex	X	X		X
Polyline	X	X	X	X
Line3d	X	X	X	X
Face3d	X	X		X

Drawing Entities Not Supported

See **Table 15-6**. Note that the Extrusion, Dimension, and Viewpoint entities are not supported. Dimensions may be seen on the displayed DXF file. Some DWG (an AutoCad drawing file) to DXF converters convert the dimension entities into lines and arcs, which are supported entities.

Files Created

The DXF Converter creates the CNC file, .M for conversational. .

A file is also created with the extension .fxd. This file saves the status of parameter settings that were used in Setup.

DXF Example

From the Program listing open the DXF file. Refer to **Figure 15-1**.

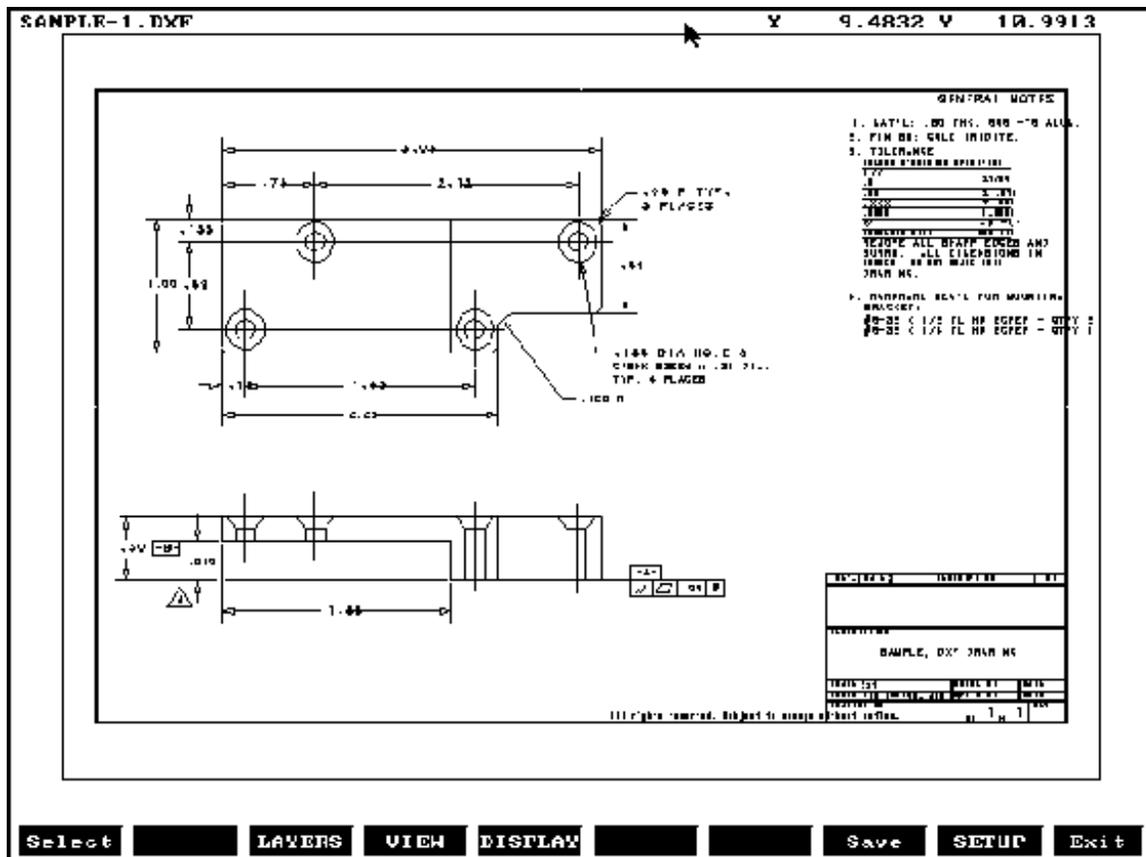


Figure 15-1, Example DXF File

Refer to **Figure 15-2**. All unneeded layers have been turned off. The Figure shows the drill locations and the contour selected.

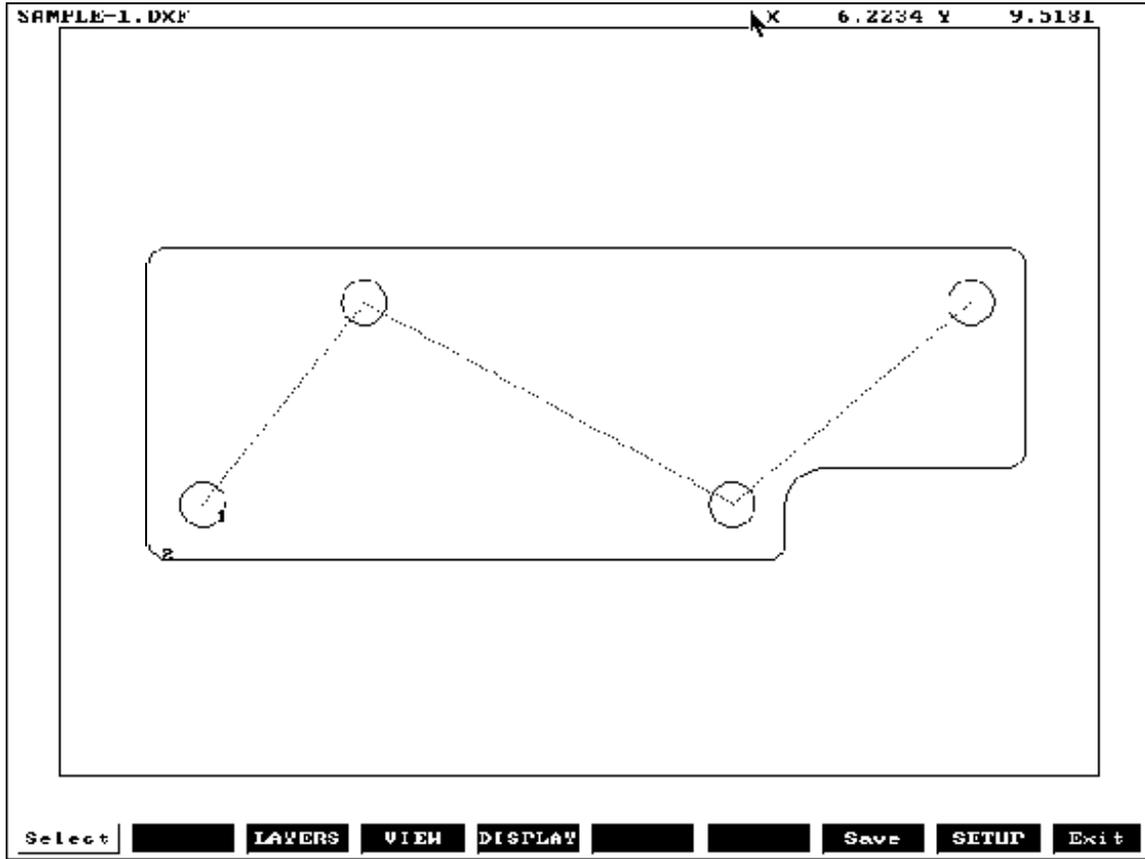


Figure 15-2, Zoomed Part with Unneeded Layers Turned Off

From **Figure 15-2**, the Output Menu (see **Table 15-4, Output Menu Descriptions**) would display as follows:

Table 15-7, Output Menu for Figure 15-2

Output program name	SAMPLE-1
Shift X.	2.41174
Shift Y.	6.16598
Output dimensions	Absolute
Create Mode.	Smart
Starting sub number.	1
Output warnings.	Yes
Re-calc Intersections.	Yes
Output resolution.	5
Create main in new	Yes
Convert values.	None
Convert polyline.	None

Unedited Conversational Program Listing

The CNC conversational program is created that must be edited to be usable. An unedited conversational program created from **Figure 15-2, Zoomed Part with Unneeded Layers Turned Off** follows. See **Table 15-8**.

Table 15-8, Unedited Conversational Program Listing

```

Call 1
Call 2
EndMain

Sub 1
Dim Abs
Rapid X 0.00000 Y 0.00000
Rapid X 1.12400 Y 1.37000
Rapid X 3.70000 Y 0.00000
Rapid X 5.36800 Y 1.37000
EndSub

Sub 2
Dim Abs
Rapid X -0.37600 Y -0.25000
Line X -0.37600 Y 1.62000
Arc Cw X -0.25100 Y 1.74500 Radius 0.12500
Line X 5.61900 Y 1.74500
Arc Cw X 5.74400 Y 1.62000 Radius 0.12500
Line X 5.74400 Y 0.37000
Arc Cw X 5.61900 Y 0.24500 Radius 0.12500
Line X 4.32500 Y 0.24500
Arc Ccw X 4.07500 Y -0.00500 Radius 0.25000
Line X 4.07500 Y -0.25000
Arc Cw X 3.95000 Y -0.37500 Radius 0.12500
Line X -0.25100 Y -0.37500
Arc Cw X -0.37600 Y -0.25000 Radius 0.12500
EndSub

```

The conversational program must be edited to add tool numbers, feed rates, cutter comp on or off, and so forth. When the edits are complete, use Draw to check the tool path. See **Figure 15-3, Edited Conversational Tool Path**.

Edited Conversational Tool Path

The edited conversational tool path is illustrated in **Figure 15-3**.

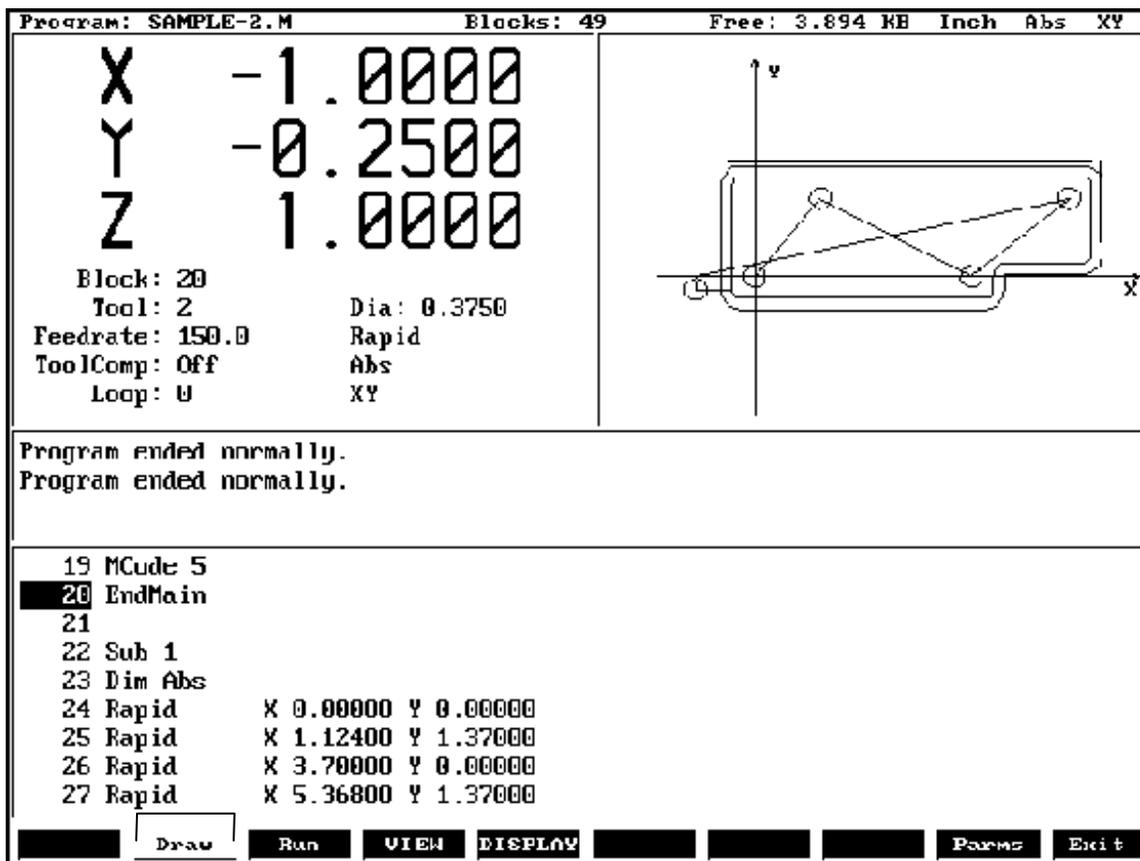


Figure 15-3, Edited Conversational Tool Path

Edited Conversational Program Listing

The edited conversational program used for the example above follows. Refer to **Table 15-9**.

Table 15-9, Edited Conversational Program Listing

Dim Abs
Unit Inch
Rapid X -1.0000 Y 0.0000
MCode 5
RPM 2000
Tool# 1
MCode 3
BasicDrill ZDepth -0.6000 StartHgt 0.1000 Feed 20.0
Call 1
DrillOff
Rapid X -1.0000 Y 0.0000
MCode 5
RPM 1500
Tool# 2

```
MCode 3
Feed 15.0
Call 2
Rapid   Z 1.0000
MCode 5
EndMain

Sub 1
Dim Abs
Rapid   X 0.00000 Y 0.00000
Rapid   X 1.12400 Y 1.37000
Rapid   X 3.70000 Y 0.00000
Rapid   X 5.36800 Y 1.37000
EndSub

Sub 2
Rapid   Y -0.2500
        Z -0.6000
Dim Abs
Rapid   X -0.3760 Y -0.2500 ToolComp Left
Line    X -0.37600 Y 1.62000
Arc Cw  X -0.25100 Y 1.74500 Radius 0.12500
Line    X 5.61900 Y 1.74500
Arc Cw  X 5.74400 Y 1.62000 Radius 0.12500
Line    X 5.74400 Y 0.37000
Arc Cw  X 5.61900 Y 0.24500 Radius 0.12500
Line    X 4.32500 Y 0.24500
Arc Ccw X 4.07500 Y -0.00500 Radius 0.25000
Line    X 4.07500 Y -0.25000
Arc Cw  X 3.95000 Y -0.37500 Radius 0.12500
Line    X -0.25100 Y -0.37500
Arc Cw  X -0.37600 Y -0.25000 Radius 0.12500
Line    X -1.0000 ToolComp Off
EndSub
```

Using DXF for Pockets with Islands

Refer to “Section 5, Pockets with Islands.” In DXF, make outside profile shape #1 or lowest number. Then all islands thereafter, the order is not important. When saving these, use Cam Shape. Refer to **Figure 15-4**.

1. Highlight the DXF filename, and press **Utility (F9)**.
2. Highlight **DXF Converter**, and press **ENTER**.
3. Press **SETUP**, select **OUTPUT format**, press **ENTER**.
4. Select **CAM Shape**, and press **ENTER**. Refer to **Figure 15-4**.
5. Select to **Convert values**, and press **ENTER**.
6. Select **Metric**, and press **ENTER**.
7. Press **Exit** twice.



Figure 15-4, DXF Output format, CAN Shape Pop-up Menu Illustration

Using **Figure 15-5, DXF Pockets with Islands Example Workpiece** as an example:

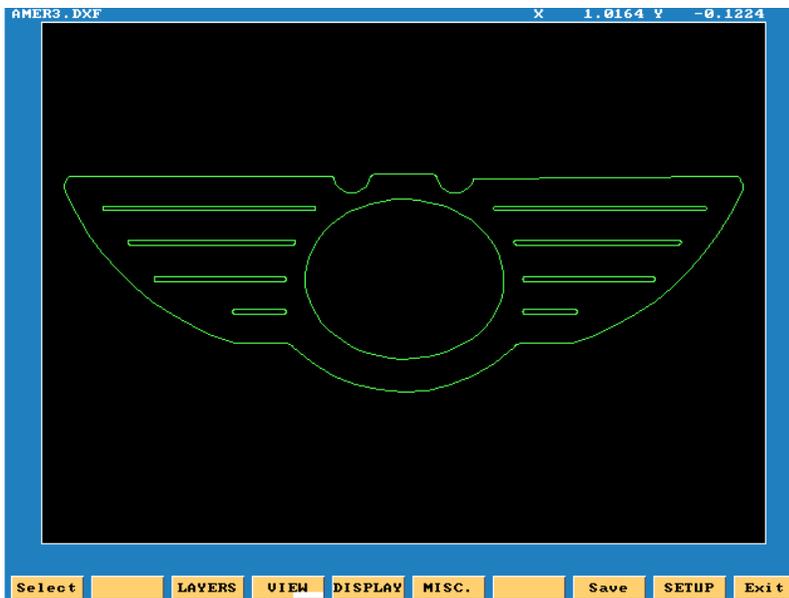


Figure 15-5, DXF Pockets with Islands Example Workpiece

1. Select a start point on outer profile and make shape #1. A good point on the workpiece illustrated is just below the radius top left.
2. Select next shape, until all 10 shapes are selected.
3. Press **Save (F9)**, it will save all 10 CAM shapes.
4. Press **Exit (F10)**.
5. Press **Yes (F1)** to exit.

An AMER3.G program needs to be created. Then the AMER3.G program is created, a pop-up list of programs is displayed. See **Figure 15-6**.

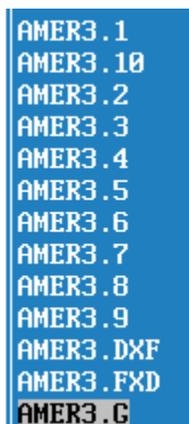


Figure 15-6, DXF Program Listing Pop-up Menu Illustration

DXF Program Example

Table 15-10, DXF Pockets with Islands Programming Example

1.	Dims Abs
2.	Offset Fixture# 1
3.	Tool #1
4.	Islands First Isl 1 Second Isl 2 Third Isl 3 Fourth Isl 4 Fifth Isl 5
5.	Islands First Isl 6 Second Isl 7 Third Isl 8 Fourth Isl 9 Fifth Isl 10
6.	Pocket Sub# 1 StartHgt 0.1000 Zdepth -0.0050 Angle 300.0000 Stepover .0100 Fin Stock 0.0010 RampFeed 30.0 Rough Feed 30.0 FinFeed 30.0
7.	RapidZ 2.0000
8.	Rapid X 0.0000 Y 0.0000
9.	EndMain

The completed DXF Pockets with Islands example is illustrated in **Figure 15-7**. The rapid moves are turned off in this illustration.

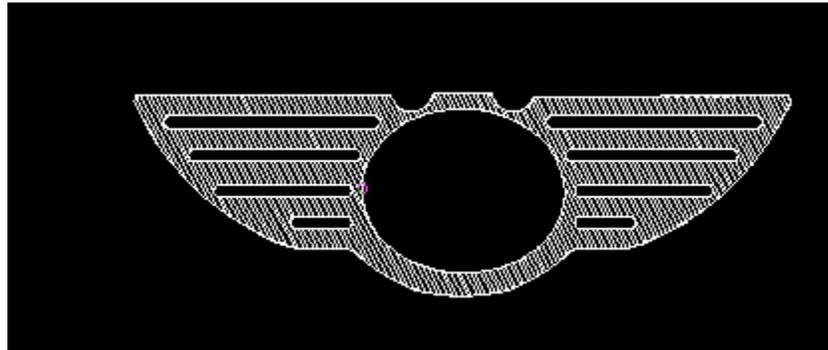


Figure 15-7, DXF Pockets with Islands Completed Example Workpiece

Section 16 - CNC Software

Machine Software Installation

To install the machine software:

1. Insert the software disk in the floppy disk drive.
2. From the CNC's startup screen, select **Setup Utility** and press **ENTER** to display the **Setup Options** menu.
3. Select **Machine/Installation** and press **ENTER** to display the **Machine/Installation** menu.
4. Select **Software Update** and press **ENTER** to begin the installation sequence. Follow the instructions on the screen.

Software Option Kit Installation

The software option kit contains new or optional features for the CNC.

Each kit typically contains the following items:

- Installation Procedure
- Installation Diskette
- Hardware Key

The hardware key enables purchased options.

Procedure

To install the software option kit:

1. Use the shipping document to verify that the kit contains the ordered options.
2. Gain access to the female DB-25 printer port connector and install the hardware key.

NOTE: Remove any previously installed hardware key before you install the new key.

3. Insert the floppy disk into the floppy disk drive.
4. Install the software as described above.

NOTE: In the Setup Utility, make any necessary configuration changes to enable the purchased options.

Refer to the *3000M CNC Setup Utility Manual*, P/N 70000499, for instructions on how to change the Setup Utility.

NOTE: After you install and verify the operation of the software upgrade, discard the original software disk.

Using Soft Keys from a Keyboard

Refer to “Section 13, “Using Soft Keys from a Keyboard.”

Keypad Equivalent Keyboard Keys

Refer to “Section 13, “Keypad Equivalent Keyboard Keys.”

Making Jog Moves from a Keyboard

Refer to “Section 13, “Using Soft Keys from a Keyboard.”

-
- #loops, 5-41
 - %, 3-4
 - % Exec Buf Done, description, 11-8
 - % Rec Buf Full, description, 11-8
 - *HALTED, 3-4
 - .DXF extension, 15-1
 - .fxd extension, created, using DXF converter, 15-11
 - .M extension, 9-2, 9-4
 - .M extension, created, using DXF converter, 15-1, 15-11
 - .S files, 9-3
 - 3000M CNC Setup Utility Manual*, P/N 70000499, referenced, 4-28, 5-66, 16-1
 - 3-D view, Iso, 7-12
 - A**
 - ABS, 3-4, 3-9
 - Abs/Inc, 6-2
 - ABS/INC key, illustration, 2-2
 - absolute mode
 - change, 4-4
 - description, 1-3, 3-9
 - hot key, 2-2
 - to activate, 3-9
 - absolute move, 4-5
 - absolute moves, 3-11
 - absolute zero
 - in X and Y, 3-10
 - locate, 3-10
 - measure from, 3-9
 - reference, 4-11
 - reference point, 1-3
 - resetting, 4-11
 - setting, 3-10
 - absolute/incremental modes, 3-9
 - accessing, communication package, 11-2
 - Action, 5-99
 - activate
 - a new tool, 4-5
 - a plane, 3-13
 - a spindle RPM, 3-14
 - a tool, 3-13, 4-5
 - background mode, 8-7
 - fixture offsets, 10-5
 - fixture offsets table, 4-10
 - manual mode, feed, 3-8
 - manual mode, rapid, 3-8
 - servos, 3-2
 - test link screen, 11-5
 - tool-diameter compensation, 4-6
 - U-axis synchronization (M900), 4-28
 - Activate, 5-99
 - active
 - feedrate, 3-8
 - fixture offset, 3-4
 - fixture offset location, 3-11
 - plane, 4-13, 6-2
 - positioning mode, 6-2
 - tool, 3-4
 - tool, choices, 3-7
 - units mode, display, 3-9
 - actual, RPM, 3-4
 - add, blocks to programs, 4-3
 - additional function, math calculator, 12-4
 - adjusting
 - Draw display, 7-10
 - feedrate, 3-8
 - rapid move, speed, 3-8
 - the spindle speed, 3-9
 - angle, measurement, 1-10
 - angle, references, 1-14
 - angle/radius template, 4-20
 - arc
 - compensation requirement, 4-6
 - direction, illustration, 1-15
 - program
 - using center - endpoint, hot keys, 4-23
 - using center - endpoint, soft keys, 4-24
 - using center - included angle, hot keys, 4-26
 - using center - included angle, soft keys, 4-26
 - using endpoint - radius, hot keys, 4-22
 - using endpoint - radius, soft keys, 4-22
 - selecting, the plane, 4-21
 - template, to display, 4-23
 - ARC, 3-4
 - ARC, hot key, 2-2
 - arc-to-arc corner, rounding, 1-12
 - ARROW, keys, 2-3
 - ASCII chart (F2), 2-7
 - ASCII chart, illustration, 2-6
 - ASCII, (F2), 2-6
 - ASCII, data type, 11-4
-

asterisk, 2-2
asterisk key, 6-5
AUTO, 3-4
auto mode
 description, 3-6
 program listing, 3-3
 program, cancel, 8-3
 program, hold, 8-3
 program, to run, 8-3
 starting block, select
 using arrow keys, 8-4
 using SEARCH, 8-4
 switch from, single-step mode, 8-3
 to activate, 3-6
AUTOEXEC.BAT, DXF
 converter, example, 15-1
automatic
 Draw restart, 7-8
 mode, defined, 8-1
 mode, Draw, 7-7
 setting, tool-length offset, from tool page,
 10-5
 tool compensation, 4-6
auxiliary
 axis, synchronous or non-synchronous,
 14-2
 keyboard, entity information, to display,
 15-7
available memory, 6-2
axes of rotation, illustration, 5-50
axis
 descriptions, 1-1
 of motion, illustration, 1-1
 types
 description, 14-1
 linear, description, 14-1
 rotary, description, 14-1
 work coordinate offset, 10-6

B

back up, the program, 4-2
background mode
 description, 8-7
 to activate, 8-7
ball-end mill
 diameter compensation, 1-10
 length offset, 1-10
 setting, tool-length offset, 10-5
basic drilling cycle
 description, 5-2
 graphic menu, illustration, 5-2

 baud rate, setting, 11-4
 before, activating a new tool, 4-5
 BINARY, data type, 11-4
 blank fields, 4-4
 blend, description, 1-11
 block
 number, 3-4
 program, execution of, 8-2
 program, to delete, 6-3
 program, to insert, 6-4
 selected, run Draw, 7-8
 BLOCK, 3-4
 bolt hole cycle
 description, 5-9
 to program, 5-9
 Boring block, 5-4
 boring cycle
 description, 5-4
 graphic menu, illustration, 5-4
 to program, 5-4
 Boss, 5-86
 Both, Draw, description, 7-5
 break and wear, BrkWearDet,
 5-67, 5-81
 BrkWearDet, break and wear,
 5-67, 5-81

C

CALC, hot key, 2-3, 12-1, 12-5,
 12-7
calculate, distance between, two
 element, 12-12
calculators
 description, 12-1
 hot key, illustration, 2-3
 hot key, to activate, 12-1, 12-5, 12-7
 listed, 12-1
 memory, 12-1
 select type, menu, illustration, 12-1
Calib Z, (F8), 10-5
CalibPtPrb, spindle probe
 calibration cycle
 defined, 5-84
 description, 5-86
calibrate, a fixture offset, 4-11
calibration keys, RefProg, 10-6
CalibTIPrb, probe calibration,
 5-67, 5-69
CalibX (F5), 10-6
CalibY (F6), 10-6
CalibZ (F7), 10-6

- call
 - a loop subprogram, 5-41
 - RMS subprogram, 5-42
 - subprogram, 5-40
 - subprograms from the main program, 5-40
- cancel
 - a comment, 6-6
 - a single step run, 8-2
 - auto mode program, 8-3
 - Draw, program, 7-3
 - feedrate change, 3-8
 - unsaved edits, 6-3
- canned cycle
 - library, 4-1
 - spindle probing cycles, description, 5-83
 - tool probe cycles, description, 5-66
- canned cycles, 1-9, 4-6
- chamfering, illustration, 1-13
- change
 - a fixture offset, 4-11
 - a tool, 4-5
 - absolute zero, 4-11
 - Draw views, 7-1, 7-12
 - drawing scales, 7-1
 - hole dimensions, 5-1, 5-4, 5-5
 - jog mode, 3-15
 - link test link screen data display, 11-5
- chip break cycle
 - description, 5-5
 - graphic menu, illustration, 5-6
 - to program, 5-6
- circle template
 - description, 12-11
 - purposes, listed, 12-11
- circular pocket cycle
 - compensation requirement, 4-7
 - description, 5-25
 - graphic menu, illustration, 5-25
 - to program, 5-25
- circular profile cycle
 - compensation requirement, 4-7
 - description, 5-21
 - graphic menu, illustration, 5-21
 - to program, 5-22
- cleaning up, inside diameter, 5-29
- clear
 - a halted program, 8-4
 - entries, 2-6
 - messages, 2-8
 - receive area, 11-6
 - tool page entries, 13-6
- CLEAR, key, 2-3, 2-6, 2-8
- CNC
 - DXF converter, description, 15-3
 - DXF converter, file creation, 15-1
 - DXF converter, files created, 15-11
 - parts counter, description, 8-6
 - power on, 3-1
 - shutting down, 3-1
 - software, 16-1
 - timer, description, 8-6
- column, of number, math calculator, 12-3
- COM1, 11-3
- COM2, 11-3
- Comm Port, selecting, 11-3
- comment
 - block, writing, 6-5
 - hot key, 2-2
 - out, existing block, 6-5
 - to cancel, 6-6
 - using, 6-5
- communication
 - description, 11-1
 - package, accessing, 11-2
 - parameters, setting, 11-3
 - port, to select, 11-3
 - screen layout, description, 11-2
 - screen, accessing, 11-2
 - screen, illustration, 11-2
- compensation
 - effects on end points, 1-8
 - effects on starting points, 1-8
 - left, 1-6
 - right, 1-6
- completed program, 3-4
- consecutive compensated moves, illustration, 1-7
- console, illustration, 2-1
- constructs, a line, 12-10
- continuous
 - jog, 3-7
 - jog, Feed mode, 3-15
 - jog, Rapid mode, 3-15
 - jogging, 3-16
- conventional, jog, 3-7
- conventional, jogging, 3-15
- conversational editor, More (F1) soft key, 6-6

- conversational programming
 - probe cycles, description, 5-66
 - spindle probe cycles, description, 5-66
 - spindle probing
 - description, 5-83, 5-85
 - designations, 5-84
 - tool probe cycle, description, 5-68
 - tool probe cycle, designations, listed, 5-67
 - tool probe cycle, listed, 5-68
 - tool probe cycles, description, 5-66
- conversion formula
 - minutes to decimal, 14-1
 - seconds to degrees, 14-1
- converting, G-Code programs to
 - conversational format, 9-8
- COOLANT, 3-4
- coolant, description, 10-3
- coolant, status, 3-4
- coordinates, 1-2
- copy
 - programs, other directories, 9-16
 - programs, to floppy disks, 9-6
- corner, rounding, 1-11
- CornerIn, inside part corner find
 - defined, 5-84
 - description, 5-91
- CornerOut, outside part corner find
 - defined, 5-84
 - description, 5-89
- CornerRad, 1-11
- CornerRad, negative value, 1-13
- counter bore, existing holes, 5-29
- create
 - new programs, 9-2
 - subdirectory, 13-6
- Create (F2), 9-2
- current
 - jog mode, 3-4
 - mode, 3-4
 - move mode, 3-4
- cursor, functions, 2-7
- cursor, tool page, description, 2-7
- cut through holes, 5-29
- cutting direction, 5-17
- cutting edge, 1-6
- cycle, compensation
 - requirements, table, 4-6

D

- data
 - bits, to set, 11-4
 - control codes
 - function, table, 11-9
 - receive mode, to use, 11-10
 - send mode, to use, 11-10
 - to use, 11-9
 - link, testing, 11-4
 - type, to set, 11-4
- date and time, 9-2
- DE-9, 11-1
- decimal point, hot key, 2-2
- decimal points, 4-4
- DECIMAL, hot key, 2-2
- default feedrate, 3-8
- default plane, 4-13
- define, a point, 12-9
- defining, absolute zero in X and Y axes, 3-10
- defining, positions, 1-2
- Del (F4), 2-8
- delete
 - a program, 9-4
 - all elements, 12-11
 - groups of programs, 9-6
 - program block, 6-3
 - sketch, elements, 12-11
 - text, 2-8
- Delete (F3), 9-4
- developing, part programs, 4-1
- DIA, 3-4
- diameter compensation, tool, 1-6
- diameter special, DiaSpecMea, 5-67, 5-78
- DiamOfStd, 5-69
- DiaSpecMea, diameter special, 5-67, 5-78
- dim block, to program, 4-4
- direct numeric control. *See* DNC
- disabled, features, off-line, 13-3
- disclaimer, iii
- DisDown, 5-86
- disengage
 - servos, 3-1
 - Z-axis drive system, 3-19
- disks, floppy, formatting, 9-7
- display
 - double size, 7-10
 - Draw adjusting, 7-10

- Draw, fit window, 7-10
- Draw, scale, 7-11
- erase, 7-8
- half size, 7-10
- modes, listed, 9-2
- system information, illustration, 9-15
- to erase, 7-11
- DISPLAY (F5), 7-2
- distance to go, position display, 3-5
- DistBack, 5-86, 5-90, 5-92, 5-93, 5-95, 5-100
- DistDown, 5-69, 5-72, 5-78, 5-81, 5-89, 5-91, 5-93, 5-95, 5-100
- DistInX, 5-86, 5-90, 5-92, 5-94, 5-96, 5-100
- DistInY, 5-86, 5-90, 5-92, 5-94, 5-96, 5-100
- DistPicks, 5-99
- DistSide, 5-89, 5-91
- DISTTOGO, distance to go, position display, 3-5
- DNC
 - put CNC, in, 11-9
 - running in, 11-8
 - screen, illustration, 11-8
- doubling, display size, 7-10
- Draw
 - automatic mode, 7-7
 - changing, views, 7-12
 - CNC code, view tool paths, 15-3
 - display
 - adjusting, 7-10
 - double size, 7-10
 - erase, 7-11
 - fit window, 7-10
 - half size, 7-10
 - size, scale, 7-11
 - end at, specific block, 7-9
 - erase display, 7-8
 - grid parameter, to set, 7-6
 - grid size, to set, 7-6
 - motion mode, 7-7
 - parameters, viewing, 7-4
 - rapid moves, description, 7-6
 - real-time mode, description, 7-1
 - restart, automatic, 7-8
 - run, selected block:, 7-8
 - S.Step, 7-7
 - simulation mode
 - cancel, program, 7-3
 - description, 7-1
 - screen, illustration, 7-3
 - to activate, 7-2
 - to pause, 7-3
 - single-step mode, 7-7
 - start at, specific block, 7-9
 - starting, 7-2
 - text, on or off, 7-4
 - tool compensation, description, 7-5
 - tool, on or off, 7-5
 - ToolComp, description, 7-5
 - using, while running programs, 8-5
 - viewing, programs, 7-1
 - zoom in, 7-11
- Draw (F10), 7-1
- Drawing Exchange File. See DXF converter
- drill
 - cycles, basic, 5-2
 - cycles, description, 5-1
 - cycles, listed, 5-1
 - pattern
 - description, 5-8
 - to program, 5-8
 - sync-off, programming example, 14-3
- drilling cycles, listed, 5-1
- Drilling Off block, 5-3, 5-5, 5-7
- DrillOff block, 5-1
- drive system, Z-axis, disengage, 3-19
- drives, log on, 9-4
- DRO mode, 4-18
- DRO, manual mode, 3-6
- dry run, 4-2
- dry run mode
 - all axes (M105), 4-28
 - M-codes, listed, 4-28
 - no Z-axis (M106), 4-28
- duplicating, manual moves, 4-18
- dwell
 - program, using hot keys, 4-7
 - program, using soft keys, 4-7
 - programming, 4-7
 - resolution, 4-7, 5-4
- DWELL, 3-4
- DWELL, hot key, 2-2
- DXF converter
 - CNC code, description, 15-3
 - contours and drill holes, 15-1

- contours, description, 15-3
- convert polyline, description, 15-9
- create, conversational file, 15-1
- display menu, descriptions, 15-9
- drilling, description, 15-3
- edited, conversational program listing, 15-14
- edited, conversational tool path, illustration, 15-14
- entities supported, table, 15-10
- entities, not supported, 15-10
- entity endpoints, toggle, 15-5
- example, 15-11
- exit (F10), description, 15-7
- feature, description, 15-1
- files, created, 15-11
- hot keys, table, 15-5
- mouse operations, table, 15-4
- output format, pop-up menu, 15-16
- output menu, descriptions, 15-8
- output menu, example, 15-12
- pockets with islands
 - description, 15-16
 - program example, 15-18
- program listing, pop-menu, 15-17
- requirements
 - machine software, 15-1
 - off-line software, 15-1
- shapes, creating, 15-2
- shapes, types, listed, 15-2
- shift X, shift Y, descriptions, 15-8
- soft keys, descriptions, 15-6
- to open, 15-2
- troubleshooting, 15-5
- unedited, conversational program listing, 15-13

E

- edge finding, EdgeFind, 5-88
- EdgeFind, single surface
 - measure/edge find
 - defined, 5-84
 - description, 5-88
- edit mode, hot keys, table, 2-2
- edited, marker, 6-2
- editing
 - keys, illustration, 2-1
 - keys, table, 2-3
 - program block, 6-4
 - program blocks, More (F1), 6-6
 - programs, 6-1

- with text editor, 13-7
- editor, conversational, More (F1), 6-6
- edits, canceling, unsaved, 6-3
- edits, saving, 6-3
- elbow cavity and core,
 - illustration, 5-55
- elbow milling cycle
 - conical elbow details, illustration, 5-58
 - core & cavity detail, illustration, 5-58
 - details, illustration, 5-56
 - graphic menu, illustration, 5-59
 - moves, illustration, 5-56
 - programming, 5-55
 - starting point, orientation, illustration, 5-57
 - starting point, size, illustration, 5-57
 - to program, 5-59
- elbow mold cycle, compensation requirement, 4-7
- elements, sketch, 12-8
- ellipse
 - compensation requirement, 4-6
 - description, 5-43
 - graphic menu, illustration, 5-44
 - programming, 5-44
 - tool compensation, illustration, 5-44, 5-45
- emergency stop
 - button, 2-4
 - reset, 3-1
 - to perform, 3-1
- End N#, parameter, 7-8
- end of main, block, 5-39
- end of program, block, 4-3
- end view, YZ, 7-12
- ending, main programs, 5-40
- ending, subprograms, 5-41
- EndMain, 3-5, 5-40
- EndMill block, to program, 5-64
- endpoint radius arc types,
 - illustration, 4-22
- EndSub, 5-41
- engraving cycle
 - description, 5-60
 - sample program, 5-62
 - to program, 5-60
- ENTER, key, 2-3
- entering, text, 2-7
- entity information
 - DXF miscellaneous menu (F6),
 - description, 15-7
 - off-line, to display, 15-7

to display, DXF miscellaneous menu (F6),
15-7

entry field

optional, defined, 4-4

required, defined, 4-4

types, defined, 4-4

erase, display, 7-11

Erase, parameter, 7-8

error messages, displayed, 2-8

errors, 8-4

EstAngle, 5-99

EstDiam, 5-71, 5-76, 5-78, 5-81

E-STOP

button, 2-4

emergency stop, 3-1

to reset, 3-1

executing, a SetZero block,

illustration, 4-11

exit (F10), DXF converter, 15-7

exit, shut down, 3-1

exiting, software, 13-1

external keyboard, 2-5

F

F1, MESSAGE, 2-8

F1, Offsets, 10-6

F1, RefProg, 10-6

F1, soft key, More,

conversational editor, 6-6

F1, Teach soft key, 4-18

F1, toggle select mode, DXF
converter, 15-6

F1, ToolPro, 5-68

F10, Draw, 7-1

F10, exit, DXF converter, 15-7

F10, Probe, 5-68

F2, ASCII, 2-6

F2, ASCII chart, 2-7

F2, Create, 9-2

F3, Delete, 9-4

F3, Ins, 2-7

F3, layers menu, DXF converter,
15-6

F3, SpinPro, 5-68

F4, Del, 2-8

F4, view menu, DXF converter,
15-6

F5, CalibX, 10-6

F5, Display, 7-2

F5, display menu, DXF
converter, 15-6

F5, Mill, 5-45, 5-68

F6, CalibY, 10-6

F6, miscellaneous menu, DXF
converter, 15-6, 15-7

F6, RMS, 5-42

F7, CalibZ, 10-6

F7, MDI, 3-18

F7, More, 5-45

F8, Calib Z, 10-5

F8, save, DXF converter, 15-6

F9, Misc soft key, 6-6

F9, Parm, 7-2

F9, setup, DXF converter, 15-6

F9, Tool, 10-6

face pocket

graphic menu, illustration, 5-17

face, compensation requirement,
4-7

face, the surface of a part, 5-17

facing cycle

description, 5-17

to program, 5-17

tool approach, illustration, 5-17

features, disabled, off-line, 13-3

Feed, 5-97

FEED, 3-4

feed block

program, using hot keys, 4-14

program, using soft keys, 4-14

feed mode, 3-8

feed moves, feedrate override

setting, 3-4

FEED, hot key, 2-2

Feed, move, 3-15

feedrate

adjusting, 3-8

change active, 3-8

change via FeedU block, 4-14

change via program, 4-14

current, 3-4

FEEDRATE OVERRIDE

range, 3-8

setting, 3-4

switch, illustration, 2-4

to change, 3-8

FeedU block, U-axis, feedrate

change, to program, 4-14

Find, 5-99

FindActive, 5-99

finding, coordinates, 4-1

first block, 4-1

FIXTURE, 3-4

fixture offset

- active, 3-4
- changing, adjust fixture offset table entry, 4-11
- changing, calibrate the fixture offset table, 4-11
- changing, manually enter a value, 4-11
- define, in fixture offset table, 4-2
- locating, illustration, 3-11
- number, 4-9
- to activate, 10-5
- to calibrate, 4-11
- to cancel, 4-10
- to change, 4-11
- to program, 4-9
- work coordinate system, 3-11

fixture offsets table

- define fixture offsets, 4-2
- fixture offset, number, 4-9
- illustration, 4-10
- to activate, 4-10
- Z-offset coordinate, 4-10

fixture offsets, pop-up menu, RefProg, 10-6

floppy disk

- checking for lost data, 9-14
- copy programs, to, 9-6
- formatting, 9-7
- printing from, 9-17
- rename programs, 9-16
- to read, 9-4

formatting, floppy disks, 9-7

four-axis programming

- description, 14-1
- examples, 14-3

frame pocket cycle

- compensation requirement, 4-7
- description, 5-27
- graphic menu, illustration, 5-27
- to program, 5-27

front view, XZ, 7-12

G

GaugeDiam, 5-86

G-code

- conversion number formats, listed, 9-9
- converter, 9-8
- converting, programs to conversational format, 9-8
- equivalents, table, 9-9

geometry calculator

- circle template, description, 12-11
- description, 12-7
- line template, description, 12-10
- point template, description, 12-9
- recall, a position, 12-12
- recalling, values, 12-15
- screen, illustration, 12-7
- templates, listed, 12-8
- to activate, 12-7
- using, 12-8

geometry list

- element numbers, 12-8
- list all, 12-12

graphic menu

- area, 6-2
- basic drilling, illustration, 5-2
- bolthole drill, illustration, 5-9
- boring drilling, illustration, 5-4
- chip breaking cycle, illustration, 5-6
- circular pocket, illustration, 5-25
- circular profile, illustration, 5-21
- elbow milling, illustration, 5-59
- ellipse, illustration, 5-44
- face pocket, illustration, 5-17
- fixture offset, illustration, 4-9
- frame pocket, illustration, 5-27
- hole-mill, illustration, 5-29
- irregular pocket, illustration, 5-34
- line move, illustration, 4-17
- machine home, activate, 4-8
- mold rotation, illustration, 5-54
- pattern drill, illustration, 5-8
- peck drilling, illustration, 5-3
- rapid move, illustration, 4-16
- Rapid Move, illustration, 4-16
- rectangular pocket, illustration, 5-23
- rectangular profile, illustration, 5-19
- RMS SUB, illustration, 5-42
- sample, illustration, 4-3
- spiral, illustration, 5-45
- tapping cycle, illustration, 5-7
- thread mill, illustration, 5-11
- using, 4-3

grid

- parameter, to set, 7-6
- size, to set, 7-6

H

HALTED, 3-4

halted program, clearing, 8-4

halving, display size, 7-10
handshaking, 11-4
handwheel
 pop-up menu, Select HW Axis, illustration, 3-16
 to operate, 3-16
 to select, 3-16
helical threads, to cut, 4-23
hiding, triangle calculator screen, 12-6
highlight, 6-3
highlight bar, 2-5
highlight, functions, 2-7
hold
 a single step run, 8-2
 an auto run, 8-3
 auto mode program, 8-3
 Draw, description, 7-3
 the execution, 8-2
 transmission / receiving operations, 11-7
HOLD, 3-4
Hold (F1), 11-7
HOLD, key, 2-5
hole mill pocket cycle
 description, 5-29
 graphic menu, illustration, 5-29
 to program, 5-29
home block, 4-8
homing
 capability, 3-12
 machines without, tool-length offset, to set, 1-5, 3-10, 3-12, 10-4
 method 1, 4-8
 method 2, 4-8
hot keys
 DXF converter, table, 15-5
 editing, program blocks, More (F1), 6-6
 program
 arc
 using center - endpoint, 4-23
 using center - included angle, 4-26
 using endpoint and radius, 4-22
 dwell, 4-7
 feed block, 4-14
 line move, 4-17
 plane block, 4-13
 rapid move, 4-16
 RPM block, 4-15
 programming, illustration, 2-2

I
Ignore, Draw, description, 7-5
INC, 3-4, 3-9
INCH, 3-4
inch mode, 3-9
inch mode, selection, 3-9
inch, hot key, 2-2
Inch/MM, 6-2
inch/MM mode, changes, 4-4
incremental
 command, 4-5
 mode
 change, 4-4
 description, 3-9
 hot key, 2-2
 to activate, 3-9
 positioning, 1-4
InOutBoss, inside or outside hole or boss center find
 defined, 5-84
 description, 5-93
InOutWeb, inside or outside web or slot center find
 defined, 5-84
 description, 5-95
IN-POSN, 3-4
Ins (F3), 2-7
insert mode, 2-7
inserting, program block, 6-4
inside corner finding, CornerIn, 5-91
inside or outside hole or boss center find, InOutBoss, 5-84, 5-93
inside or outside web or slot center find, InOutWeb, 5-84, 5-95
inside part corner find, CornerIn, 5-84, 5-91
inside profile, 5-19
inside/outside hole/boss finding, InOutBoss, 5-93
inside/outside web finding, InOutWeb, 5-95
inspecting, programmed moves, 7-1
install, RS-232 cable, 11-1
installation, Windows, off-line, 13-1
intersection radius, 1-11

irregular pocket
 (M9367) correct tool diameter, from a subprogram, 5-35
 cycle
 description, 5-31
 programming, 5-31
 cycle, to program, 5-34
 dycle, compensation requirement, 4-7
 execution, 5-33
 graphic menu, illustration, 5-34
island, in rectangles, 5-27
Iso, 3-D view, 7-12
isometric view, 7-12

J

jog
 :1, mode, 3-15
 :10, mode, 3-15
 :100, mode, 3-15
JOG, 3-4
jog mode
 changing, 3-15
 current, 3-4
 descriptions, 3-8
 listed, table, 3-15
jog moves, enable, 3-14
jog moves, from a keyboard, 13-4
JOG, key, 2-3
jogging
 machine (continuous), 3-16
 machine (conventional), 3-15
jumping, program listing, 6-5

K

keyboard
 auxiliary, to display entity information, 15-7
 equivalent keypad keys, table, 13-4
 off-line, 2-5
keypad
 equivalent keyboard keys, table, 13-4
 illustration, 2-1
keys, editing, 2-3
knob pointer, override, 3-8, 3-9

L

LCD, defined, 2-1
left-hand, tool compensation, illustration, 1-6

LenDiamMea, length and diameter, 5-67, 5-71
Length, 5-93, 5-95
length and diameter, LenDiamMea, 5-67, 5-71
length offset, 4-2
length special, LenSpecMea, 5-67, 5-76
LenSpecMea, length special, 5-67, 5-76
limit switch, 3-2, 4-8
line (feed) moves, 4-17
line move
 description, 4-19
 program, using hot keys, 4-17
 program, using soft keys, 4-17
LINE move, hot key, 2-2
line template
 description, 12-10
 purposes, listed, 12-10
line, compensation requirement, 4-6
line-to-arc corner, rounding, 1-12
line-to-line corner, rounding, 1-11
link or new shape, DXF miscellaneous menu (F6), description, 15-7
link test, screen data display, to change, 11-5
link, testing, 11-6
list, all elemental, 12-12
loaded, program, 3-4
loading, program for running, 9-2
locating, tool #, 3-12
logging, to other drives, 9-4
LOOP, 3-4
looping, subprograms, 5-39, 5-41
loops remaining, 3-4

M

M105, dry run, all axes, 4-28
M106, dry run, no Z-axis, 4-28
M900, activate, U-axis synchronization, 4-28
M9367, correct tool diameter, irregular pocket milling, 5-35
machine
 home, 3-10, 4-8
 home, activate, graphic menu, 4-8

- position, 3-5
- position display, description, 3-3
- position display, illustration, 3-3
- power down, 3-10
- setup, 3-1
- software installation, 16-1
- software, DXF converter, 15-1
- without homing, tool-length offset, to set, 1-5, 3-10, 3-12, 10-4
- zero, 4-8
- manual
 - data input, 3-7
 - data input, screen illustration, 3-18
 - machine operation, types, 3-6
 - mode
 - description, 3-6
 - feed, 3-8
 - hot keys, table, 2-2
 - move types, 3-7
 - rapid, 3-8
 - screen illustration, 3-3
 - settings, 3-7
 - to activate, 3-6
 - mode, to activate, 3-1
 - operation, 3-1
 - operation keys
 - illustration, 2-1, 3-14
 - table, 2-3
 - restart, 4-7
 - setting, tool-length offset, from tool page, 10-5
- MANUAL, 3-4
- Manual Data Input. *See* MDI
- manual tool diameter
 - measure for special tools, DiaSpecMea, 5-78
 - preset, DiaSpecMea, 5-67, 5-78
- manual tool length
 - measure for special tools, LenSpecMea, 5-76
 - offset preset, LenSpecMea, 5-67, 5-76
- mark
 - all programs, 9-5
 - an item, 2-7
 - program, 9-5
- marker, edited, 6-2
- math calculator
 - additional function, 12-4
 - basics, 12-2
 - column, of number, 12-3
 - description, 12-1
 - operations, two numbers, 12-3
 - order of operations, 12-4
 - parentheses, using, 12-3
 - recalling, values, 12-13
 - screen illustration, 12-2
 - soft keys, listed, 12-3
 - to activate, 12-1
- MaxDiaAdj, 5-81
- maximize, program storage, 9-3
- maximum memory allocated
 - description, 13-3
 - to change, 13-3
- MaxLenAdj, 5-81
- M-code
 - block, programming, 4-27
 - blocks, programming, 4-27
 - dry run, 4-28
 - functions, listed, 4-27
 - program parts counter, program time, table, 8-7
 - U-axis synchronization, 4-28
- MCODE, hot key, 2-2
- MDI
 - blocks, 3-17
 - cancel, 3-18
 - defined, 3-17, 8-1
 - editor, 3-17
 - execute, a command, 3-18
 - manual mode, 3-7
 - screen, illustration, 3-18
 - U-axis synchronization, 4-28
- MDI (F7), 3-18
- MDI.M program, 3-18
- MeasTyoe, 5-71
- measurement mode, current, 6-2
- memory, available, 6-2
- memory, maximum allocated
 - description, 13-3
 - to change, 13-3
- menus
 - bolthole drill, graphic menu, illustration, 5-9
 - Display, pop-up, illustration, 7-10
 - Draw parameters, pop-up, 7-4
 - ellipse, graphic illustration, 5-44
 - End N#, pop-up, 7-9
 - face pocket, graphic, illustration, 5-17
 - graphic
 - basic drilling, illustration, 5-2
 - boring drilling, illustration, 5-4
 - chip breaking cycle, illustration, 5-6

- circular pocket, illustration, 5-25
- circular profile, illustration, 5-21
- elbow milling, illustration, 5-59
- frame pocket, illustration, 5-27
- hole-mill, illustration, 5-29
- irregular pocket, illustration, 5-34
- mold rotation, illustration, 5-54
- pattern drill, illustration, 5-8
- peck drilling, illustration, 5-3
- rapid move, illustration, 4-16
- rectangular pocket, illustration, 5-23
- rectangular profile, illustration, 5-19
- RMS SUB, illustration, 5-42
- tapping cycle, illustration, 5-7
- pop-up
 - description, 2-5
 - Draw view, illustration, 7-12
 - parameter settings, illustration, 11-3
 - pocket, illustration, 5-16
 - recall values, illustration, 12-13
 - select type of calculator, hot key, illustration, 2-3
 - select type of calculator, hot key, to activate, 12-1, 12-5, 12-7
 - select type of calculator, illustration, 12-1
- Software Options, 3-1
- spiral, graphic menu, illustration, 5-45
- Start N#, pop-up, 7-9
- thread mill, graphic menu, illustration, 5-11
- message
 - area, description, 3-3
 - area, illustration, 3-3
 - displayed, 2-8
 - error, displayed, 2-8
 - priority, 2-8
 - storage, 2-8
- MESSAGE (F1), 2-8
- metric mode, selection, 3-9
- Mill (F5), 5-45
- mill (sync-on), programming
 - example, 14-5, 14-6
- mill cycle
 - description, 5-64
 - sample program, 5-65
 - to program, 5-64
- Mill, F5, 5-68
- millimeter mode, selection, 3-9
- millimeters, hot key, 2-2
- millimeters, MM mode, 3-9
- milling, cavities, 5-47
- milling, cores, 5-47
- minutes to decimal, conversion
 - formula, 14-1
- mirroring and scaling,
 - subprograms, 5-42
- mirroring, subprograms, 5-39
- Misc, soft keys, 6-3
- miscellaneous menu, DXF
 - converter, F6, 15-6, 15-7
- MM, 3-4
- MM mode, 3-9
- modal
 - compensation requirement, 4-6
 - functions, 3-7
 - moves, programming, 4-17
 - setting, 4-1
- mode selection, to activate, 3-9
- mode, changes, 4-4
- mold cycles, 5-47
- mold cycles, listed, 5-47
- mold rotation
 - around X and Y axes
 - small radius, 5-48
 - axes of rotation, illustration, 5-50
 - categories, listed, 5-47
 - cutting a cavity, illustration, 5-51
 - cycle, compensation requirement, 4-7
 - graphic menu, illustration, 5-54
 - programming, 5-47
 - rotation around the Z-axis, 5-53
 - rotations around X and Y axes
 - large radius, 5-52
 - XY rotations about Z, illustration, 5-51
 - XY symmetrical axis, illustration, 5-47
 - Z-axis, illustration, 5-53
- MoldRot block, to program, 5-54
- More (F7), 5-45
- More pop-up menu, to display, 6-6
- motion mode
 - defined, 8-1
 - description, 8-2
 - Draw, 7-7
- move
 - absolute zero in a program, 4-11
 - angle, specifying, illustration, 5-33
 - compensation requirements, table, 4-6
 - direction, determining, 5-32
 - mode, choices, 3-7
 - mode, current, 3-4

orientation, illustration, 4-19
toward a part, 4-2
moving, the machine with servos
off, 3-6

N

negative
radius value, 4-22
signs, 4-4
nesting subprograms, 5-39
New Blocks, 6-4
new program, creating, 9-2
new shape, DXF miscellaneous
menu (F6), description, 15-7
no move blocks, description, 4-4
Nominal probe stylus diameter,
5-70
non-modal arc center points,
13-7
non-synchronous rotary,
description, 14-2
number of parts, counter, 8-6

O

off-line
entity information, to display, 15-7
keyboard, 2-5
program file directory utilities, 13-6
tool page utilities, 13-6
passwords, listed, 13-1
software
DXF converter, 15-1
installation, 13-1
Windows
icon, set up, 13-2
to install, 13-1
to run, 13-2
Offset, 5-88, 5-90, 5-92, 5-94,
5-96
Offsets (F1), 10-6
offsets, tool-length, 1-5
one-shot, moves, 3-17
operating mode, current, 3-4
operations, two numbers, 12-3
operator keys, illustration, 2-1
operator keys, table, 2-4
operator prompts, 2-6
optional, entry fields, 4-4
order of operations, math
calculator, 12-4
organize, the tooling, 4-2

organizing programs, containing
subprograms, 5-40
output format pop-up menu,
illustration, 15-16
outside corner finding,
CornerOut, 5-89
outside part corner find,
CornerOut, 5-84, 5-89
outside profile, 5-21
outside profile ramp moves,
illustration, 5-19
overriding, programmed spindle
RPM, 3-9
OvrFstFeed, 5-72
OvrMedFeed, 5-72, 5-76, 5-78,
5-82
OvrRPM, 5-72, 5-76, 5-79, 5-82
OvrSlwFeed, 5-72, 5-76, 5-78,
5-82

P

P/N 70000499, 3000M CNC
Setup Utility Manual,
referenced, 4-28, 5-66, 16-1
paging through, program listing,
6-5
parameter settings, pop-up
menu, illustration, 11-3
parameters
protected, to access, 13-1
viewing, 7-4
parentheses, using, 12-3
parity, to set, 11-4
Parms (F9), 7-2
part programs, developing, 4-1
part programs, display, 9-4
part zero
location, 4-1
resetting, 4-11
setting, 1-3
Part Zero position, 3-10
PARTS, 3-5
parts counter, CNC, description,
8-6
passwords, off-line, listed, 13-1
Pattern Drill. See drill pattern
pattern drill cycle
description, 5-8
graphic menu, illustration, 5-8
pause, any running program, 2-5

- peck drilling cycle
 - description, 5-3
 - graphic menu, illustration, 5-3
 - to program, 5-3
- pending, messages, 2-8
- permanent, absolute zero, 3-10
- permanent, memory, 4-1
- plane
 - active, 6-2
 - block
 - program, using hot keys, 4-13
 - program, using soft keys, 4-14
 - choices, 3-7
 - illustration, 1-14
 - selecting, for arcs, 4-21
 - selection, description, 1-14
 - selection, ellipse and spiral, 5-43
- PLANE, hot key, 2-2
- pocket
 - cycle, description, 5-16
 - cycle, tool compensation, 5-16
 - cycles, listed, 5-16
 - pop-up menu, illustration, 5-16
 - with islands
 - description, 5-36
 - DXF, program example, 15-18
 - subroutines, example, 5-36
 - using DXF, 15-16
- point identification, tools, 12-8
- point of rotation, 5-42
- point template
 - description, 12-9
 - purposes, listed, 12-9
- polar coordinates, description, 1-3
- pop-up menus
 - description, 2-5
 - editing, program blocks, More (F1), 6-6
 - miscellaneous menu, DXF converter, F6, 15-6, 15-7
 - More (F1), illustration, 6-7
 - More (F7) engraving cycle, illustration, 5-60
 - output format, 15-16
 - program listing, 15-17
 - RefProg, fixture offsets, illustration, 10-6
 - Select HW Axis, illustration, 3-16
- pop-up, ASCII chart, illustration, 2-6
- position
 - coordinates, 3-3
 - defining, 1-2
 - display, 3-5
 - display mode, 3-4
 - display options, illustration, 3-5
 - display, illustration, 3-3
 - locating, illustration, 1-2
 - mode, active, 3-9
 - mode, choices, 3-7
 - mode, current, 3-4
 - prompt, 4-4
- positive, radius value, 1-11, 4-22
- POSN, 3-4, 3-5
- power up, 3-10, 3-12
- powering
 - off, 3-1
 - on, the CNC, 3-1
- presetting, X or Y-axis, 3-10
- presetting, Z-axis, 3-13
- primary display area
 - description, 3-3
 - illustration, 3-3
 - labels, 3-4
- printing
 - from floppy drives, 9-17
 - programs, 9-7
- probe
 - calibration, CalibTIPrb, 5-67, 5-69
 - cycles, conversational programming, description, 5-66
 - orientation, description, 5-70
 - spindle cycles
 - conversational programming, description, 5-66
 - description, 5-83
 - listed, 5-84
 - tool cycles
 - description, 5-66, 5-68
 - designations, 5-67
 - listed, 5-68
- Probe, F10, 5-68
- ProbeMove, protected
 - positioning move
 - defined, 5-85
 - description, 5-97
- program
 - arc
 - using center - endpoint, 4-23
 - using center - endpoint, hot keys, 4-23
 - using center - endpoint, soft keys, 4-24
 - using center - included angle, 4-25

- using center - included angle, hot keys, 4-26
- using center - included angle, soft keys, 4-26
- using endpoint - radius, hot keys, 4-22
- using endpoint - radius, soft keys, 4-21, 4-22
- basics, 4-1
- block
 - deleting, 6-3
 - editing, 6-4
 - single-step execution of, 8-2
 - to insert, 6-4
- blocks
 - current, number, 3-4
 - displaying, 9-4
 - editing, More (F1), 6-6
 - writing, 4-3
- cancel, Draw, 7-3
- copy, other directories, 9-16
- copying, to floppy disk, 9-6
- create, new program, 9-2
- definition, 1-1
- delete, groups, 9-6
- dwel, 4-7
- dwel, using hot keys, 4-7
- dwel, using soft keys, 4-7
- editing, 6-1
- editor, 4-3
- elbow milling cycle, 5-59
- ending, main, 5-40
- file utilities, 9-3
- fragment, checking disk, 9-14
- fragments, 9-14
- G-Code, to conversational format, 9-8
- halted, clearing, 8-4
- listing, 6-2
- listing pop-up menu, illustration, 15-17
- loading, 9-2
- management, description, 9-1
- mark, 9-5
- mark all, 9-5
- move, using XY location, radii, or angles, 4-20
- name, 3-4, 6-2
- names, choosing, 9-2
- parts counter, description, 8-6
- parts counter, illustration, 8-6
- parts counter, M-codes used with, 8-7
- position display, 3-5
- printing, 9-7
 - printing, from floppy drives, 9-17
 - rename, 9-7, 9-16
 - restore, 9-6
 - return, to machine home, 4-8
 - run time, timer, 3-5
 - running, 8-1
 - running, one step at a time, 8-1
 - run-time timer, description, 8-6
 - select for running, 8-1
 - select, for editing, 9-3
 - sending, description, 11-6
 - SetZero block, 4-13
 - size, 6-2
 - storage, maximize, 9-3
 - timer, description, 8-6
 - timer, illustration, 8-6
 - timer, M-codes used with, 8-7
 - to delete, 9-4
 - to list contents, 9-4
 - to receive, 11-7
 - undeleting, 9-6
 - unmark, all marked, 9-5
 - unmark, marked, 9-5
 - using real-time Draw, while running programs, 8-5
 - viewing, with Draw, 7-1
 - writing, 4-1
- program directory
 - description, 9-1
 - display, changing, 9-2
 - illustration, 9-1
- program editor
 - activate from manual screen, 6-1
 - activate from program directory, 6-1
 - screen, illustration, 6-2
- program listing
 - description, 3-3
 - illustration, 3-3
 - jumping, 6-5
 - paging through, 6-5
 - scrolling, 6-4
- programmed
 - hold, 3-4
 - path, 1-6
 - RPM, 3-4
 - RPM, adjust, 3-9
 - spindle RPM, 3-9
- programming
 - an elbow milling cycle, 5-55
 - an ellipse, 5-44
 - arcs, 4-21

- basic drill cycles, 5-2
 - circular pocket cycle, 5-25
 - circular profile cycles, 5-21
 - concepts, 1-1
 - conventions, rotary/U-axis, 14-2
 - drill patterns, 5-8
 - examples
 - 4-axis, description, 14-3
 - drill (sync-off), 14-3
 - mill (sync-on), 14-5, 14-6
 - facing cycles, 5-17
 - feed moves, 4-17
 - feedrate, change, 4-14
 - fixture offsets, 4-9
 - four-axis, description, 14-1
 - frame pocket cycles, 5-27
 - from the part, 4-18
 - hole, 5-29
 - hole mill pocket cycle, 5-29
 - hot keys, illustration, 2-1, 2-2
 - inch/MM mode changes, 4-4
 - irregular pocket cycle, 5-34
 - irregular pocket cycles, 5-31
 - line (feed) moves, 4-17
 - line moves, 4-17
 - M-code blocks, 4-27
 - modal moves, 4-17
 - mold rotation, 5-47
 - MoldRot block, 5-54
 - plane, change, 4-13
 - process, 4-1
 - rapid move, 4-16
 - rectangular pocket cycles, 5-23
 - rectangular profile cycles, 5-19, 5-20
 - single moves, 3-17
 - spindle RPM, 4-15
 - spiral, 5-45
 - the part's edge, 1-6
 - tool change, 4-5
- prompts, 3-3
- prompts, operator, 2-6
- protected parameters, to access, 13-1
- protected positioning move, ProbeMove, 5-85, 5-97
- protected probe positioning, ProbeMove, 5-97
- R**
- radius center, 1-11, 1-12
- ramp move, 1-7
- ramp moves, 4-6
- ramping, into a compensated move, 1-8
- range of travel, 3-10
- RAPID, 3-4
- rapid mode, 3-8
- rapid move
 - description, 4-19
 - feedrate override setting, 3-4
 - graphic menu, 4-16
 - override range, 3-8
 - program, using hot keys, 4-16
 - program, using soft keys, 4-16
 - speed adjusting, 3-8
 - speed override, 3-8
 - viewing, 7-6
- RAPID move, hot key, 2-2
- rapid, compensation
 - requirement, 4-6
- Rapid, mode, 3-15
- real-time Draw
 - mode, 7-1
 - using while running programs, 8-5
- real-time mode, Draw screen, 8-5
- recall values
 - geometry calculator, 12-15
 - math calculator, 12-13
 - one calculator, to another, 12-15
 - pop-up menu, illustration, 12-13
 - triangle calculator, 12-14
- recalling, calculator values, 12-13
- receive area, to clear, 11-6
- receive mode, data control codes, to use, 11-10
- receiving
 - display, to set, 11-7
 - operations, to hold, 11-7
 - program, 11-7
 - programs, through RS-232 communications, 8-7
- recover, deleted program, 9-6
- rectangular pocket cycle
 - compensation requirement, 4-7
 - description, 5-23
 - graphic menu, illustration, 5-23
 - to program, 5-23
- rectangular profile cycle
 - compensation requirement, 4-7
 - description, 5-19
-

- graphic menu, illustration, 5-19
 - to program, 5-20
 - reference point, 12-9
 - RefProg
 - fixture offsets, pop-up menu, 10-6
 - offset, description, 10-6
 - RefProg (F1), 10-6
 - reminders, 3-3
 - rename
 - program, 9-7
 - programs, 9-16
 - repeat cycle
 - description, 5-62
 - sample program, 5-63
 - to program, 5-62
 - RepeatMeas, 5-94
 - repetitive
 - drilling cycle, subprograms, 5-39
 - operations, 5-16
 - operations, subprograms, 5-39
 - required, entry fields, 4-4
 - Reset Rotary at 360, parameter, 14-2
 - reset, after stop, 3-1
 - resetting, absolute zero, 4-11
 - resetting, the servos, 3-1, 3-2
 - restarting
 - a program, 8-2
 - Draw automatic, 7-8
 - spindle, 3-2
 - restore original plane, 4-13
 - restore, deleted program, 9-6
 - Resume (F2), 11-7
 - return, to machine home, 4-8
 - reviewing, messages, 2-8
 - right triangle calculator. See triangle calculator
 - right-hand, tool compensation, illustration, 1-7
 - RMS
 - description, 5-42
 - SUB, graphic menu, illustration, 5-42
 - subprogram, entry fields, 5-42
 - subprogram, to call, 5-42
 - subprograms, 5-42
 - RMS (F6), 5-42
 - rotary
 - axis
 - programming conventions, 14-2
 - programming, in absolute, 14-2
 - programming, in incremental, 14-2
 - non-synchronous, description, 14-2
 - synchronous, description, 14-2
 - rotating, mirroring, and scaling. See RMS
 - rotating, subprograms, 5-39
 - rotation
 - around the Z-axis, 5-53
 - around X and Y axes
 - large radius, 5-52
 - small radius, 5-48
 - Z-axis rotation, start and end angles, 5-54
 - rough and finish cycles, subprograms, 5-39
 - rounding
 - arc-to-arc corner, 1-12
 - corner, 1-11
 - line-to-arc corner, 1-12
 - line-to-line corner, 1-11
 - RPM, 3-14, 4-15
 - RPM block
 - program, using hot keys, 4-15
 - program, using soft keys, 4-15
 - RPM for calibration and tool measurement, 5-70, 5-72, 5-76, 5-79, 5-82
 - RPM, spindle, 3-4, 10-2
 - RS 274D, 9-8
 - RS-232
 - cable, installing, 11-1
 - communication connector, illustration, 11-1
 - communications, 8-7
 - Run parameter, 7-8
 - running
 - program, one step at a time, 8-1
 - programs, 8-1
 - RUNNING, 3-4
- ## S
- S.STEP, 3-4
 - S.Step mode, 3-3
 - S.Step, Draw, 7-7
 - saving, a program block, 4-3
 - saving, edits, 6-3
 - scale, Draw, display size, 7-11
 - scaling, subprograms, 5-39, 5-42
 - scientific, calculator, 12-1
 - screen clutter, eliminate, 7-6
 - screen saver, description, 2-6
 - screens
 - communication, illustration, 11-2

- DNC, illustration, 11-8
- Draw (real-time mode), 8-5
- Draw, simulation mode, 7-3
- engraving cycle, illustration, 5-61
- geometry calculator, illustration, 12-7
- manual
 - data input, illustration, 3-18
 - manual mode, illustration, 3-3
 - math calculator, illustration, 12-2
 - mill cycle, illustration, 5-64
 - program editor, illustration, 6-2
 - repeat cycle, illustration, 5-62
 - system information, illustration, 9-15
 - Test Link, illustration, 11-5
 - tool page, illustration, 10-2
 - triangle calculator, illustration, 12-5
- scrolling, program listing, 6-4
- SEARCH
 - to select, auto mode, starting block, 8-4
 - using to select, single-step mode, starting block, 8-2
- SearchDir, 5-88
- searching, blocks, for words or numbers, 6-4
- SearchQuad, 5-89, 5-91
- secondary display area
 - absolute/incremental mode, 3-9
 - description, 3-3
 - illustration, 3-3
 - labels, 3-4
- secondary, status area, 3-9
- seconds remaining in a dwell, 3-4
- seconds to degrees, conversion formula, 14-1
- selecting
 - communication port, 11-3
 - Draw view, 7-12
 - program for editing, 9-3
 - program for utilities, 9-3
 - programs, 8-1
- send mode, data control codes, to use, 11-10
- sending
 - program, 11-6
 - programs, through RS-232 communications, 8-7
- SERVO RESET, key, 2-4
- servos
 - activating, 3-2
 - disengage, 3-1
 - emergency stop, 3-1
 - reactivating, 3-1
 - reset, 3-2
- set plane graphic menu, 4-14
- set shift, DXF miscellaneous menu (F6), description, 15-7
- setting
 - a feedrate, 3-8
 - absolute zero, 3-10
 - automatic, tool-length offset, from tool page, 10-5
 - baud rate, 11-4
 - data bits, 11-4
 - data type, 11-4
 - manually, tool-length offset, from tool page, 10-5
 - parity, 11-4
 - part zero, 1-3
 - receiving display, 11-7
 - software, 11-4
 - stop bits, 11-4
 - test link display modes, 11-5
 - tool change position, 3-12
 - tool-length offset, 10-4
 - tool-length offset, for ball-end mill, 10-5
 - transmission display, 11-7
- Setup Utility, 16-1
- SetZero block, 4-11
- SetZero block, to program, 4-13
- shape
 - CAM, extension, 15-11
 - to delete, 15-3
- shutting down, CNC, 3-1
- Side, 5-93, 5-95
- side view, YZ, 7-12
- sign change, hot key, 2-2
- simulation mode, Draw, 7-1
- single surface measure/edge
 - find, EdgeFind, 5-84, 5-88
- single-step mode
 - auto mode, switch to, 8-3
 - defined, 8-1
 - description, 8-2
 - Draw, 7-7
 - MOTION mode, 8-1
 - program, to run, 8-1
 - S.STEP mode, 8-1
 - starting block, select
 - using arrows, 8-2
 - using SEARCH, 8-2
 - switch, to motion mode, 8-2

- sketch, delete, elements, 12-11
- sketch, elements, 12-8
- skew error find, SkewComp, 5-98
- skew error or angle find, SkewComp, 5-85, 5-98
- SkewComp, skew error or angle find
 - defined, 5-85
 - description, 5-98
- soft keys
 - DXF converter, descriptions, 15-6
 - labels, 6-3
 - labels, description, 2-5, 3-3
 - labels, illustration, 3-3
 - math calculator, listed, 12-3
 - More (F1), editing, program blocks, 6-6
 - program
 - arc
 - using center - endpoint, 4-24
 - using center - included angle, 4-26
 - using endpoint - radius, 4-22
 - dwll, 4-7
 - feed block, 4-14
 - line move, 4-17
 - plane block, 4-14
 - rapid move, 4-16
 - RPM block, 4-15
 - U-axis feedrate change, 4-14
 - using from a keyboard, 13-4
- software
 - basics, 2-5
 - installation, machine, 16-1
 - option kit, to install, 16-1
 - options, menu, 16-1
 - parameters, 11-1
 - setting, 11-4
 - to exit, 13-1
 - upgrades, 16-1
- Software Options Menu, 3-1
- solved triangle, 12-6
- SPDL FWD, key, 2-4
- SPDL OFF, key, 2-4
- SPDL REV, key, 2-4
- spindle
 - direction, 10-3
 - override, setting, 3-4
 - probe calibration
 - CalibPtPrb, 5-86
 - cycle, CalibPtPrb, 5-84, 5-86
 - probe cycles
 - conversational programming, description, 5-66
 - description, 5-83
 - listed, 5-84
 - RPM, 3-4, 10-2
 - RPM, enter, 4-15
 - RPM, hot key, 2-2
 - RPM, programming, 4-15
 - RPM, to activate, 3-14
 - speed, 3-9
 - status, 3-4
- SPINDLE, 3-4
- SPINDLE FORWARD, key, 2-4
- SPINDLE OFF, key, 2-4
- SPINDLE OVERRIDE, 3-9
- SPINDLE OVERRIDE range, 3-9
- SPINDLE OVERRIDE switch, illustration, 2-4
- SPINDLE REVERSE, key, 2-4
- SpinPro, F3, 5-68
- spiral
 - compensation requirement, 4-6
 - geometry, 5-45
 - graphic menu, illustration, 5-45
 - inward, 5-45
 - outward, 5-45
 - programming, 5-45
- standard, starting height, 4-2
- Start N#, parameter, 7-8
- START, key, 2-5
- starting
 - Draw, 7-2
 - spindle, 3-2
 - subprograms, 5-41
- starting block, select
 - auto mode, using SEARCH, 8-4
 - single-step mode, using SEARCH, 8-2
 - using arrow keys, 8-4
 - using arrows, 8-2
- stepover
 - face pocket, value, 5-18
 - facing cycle, 5-17
 - move direction, 5-32
 - move orientation, illustration, 5-31
 - orienting from subprogram, illustration, 5-32
- Stepover values, 5-18
- stop bits, to set, 11-4
- stop, emergency, 3-1

storing, results, triangle
 calculator, 12-5, 12-6
straight moves, 4-16
Sub block, to program, 5-41
Sub#, 5-41
Sub, soft keys, 6-3
subdirectory, create, 13-6
subprogram
 call, 5-40
 call a loop, 5-41
 calling, 5-40
 description, 5-39
 ending, 5-41
 example, 5-39
 looping, 5-41
 M9367, correct tool diameter, irregular
 pocket milling, 5-35
 orientation, illustration, 5-48
 program structure, 5-39
 programs, containing, 5-40
 repetitive drilling cycle, 5-39
 RMS, to call, 5-42
 rotating, mirroring, and scaling, 5-42
 rough and finish cycles, 5-39
 specific, illustration, 5-49
 to program, 5-41
 to write, 5-40
 Z-axis rotation, illustration, 5-53
subroutines, pockets with
 islands, example, 5-36
switching, selections, 2-6
synchronous rotary, description,
 14-2
system, information, displaying,
 9-15

T

table motion, 1-1
tangent radius, description, 1-11
tapping cycle
 description, 5-7
 graphic menu, illustration, 5-7
 to program, 5-7
Teach (F1) soft key, 4-18
teach mode, activate, from edit
 screen, 4-18
teach mode, description, 4-18
teach move, to program, 4-18
templates
 circle, description, 12-11
 line, description, 12-10

 point, description, 12-9
test link
 display modes, to set, 11-5
 screen, to activate, 11-5
testing, data link, 11-4
testing, link, 11-6
text
 deleting, 2-8
 Draw, on or off, 7-4
 editor, rules for CNC blocks, 13-7
 entering, 2-7
thread milling cycle
 description, 5-11
 graphic menu, illustration, 5-11
 sample program, 5-15
 to program, 5-11
TIMER, 3-5
timer, CNC, description, 8-6
toggle
 endpoints, DXF miscellaneous menu (F6),
 description, 15-7
 hot key, 2-2
 key, 2-6
tool
 # block, 4-5
 #0, Z0, 3-12
 center, 1-6
 change position, setting, 3-12
 compensation, 1-6
 compensation, Draw, description, 7-5
 compensation, ellipse, illustration, 5-44
 diameter, 3-4, 4-2
 diameter compensation
 activating, 4-6
 description, 1-6
 plane you select, 1-14
 tool change, 4-5
 diameter, correct for irregular pocket
 milling, M9367, 5-35
 Draw, on or off, 7-5
 edge, 1-6
 management, description, 10-1
 motion, 1-1
 number, 4-5, 10-2
 number, to find, 10-3
 to activate, 4-5
 to change, 4-5
TOOL, 3-4
Tool (F9), 10-6
tool approach, facing cycle,
 illustration, 5-17

- tool breakage, length and
 - diameter wear detection,
BrkWearDet, 5-67, 5-81
 - tool length and diameter offset
 - preset, LenDiamMea, 5-67,
5-71
 - tool page
 - cursor, description, 2-7
 - definition, 10-1
 - description, 10-2
 - features, listed, 10-2
 - row, to clear, 10-4
 - RPM entry, 4-15
 - screen, illustration, 10-2
 - single value, to adjust, 10-4
 - single value, to clear, 10-4
 - specific tool number, to find, 10-3
 - to enter, 10-1
 - using, 10-3
 - utilities, off-line, keyboard, 13-6
 - values, changing, 10-3
 - tool probe calibration cycle,
CalibTIPrb, 5-67, 5-69
 - tool probe cycles
 - description, 5-66, 5-68
 - designations, listed, 5-67
 - listed, 5-68
 - Tool#, 5-41
 - TOOL, hot key, 2-2
 - ToolComp
 - command, 4-6
 - Draw, description, 7-5
 - tool-length offset
 - active, 3-10
 - automatic setting, from tool page, 10-5
 - description, 1-5
 - machines without homing, to set, 1-5,
3-10, 3-12, 10-4
 - manually setting, from tool page, 10-5
 - setting, 10-4
 - setting, for ball-end mill, 10-5
 - settings, 3-12
 - ToolPro, F1, 5-68
 - Top, 5-86, 5-89, 5-91, 5-93,
5-95, 5-99
 - top view, XY, 7-12
 - Total Blks Rec, description, 11-8
 - tracking, mode changes, 6-3
 - transmission display, to set, 11-7
 - transmission operations, to hold,
11-7
 - triangle calculator
 - hide, the screen, 12-6
 - recalling, values, 12-14
 - screen illustration, 12-5
 - sides and angles, to solve for, 12-6
 - storing, results, 12-5, 12-6
 - to activate, 12-5
 - using, 12-6
 - troubleshoot, finished programs,
4-2
 - troubleshooting, DXF converter,
15-5
 - typeover mode, 2-7
- ## U
- U negative, key, 2-3
 - U positive, key, 2-3
 - U-axis
 - feedrate, change via FeedU block, 4-14
 - four-axis systems, 14-1
 - hot key, 2-2
 - programming
 - conventions, 14-2
 - in absolute, 14-2
 - in incremental, 14-2
 - synchronization
 - M900, activate, 4-28
 - M-codes, listed, 4-28
 - undeleting, a program, 9-6
 - unit block
 - description, 4-4
 - to program, 4-4
 - UNIT, hot key, 2-2
 - units
 - conversion, 12-1
 - display setting, 3-9
 - mode, choices, 3-7
 - mode, current, 3-4, 6-2
 - mode, to change, 3-9
 - of measurement, 3-9
 - unmark
 - all, marked programs, 9-5
 - marked programs, 9-5
 - unsaved edits, to cancel, 6-3
 - Update, 5-82
 - Use, Draw, description, 7-5
 - USER listing, 9-1
 - using, comments, 6-5
 - using, data control codes, 11-9
 - utilities, program file, 9-3
-

V

view, Draw, selecting, 7-12
viewing, programs with Draw,
7-1

W

Width, 5-93, 5-95
window, sized, 7-11
Windows, off-line, software
icon, set up, 13-2
to install, 13-1
to run, 13-2
work coordinate offset, axis, 10-6
work coordinate system, 3-11
work offset, Z-axis, 10-7
writing
comment block, 6-5
program blocks, 4-3
programs, 4-1

X

X negative, key, 2-4
X positive, key, 2-4
X0, Y0, Z0 Position, 1-2
X-axis
description, 1-2
hot key, 2-2
part zero, position, 3-10
Xincr, 5-41
XMinus, 5-88
XMinusYMinus, 5-89, 5-91
XMinusYPlus, 5-89, 5-91
Xoff, 11-4
X-offset coordinate, 4-9
Xon, 11-4
XPlus, 5-88
XPlusYMinus, 5-89, 5-91
XPlusYPlus, 5-89, 5-91
XStepover, 5-18
XY plane, 4-13
XY, top view, 7-12
XZ, front view, 7-12

Y

Y negative, key, 2-3
Y positive, key, 2-3

Y-axis

description, 1-2
hot key, 2-2
part zero, position, 3-10

Yincr, 5-41

YMinus, 5-88

Y-offset coordinate, 4-9

YPlus, 5-88

YStepover, 5-18

YZ, end view, 7-12

YZ, side view, 7-12

Z

Z final pick, SLOW feedrate,
5-72, 5-76, 5-78, 5-82

Z first pick

FAST feedrate, 5-72

MEDIUM feedrate, 5-69, 5-72, 5-76, 5-78,
5-82

Z negative, key, 2-3

Z positive, key, 2-3

Z0 quill position, 1-5

Z-axis

description, 1-2

drive system, disengage, 3-19

drive system, to engage, 5-1

hot key, 2-2

part zero, 1-5, 3-10

position display, 1-5

presetting, 3-13

Z-axis, work offset, 10-7

zero crossing, 4-8

zero degree reference, 1-10

ZFeed, 5-41

Zincr, 5-41

ZMinus, 5-88

Zmode

description, 5-1

ThreeAxes, 3-19, 5-1

TwoAxAuto, 5-1

TwoAxes, 3-19

TwoAxStep, 5-1

Z-offset coordinate, 4-10

zoom, Draw, 7-11

zoom, in or out, 7-1

ZPlus, 5-88

HEIDENHAIN CORPORATION

333 East State Parkway

Schaumburg, IL 60173-5337 USA

 +1 (847) 490-1191

 +1 (847) 490-3931

E-Mail: info@heidenhain.com

www.heidenhain.com
