ANILAM

3000M CNC
Programming and Operations
Manual for
Two-Axis Systems

Warranty

ANILAM warrants its products to be free from defects in material and workmanship for one (1) year from date of installation. At our option, we will repair or replace any defective product upon prepaid return to our factory.

This warranty applies to all products when used in a normal industrial environment. Any unauthorized tampering, misuse or neglect will make this warranty null and void.

Under no circumstances will ANILAM, any affiliate, or related company assume any liability for loss of use or for any direct or consequential damages.

The foregoing warranties are in lieu of all other warranties expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose.

The information in this manual has been thoroughly reviewed and is believed to be accurate. ANILAM reserves the right to make changes to improve reliability, function, or design without notice. ANILAM assumes no liability arising out of the application or use of the product described herein.

Section 1 - CNC Programming Concepts

Section 2 - CNC Console and Software Basics

Section 3 - Manual Operation and Machine Setup

Section 4 - Writing Programs

Section 5 - Programming Canned Cycles, Ellipses, and Spirals

Section 6 - Editing Programs

Section 9 - Program Management

Section 10 - Tool Management

Section 11 - Communications and DNC

Section 12 - Calculators

Section 13 - Off-line Software

Section 14 - DXF Converter Feature

Section 15 - CNC Software

Section 1 - CNC Programming Concepts

Programs

This manual describes CNC programming and operations for 3000M two-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.)
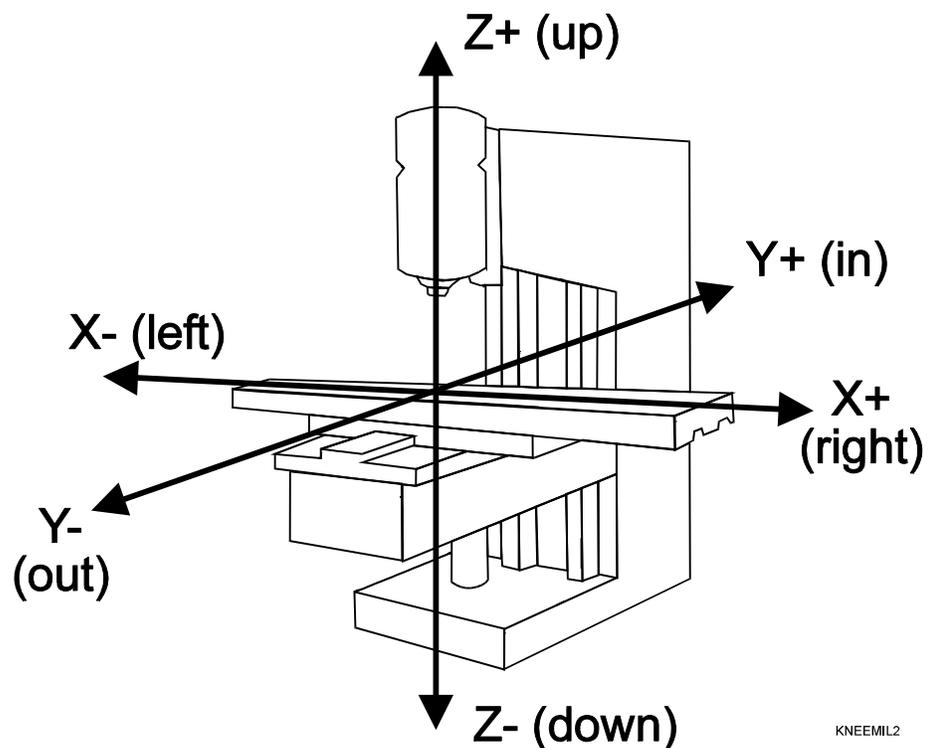
Figure 1-1, Mill Axes of Motion (Tool Motion Orientation)

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

NOTE: Z Moves are manual.
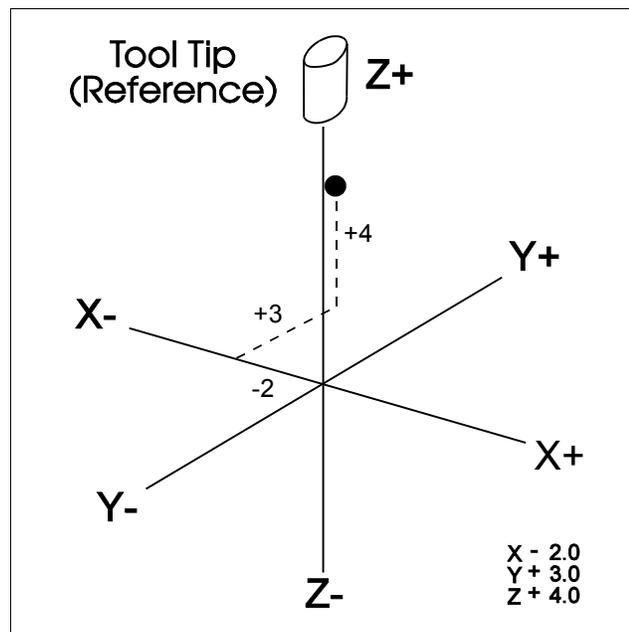
Figure 1-2, Locating Positions

Polar Coordinates

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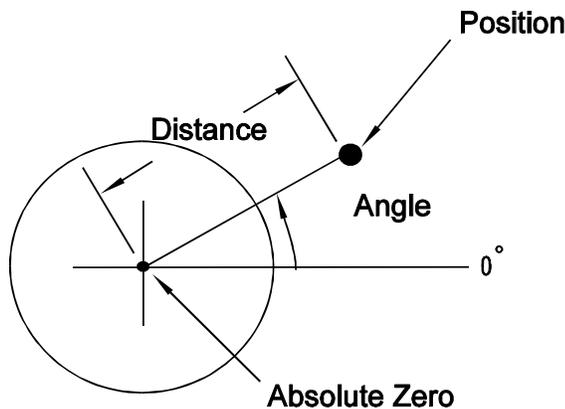
POLAR

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 you set the Absolute Zero Reference at a position that makes it easy use the dimensions from the blueprint. This is also called setting the Part Zero.
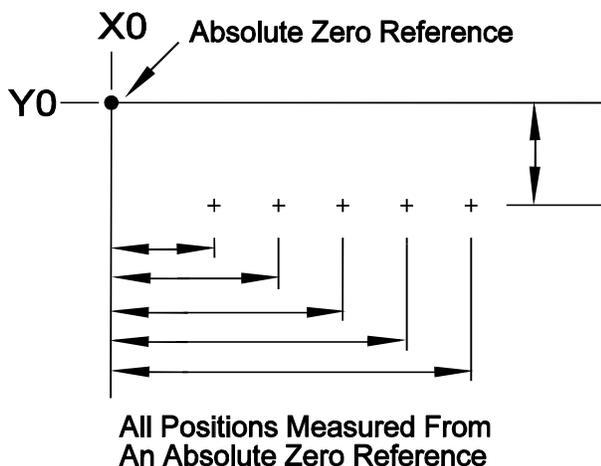
ABSOLUTE

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.
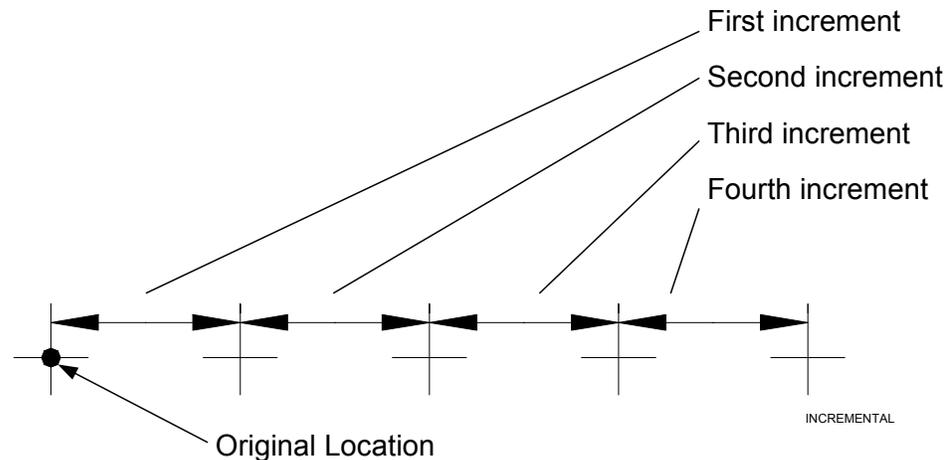
Figure 1-5, Incremental Positioning

Tool-Length Offsets

You set the Z0 position of the quill, from which the CNC applies Tool Length Offsets. Usually, it is the fully retracted position of the quill.

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, Tool Length Offset.

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. Manually move to a position 0.5-in. into the work.
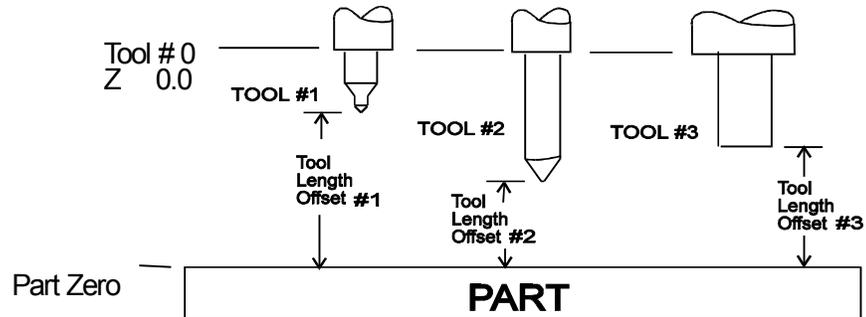
Figure 1-6, Tool Length Offset

Tool Diameter Compensation

When tool compensation is not active, the CNC positions the tool's 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 of CNC 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 enables you to program the coordinates along the part profile. You do not need to adjust 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 only be used 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, Left-Hand Tool Compensation.
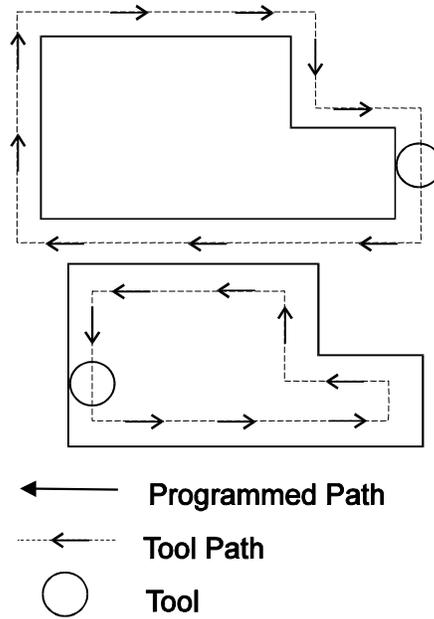
Figure 1-7, Left-Hand Tool Compensation

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**.
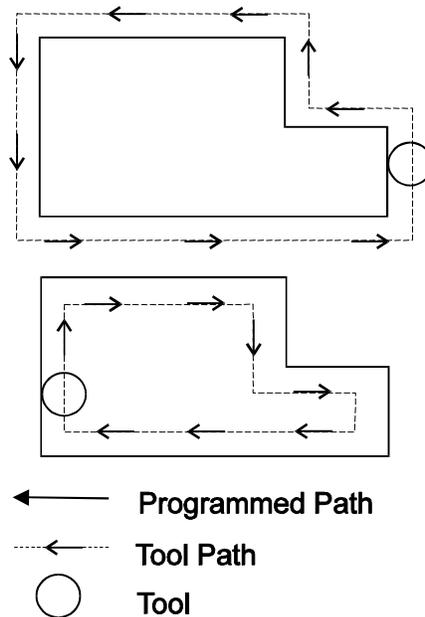
Figure 1-8, Right-Hand Tool Compensation

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**.
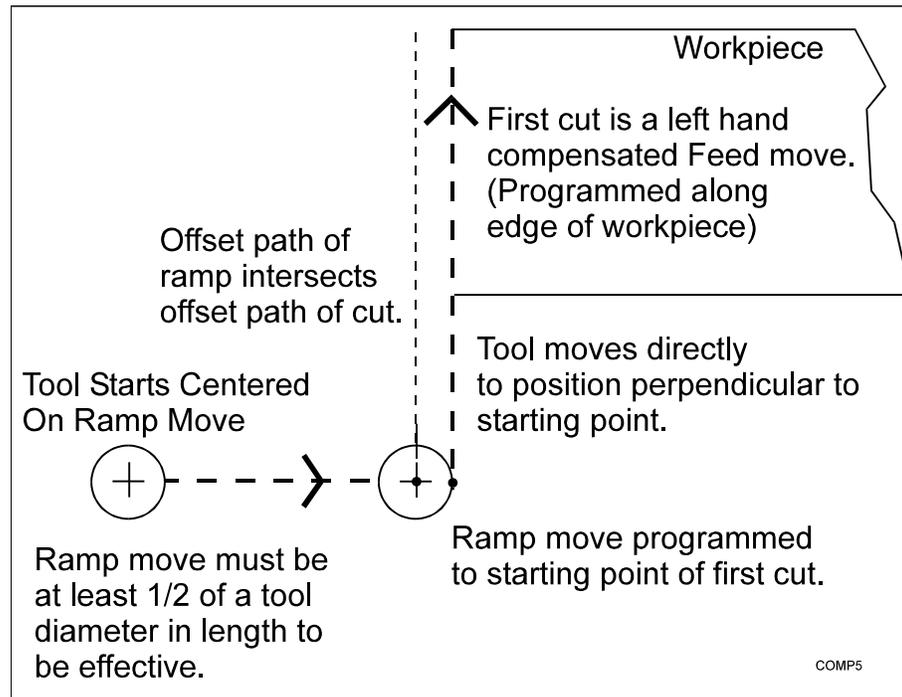
Figure 1-10, Ramping into a Compensated Move

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

ANILAM

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.
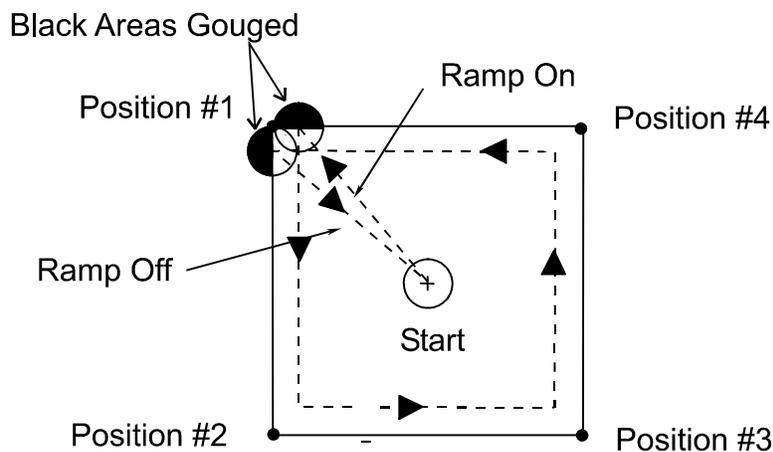
Poorly Chosen Starting & End Points.
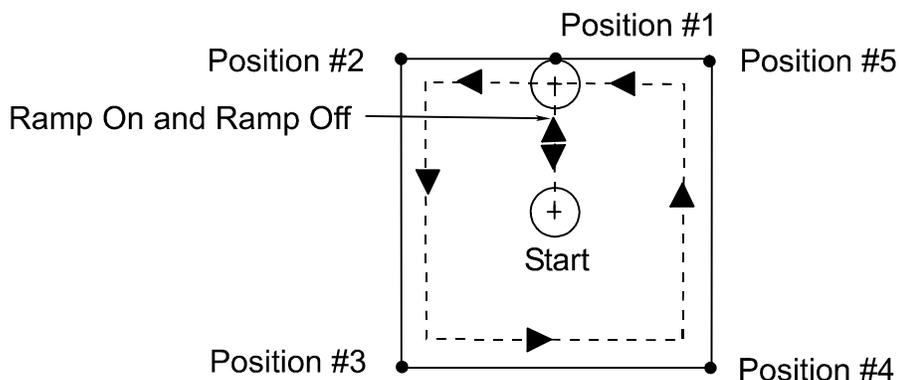
Preferred Method

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."

Ball End Mill

Quill At Tool# 0, Z 0 Position

$\frac{1}{2}$ Tool Diameter From Tip

Tool Length Offset
Adjusted To Ball's Center

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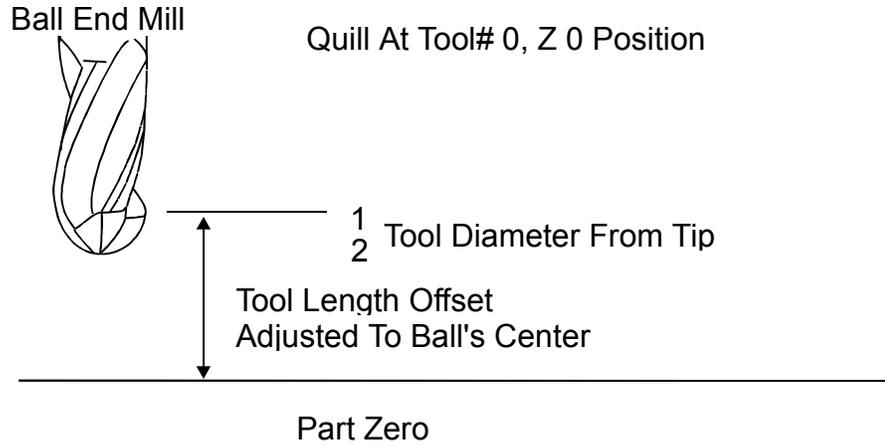
Figure 1-12, Setting Tool Length Offset for Ball End Mill

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**.
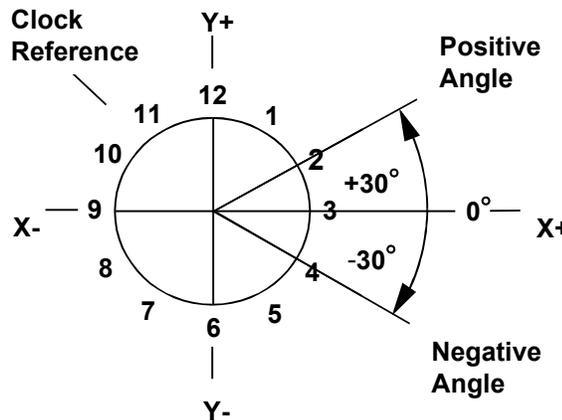
Figure 1-13, Absolute Angle Measurement

ANILAM

Corner Rounding

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

To activate corner rounding, enter 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, two arcs, or 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 in the figure. Refer to **Figure 1-14**.

Second Move
End Point

Radius Tangent
to Second Move

(Radius = **CornerRad** Value)

Intersection Between
Two Line Moves

Radius Tangent
to First Move

First Move
Starting Point

Solid Line = Path of Tool

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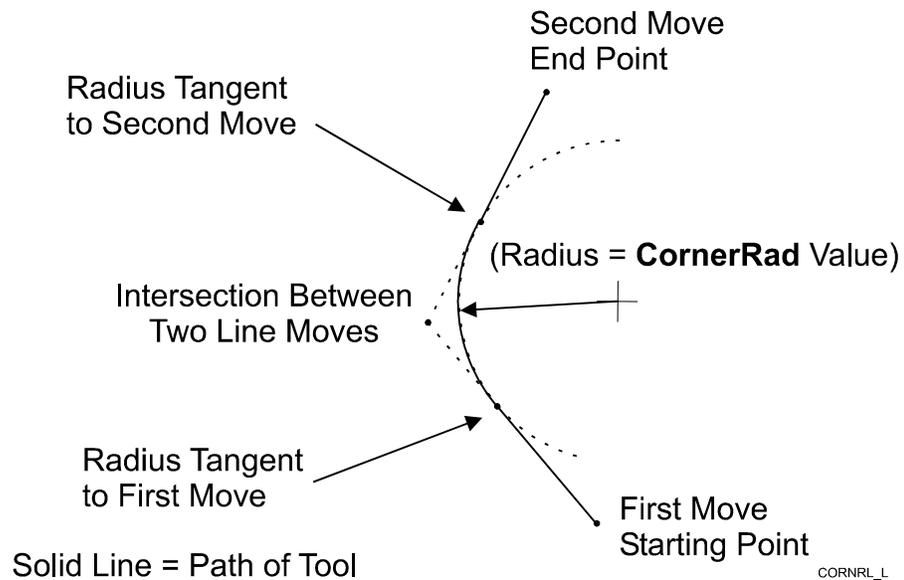
Figure 1-14, Line-to-Line Corner Rounding

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 in the figure. Refer to **Figure 1-15**.

> **NOTE:** If the line move is already tangent to the arc move, the CNC ignores corner rounding.
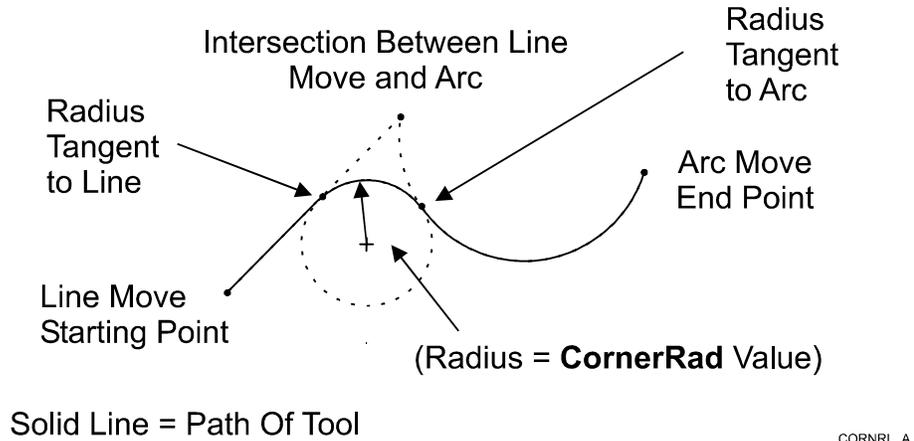
Figure 1-15, Line-to-Arc Corner Rounding

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 in the figure. Refer to **Figure 1-16**.
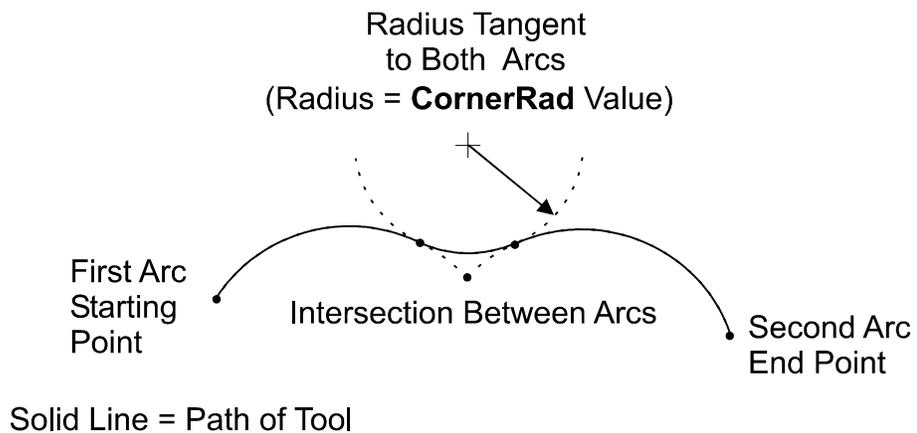
Figure 1-16, Arc-to-Arc Corner Rounding

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 in the figure. Refer to **Figure 1-17**.

First Move End Point
(Second Move Starting Point)

Distance

Distance

Chamfer

(Distance = Negative
CornerRad Value)

Solid Line = Path of Tool
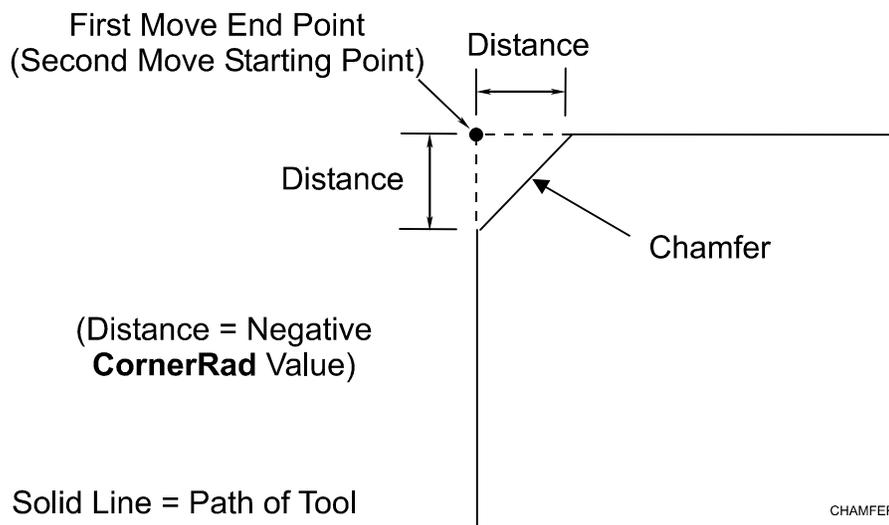
CHAMFER

Figure 1-17, Chamfering

Arc Direction

Arc direction is either clockwise (Cw) or counterclockwise (Ccw). The angle is measured from the three o'clock (0 degree) position in both Absolute and Incremental Modes.

Refer to **Figure 1-18** for Cw/Ccw arc directions in Absolute Mode. Refer to **Figure 1-19** for Cw/Ccw arc directions in Incremental Mode.
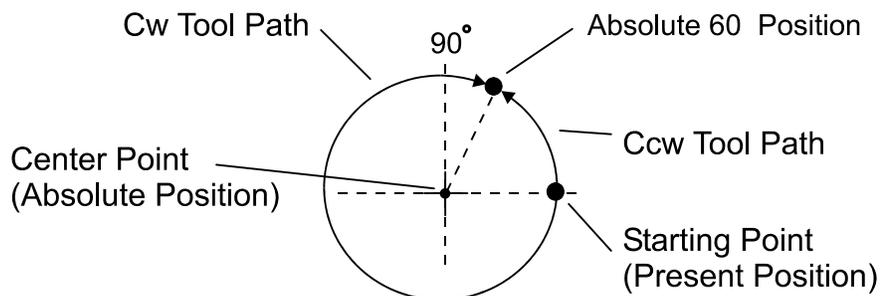
Figure 1-18, Absolute XY Arc Directions
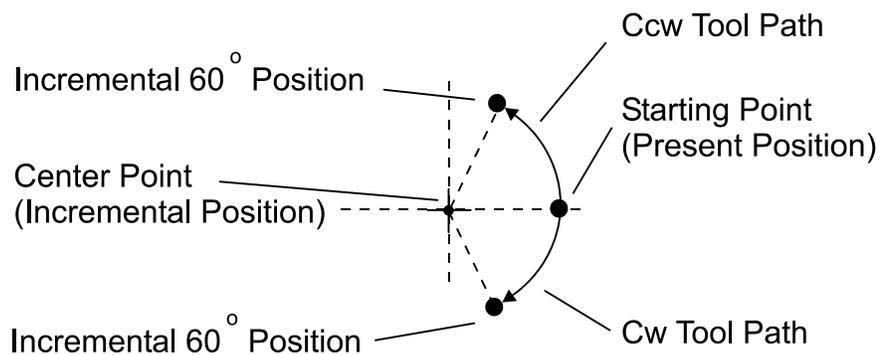
Figure 1-19, Incremental XY Arc Directions

Section 2 - CNC Console and Software Basics

Console

The CNC console consists of a 12.1" color, flat-panel Liquid Crystal Display (LCD) and two keypads, one to the right of the monitor, the other underneath the monitor. Refer to **Figure 2-1**.
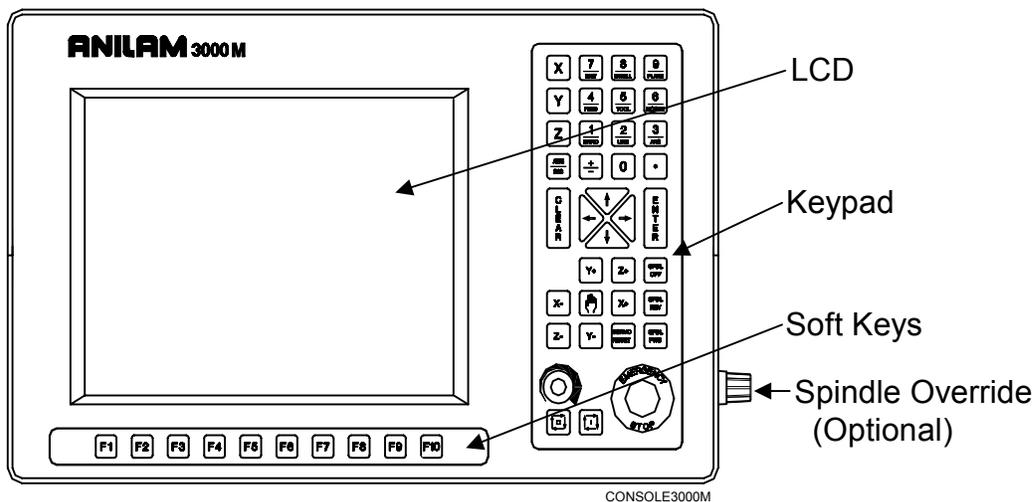
Figure 2-1, CNC Console

Keypad
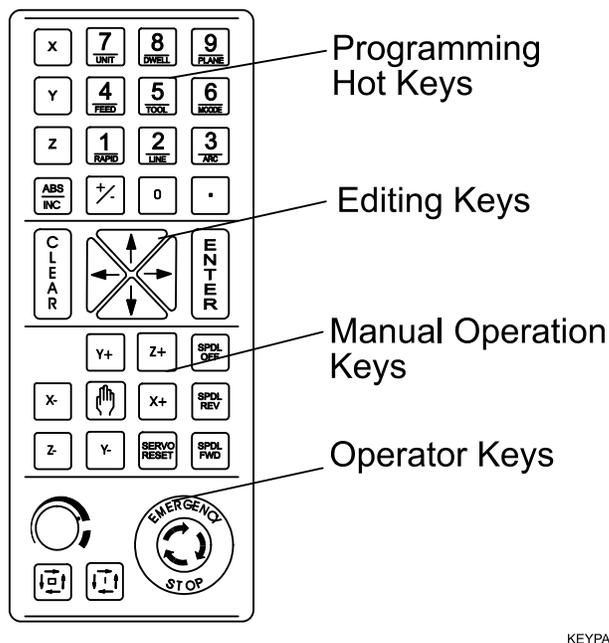
Figure 2-2, Keypad

Refer to Figure 2-2, Keypad. The keypad to the right of the monitor has four key types:

❑   Programming Hot Keys

❑   Editing Keys

❑   Manual Operation Keys

❑   Operator Keys

Programming Hot Keys

Refer to **Table 2-1**. Programming Hot Keys have two purposes. They enable you to enter position coordinates and provide quick access to frequently used functions. Hot keys are active in the Edit and Manual Modes.

Table 2-1, Programming Hot Keys

Label or Name	Key Face	Purpose
X	X	Selects X-axis for position inputs.
Y	Y	Selects Y-axis for position inputs.
Z	Z	Selects Z-axis for position inputs. **NOTE:** All Z moves are manual.
ABS/INC	ABS INC	Switches CNC between Absolute and Incremental Modes.
0	0	Zero / Switches comment asterisk in edit mode. Switches resolution display between program and Dist. To Go.
1/RAPID	1 RAPID	One / Hot key for programming a Rapid move.
2/LINE	2 LINE	Two / Hot key for programming a Line move.
3/ARC	3 ARC	Three / Hot key for programming an Arc.
4/FEED	4 FEED	Four / Hot key for changing feedrate.
5/TOOL	5 TOOL	Five / Hot key for programming a tool.
6/MCODE	6 MCODE	Six / Hot key for programming an M-code.
7/UNIT	7 UNIT	Seven / Hot key for switching between inches (Inch) and millimeters (mm).
8/DWELL	8 DWELL	Eight / Hot key for programming a Dwell.

(Continued…)

ANILAM

Table 2-1, Programming Hot Keys(Continued)

Label or Name	Key Face	Purpose
9/PLANE	9 PLANE	Nine / Plane hot key not used on 3000M Two-Axis CNC.
+/-	+/-	Sign change / Toggle key.
DECIMAL	.	Decimal point.

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	CLEAR	Clears selected messages, values, commands, and program blocks.
ARROW	(arrows)	Enables you to move highlight bars and cursor around the screen.
ENTER	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	(hand)	Cycles the CNC through manual movement modes (**JOG: RAPID**, **JOG: FEED**, **JOG: 100**, **JOG: 10**, **JOG: 1**).
Y+	Y+	Manually moves machine in positive Y direction.
Y-	Y-	Manually moves machine in negative Y direction.
X+	X+	Manually moves machine in positive X direction.
X-	X-	Manually moves machine in negative X direction.
SERVO RESET	SERVO RESET	Activates servo motors.

(Continued…)

Table 2-3, Manual Operation Keys (Continued)

Label or Name	Key Face	Purpose
SPINDLE FORWARD	SPDL FWD	Starts spindle in a clockwise direction (viewed from the top of the motor). Optional.
SPINDLE REVERSE	SPDL REV	Starts spindle in a counterclockwise direction (viewed from the top of the motor). Optional.
SPINDLE OFF	SPDL 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		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.
E-STOP	EMERGENCY STOP	The emergency stop button disconnects the servos, preventing spindle and machine motion.
SPINDLE OVERRIDE (Optional)	∩% SPINDLE	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.
START		The green **START** key initiates all machine moves except jog.
HOLD		The red **HOLD** key pauses a running program or programmed move. (Press **START** to resume.)

Soft Keys (F1) to (F10)

Labeled soft keys (**F1** to **F10**), also called function keys, are located just below the monitor. Soft key functions are not hardwired; their functions change with changes in mode.

Labels indicate the function of each soft key. Unlabeled soft keys are inactive.

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** have assigned keyboard equivalents.

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 enable 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 a highlighted selection. Press the soft key again or press **CLEAR** to deactivate the function.

```
1 Arc Cw      X 0.000
2 Dim Abs
3 * .5 DRILL
4 BasicDrill      ▶Basic      a
5 Rapid        Drilling Off   0
6 Rapid          Pattern
7 Dim Incr      Bolt Hole
8 Dim Abs
9 <End Of Program>

[Teach]  [Draw]  [Drill]  [Pocket]  [Mill]  [Too
```
POPUP
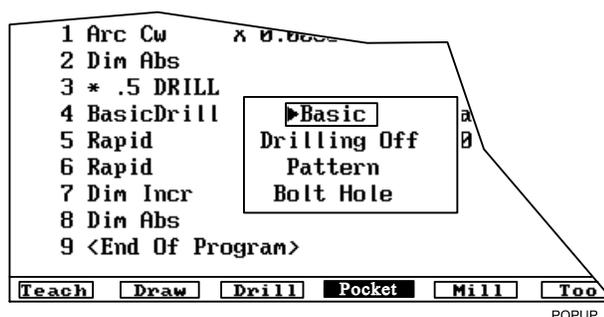
Figure 2-3, Pop-Up Menu

Screen Saver

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

Switching Selections with the Toggle Key

Press the (**+/-**) key (toggle key) to switch 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**.

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**. Selected character is displayed 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 are displayed 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 Letters

To delete letters:

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

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

Messages/Error Messages

Messages generated by the CNC are displayed 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 current message clears and the next message is displayed.

❑ From the Manual screen, press **MESSAGE** (**F1**). Displayed messages are displayed in the Program Area of the screen.

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

```
                                    MACHINE      PROGRAM: 2AX-SP5.M

  X+      0.0000           X +    0.0000    MANUAL   HALTED
  Y+      0.0000           Y +    0.0000    IN-POSN  FEED
                                                      ABS
  Z+      0.0000                                      INCH

                                     LOOP: 0        JOG: RAPID
                                    DWELL: 0.0   SPINDLE: OFF
 BLOCK:          POSN: PROGRAM        RPM: 0     COOLANT: OFF
  TOOL: 0         DIA: 0.0000
  FEED: 0.0        %: 100           PARTS: 0
                                    TIMER: 00:00:00




            No program selected!
            Illegal program name!




 MESSAGE:

 MESSAGE PROGRAM  EDIT   MANUAL  S.STEP  AUTO   MDI           TOOL   EXIT
```
MESSAG
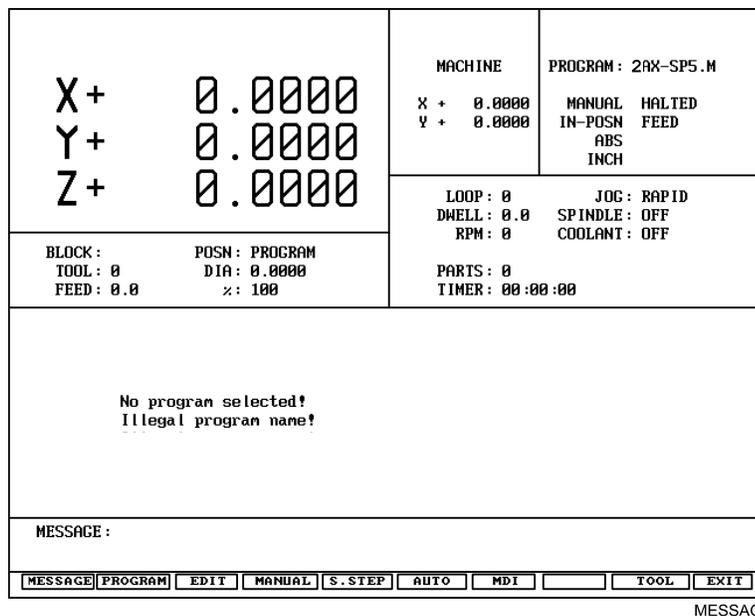
Figure 2-5, Messages Display

ANILAM

Section 3 - Manual Operation and Machine Setup

Powering On the CNC

To power on the CNC:

1. Use the power switch located on the CNC cabinet to start the CNC. The Startup screen activates and prompts you to "Press F10 to continue."

2. Press **Cont.** (**F10**). The CNC displays the **Software Options** menu.

3. Highlight **CNC Control**, and press **ENTER**. Manual Mode activates.

Shutting Down the CNC

To power down the CNC:

1. Press **E-STOP**. Servos disengage and control goes to Manual Mode.

2. Press **EXIT** (**F10**). The CNC displays the **Software Options** menu.

3. Use the power switch located on the CNC cabinet to shut down the CNC.

Emergency Stop Button

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

To reset **E-STOP**:

1. Rotate the switch clockwise, in the direction of the arrows on the E-STOP switch. The switch clicks when reset. The CNC does not automatically reactivate the servos following an E-STOP reset. 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**. Servos disengage and Manual Mode activates.

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

3-1
31-July-05

Activating/Resetting the Servos

For safety reasons, the control powers up with the servo motors off. While the servos are off, the CNC cannot move the machine and the spindle will not start. The CNC generates a "SERVO OFF!" message when the servos turn off. Refer to "Section 2 - CNC Console and Software Basics" to review 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. When a limit switch trips and the servos turn off, manually reposition the machine within its range of travel.

2. Press **E-STOP**.

3. Rotate the **E-STOP** switch in the direction of the arrows to reset it. The switch clicks when reset.

4. Press **SERVO RESET**. Servos reset.

> **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 activated or if the servos go off-line.

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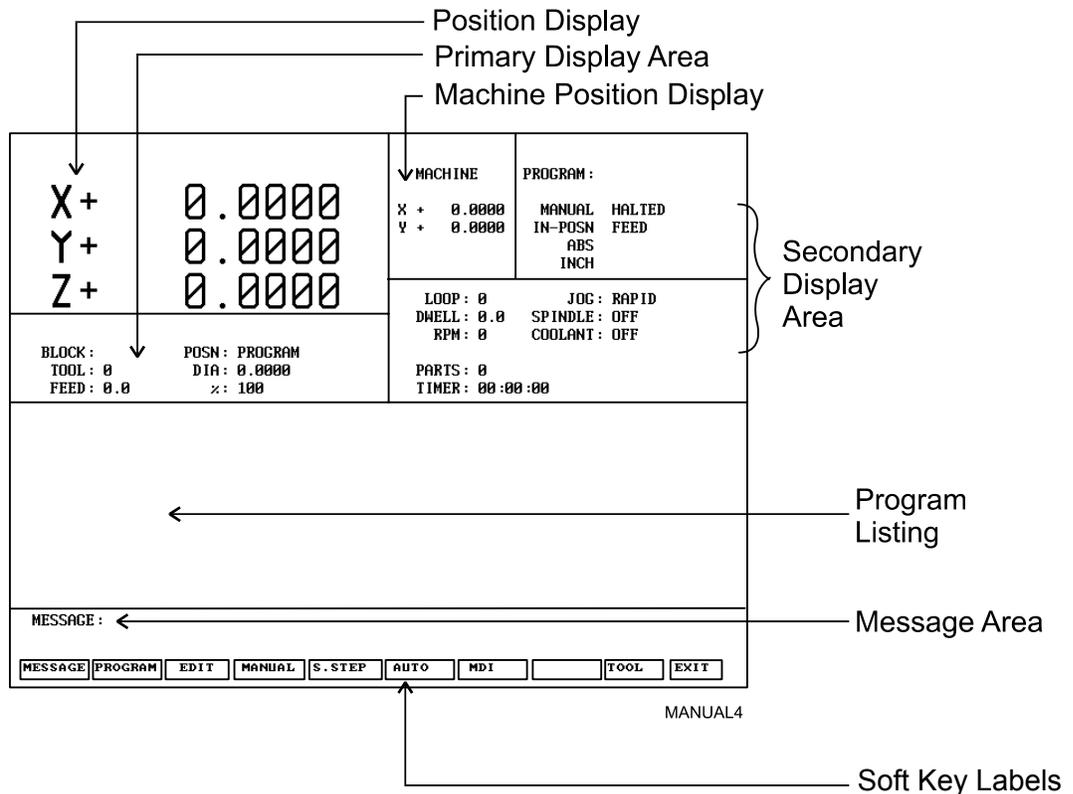
Figure 3-1, Manual Mode Screen

The Manual screen features:

Position Display   Displays X and Y position coordinates.

Machine Position Display

This area of the screen shows the axis positions in reference to Machine Home. (Not shown when Draw activates.)

Primary Display Area

Displays essential operating information.

Secondary Display Area

Displays additional operating information. (Not shown when Draw activates.)

Message Area   Displays messages, prompts, and reminders.

Soft Key Labels   Identifies 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:	Tells operator whether machine has reached target (in-posn) or not.
ABS / INC:	Current positioning mode.
INCH / MM:	Current units mode.
HALTED/*HALTED/RUNNING:	Without asterisk: machine is in a programmed hold, or has completed its program. With asterisk: 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.
RPM:	Spindle RPM (Optional). May display programmed RPM or actual RPM. Refer to builder's documentation for details.
JOG:	Current jog mode.
SPINDLE:FWD/REV/OFF:	Spindle status (**FWD**/**REV**/**OFF**). Optional.
COOLANT:	Coolant status. Optional.

ANILAM

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 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.

To switch the **POSN:** setting:

1.   In Manual, S. Step, or Auto Mode, press **0** to switch the setting.
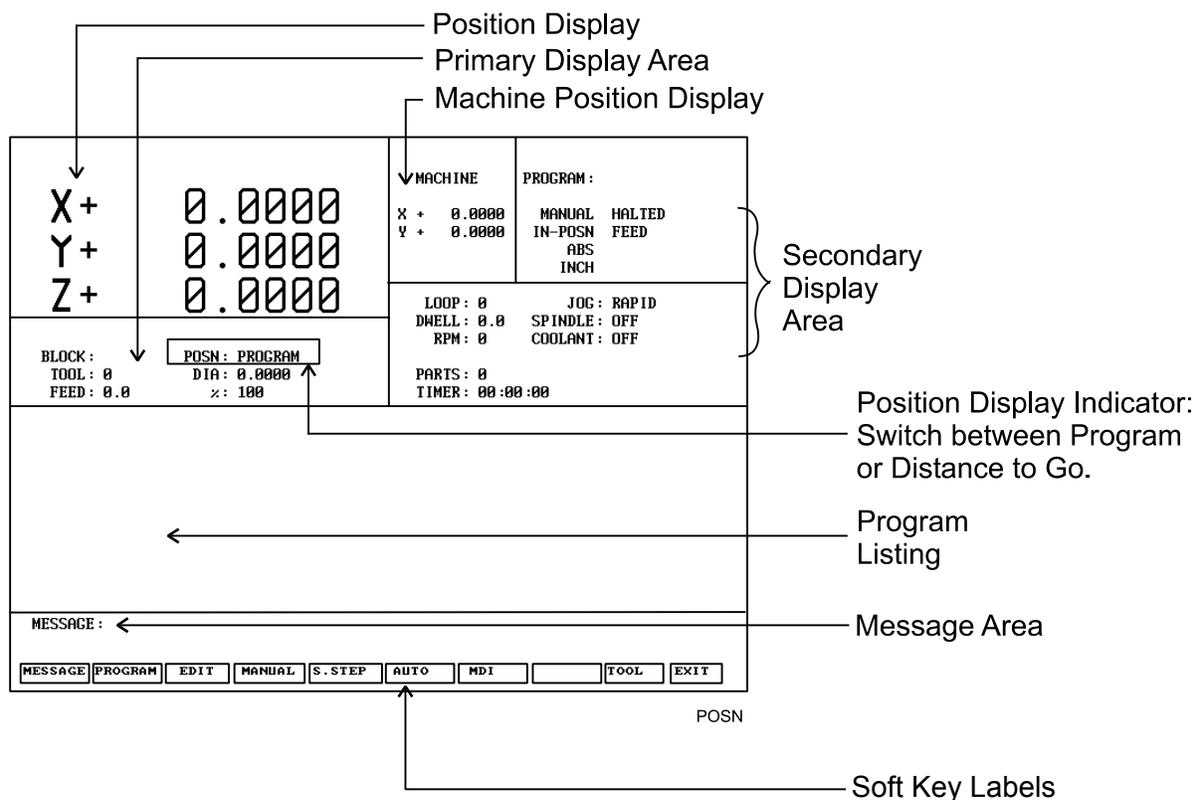
Figure 3-2, Position Display Options

Manual Machine Operation

Two types of manual operation are available:

Auto Mode   Operator controls the machine via the keypad. Servos move the machine.

Manual Mode   Digital Read Out. Operator uses handles to move the machine. Servos deactivate.

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**. Servos reactivate and the CNC operates the machine.

Manual Mode

In Manual Mode, the CNC acts as a digital read out 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**. Spindle reactivates.

Mode Settings

You control 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.

Make all of the modal adjustments required for the machine to move properly before you make a manual move. Modes set from the Manual screen remain active until changed.

Refer to Table 3-1, Manual Mode Settings 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
Active Tool	Tool Diameter Compensation and Tool Length Offsets

You can use the Manual screen to program the locations of the Absolute Zero Reference (X0, Y0).

There are four types of moves available in the 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

The user 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**. The CNC returns to the Manual screen.

3. Press **START** to activate the feedrate change.

– *or* –

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

CAUTION:	If the CNC is powered down, then restarted, the setup file will reload the CNC's default feedrate.

Adjusting Rapid Move Speed

Use **FEEDRATE OVERRIDE** to adjust the Rapid move speed. Each 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 (Option)

NOTE:	If **SPINDLE OVERRIDE** is not installed in your system, the spindle percent label (**%:**) is not displayed in the secondary display area of the Manual Mode screen. Refer to Figure 3-2, Position Display Option.

If **SPINDLE OVERRIDE** is installed in your system, 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. You 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 current position. The active position mode appears in the Secondary Display Area.

❑ **ABS** indicates Absolute Mode.
❑ **INCR** indicates Incremental Mode.

To activate the Absolute or Incremental Mode:

1. Put the CNC in Manual Mode.

2. Press **ABS/INC** to switch 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 units 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 switch the active units to Inch or MM Mode. Secondary Status Area displays 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 position the Absolute Zero at any convenient location.

At power up, the current machine location becomes Absolute Zero. At power down, the Absolute Zero is lost. Setting the Absolute Zero to a location on the part is called setting Part Zero.

NOTE: Use the 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 the **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**. The CNC accepts 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 some known distance from the desired Part Zero.

2. Press **X**. CNC prompts for **X axis Part Zero** position and displays 0.00.

3. Enter the desired **X-axis** position, including the sign, and press Y. The CNC prompts for **Y-axis Part Zero** position and displays 0.00.

4. Enter the desired **Y-axis** position, including the sign, and press **ENTER**. New position coordinates are set.

> **CAUTION:** To enter positions, press **ENTER**. Do not press **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**.
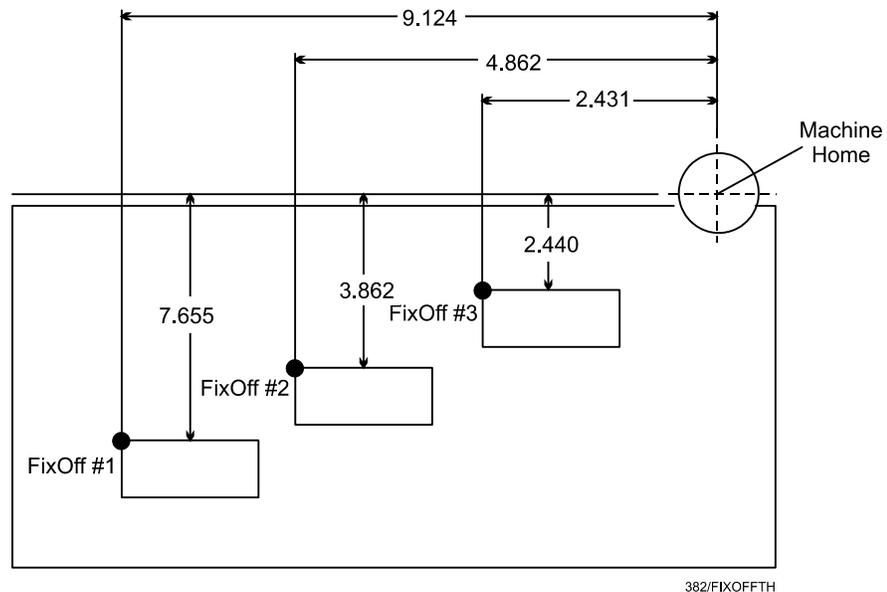
Figure 3-3, Locating Fixture Offsets

ANILAM

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 positive or negative direction of movement. For the diagram above:

Fixture Offset #	Coordinates Entered in Fixture Offset Table
FixOff #1	X-9.124, Y-7.655
FixOff #2	X-4.862, Y-3.862
FixOff #3	X-2.431, Y -2.440

Fixture Offset locations can also 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 during tool changes or as a safe place from which to make rapid XY moves.

NOTE:	Tool Length Offsets are only available when the Z-axis has been set up as a digital read-out.

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

Set Tool #0, Z0 before you set 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:	Program a move to the tool change position 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.

NOTE:	All Z moves are made manually.

Activating a Tool

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

To activate a tool manually:

1. In Manual Mode, press **5/TOOL**. The CNC prompts for tool number.

2. Enter tool number and press **Save** (**F10**). The CNC returns to Manual Mode.

3. Press **START**. The active tool number appears in the Primary Display Area.

   – 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.
❑ The axis being moved is a controlled axis (X and Y).
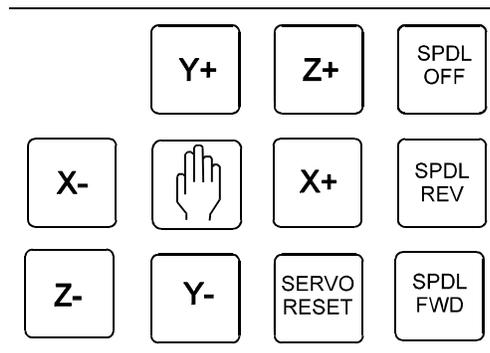
Figure 3-4, Manual Operation Keys

NOTE:	The Z+ and Z- keys will not initiate a Jog move. However, your 2-axis system can be upgraded to a 3-axis system. Contact ANILAM for prices.

There are five move modes available. 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 is determined at machine setup.
Feed	Continuous jog at current feedrate.
Jog: 100	Conventional Jog Mode, increment set to 100 times the resolution of machine.
Jog: 10	Conventional Jog Mode, increment set to 10 times the resolution of machine.
Jog: 1	Conventional Jog Mode, increment set to the actual resolution of machine.

In Manual Mode, you can change the Jog Mode at any time.

Changing the Jog Mode

To change the Jog Mode:

1. In Manual Mode, press **JOG**. The mode selection changes. The selected mode appears in the Secondary Display Area. 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 Manual Mode, Teach Mode or Tool Page, press JOG to cycle through the available settings: **JOG: 100**, **JOG: 10** or **JOG: 1**. The selected setting appears in the Secondary Display Area.

2. Press the required axis key (**X+**, **X-**, **Y+**, or **Y-**). Every time you press an axis key, the machine jogs along the selected axis at the indicated increment (100, 10 or 1 times the 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 selected setting appears in the Secondary Display Area.

2. Press the required axis key (**X+**, **X-**, **Y+**, or **Y-**). 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 the user to jog a selected axis via 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 selected Jog Mode is displayed on the screen.

Use the **JOG** key to cycle through the available Jog Modes. Refer to "Jog Moves" in this Section for more information on how to select a Jog Mode. The handwheel only works in conventional Jog Modes (100, 10 and 1).

To operate the handwheel:

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 Pop-Up menu activates. Refer to **Figure 3-5**.
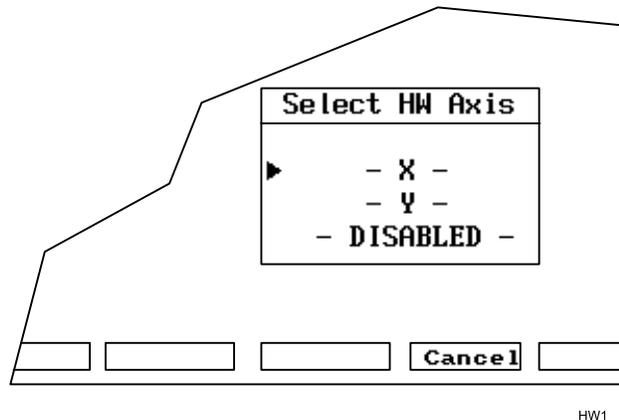
HW1

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 via the remote handwheel. Press **ENTER**. The selected axis can now be moved via the remote handwheel.

31-July-05

ANILAM

3. Use the JOG key to select a Jog Modes: 100, 10, 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 use one-shot moves to position the machine. One-shot moves can be in Absolute or Incremental Mode and in Rapid or Feed Mode.

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. Use **ABS/INC** to select a Positioning Mode (ABS/INC).

3. Enter the appropriate **X** and **Y** 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 to initiate a one-shot move. Do not press ENTER. If you press ENTER, Absolute Zero moves to the entered position.**

NOTE:	The **1/RAPID** or **2/LINE** hot keys can also 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 you would write program blocks. The **MDI** editor has the same features as the regular Program Editor. Use **Draw** to test MDI moves.

Refer to Figure 3-6, Manual Data Input (MDI) Screen. The moves in MDI remain in the MDI.M program until cleared or edited. Run the MDI.M program at any time.

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

The MDI screen resembles the Edit screen with MDI.M listed as the active program.

```
Program: MDI.M                Blocks: 2        Free: 1,879 KB  Inch  Abs  XY




















1            X 2.0000
2 Line       X 1.0000
3 <End Of Program>





Teach   Draw   Drill  Pocket   Mill   Tool   Calc   Sub   Misc  Exit
```
MDIL
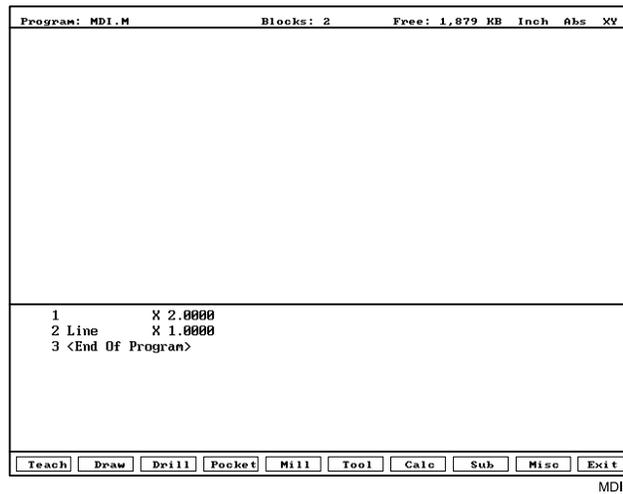
Figure 3-6, Manual Data Input (MDI) Screen

To execute an MDI command:

1. From the Manual screen, press **MDI** (**F7**). The CNC displays the Program Editor. The CNC automatically loads the MDI.M program and displays the program name on the bottom right portion of the screen.

2. Enter the required program blocks.

3. Verify moves in Draw, if necessary.

4. Press **Exit** (**F10**) to return to the Manual screen.

5. Press **START**. The CNC executes an MDI move.

    – *or* –

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

ANILAM

Section 4 - Writing Programs

2-Axis CNC Operator's Role

Manually position the Z-axis. If the Z-axis is equipped as a digital readout, the control displays the Z-axis position, but does not control movement along Z.

To prevent machine damage or bodily injury, the CNC has a number of built-in machine holds.

Operator, CNC roles, and restrictions for running programs follow:

❑ The CNC will hold and prompt, "POSITION Z AXIS AND PRESS START." whenever it encounters a Z-axis command.

❑ In cycles that use a programmed starting height (**StartHgt**) and cutting depth (**DepthCut**), the control calculates the number of passes needed to peck from the StartHgt to the pocket depth (ZDepth) when incrementing the DepthCut on each pass. The CNC will hold and prompt, "POSITION Z AXIS AND PRESS START." on each pass.

❑ In cycles that use a programmed finish stock (**FinStock**), the control calculates the number of passes required to peck to the pocket depth (**ZDepth**) and leave enough stock for a finish pass. Leave finish stock on the pass before the finish pass and be sure to accurately position the Z-axis on the finish pass.

❑ The CNC will hold and prompt, "PREPARE FOR FEED MOVE AND PRESS START." every time the CNC changes from Rapid to Feed. This enables you to verify the Z-axis position before feeding into the part. This hold also occurs in canned cycles.

❑ The CNC will hold and prompt, "PREPARE FOR RAPID MOVE AND PRESS START." every time the CNC changes from Feed to Rapid. During this hold, verify the clearance before making a Rapid move to another location. This hold also occurs in canned cycles.

❑ The CNC will hold and prompt, "Program stopped on tool mount, press START." at the start of every block that activates a tool. This hold enables you to verify the Z position (and length of the new tool) before executing the block.

> **NOTE:** Following a Rapid/Feed Mode change programmed after a Z move, more than one hold may occur.

Program Basics

A program is a sequence of precise machine instructions. Each program consists of blocks of instructions that direct machine movements. Every program is given a unique name for identification. Every block is automatically numbered by the CNC.

Features (or settings) that remain active for more than one operation or event are said to be modal. Modal settings are active until changed or turned off.

Many of the CNC's features are modal: the move type (Rapid or Feed), the actual feedrate (IPM) and the units (Inch or MM).

Write programs using combinations of moves, mode changes, and canned cycles. Canned cycles are stored in the CNC's 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 move to Z0. Treat tool information like part of the program.

Refer to the blueprint when you choose 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. Enter the Program Directory (the **PROGRAM** screen) and create the program for the part (refer to "Section 9 - Program Management").

2. Enter the Program Editor (the **Edit** screen) to open the new program and begin to write blocks (refer to "Section 6 - Editing Programs").

3. Put the CNC in Absolute Mode in the first block 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, rapid to Tool#0 and move to Z0 manually to fully retract the quill for the next move.

6. In the second move, rapid to a convenient part-change position.

7. Execute moves toward a part in two steps: A Rapid X, Y move toward the part, followed by a manual 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.

8. Subsequent blocks are the moves, cycles, and tool changes required to machine the part.

9. Make the last three blocks of the program as follows: a) a Rapid move to Tool #0 (and manual move to Z0), b) a Rapid XY move to the same part change position used at the start of the program and c) an **EndMain** block.

10. Once the program is written, run the program in Draw to check for errors. Refer to "Section 7 - Viewing Programs with Draw."

11. Secure the work on the table with the appropriate work-holding device.

12. Set Part Zero from the Manual screen. Refer to "Section 3 - Manual Operation and Machine Setup."

13. 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."

14. 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, holding between each move; or run the program from in Single-Step Mode, holding 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.

15. 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 via 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**.
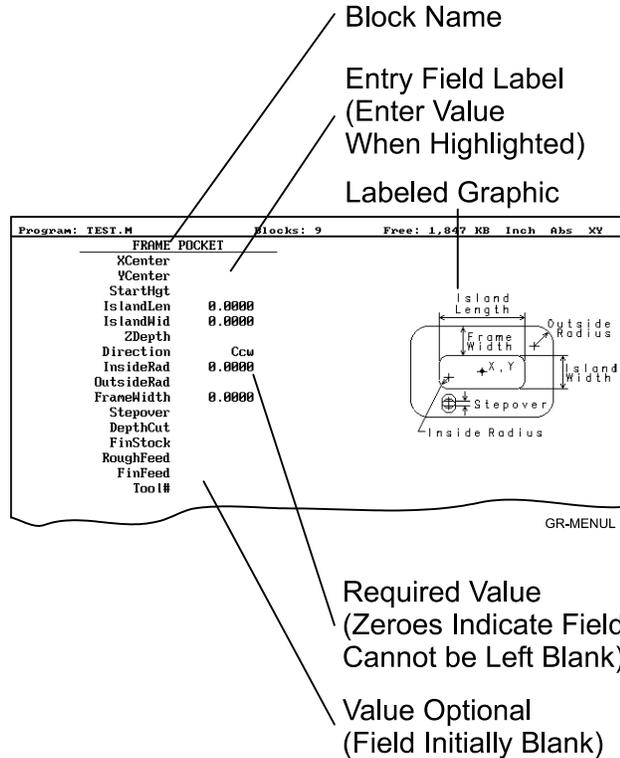
Figure 4-1, Sample Graphic Menu

Graphic Menus activate with the first entry field highlighted. To enter values, highlight the appropriate entry field. 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.

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

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

No Move Blocks

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

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 the operator to select **Abs** or **Incr**.

2. Press (**+/-**) to switch 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 puts the CNC in either 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 for the **Inch** or **MM** selection.

2. Press the (**+/-**) key to switch settings.

3. Press **Save** (**F10**) or **ENTER** to add the block to the Program Listing.

Programming a Tool Change (Activating a Tool)

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 perform a typical tool change:

1. Program a Tool #0 to cancel offsets.

2. Program a **ZMove** to Z0, the fully retracted quill position. (Do this manually during program execution.)

3. After the **ZMove**, the program resumes, and the CNC moves to the tool change position. Once in position, the CNC activates the next tool number and holds for a tool change.

4. Press **START** when the tool change is completed. When the program restarts, the new diameter and length offsets activate.

> **NOTE:** To easily find tool changes in the program, program them in separate blocks.

To program a **Tool#** block:

1. In Edit Mode, press **5/TOOL**. The **TOOL MOUNT** Graphic Menu prompts for the **Tool #**.

2. Type tool number, and press **ENTER**. The **Tool#** block is added to the Program Listing.

Setting Length Offsets at Tool Change

When collet-type tool holders are used, set the length offsets at tool change. Be sure to use the correct tool diameter.

To set tool length offsets at tool change (not in the program):

1. Execute a move to the tool change position and activate a programmed hold.

2. Mount the new tool.

3. Touch the tip of the tool to the part Z0 position and press **SET-Z** (**F8**).

4. Press **START** to continue the program.

ANILAM

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 in effect until canceled or changed.

Turn compensation on/off in Rapid or Line move blocks. Ramp moves offset the tool on the programmed path by half the tool diameter.

To activate tool compensation, switch the Rapid or Line **ToolComp** selection to one of the following three choices:

❑ **Left** (of the path)
❑ **Right** (of the path)
❑ **Off** (cancels tool compensation)

Once tool compensation is activated, it remains in effect until you turn it off. Before activating a new tool, turn tool compensation off and ramp off.

Cycles with built-in compensation activate and deactivate compensation automatically. In order for these cycles to be accurate, the correct tool diameter must be active. Activate the required tool within the cycle.

31-July-05

Refer to **Table 4-1** for a list of move and cycle 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.	Compensation is activated and deactivated 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")	X	---
Spiral (no compensation available)	---	---
Face (only affects stepover)	---	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

ANILAM

Programming a Dwell

A **Dwell** block pauses the program for a specified length of time. Dwell resolution is 0.1 sec. If you enter 0.0, the CNC will dwell until restarted (called an "infinite dwell"). Press **START** to resume.

To program a **Dwell** block using hot keys:

1. In Edit Mode, press **8/DWELL**. The **DWELL** Graphic Menu activates.

2. Type time, and press **ENTER** to add the block to the Program Listing.

To program a **Dwell** block using the soft keys:

1. In Edit Mode, press **Sub** (**F8**) to display the Secondary soft key functions.

2. Press **Dwell** (**F7**). The **DWELL** Graphic Menu activates.

3. Type the time, and press **ENTER** to add the block to the Program Listing.

Programming a Return to Machine Zero

> **NOTE:** The CNC measures all entered 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 in one of 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**-axis keys. On each axis selected, the machine feeds from the current position to the limit switch, reverses direction, travels to the first detected zero crossing and sets Machine Zero at that point.
2. Enter coordinates	Activate Machine Home Graphic Menu. For each required axis, highlight the axis entry field and enter a coordinate (example: X0, Y-1). The machine rapids to the entered coordinate, 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 entered coordinate, then initiates the homing sequence.

To activate the Machine Zero Graphic Menu:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft keys are displayed.

2. Press **More** (**F7**). The More Menu is displayed.

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.
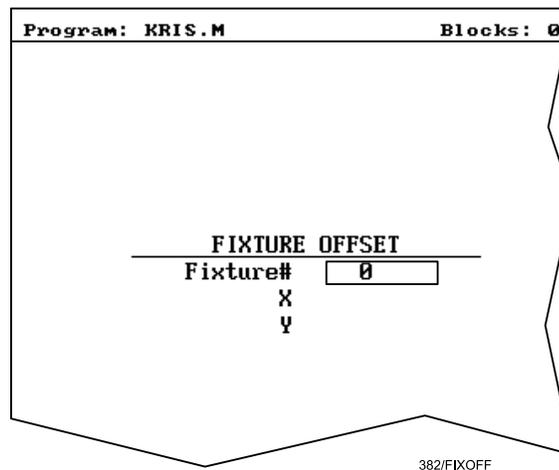
Figure 4-2, Fixture Offset Graphic Menu

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**. The Fixture Offset Graphic Menu is displayed.

4. Fill in the labeled entry fields, as follows:

Fixture#   The Fixture-Offset number. Indicates which set of values from the Fixture Offset Table will be activated or changed. Enter a number 1 to 9, corresponding to the Fixture Offset Table, to activate or change an offset. Enter 0 to cancel fixture offsets.

X   X-offset coordinate. If no value is entered, the CNC activates the offsets listed in the Fixture Offset Table for the entered **Fixture#**. If a value is entered, the CNC applies the entered offset. When the program runs, the CNC updates the Fixture Offset Table with the given X offset and clears the previous value. Optional.

Y   Y-offset coordinate. If no value is entered, the CNC activates the offsets listed in the Fixture Offset Table for the entered **Fixture#**. If a value is entered, the CNC applies the entered offset. When the program runs, the CNC updates the Fixture Offset Table with the given Y 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, enter **0** to cancel Fixture Offsets. Do not fill in the other entry fields.

Fixture Offset Table

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

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

Figure 4-3, Fixture Offset Table

Activating the Fixture Offset Table

To activate the Fixture Offset Table:

1. In the Tool Page, press **OFFSETS** (**F1**). The Fixture Offset Table activates.

Changing Fixture Offsets in the Table

There are two ways to change the values in the table, manually enter a value or calibrate the table entry to the machine's current location (shown on the axis display).

To change a fixture offset to a manually entered coordinate:

1. Highlight a Fixture Offset (row 1 to 9).

2. Press an axis key (**X** or **Y**).

3. Type a value, and press **ENTER**. The CNC stores the value in the table.

To change a fixture offset to the current machine position:

1. Highlight a Fixture Offset (row 1 to 9).

2. Press **CalibX** (**F5**) or **CalibY** (**F6**). The CNC stores the current machine position for the selected axis 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 entered coordinates.

In axis presetting, non-zero XY values set the machine's present position to the entered coordinates. In axis resetting, X0 and Y0 values set the machine's present 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.

Refer to **Figure 4-4**.

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.
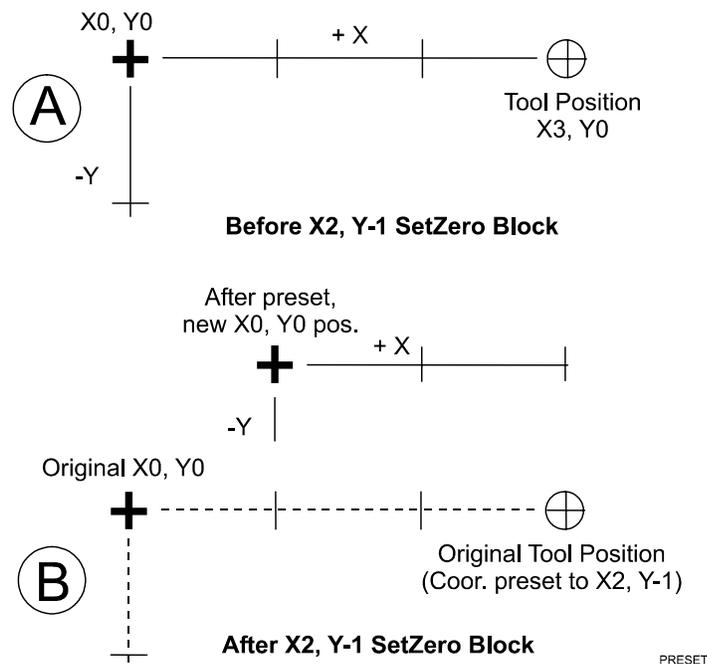
Figure 4-4, Executing a SetZero Block

Refer to Figure 4-5, Using SetZero in a Program. Use **SetZero** to cut more than one part with the same programming steps. To do this, change the Absolute Zero Reference before you machine each part.

Restore the location of the original X0, Y0 reference at the end of the program.
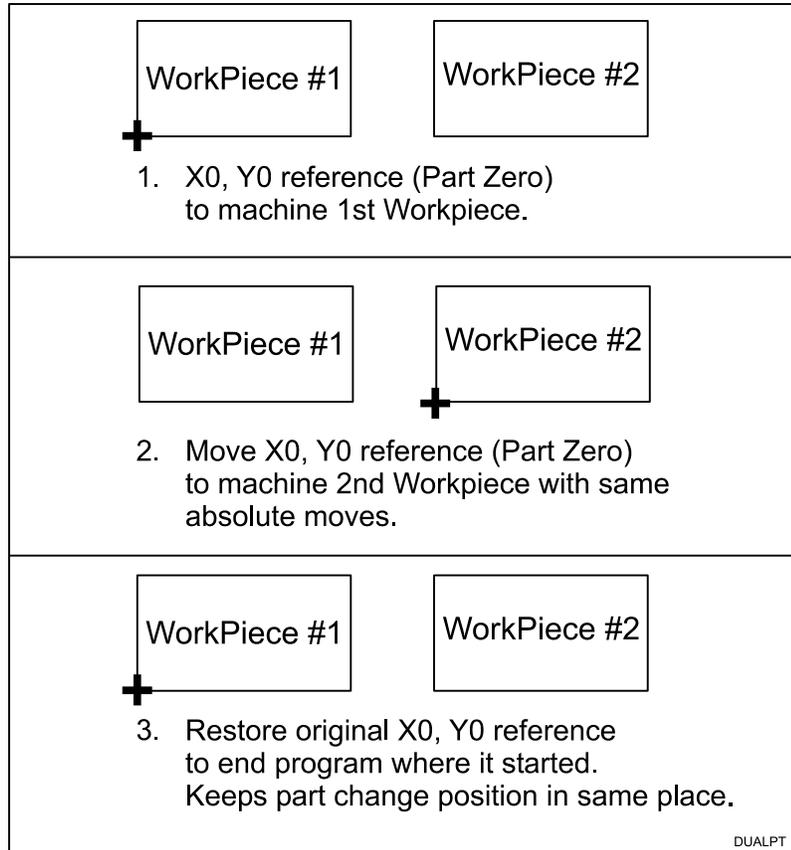
Figure 4-5, Using SetZero in a Program

When the X or Y values are left blank, no coordinate change is made. Refer to **Figure 4-6**.
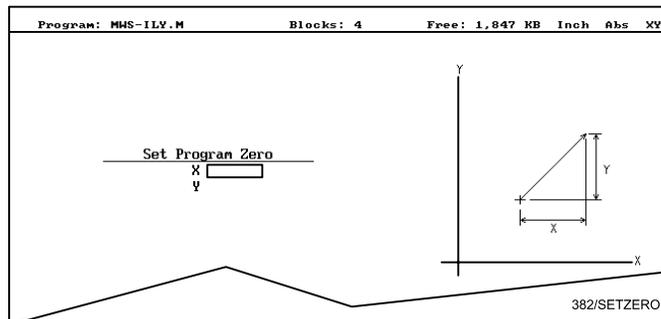
Figure 4-6, Set Zero Graphic Menu

ANILAM

To program a **SetZero** block:

1. In Edit Mode, press **Mill** (**F5**) to display the Mill soft key functions.

2. Press **More** (**F7**) to a pop-up menu.

3. Position the highlight to select **SetZero**, and press **ENTER**. The **Set Program Zero** Graphic Menu prompts for the absolute coordinates of the present position.
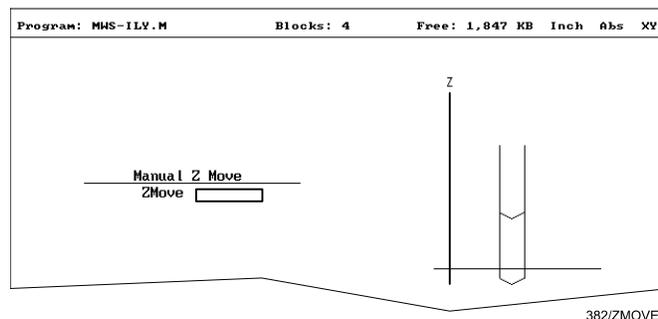
4. Enter the required X and Y coordinates.

Programming Manual Z Moves

A **ZMove** block holds the machine and prompts the operator to execute the manual move. Press **START** to continue the program after you complete the Z move. The **ZMove** block reminds the operator where to move in Z. Refer to **Figure 4-7**.


```
Program: MWS-ILY.M          Blocks: 4       Free: 1,847 KB  Inch  Abs  XY



                                              Z



                  Manual Z Move
                    ZMove  [        ]




```
382/ZMOVE

Figure 4-7, Manual Z Move Graphic Menu

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

1. In Edit Mode, press **Z**. The Manual Z Move Graphic Menu prompts for the Z position.

2. Type the Z position, and press **ENTER**. The manual move block is inserted into the program.

To program a **ZMove** block using soft keys:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft keys are displayed.

2. Press **ZMove** (**F5**). The Manual Z Move Graphic Menu prompts for the Z position.

3. Type the Z position, and press **ENTER**. The manual **ZMove** block is inserted into the program.

> **NOTE:** The CNC will hold and prompt, "POSITION Z AXIS AND PRESS START." whenever a Z-axis move is encountered.

Programming a Feedrate Change

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

> **NOTE:** Feed blocks do not activate the Feed Mode.

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

1. In Edit Mode, press **4/FEED**. The FEEDRATE Graphic Menu prompts for a new feedrate.

2. Type the required feedrate, and press **ENTER**. The **Feed** block is added to the Program Listing.
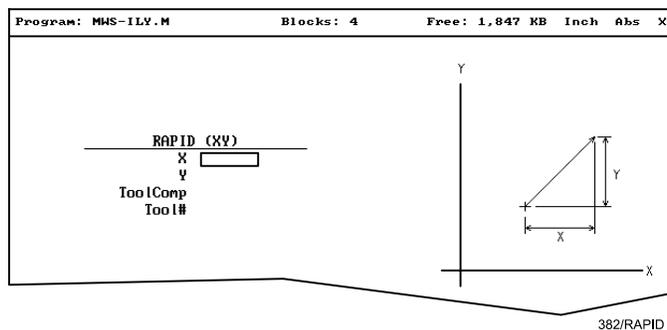
To program a **Feed** block using the soft keys:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft key labels are displayed.

2. Press **More** (**F7**). The More pop-up menu is displayed.

3. Select **Feed**, and press **ENTER**. The FEEDRATE Graphic Menu prompts for feedrate.

4. Press **Save** (**F10**) or **ENTER**. The **Feed** block is added to the Program Listing.

Straight Moves

Programming a Rapid Move

Rapid moves run in Rapid rate. Rapid moves save time when positioning for a cut or a canned cycle. Use Rapid moves to activate and deactivate tool diameter compensation (cutter compensation).

Use Rapid moves to activate/deactivate tool diameter compensation and cutter compensation. Refer to **Figure 4-8**.

Figure 4-8, Rapid Move Graphic Menu

ANILAM

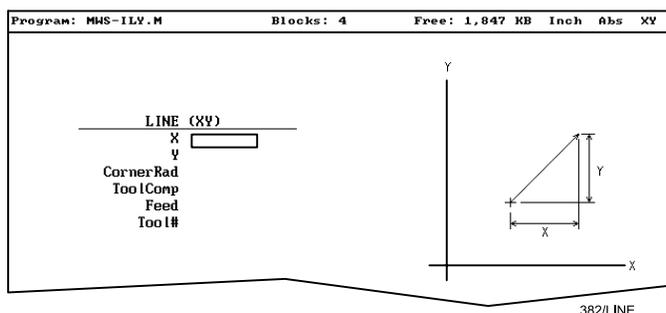To program a Rapid move using hot keys (the keypad):

1. In Edit Mode, press **1/RAPID**. The RAPID (XY) Graphic Menu activates.

2. Enter the appropriate settings or values in the entry fields.

To program a Rapid move using soft keys:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft key labels are displayed.

2. Press **Rapid** (**F2**). The RAPID (XY) Graphic Menu prompts for **X** position.

3. Enter the appropriate settings or values in the entry fields.

Programming a Line Move

Line moves are executed in Feed. Refer to **Figure 4-9**.

Figure 4-9, Line Move Graphic Menu

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

1. In Edit Mode, press **2/LINE**. The LINE (XY) Graphic Menu prompts operator for the **X** position.

2. Enter the appropriate settings or values in the entry fields.

To program a Line move using the soft keys:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft keys activate.

2. Press **Line** (**F3**). The LINE (XY) Graphic Menu prompts operator for **X** position.

3. Enter the appropriate settings or values in the entry fields.

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** or **Y**. The MODAL MOVE Graphic Menu prompts the operator for the X or Y position.

2. Enter the required position.

3. Press **ENTER** or **Save** (**F10**). The modal move block (Xn Yn) is added to the program.

> **NOTE:** When using modal moves, be sure the CNC is in the required Rapid or Line Mode. Line Mode moves are executed in Feed Mode.
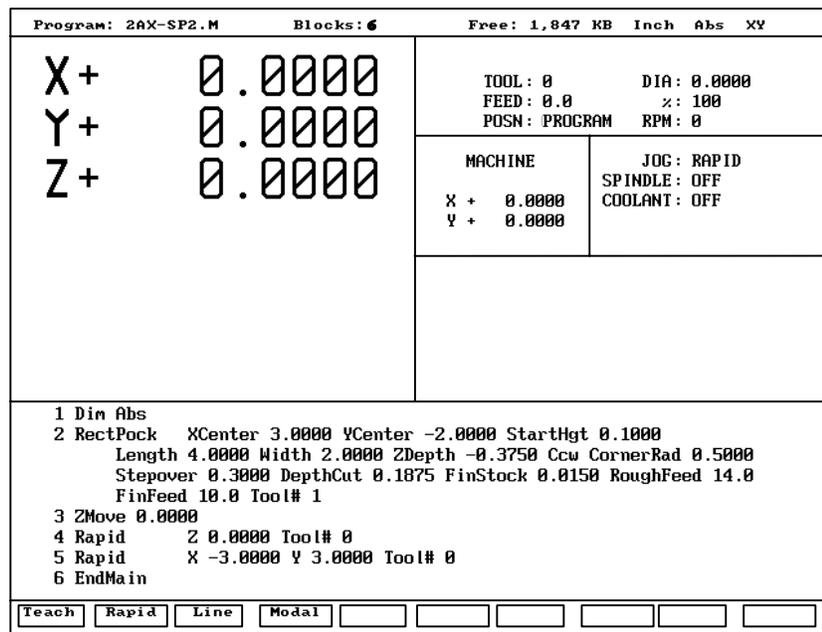
Teach Mode (Programming from the Part)

In the Teach Mode, the CNC writes program blocks that duplicate manually executed moves. You can use the Teach Mode to pick endpoints off a part diagram so that you can 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.

```
Program: 2AX-SP2.M        Blocks:6        Free: 1,847 KB  Inch  Abs  XY

X+        0.0000           TOOL: 0         DIA: 0.0000
                           FEED: 0.0          %: 100
Y+        0.0000           POSN: PROGRAM   RPM: 0

Z+        0.0000           MACHINE        | JOG: RAPID
                                          | SPINDLE: OFF
                           X +    0.0000  | COOLANT: OFF
                           Y +    0.0000




1 Dim Abs
2 RectPock   XCenter 3.0000 YCenter -2.0000 StartHgt 0.1000
             Length 4.0000 Width 2.0000 ZDepth -0.3750 Ccw CornerRad 0.5000
             Stepover 0.3000 DepthCut 0.1875 FinStock 0.0150 RoughFeed 14.0
             FinFeed 10.0 Tool# 1
3 ZMove 0.0000
4 Rapid      Z 0.0000 Tool# 0
5 Rapid      X -3.0000 Y 3.0000 Tool# 0
6 EndMain

Teach   Rapid   Line   Modal
```
382/TEACH

Figure 4-10, Teach Mode Screen

Use the Teach Mode to program a Line, Rapid, or Modal move. Refer to **Figure 4-10**.

To program a Teach move:

1. In Edit Mode, press **Teach** (**F1**). Teach Mode activates. Display shows 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

Given the X, Y, or XY endpoint, 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 4-11**.
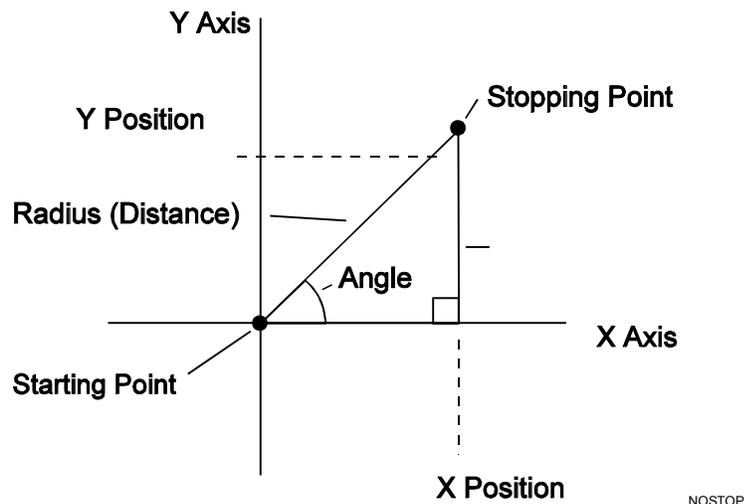
Figure 4-11, Move Orientation

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 menu does not contain **CornerRad** or **Feed** entry fields. Use either the Absolute or Incremental Mode.
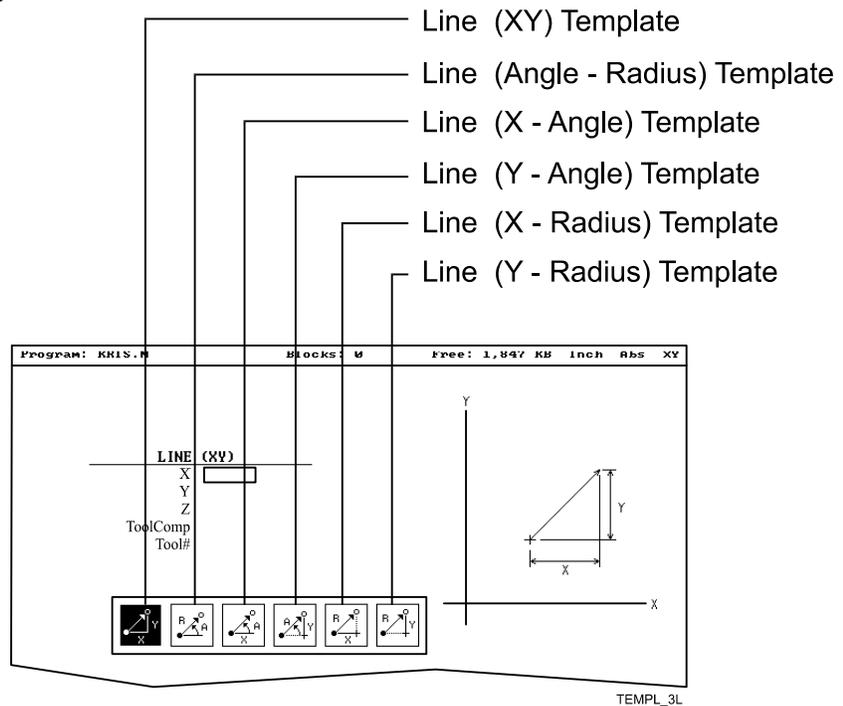
Programming a Move Using XY Location, Radii or Angles

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**. The **RAPID (XY)** or **LINE (XY)** Graphic Menu is displayed.

2. Press **More. . .** (**F4**). Template pop-up menu is displayed. Refer to **Figure 4-12**.


Line (XY) Template
Line (Angle - Radius) Template
Line (X - Angle) Template
Line (Y - Angle) Template
Line (X - Radius) Template
Line (Y - Radius) Template

Figure 4-12, Rapid/Line Templates

3. Highlight the appropriate template, and press **ENTER**. Graphic Menu activates.

4. Enter the required values/settings in the entry fields.

ANILAM

Arcs

You can program Arc moves three different ways:

❑ Using the endpoint and radius.

❑ Using the center and endpoint.

❑ Using the center and angle.

Refer to **Figure 4-13**. Use the templates to access the Graphic Menus for the three arc types.

Arc (Center - Angle) Template

Arc (Center - End Point) Template

Arc (Endpoint - Radius) Template
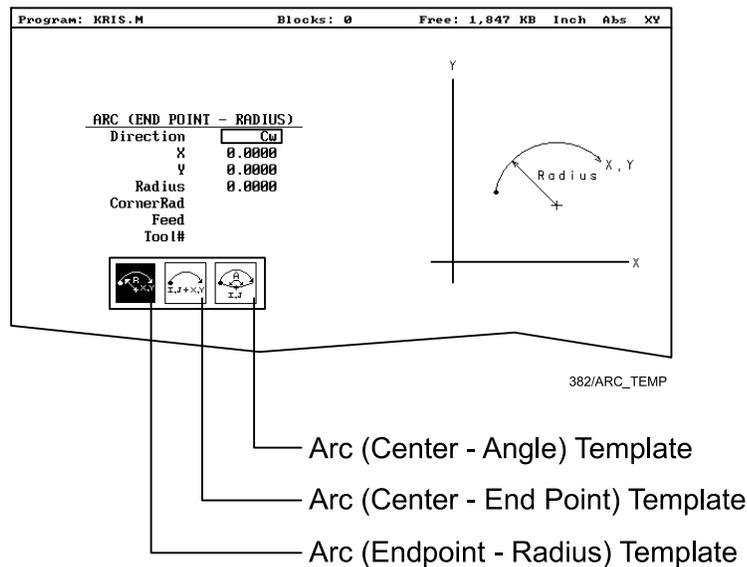
Figure 4-13, Arc Template Menu

To access the Arc Template Menu:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft keys are displayed.

2. Press **Arc** (**F4**). The Arc soft keys are displayed.

3. Press **More…**(**F4**). The Arc Template Menu is displayed.

4. Select the appropriate Arc Template.

NOTE:	Use either the keypad hot keys or soft keys to program arcs. The following sections provide step-by-step instructions on both methods.

Programming an Arc Using an Endpoint and Radius

To define the Endpoint-Radius Arc, enter the Arc direction, the endpoint and the radius. The CNC cuts an Arc of the proper radius, from the present position to the endpoint. The endpoint coordinates are modal values. Modal values must be defined correctly so as to agree with the active mode of the CNC when the cycle runs.

Refer to **Figure 4-14**. There are always two Arcs that 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.
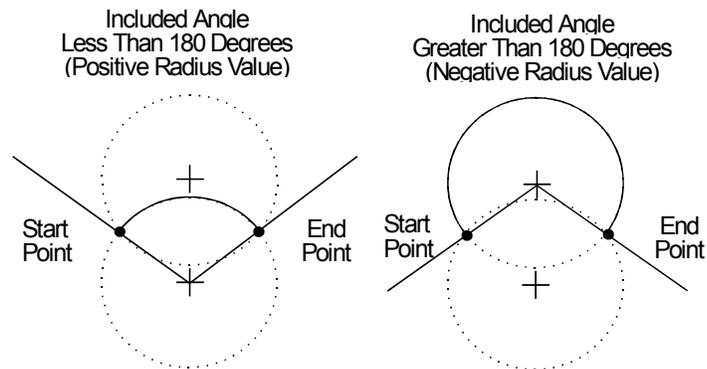
Figure 4-14, Endpoint Radius Arc Types

To program the Arc with an included angle less than 180 degrees, enter a positive radius value. To program the Arc with an included angle greater than 180 degrees, enter a negative radius value. The CNC chooses one of the two possible Arc centers based on the sign of the entered 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**). The Mill secondary soft keys are displayed.

2. Press **Arc** (**F4**) to display the **ARC (END POINT - RADIUS)** Graphic Menu. Refer to Figure 4-15, ARC (END POINT-RADIUS) Graphic Menu.

ANILAM

382/END_RAD
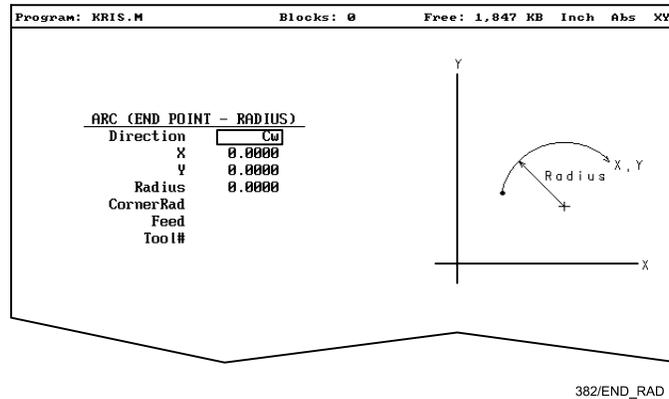
Figure 4-15, ARC (END POINT-RADIUS) Graphic Menu

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

Direction	Direction of the Arc move, counterclockwise (Ccw) or clockwise (Cw). Selection required.
X	The X coordinate of Arc endpoint. Value required.
Y	The Y coordinate of Arc endpoint. Value required.
Radius	The radius of the Arc. Value required, can be 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

To define the Center-Endpoint Arc enter the endpoint, arc center and direction. The CNC will cut the Arc from the present position to the endpoint.

In Absolute Mode, the Arc center and endpoint are measured from Absolute Zero. In Incremental Mode, they are measured from the starting position of the arc.

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

The CNC calculates the radius from the given 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 control will adjust the center or endpoint. 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-Endpoint Arc using hot keys** (the keypad):

1. In Edit Mode, press **3/ARC**. The ARC (END POINT-RADIUS) Graphic Menu is displayed.

2. Press **More...** (**F4**). The Arc Template Pop-Up menu is displayed. (Refer to Figure 4-13, Arc Template Menu).

3. Select the **CENTER-ENDPOINT** template, and press **ENTER**. The ARC (CENTER-END POINT) Graphic Menu prompts for labeled values.

4. Fill in the entry fields as labeled.

To program a **Center-Endpoint Arc using the soft keys**:

1. In Edit Mode, press **Mill** (**F5**). The Mill secondary soft keys are displayed.

2. Press **Arc** (**F4**). The ARC (END POINT-RADIUS) Graphic Menu is displayed.

3. Press **More...** (**F4**). The Arc Template Pop-Up menu is displayed. (Refer to Figure 4-13, Arc Template Menu).

4. Select the CENTER-END POINT template, and press **ENTER**. The **ARC (CENTER-END POINT)** Graphic Menu prompts for the labeled values.

Figure 4-16, Arc (Center-End Point) Graphic Menu

ANILAM
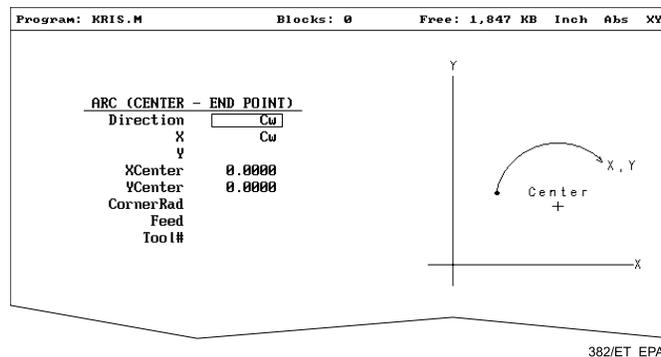
5. Refer to Figure 4-16, Arc (Center-End Point) Graphic Menu and fill in the entry fields labeled as follows:

Direction	Direction of the Arc, counterclockwise (**Ccw**) or clockwise (**Cw**). Selection required.
X	The X coordinate of the Arc endpoint. Optional.
Y	The Y coordinate of the Arc endpoint. Optional.
XCenter	The X coordinate of the Arc's center. Value required.
YCenter	The Y coordinate of the Arc's center. Value required.
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 the Included Angle

Use the Center-Angle Arc to define an Arc, given center position and the number of degrees in the included angle. The CNC cuts the Arc from the machine's present position until the Arc travels the specified number of degrees. The CNC calculates the radius, the distance between the starting position and the center.

Refer to **Figure 4-17** and Figure 4-18, Incremental Mode, Center-Angle Arc. The angle and center position are modal values and must be correctly defined for the current Absolute/Incremental Mode of the CNC. The direction of the Arc and the sign (**+/-**) of the angle control the path of the tool.
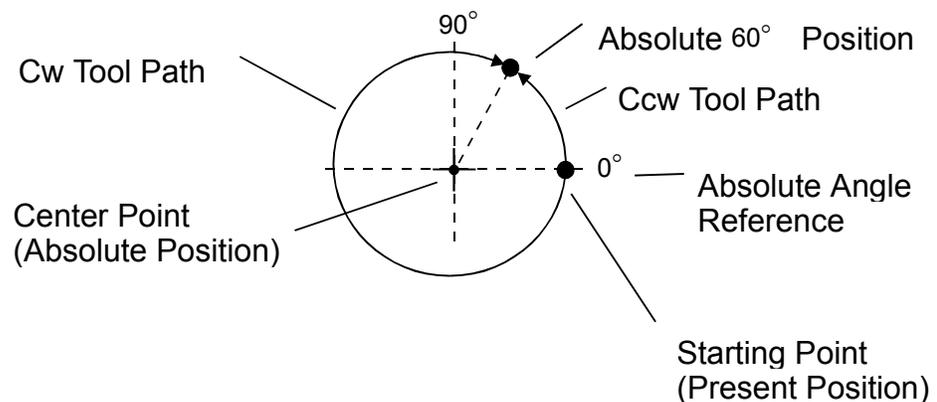
Figure 4-17, Absolute Mode, Center-Angle Arc

Ccw Tool Path

Incremental 60° Position

Starting Point
(Present Position)

Center Point
(Incremental Position)

Incremental 60° Position

Cw Tool Path
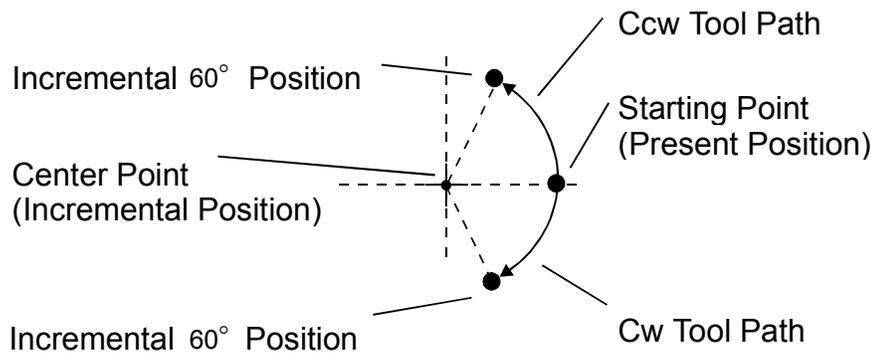
Figure 4-18, Incremental Mode, Center-Angle Arc

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

1. In Edit Mode, press **3/ARC**. The ARC (ENDPOINT-RADIUS) Graphic Menu is displayed.

2. Press **More...** (**F4**). The Arc Template Pop-Up menu is displayed (refer to Figure 4-13, Arc Template Menu).

3. Select the **CENTER-ANGLE** template, and press **ENTER**. The ARC (CENTER-ANGLE) Graphic Menu prompts for the labeled values.

4. Fill in the entry fields as labeled.

To program a **Center-Angle Arc using the soft keys**:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft keys are displayed.

2. Press **Arc** (**F4**). The ARC (END POINT-RADIUS) Graphic Menu is displayed.

3. Press **More...** (**F4**). The Arc Template Pop-Up menu is displayed (refer to Figure 4-13, Arc Template Menu).

4. Select the **CENTER-ANGLE** template, and press **ENTER**. The ARC (CENTER-ANGLE) Graphic Menu prompts for the labeled values.

382/CTR_ANG
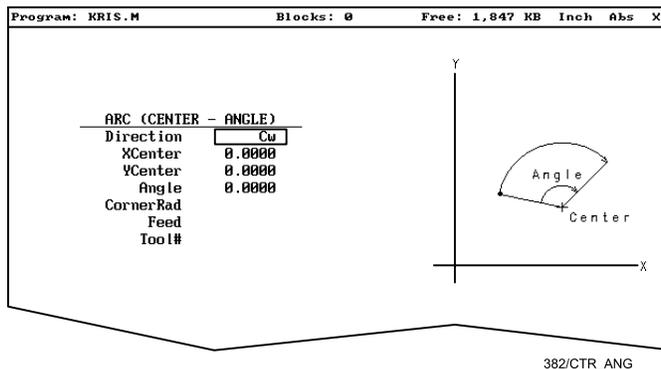
Figure 4-19, Arc (Center-Angle) Graphic Menu

5. Refer to **Figure 4-19** and fill in the entry fields labeled as follows:

Direction	Direction of Arc (clockwise or counterclockwise). Required.
XCenter	The X coordinate of Arc center. Value required.
YCenter	The Y coordinate of Arc center. Value required.
Angle	Included angle of the Arc. Value required.
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 control supports spindle control and a broad range of M-code functions. The machine builder determines which M-Codes are set up. Refer to the builder's technical data to find out exactly, which M-Codes are available.

Some programmed events perform the same function as M-Codes. Refer to **Table 4-3** for the most commonly used M-code functions.

Table 4-3, Commonly Available M-Codes and Functions

M-Function	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**). Graphic Menu prompts for **MCode** number and any required X, Y values.
2. Type values and press **Save** (**F10**) or **ENTER** to the **MCode** block is added to the program.

Dry Run M-Codes

In Dry Run Mode, the machine axes (X and Y) 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 M105. 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 on	Enables machine Dry Run Mode. Program runs at dry run feedrates specified in the Setup Utility.
M107	Cancel dry run	Cancels active Dry Run Mode.

Section 5 - Programming Canned Cycles, Ellipses, and Spirals

Drilling Cycles

Drill Cycles simplify the programming required for repetitive drilling operations. Drill Cycles, selected from the Program Editor's **Drill** (**F3**) Pop-Up Menu, include:

- ❑ Basic
- ❑ Drilling Off
- ❑ Pattern
- ❑ Bolt Hole

Drill Cycles are modal. When the CNC runs a **BasicDrill** block, it runs that cycle at the endpoint of each subsequent move, until it encounters a **DrillOff** block. Drill Cycle characteristics cannot be changed between moves. It must be shut off and a new one programmed.

Basic Drill Cycles

A Basic Drill Cycle is similar to a Manual Z move. It holds the program and prompts you to move the Z-axis. When the Z move is completed, press **START** to continue the program. The **ZDepth** is only a reminder. The CNC will run the Drill Cycle at the endpoint of every move that follows the cycle, until turned off.

The **StartHgt** (Start Height) is the Z height that the tool must reach before it proceeds to the next drilling location. It is designed to prevent the quill from attempting to move to the next position before the tool has cleared the surface of the work. The **StartHgt** can be any positive or negative value but, when used, should be 0.1 in. (2 mm) above the work surface. The default is 0.

> **NOTE:** The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

> **NOTE:** Do not use the **StartHgt** feature on offline software.
>
> On 2-axis systems without a DRO on the Z-axis, do not fill in a **StartHgt** for drilling operations.

A **DrillOff** block must be programmed to turn off a Basic Drill Cycle. You can program any number of patterns and moves before turning off the cycle.

After a **BasicDrill** block, program moves to tell the CNC where the cycle will run. The CNC will run the Basic Drill Cycle at the endpoint of all subsequent moves until a **DrillOff** block is run.

To program a **BasicDrill** block:

1. In Edit mode, press **Drill** (**F3**). The Drill cycle pop-up menu is displayed.

2. Highlight **Basic**, and press **ENTER**. The **BASIC DRILLING** Graphic Menu is displayed. Refer to **Figure 5-1**.
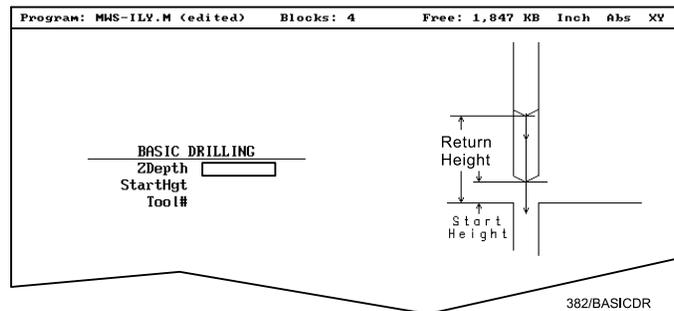
Figure 5-1, Basic Drilling Graphic Menu

3. Enter required values and settings in the entry fields. With the last entry field highlighted, press **ENTER**. Display clears and 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**). The Drill cycle pop-up menu is displayed.

6. Highlight **Drilling Off**, and press **ENTER** to cancel the Drilling Mode.

Drill Pattern

The Pattern Cycle instructs the CNC to run a series of regularly spaced moves. At the endpoint of each of these moves the CNC will run a Drill Cycle. The Drill Cycle used with the pattern must appear in the program before the Pattern Cycle. In a Pattern Cycle, you enter the size, location, spacing, tool # and rotation of the pattern.

You should first program a Basic Drill Cycle to describe the hole being drilled. The Drill Cycle is followed by one or more moves or patterns. The moves and patterns position the CNC everywhere the Drill Cycle is to be run. Program a **DrillOff** block to cancel the Drill Cycle.

NOTE:	The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle.

To program a Pattern Cycle:

1. In Edit Mode, press **Drill** (**F3**). The Drill Cycle Pop-Up Menu is displayed.

2. Select **Pattern**, and press **ENTER**. The PATTERN DRILL Graphic Menu is displayed. Refer to Figure 5-2, Pattern Drill Graphic Menu.
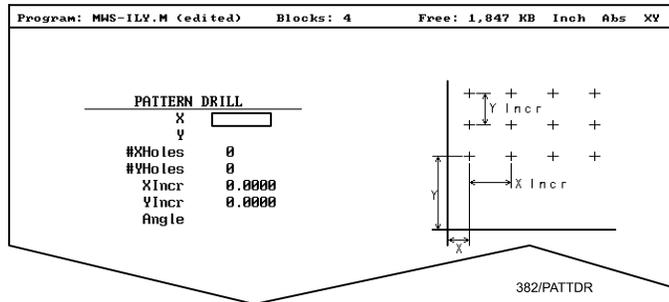
Figure 5-2, Pattern Drill Graphic Menu

3. Fill in the entry fields labeled as follows:

X	X coordinate of corner hole, no entry puts corner hole at current location. Optional.
Y	Y coordinate of corner hole, no entry puts corner hole at current location. Optional.

NOTE: Use absolute center point coordinates whenever possible.

#XHoles	Number of rows that lie along the X-axis. Value must be greater than 0. Value required.
#YHoles	Number of rows that lie along the Y-axis. Value must be greater than 0. Value required.

NOTE: Entering a one for either of these values writes a block to drill a single row or column.

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

NOTE: Remember to program a **DrillOff** block after drilling is completed.

Bolt Hole Pattern

Refer to **Figure 5-3**. 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, a previously programmed Drill Cycle can be run.

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

> **NOTE:** The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle.

To program a Bolt Hole Cycle:

1. In Edit Mode, press **Drill** (**F3**). The Drill Cycle Pop-Up Menu is displayed.

2. Select **Bolt Hole**, and press **ENTER**. The BOLTHOLE DRILL Graphic Menu prompts for labeled values.
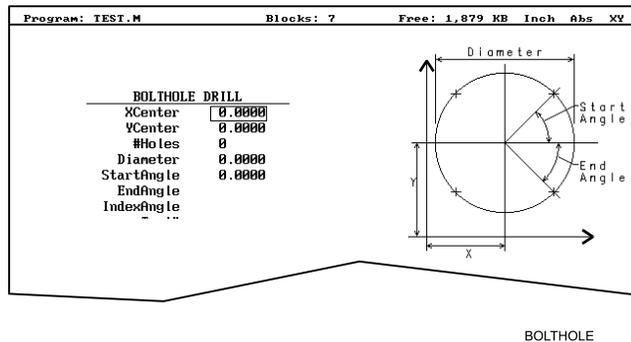
Figure 5-3, BOLTHOLE DRILL Graphic Menu

3. Fill in the entry fields as labeled:

XCenter	Absolute X position of bolt hole pattern center. No entry puts center of Bolt Hole pattern at X0. Value required.
Ycenter	Absolute Y position of bolt hole pattern center. No entry puts center of Bolt Hole pattern at Y0. Value required.

> **NOTE:** Use absolute center point coordinates whenever possible.

#Holes	Number of equally spaced holes to be drilled in the circular pattern. Value required.
Diameter	Diameter of the circular pattern. Value required.

StartAngle	The number of degrees (from the 3 o' clock position) to the first hole. Value required.
EndAngle	The number of degrees (from the 3 o'clock position) to the first hole. Value required.
IndexAngle	The number of degrees the 3 o'clock reference position (the entire pattern) is rotated around the center. Optional.

> **NOTE:** Remember to program a **DrillOff** block after the drilling is complete.

Pocket Cycles

Pocket canned cycles simplify the programming required to mill out pockets. Select one of the following pocket canned cycles from the Program Editor's **Pocket** (**F4**) Pop-Up Menu:

- ❑ Face Pocket
- ❑ Rectangular Profile
- ❑ Circular Profile
- ❑ Rectangular Pocket
- ❑ Circular Pocket
- ❑ Frame Pocket
- ❑ Hole
- ❑ Irregular Pocket

> **NOTE:** A **Tool#** programmed in a Pocket Cycle automatically activates the necessary tool compensation for that cycle. Ensure that the length offset and diameter listed on the Tool Page are correct.

Facing Cycle

Face Cycles simplify the programming required to face the surface of a part. When a Face Cycle runs, the CNC rapids to the X, Y start position, holds and prompts for a move to the Z position (**ZDepth**) for facing.

Position Z and press **START**.   The CNC faces the part to the **Length** and **Width** values entered. The CNC then rapids back to the starting position.

The tool begins its cut one tool radius from the start point, moving 90 degrees to the right of the step-over direction (viewed from the rear), refer to Figure 5-4, Face Cycle Tool Approach. Face Cycles can start in any corner and cut in any direction, depending on the sign of the step-over. To ensure complete facing, a slightly oversize **Length** and **Width** value is recommended.
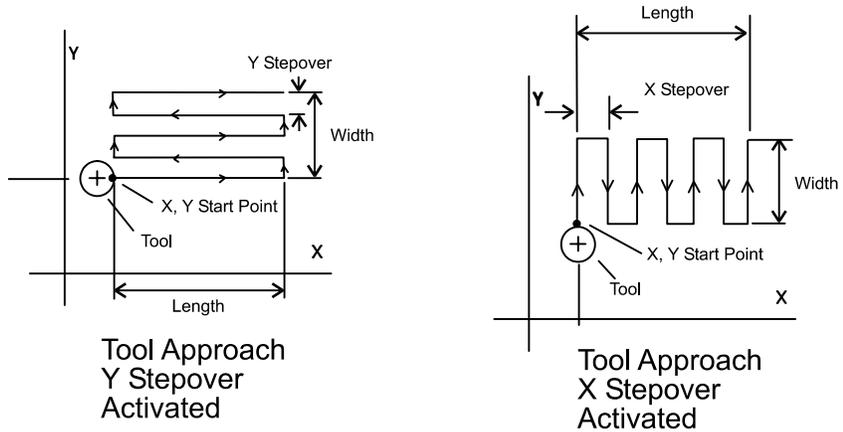
Figure 5-4, Face Cycle Tool Approach

NOTE:	The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

To program a Face Cycle:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Move the highlight to select **Face**, and press **ENTER**. The FACE POCKET Graphic Menu prompts for labeled values.
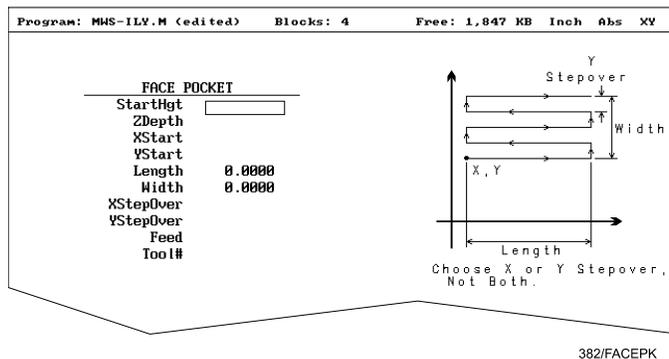
Figure 5-5, Face Pocket Cycle Graphic Menu

3. Refer to **Figure 5-5** and fill in the entry fields labeled as follows:

StartHgt	Z starting position. Optional.
ZDepth	The Z position of the faced surface. Optional.
XStart	X coordinate of the starting point. Modal, optional.
YStart	Y coordinate of the starting point. Modal, optional.

NOTE: Use absolute starting point coordinates whenever possible.

Length	The X-axis length of face. Value required.
Width	The Y-axis length of face. Value required.
XStepover	Length of the cut in the X-axis direction. 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 radius. Optional.
YStepover	Width of the cut in the Y-axis direction. 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 radius. Optional.

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

Tool#	Active tool. Optional.

Rectangular Profile Cycle

The Rectangular Profile Cycle simplifies the programming needed to clean up the inside or outside profile of a rectangle. When a Rectangular Profile Cycle runs, the CNC rapids to the ramp starting position, holds and prompts for the required Z move.

Refer to **Figure 5-6**. Once the Z-axis is positioned, press **START**. The machine ramps into the work along Ramp #1, cuts around the rectangle to **Length** and **Width** and ramps away from the work along Ramp #2. If **DepthCut** is used, the machine rapids from the endpoint of Ramp #2 to the starting point of Ramp #1 to the start of each pass.
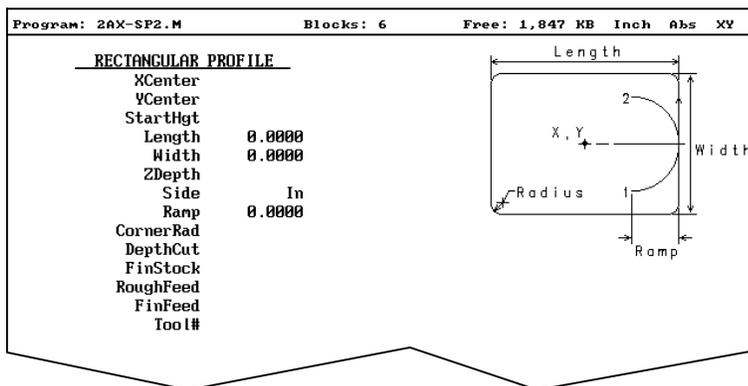
Figure 5-6, Rectangular Profile Graphic Menu

Refer to **Figure 5-7**. When cutting an outside profile, the tool ramps into the work along Ramp #1 and away from the work along Ramp #2 as shown.
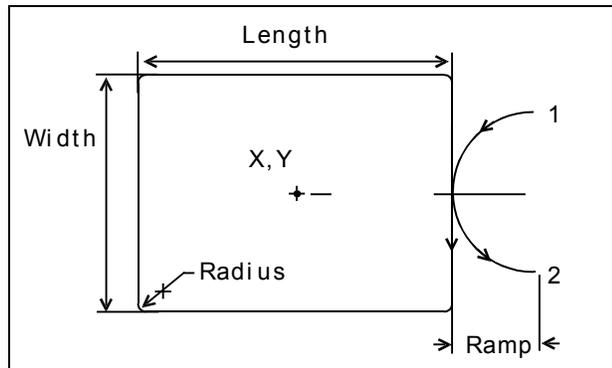
Figure 5-7, Ramp Position for Outside Profile

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC calculates and runs the number of passes required to get from the **StartHgt** to the **Zdepth**. It cuts to the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds an additional **ZDepth** pass to cut to the finished **Length** and **Width** dimensions. You must calculate the required Z-axis finish stock. If a negative **FinStock** value is entered, the CNC will leave the finish stock, but not execute a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC runs Feed moves at the current feedrate. If **RoughFeed** is entered, only the rough feed is affected. When **FinFeed** is entered, only the feedrate is affected.

NOTE:	The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

ANILAM

To program a Rectangular Profile Cycle:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Select **Rect.Profile**, and press **ENTER**. The PROFILE RECTANGULAR Graphic Menu prompts for labeled values.

3. Fill in the entry fields as labeled:

XCenter	X coordinate of the center. If no coordinate is entered, pocket is centered at current position. Optional.
YCenter	Y coordinate of the center. If no coordinate is entered, pocket is centered at current position. Optional.
StartHgt	Z starting position, used by the CNC to calculate the number of passes when cutting to ZDepth on each pass. Not required if only doing one pass, or when ZDepth is not used. Optional.
Length	Finished length of rectangle. Value required.
Width	Finished width of rectangle. Value required.
ZDepth	Finished Z position of milled area. Optional.
Side	Setting for cutting on the inside of the profile, (**In**) or the outside (**Out**). This setting is switched between selections using the (**+/-**) key.
Ramp	Radius of the ramping move. Value required.
CornerRad	Corner radius setting. When negative value is used, both the direction of cut and the starting and endpoints are reversed. Optional.
DepthCut	Z-axis increment for each pass. Optional.
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make a finish pass. Optional.
RoughFeed	Rough pass feedrate. Optional.
FinFeed	Finish pass feedrate. Optional.
Tool#	Active tool. Optional.

Circular Profile Cycle

The Circular Profile Cycle simplifies the programming needed to clean up the inside or outside profile of a circle. When a Circular Profile Cycle runs, the CNC rapids to the ramp starting position, holds and prompts for the required Z move.

Position the Z-axis and press **START**. The machine ramps into the work along Ramp #1, cuts the circle to the **Diameter** and ramps away from the work along Ramp #2 (refer to **Figure 5-8**). If **DepthCut** is used, the machine rapids from the endpoint of Ramp #2 to the starting point of Ramp #1 to starting point of each pass.
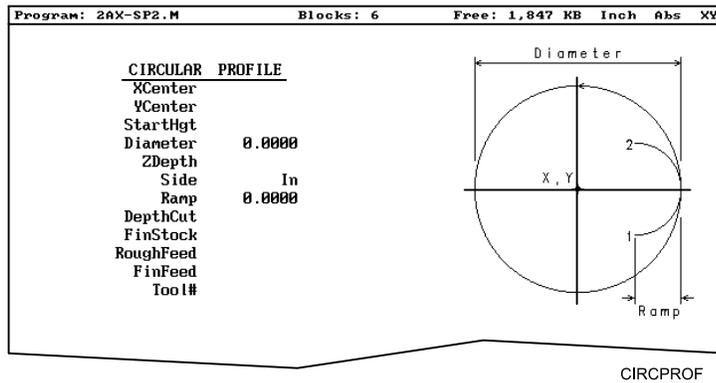
Figure 5-8, Circular Profile Graphic Menu

When cutting an outside profile, the tool ramps into the work along Ramp #1 and away from the work along Ramp #2 as shown. (Refer to **Figure 5-9**.)
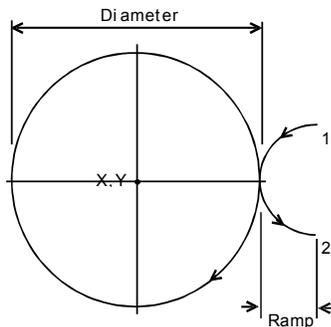
Figure 5-9, Ramp Position for Outside Profile

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC runs the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds an additional **ZDepth** pass to cut to the **Diameter**. You must calculate the required Z-axis finish stock. If a negative **FinStock** value is entered, the CNC will leave the finish stock, but not execute a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC runs feed moves at the current feedrate. If **RoughFeed** is entered, only the rough feed is affected. When **FinFeed** is entered, only the feedrate is affected.

NOTE:	**ZDepth** must be lower than **StartHgt**. A **ZDepth** entry is not necessary if you program only one level plus finish stock.

NOTE:	The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

To program a circular profile:

1. In Edit Mode, press **Pocket** (**F4**). The **Pocket** Pop-Up Menu is displayed.

2. Select **Circ.Profile**, and press **ENTER**. The PROFILE CIRCULAR Graphic Menu prompts for labeled values.

3. Refer to Figure 5-8, Circular Profile Graphic Menu and fill in the entry fields labeled as follows:

XCenter	X coordinate of pocket center. If no coordinate is entered, pocket is centered at current position. Optional.
YCenter	Y coordinate of pocket center. If no coordinate is entered, pocket is centered at current position. Optional.
StartHgt	Z starting position, used by the CNC to calculate the number of passes when cutting to **ZDepth** on each pass. Not required if only doing one pass, or when **ZDepth** is not used. Optional.
Diameter	Finished diameter of circle. If a negative value is used, both the direction and the starting and endpoints of the cut are reversed. Value required.
ZDepth	Finished Z position of milled area. Optional.
Side	Setting for cutting on the inside (**In**) of the profile or the outside (**Out**). Press the (**+/-**) key to switch the setting.
Ramp	Radius of the ramp into and away from the cut, may be set to zero. Value required.

DepthCut	Depth the machine takes in a single pass, defaults to **ZDepth** minus the finish stock if no position is entered. Optional.
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make a finish pass. 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. The CNC rapids to the center of the lower-left radius, holds, and prompts operator to position the Z-axis. Position the Z-axis and press **START** to resume. The CNC feeds to the narrow center and mills successively larger rectangles until **Length** and **Width** are reached.

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC runs the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds an additional **ZDepth** pass to cut to the finished **Length** and **Width**. You must calculate the required Z-axis finish stock. If a negative **FinStock** value is entered, the CNC will leave the finish stock, but not execute a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC runs feed moves at the current feedrate. If **RoughFeed** is entered, only the rough feed is affected. When **FinFeed** is entered, only the feedrate is affected.

NOTE: **ZDepth** must be lower than **StartHgt**. A **ZDepth** entry is not necessary if you program only one level plus finish stock.

NOTE: The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

To program a Rectangular pocket:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Select **Rectangular**, and press **ENTER**. The RECTANGULAR POCKET Graphic Menu prompts for labeled values.
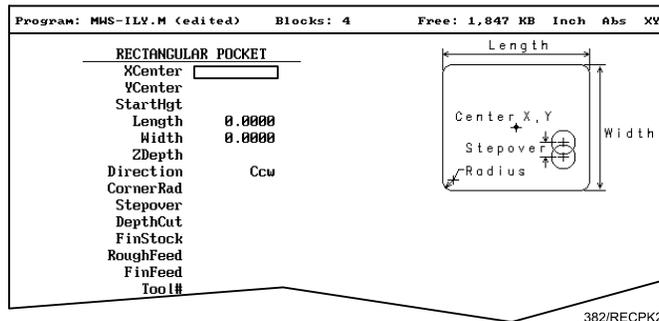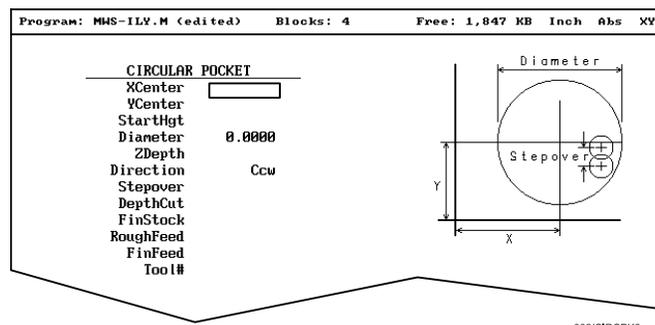

Figure 5-10, Rectangular Pocket Graphic Menu

3. Refer to **Figure 5-10** and fill in the labeled entry fields as follows:

XCenter X coordinate of the pocket's center. If no coordinate is entered the pocket is centered at current position. Optional.

YCenter Y coordinate of the pocket's center. If no coordinate is entered the pocket is centered at current position. Optional.

NOTE: Use absolute center point coordinates whenever possible.

StartHgt Z starting position, used by the CNC to calculate the number of passes when cutting to **ZDepth** on each pass. Not required if only doing one pass, or when **ZDepth** is not used. Optional.

Length The inside (Y length) of the finished pocket. Value required.

Width The inside (X width) of the finished pocket. Value required.

ZDepth The depth (Z position) of the finished pocket. Optional.

Direction This setting enables you to choose between a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. The setting is switched by using the (+/-) key. Selection required.

Stepover Step-over is the size of cut. If no value is entered, CNC defaults to 70% of the active tool radius. Maximum step-over is 70% of the active tool radius. Optional.

> **NOTE:** The CNC will enable you to enter incorrect step-over values, but the program may not run. The program will stop and generate an error message if the step-over value is too large.

DepthCut	Depth the machine takes in a single pass, defaults to a single **ZDepth** cut minus the finish stock if no position is entered. Optional.
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make a finish pass. Optional.
RoughFeed	Rough pass feedrate. Optional.
FinFeed	Finish pass feedrate. Optional.
Tool#	Active tool. Optional.

Circular Pocket Cycle

Circular Pocket Cycles simplify the programming required to mill out circular pockets. The CNC rapids to the center, holds, and prompts operator to position the Z-axis. Position the Z-axis and press **START**. The CNC spirals from the center outward until it reaches the **Diameter**.

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC runs the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds an additional **ZDepth** pass to cut to the **Diameter**. You must calculate the required Z-axis finish stock. If you enter a negative **FinStock** value, the CNC will leave the finish stock, but not execute a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC runs feed moves at the current feedrate. If you enter a **RoughFeed**, only the rough feed is affected. When you enter a **FinFeed**, only the feedrate is affected.

NOTE: The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

NOTE: **ZDepth** must be lower than **StartHgt**. A **ZDepth** entry is not necessary if you program only one level plus finish stock.

To program a Circular pocket:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Select **Circular**, and press **ENTER**. The **CIRCULAR POCKET** Graphic Menu prompts for labeled values.

Figure 5-11, Circular Pocket Cycle Graphic Menu

3. Refer to <u>Figure 5-11, Circular Pocket Cycle Graphic Menu</u> and fill in the entry fields, labeled as follows:

XCenter	X coordinate of the pocket center. If you enter no coordinate, the pocket is centered at current position. Optional.
YCenter	Y coordinate of the pocket center. If no coordinate is entered the pocket is centered at current position. Optional.

> **NOTE:** Use absolute center point coordinates whenever possible.

StartHgt	Z starting position. Used by the CNC to calculate the number of passes when cutting to **ZDepth** on each pass. Not required if only doing one pass, or when **ZDepth** is not used. Optional.
Diameter	Diameter of the pocket. Value required.
ZDepth	The depth (Z position) of the finished pocket. Optional.
Direction	This setting enables you to choose between a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. The setting is switched by using the (+/-) key. Selection required.
Stepover	Step-over is the size 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 radius. Optional.

> **NOTE:** The CNC will enable the operator to enter incorrect step-over values, but the program may not run. A program will stop and generate an error message if the step-over value is too large.

DepthCut	Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock if no position is entered. Optional.
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make a finish pass. Optional.
RoughFeed	Rough pass feedrate. Optional.
FinFeed	Finish pass feedrate. Optional.
Tool#	Active tool. Optional.

Frame Pocket Cycle

Frame Pocket Cycles simplify the programming required to mill out a framed pocket. The CNC rapids from the center to a starting position adjacent to the island, holds, and prompts the operator to position the Z-axis. Position the Z-axis and press **START**. The CNC mills across the frame, around the outside edge then toward the island in ever-decreasing rectangles. When the frame is completed, the CNC returns to the starting position and holds for a Z move out of the frame. Move the Z-axis and press **START**. The CNC finishes the cycle with a Rapid move back to the center position.

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC runs the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds an additional **ZDepth** pass to cut to the finished dimensions. You must calculate the required Z-axis finish stock. If a negative **FinStock** value is entered, the CNC will leave the finish stock, but not execute a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC runs feed moves at the current feedrate. If **RoughFeed** is entered, only the rough feed is affected. When **FinFeed** is entered, only the feedrate is affected.

> **NOTE:** **ZDepth** must be lower than **StartHgt**. A **ZDepth** entry is not necessary if you program only one level plus finish stock.

> **NOTE:** The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

To program a Frame Cycle:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Select **Frame**, and press **ENTER**. The **FRAME POCKET** graphic prompts for labeled values.
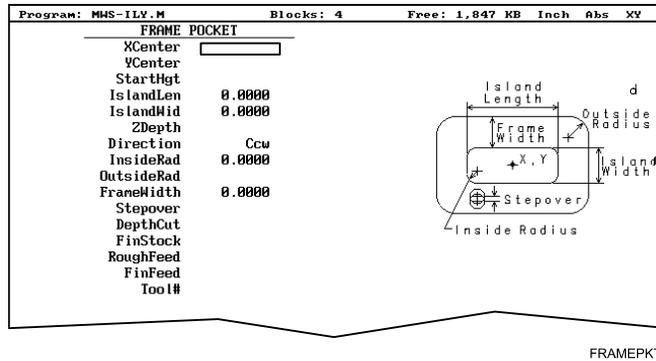
FRAMEPKT

Figure 5-12, Frame Pocket Cycle Graphic Menu

3. Refer to **Figure 5-12** and fill in the entry fields, labeled as follows:

XCenter	X coordinate of the center. If no coordinate is entered, pocket is centered at the machine's current position. Optional.
YCenter	Y coordinate of the center. If no coordinate is entered, pocket is centered at the machine's current position. Optional.

NOTE: Use absolute center point coordinates whenever possible.

StartHgt	Z starting position. Used by the CNC to calculate the number of passes when cutting to **ZDepth** on each pass. Not required if only doing one pass, or when **ZDepth** is not used. Optional.
IslandLen	The outside length (X-axis) of finished island. Value required.
IslandWid	The outside width (Y-axis) of finished island. Value required.
ZDepth	Depth of the finished pocket. Optional.
Direction	This setting enables you to choose between conventional and climb milling for the pocket. The choices are clockwise (**Cw**) or counterclockwise (**Ccw**). Press the (**+/-**) key to switch the setting.
InsideRad	Radius of the island corners. Value required.
OutsideRad	Outside radius of the frame corners. Optional.
FrameWidth	Width of finished frame. Value required.
Stepover	Size of cut. If no dimension is entered, CNC defaults to 70% of the active tool radius. Maximum permitted step-over is 70% of active tool radius. Optional.

NOTE:	It is possible to inadvertently program an incorrect step-over value. If the step-over value is too large, the CNC halts the program and generates an error message.

DepthCut	Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock if no position is entered. Optional.
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make a finish pass. Optional.
RoughFeed	Rough pass feedrate. Optional.
FinFeed	Finish pass feedrate. Optional.
Tool#	Active Tool. Optional.

Hole-Mill Pocket Cycle

Hole Cycles simplify the programming required to cut through holes and clean up existing holes. The CNC rapids to the center, holds, and prompts operator to position the Z-axis. Position the Z-axis and press **START**. The CNC feeds away from the center and into the circumference along the ramp shown on the screen. (Refer to Figure 5-13, Hole-Mill Pocket Graphic Menu.) After it cuts the hole, it ramps away from the circumference then moves radially back to the center.

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC runs the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds an additional **ZDepth** pass cutting to finished **Diameter**. You must calculate the required Z-axis finish stock. If you enter a negative **FinStock** value, the CNC will leave the finish stock, but not execute a finish pass.
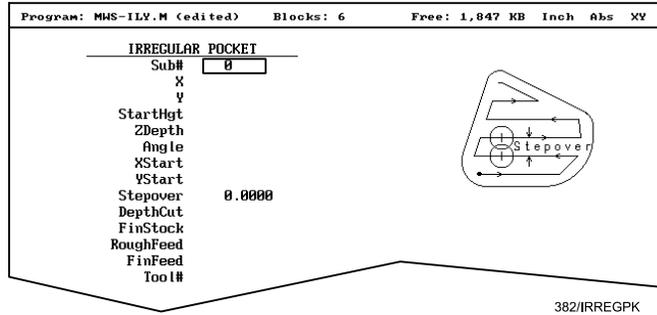
If **RoughFeed** and **FinFeed** are left blank, the CNC runs feed moves at the current feedrate. If **RoughFeed** is entered, only the rough feed is affected. When **FinFeed** is entered, only the feedrate is affected.

NOTE:	**ZDepth** must be lower than **StartHgt**. A **ZDepth** entry is not necessary if you program only one level plus finish stock.

31-July-05

> **NOTE:** The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

To program a Hole Cycle:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Select **Hole**, and press **ENTER**. The **HOLE-MILL POCKET** Graphic Menu prompts for labeled values.
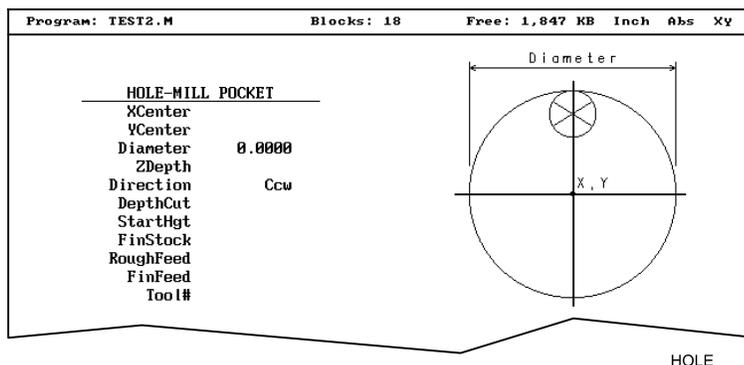
Figure 5-13, Hole-Mill Pocket Graphic Menu

3. Refer to **Figure 5-13** and fill in the entry fields labeled as follows:

XCenter	X coordinate of the center. If no coordinate is entered the pocket is centered at current position. Optional.
YCenter	Y coordinate of the center. If no coordinate is entered the pocket is centered at current position. Optional.

> **NOTE:** Use absolute center point coordinates whenever possible.

Diameter	Diameter of the pocket. Value required.
ZDepth	The depth (Z position) of the finished pocket. Optional.
Direction	This setting enables you to choose between a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press the (**+/-**) key to switch the setting. Selection required.
DepthCut	Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock if no position is entered. Optional.

StartHgt	Z starting position. Used by the CNC to calculate the number of passes when cutting to **ZDepth** on each pass. Not required if only doing one pass, or when **ZDepth** is not used. Optional.	
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make 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 required to mill out an irregular pocket. An Irregular Pocket Cycle must call a subprogram. The moves in the subprogram define the pocket's outline. The information in the **Pocket** block specifies how the pocket is milled.

The subprogram should start with an absolute rapid move to the profile starting position, followed by the other moves and arcs necessary to define the outline. The subprogram must end at 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 "Section 4 - Writing Programs" for information on how to program a subprogram. Refer to "Section 6 - Editing Programs" for information on how to comment out a block.

Refer to Figure 5-14, Step-over Move Orientation. The pocket interior is milled out with consecutive side-to-side moves that step-over across the pocket. The machine reverses direction when it reaches any portion of the outline. The moves must be oriented so that an irregularity in the outline does not prevent the entire pocket from being milled out.

31-July-05
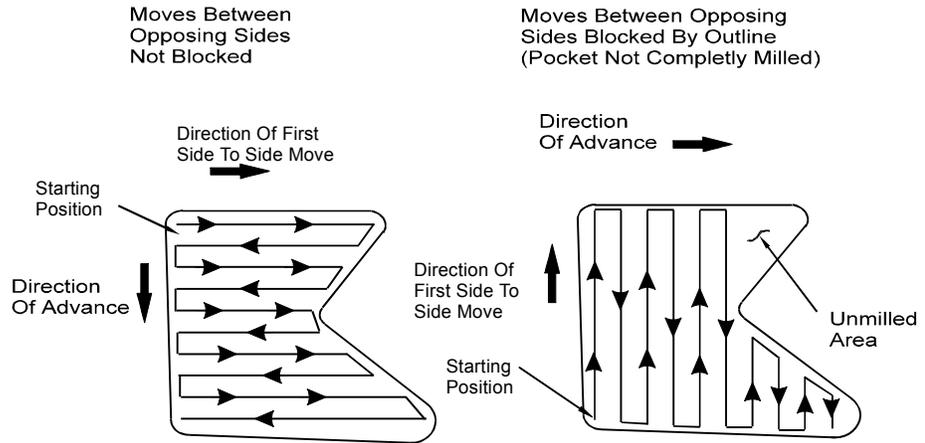
Figure 5-14, Step-over Move Orientation

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 5-15**.
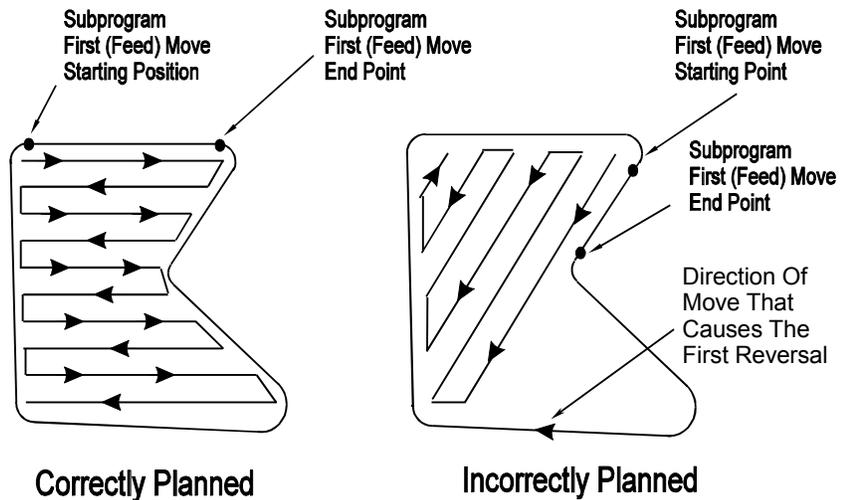
Figure 5-15, Orienting Step-over from the Subprogram

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. For most irregular pockets, leave the **Angle**, **X** and **Y** fields blank.

NOTE:	Use a Line move for the first move in the subprogram when possible.
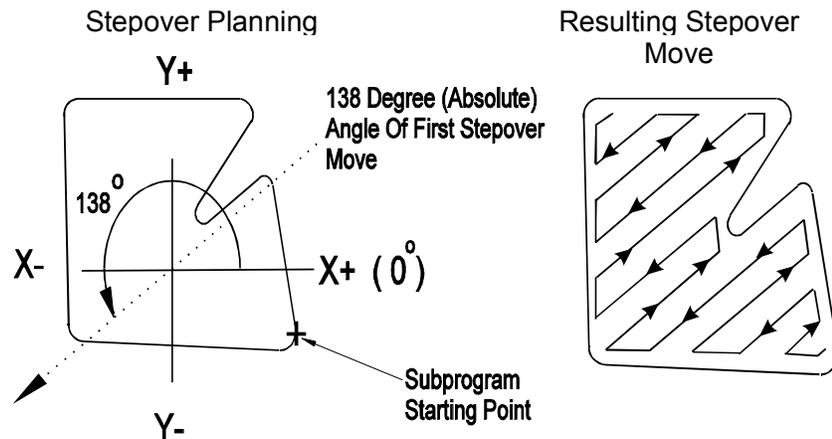
Figure 5-16, Specifying Move Angle

Refer to **Figure 5-16**. When you enter an **Angle** value, the direction of the first move is forced to the absolute **Angle** entered. The **Angle** must point toward the inside of the pocket.

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

Enter X and Y values to force the starting position of the first move to the position entered. This position must be adjusted for tool diameter compensation and **FinStock** if used.

Irregular Pocket Execution

During an irregular Pocket Cycle, the CNC rapids to the starting position, holds and prompts you to position the Z-axis. Position the Z-axis and press **START**. The CNC mills out the pocket with side-to-side moves that step-over across the pocket.

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

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

StartHgt and **ZDepth** are optional, but must be used together if at all. If you enter a **DepthCut**, the CNC runs the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass. The **StartHgt** is subtracted from the total distance before the CNC calculates the required number of passes.

When **FinStock** is used, the CNC leaves the specified amount on the profile and adds a **ZDepth** finish pass. The finish pass has two parts: the milling moves that step-over across the pocket (to cut to depth) and a pass around the profile written into a subprogram. You must calculate the required Z-axis finish stock. If you enter a negative **FinStock** value, the CNC will leave the finish stock, but not execute a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC runs feed moves at the current feedrate. If you enter a **RoughFeed**, only the rough feed is affected. When **FinFeed** is entered, only the feedrate is affected.

NOTE:	**ZDepth** must be lower than **StartHgt**. A **ZDepth** entry is not necessary if you program only one level plus finish stock.

NOTE:	The Z-axis is not a controlled axis. You must make Z moves manually. Make the Z move prior to running the cycle. The **StartHgt** entry field serves as a reminder of the appropriate Z starting height. Recommended **StartHgt** is 0.100 inch (2 mm) above the surface being cut.

To program an irregular pocket:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu is displayed.

2. Select **IRRegular**, and press **ENTER**. The **IRREGULAR POCKET** Graphic Menu prompts for labeled values. Refer to .

Figure 5-17, Irregular Pocket Graphic Menu

3. Fill in the **IRREGULAR POCKET** entry fields labeled as follows:

Sub#   Subroutine that defines shape of pocket. Value required.

X   X coordinate of the step-over starting position. If no value is entered, it is located at 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, it is located at the starting position of the first move of the subprogram. Optional.

NOTE: **X** and **Y** values must be used together, if at all.

StartHgt   Z starting position. Used by the CNC to calculate the number of passes when cutting to **ZDepth** on each pass. Not required if only doing one pass, or when **ZDepth** is not used. Optional.

ZDepth   The depth (Z position) of the finished pocket. Optional.

Angle   Absolute angle of the first move. If left blank, the first move of 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   Size of cut. If no dimension is entered, CNC defaults to 70% of the active tool radius. Maximum permitted step-over is 70% of active tool radius. Value required.

DepthCut	Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock if no position is entered. Optional.
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is entered. If a negative value is entered, the CNC will leave the stock, but not make a finish pass. 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.

Subprogram

Repetitive operations can be programmed in a subprogram, called from the main program. Subprograms also provide some useful programming options:

❑   They can be called or nested (up to ten levels) from other subprograms.

❑   They can repeat or loop, incrementally moving along any axis each time the loop runs. Looped subprograms can be rotated, scaled or mirrored.

The following examples describe two situations where subprograms save time.

Situation 1 (Repetitive Drilling Cycle)

A workpiece must be center-drilled, drilled, and then counterbored. Each of the three tools must go to the same ten hole positions in a pattern, for a total off thirty-programmed hole locations. Use a subprogram to program the ten hole locations. Call the subprogram three times from the main program (once for each tool).

Situation 2 (Rough and Finish Cycles)

The outside of a workpiece must be roughed out with a roughing mill, then finished with a finishing mill. A subprogram contains the profile. The subprogram is called from the main program twice, once for each tool. The tool diameter can be set to 0.5300 inch for the 0.5 inch roughing mill and 0.5000 inch for the 0.5 inch finishing mill. Tool #1 will leave 0.0150 inch per side, for tool #2 to clean up to size.

Subprogram Structure

When you use subprograms, define the end of the main program and the start and end of each subprogram. Main and subprogram end blocks are inserted in addition to the **<End Of Program>** block that is displayed at the end of every program.

Program Structure Example

Main programs must have an **EndMain** block. Subprograms must start at a **Sub** block and end at an **EndSub** block.

```
 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
 9   Sub 1
10   ZMove      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   ZMove      Z  0.1000
18   EndSub
19   <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 to execute the subprogram through to Block 18. It then returns to the main program at Block 4.

Organizing Programs Containing Subprograms

To organize a program containing a subprogram:

1. Write the main program, including the subprogram **Call** block. At this point, you only need a general idea of the subprogram's purpose, but you do need to keep track of the call number. The same **Call** number is given to the matching subprogram when it is written. A subprogram can be called more than once.

2. Insert an **EndMain** block at the end of the main program.

3. Start writing the subprogram by adding a **Sub** block. Each **Sub** block is followed by the subprogram **Call** number. No two subprograms can have the same number. A subprogram number can be any number from 1 to 9999.

4. Write the subprogram blocks.

5. Finish the subprogram with an **EndSub** block.

Calling Subprograms from the Main Program

To call a subprogram from the main program:

1. In Edit Mode, press **Sub** (**F8**). Subprogram soft key labels are displayed.

2. Press **Call** (**F3**). The CNC prompts for the subprogram number.

3. Type the subprogram number, and press **ENTER**. The CNC adds a **Call** block to the program.

Ending Main Programs

To write an **EndMain** block:

1. In Edit Mode, press **Sub** (**F8**). Soft key subprogram labels are displayed.

2. Press **EndMain** (**F4**). The **EndMain** is added to the program.

Starting Subprograms

To write an **Sub** block:

1. In Edit Mode, press **Sub** (**F8**). Subprogram soft keys are displayed.

2. Press **Sub** (**F2**). The CNC prompts for the subprogram number.

3. Type the **Sub** number, and press **ENTER**. The **Sub** block is added to the program. The **Sub** number must agree with the matching **Call** number.

Ending Subprograms

To program an **EndSub** block:

1. In Edit Mode, press **Sub** (**F8**). The subprogram soft keys are displayed.

2. Press **EndSub** (**F2**). The **EndSub** block is added to the program.

Looping Subprograms

Looping subprograms repeat themselves a set number of times before returning to the main program. The CNC can also increment the position each time the loop runs.

> **NOTE:** Only subprograms loop. The subprogram must be written in Incremental Mode if you use X and Y incrementing.

A looping subprogram is called from the main program by a **Loop** block.

To call a looped subprogram from a main program:

1. In Edit Mode, press **Sub** (**F8**). Subprogram soft keys are displayed.

2. Press **Loop** (**F5**). The Graphic Menu prompts for the looping subprogram number.

3. Enter values labeled as follows:

Sub#	Subprogram identification number. Value required.
#Loops	Number of times loop shall run before returning to main program. Value required.
Xincr	Distance X-axis motion increments every cycle; zero if no entry is made. Optional.
Yincr	Distance Y-axis motion increments every cycle; zero if no entry is made. Optional.
Tool#	Active tool. Optional.

Rotating, Mirroring and Scaling Subprograms

Use **RMS** to scale, rotate or mirror subprograms. To call **RMS** subprograms from the main program, insert an **RMS** block. Rotation, scaling and mirroring functions turn off when the subprogram ends.

NOTE: Only subprograms can be rotated, mirrored or scaled.

To program a RMS subprogram call:

1. In Edit Mode, press **Sub** (**F8**). Subprogram soft keys activate.

2. Press **RMS** (**F6**). The **ROTATE/MIRROR/SCALE SUB** Graphic Menu display prompts for labeled values. Refer to **Figure 5-18**.


```
Program: KRIS.M              Blocks: 0         Free: 1,847 KB   Inch   Abs   XY



           ROTATE/MIRROR/SCALE SUB
                 Sub#     [ 0 ]
                #Loops
             StartAngle
                 Angle
                XCenter
                YCenter
                MirrorX
                MirrorY
                 XScale
                 YScale
                 Tool#
```

382/RMS
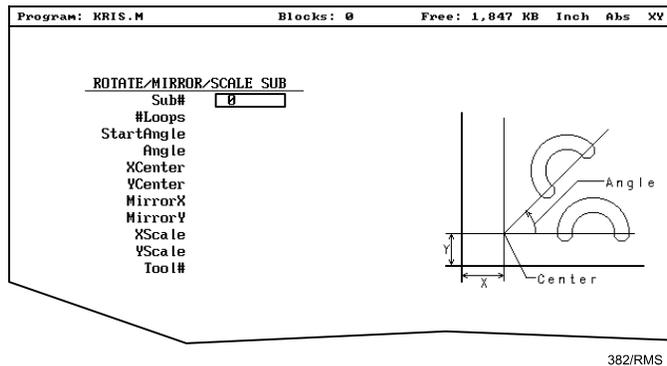
Figure 5-18, Rotate/Mirror/Scale Graphic Menu

3. Fill in the entry fields labeled as follows:

Sub#   Subprogram number. Value required.

#Loops   Number of times the subprogram will loop before returning to the main program. A value of one is assumed if no entry is made. Optional.

NOTE: **RMS** subprograms only loop if rotating.

StartAngle   The number of degrees the pattern rotates for the first loop. Zero if no entry is made. Optional.

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

Angle   The number of degrees of pattern rotation per loop. Zero if no entry is made. Optional.

XCenter   Point of rotation X coordinate. Current position if no entry is made. Optional.

YCenter   Point of rotation Y coordinate. Current position if no entry is made. Optional.

MirrorX   Activates mirroring around the X-axis. Switch **Yes** or **No** by pressing the (**+/-**) key. Optional.

MirrorY	Activates mirroring around the Y-axis. Switch **Yes** or **No** by pressing the (**+/-**) key. Optional.
XScale	X-axis scale factor. Multiplies all X-axis positions by the value entered. A value of one is assumed if no entry is made. Optional.
	NOTE: If scaling is used and the subprogram contains arcs, X and Y must use the same scaling factor.
YScale	Y-axis scale factor. Multiplies all Y-axis positions by the value entered. A value of one is assumed if no entry is made. Optional.
	NOTE: If scaling is used and the subprogram contains arcs, X and Y must use the same scaling factor.
Tool#	Active tool. Optional.

Ellipses and Spirals

> **NOTE:** It is possible for you to inadvertently write a block containing illogical entries. For safety, the control will stop a program and generate a message when it detects an illogical block. Test programs in Draw to detect these blocks prior to machining a part.

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.

Refer to Figure 5-20, Ellipse Tool Compensation. 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.

Use the (**+/-**) key to switch the setting.

> **NOTE:** To clear the entry field, press **CLEAR**.
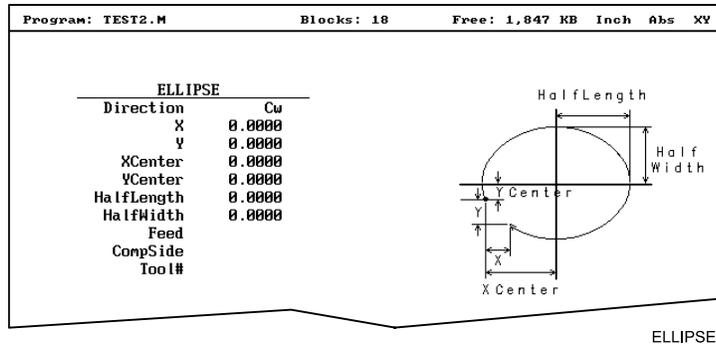
Figure 5-19, Ellipse Graphic Menu

To program an ellipse:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft key labels are displayed.

2. Press **More** (**F7**). The More Pop-Up Menu is displayed.

3. Select **Ellipse**, and press **ENTER**. The ELLIPSE Graphic Menu prompts for labeled values.

4. Refer to **Figure 5-19** and fill in the entry fields labeled as follows:

Direction	Direction of the Arc move: Clockwise (**Cw**) or counterclockwise (**Ccw**). Selection required.
X	X coordinate of endpoint. Value required, incremental.
Y	Y coordinate of endpoint. Value required, incremental.
XCenter	X coordinate of ellipse centerline. Value required.
YCenter	Y coordinate of ellipse centerline. Value required.
HalfLength	Half of the ellipse overall X dimension. Value required, incremental.
HalfWidth	Half of the ellipse overall Y dimension. Value required, incremental.
Feed	Feedrate. Optional.
CompSide	Offsets tool path to the inside (**In**) or outside (**Out**) of the ellipse by the tool radius. Refer to Figure 5-20, Ellipse Tool Compensation. If you leave the entry **blank**, no tool compensation is activated during the cycle. Optional.
Tool#	Active tool. Optional.
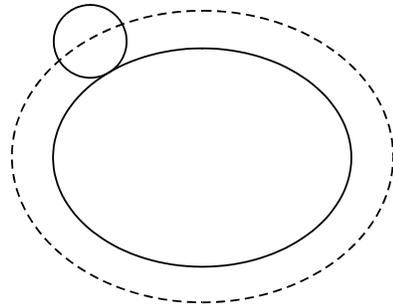
Outside Tool Compensation   Inside Tool Compensation
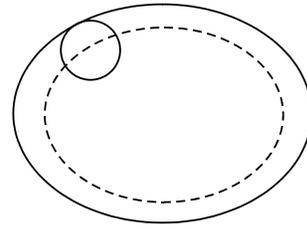
———— programmed path

------ compensated path (by tool radius)

ELLICOMP

Figure 5-20, Ellipse Tool Compensation

Programming a Spiral

A spiral is an Arc with a continuously changing radius. To program a Spiral Cycle, select 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). Use incremental values to program a spiral.

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

A) **Outward spirals** in which the present position is closer to the center than to the endpoint.

B) **Inward spirals** in which the endpoint is closer to the center than to the machine's current position.
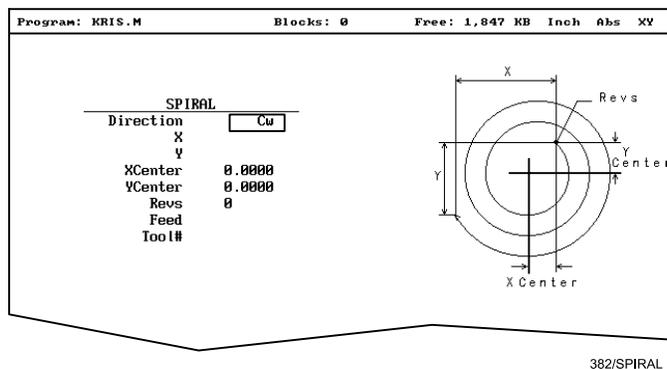
Figure 5-21, Spiral Graphic Menu

To program a Spiral Cycle:

1. In Edit Mode, press **Mill** (**F5**). The Mill soft keys are displayed.

2. Press **More** (**F7**). The More Pop-Up Menu is displayed.

3. Select **Spiral**, and press **ENTER**. The SPIRAL Graphic Menu prompts for labeled values.

4. Refer to **Figure 5-21** and fill in the entry fields labeled as follows:

Direction	Allows you to select a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the selection. Selection required.
X	X coordinate of endpoint. Optional, incremental.
Y	Y coordinate of endpoint. Optional, incremental.
XCenter	X coordinate of center. Value required, modal.
YCenter	Y coordinate of center. Value required, modal.
Revs	Number of complete revolutions in the cycle. Value required.
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 5-46**. Highlight **Engrave** and press **ENTER** to display the Engraving Cycle screen, Figure 5-47, Engraving Cycle Screen.

2. Complete the entry fields (refer to Table 5-1, Engraving Cycle Entry Fields), and press **ENTER**.
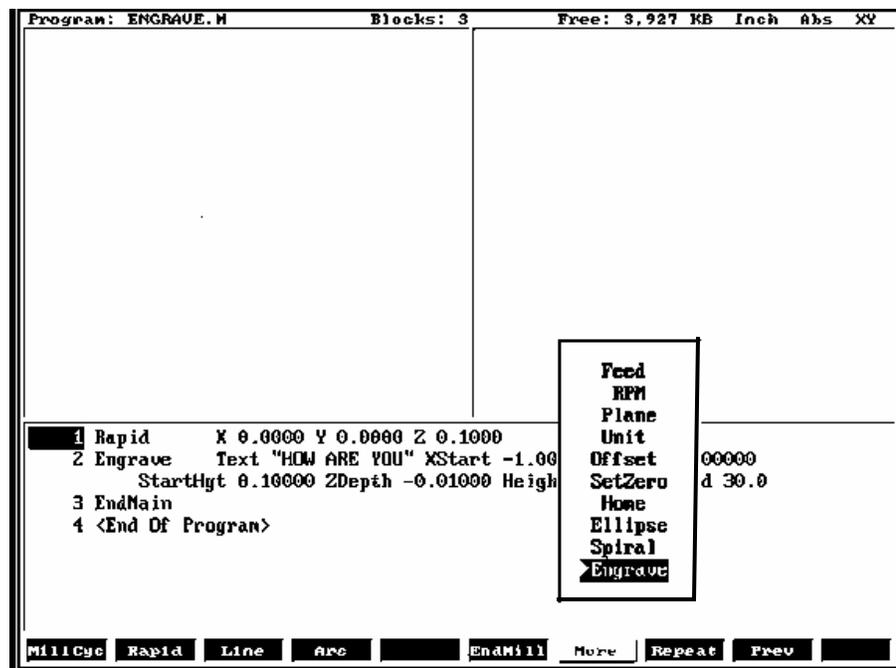
Figure 5-46, More Pop-up Menu
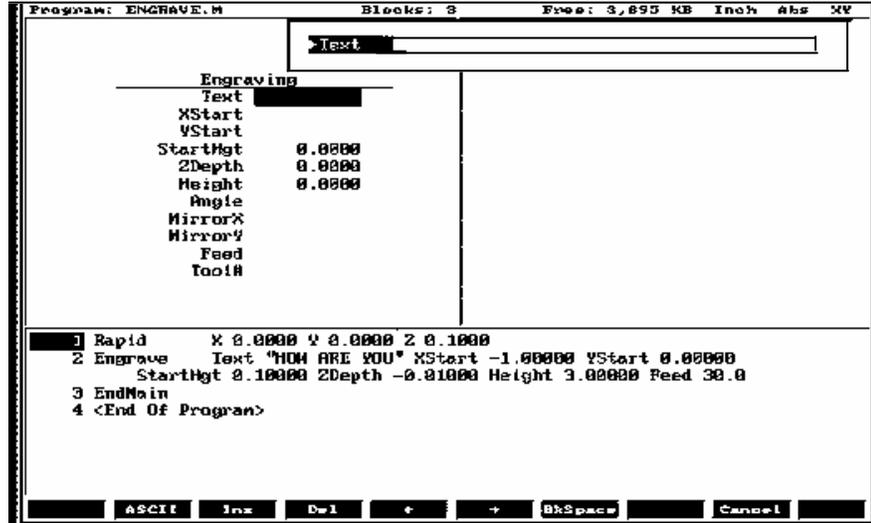
Figure 5-47, Engraving Cycle Screen

Table 5-1, 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 5-48**.
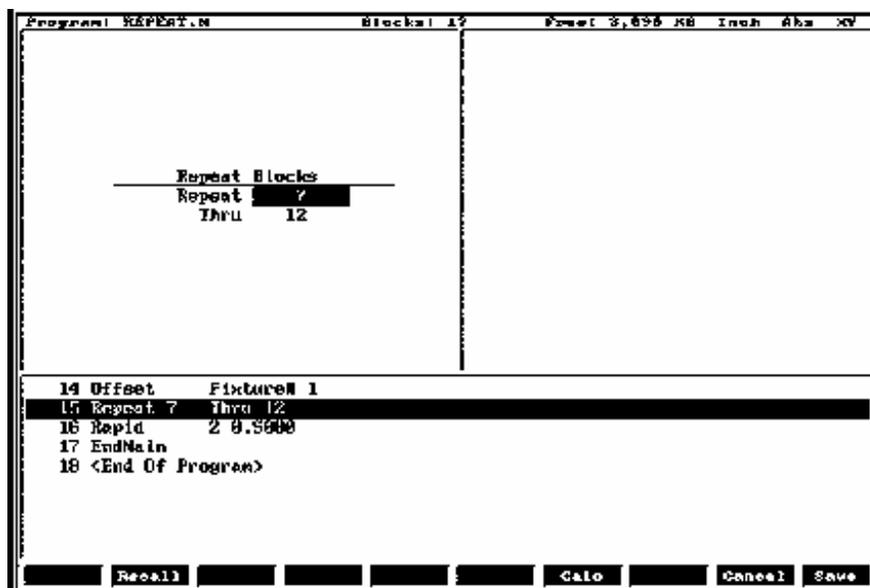
Figure 5-48, Repeat Cycle Screen

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

Table 5-2, 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 not Cutter Comp. 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 5-49**.

2. Complete the entry fields (refer to Table 5-3, 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.
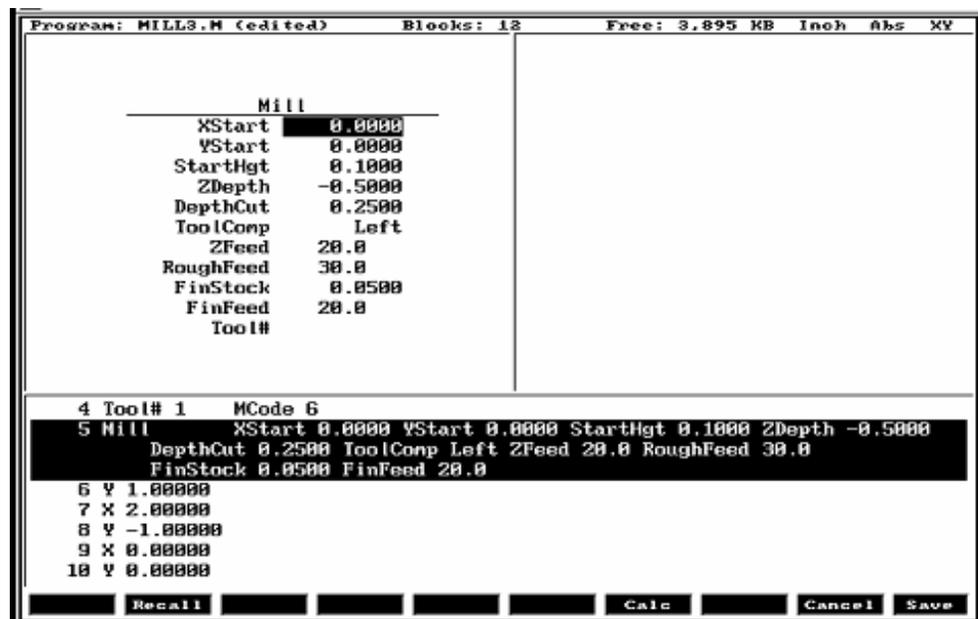
Figure 5-49, Mill Cycle Screen

Table 5-3, Mill Cycle Entry Fields

Entry Field	Description
XStart	X coordinate for start of Mill cycle. Defaults to current position if not given. [Optional]
YStart	Y coordinate for start of Mill cycle. Defaults to current position if not given. [Optional]
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".

Section 6 - Editing Programs

Write and edit program blocks in the CNC's Program Editor (the Edit screen). Activate the Program Editor to put the CNC in the Edit Mode.

The Program Editor activates from the Program Directory or the Manual screen. When the Program Editor activates from the Program Directory, the highlighted program opens for editing. When the Program Editor activates from the Manual screen, the loaded program opens for editing.

Activating the Program Editor

There are two ways to activate the Program Editor screen.

To activate the Program Editor from the Program Directory:

1. From the Program Directory, highlight a program.

2. Press **Edit** (**F4**). The Program Editor opens selected program for editing.

To activate the Program Editor from the Manual screen:

1. From the Manual screen, press **EDIT** (**F4**). The Editor opens the loaded program. If no program is loaded, the CNC will display a "**No Program Selected !**" message.

2. Press **PROGRAM** (**F2**) to activate the Program Directory.

3. Highlight a program.

4. Press **Edit** (**F4**). The Program Editor opens 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, Program Editor.

Program Name

(edited) Marker

Program Size (Blocks)

```
Program: TEST.M (edited)      Blocks: 7      Free: 1,879 KB   Inch  Abs  XY
```

Active Plane

Active Positioning Mode

Active Units Mode

Available Memory

Graphic Menu
Display Area

```
1 Dim Abs
2 * .5 DRILL
3 BasicDrill ZDepth -0.6500 StartHgt -0.6000 Tool# 1
4 Rapid     X 1.0000 Y -8.0000
5 Rapid     X 2.0000
6 Dim Incr
7 Dim Abs
8 <End Of Program>
```

```
[ Teach ] [ Draw ] [ Drill ] [ Pocket ] [ Mill ] [ Tool ] [ Calc ] [ Sub ] [ Misc ] [ Exit ]
```

EDITORL

Program Listing

Soft Key Labels

Figure 6-1, Program Editor

Program Name	Name of program opened for editing.
(edited) Marker	Indicates that the programmer has edited the program, but the edits have not been saved.
Program Size	Number of blocks in the program.
Active Plane	Current operating plane (XY).
Active Positioning Mode	
	Current measurement mode (**Abs/Inc**).
Active Units Mode	
	Current units mode (**Inch/MM**).
Available Memory	Space remaining in the Editor's memory.
Graphic Display Menu Area	
	Area for displaying Graphic Menus.
Program Listing	Current listing of the blocks in the open program.
Highlight	Enables you to select a block to edit and acts as an insertion marker to add new blocks. The CNC tracks program mode changes up to this point in the Program Listing.

ANILAM

Soft Key Labels   These labels define the soft key functions. Three sets of soft keys are available:

- ❑ 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**). A "**WARNING: Program has been edited. Sure you want to QUIT?**" message is displayed and soft key labels change.

3. Press **Yes** (**F1**). The CNC returns to the Program Directory. No edits are saved. Press **No** (**F2**) to cancel.

Deleting a Block

To delete a program block:

1. In Edit Mode, highlight a block.

2. Press **CLEAR**.

Inserting a 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.

31-July-05

Editing Blocks

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 existing values.

4. Make the appropriate changes. Press **Save** (**F10**) to close the block.

NOTE:	When the program block's Graphic Menu offers a choice of two modes (example, Cw/Ccw), highlight the block and press the (**+/-**) key to change the selection.

Searching Blocks for Words or Numbers

Use **Search** to find a block number or word. **Search** hunts forward from the cursor position only. To search an entire program, place the cursor at the beginning of the program, 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 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**). Soft key secondary functions are displayed.

2. Press the **PgUp** (**F4**) or **PgDn** (**F5**) keys to go forward or backward.

3. Press **Prev** (**F9**). The Program Editor default soft keys return.

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**). Soft key secondary functions are displayed.

2. Press **Begin** (**F6**). The CNC displays the first block of the program.

    – *or* –

    Press **End** (**F7**). The CNC 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**). Soft key secondary functions are displayed.

2. Press **Comment** (**F2**). The CNC prompts for a comment and the **ASCII** soft key label is displayed.

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. An asterisk is displayed after block number.

> **NOTE:** Off-line keyboard users must use the **0** key (not the asterisk) to produce a comment block.

Canceling a Comment

To cancel a comment:

1. In Edit Mode, highlight the comment block to be canceled.

2. Press **0**. The asterisk disappears. The block is no longer ignored during program execution.

> **NOTE:** Off-line keyboard users must use the **0** key to switch the asterisk on or off.

ANILAM

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**. 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.
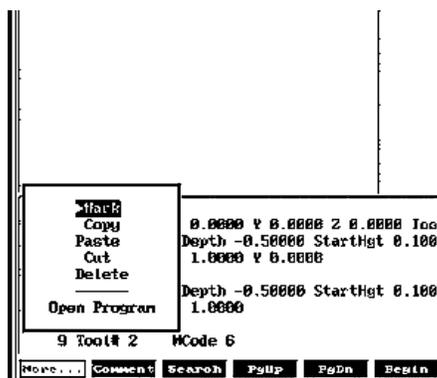
Figure 6-2, More Pop-up Menu

Section 7 - Viewing Programs with Draw

The CNC has two Draw Modes:

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 moves in the viewing area as it makes them.

In this manual, "Draw Mode" (upper and lowercase letters) refers to the CNC's Draw Simulation Mode. "DRAW" (all uppercase letters) refers to the CNC's Real-Time Draw Mode. The same spelling convention is used for 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, Single-Step) with a viewing area in the upper right corner. The **DRAW** (**F10**) soft key highlights and the soft keys change. Simulation Draw looks like an **Edit** screen with a viewing area in the upper right corner, with the Draw soft keys active.

This section of the manual explains how to use the Draw Simulation Mode to view programs. It also explains how to change viewing settings to permit a detailed inspection of programmed moves.

Use the Draw Simulation Mode soft keys to change Draw Mode view settings. Refer to "Section 8 - Running Programs" for instructions on how to run DRAW while cutting a part.

Draw Mode allows you to verify a program before you cut any material. In Draw, you can zoom in or out, or scale the drawing as required.

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, S. Step or Auto Mode.

Starting Draw Mode

Start Draw Simulation Mode from the Edit or MDI Mode. The **DISPLAY** (**F5**) and **Parms** (**F9**) settings determine how Draw Mode looks and runs. Adjust view settings before you start the simulation. Make setting changes via the soft keys.

In Draw Simulation Mode, the CNC does not halt for dwells and tool mounts.

To activate the Draw Simulation Mode:

1. In Edit Mode, with a program selected, press **Draw Mode** (**F2**). The viewing area is displayed in the upper right corner of the screen. Draw Mode soft keys activate.

2. Press **DISPLAY** (**F5**). A pop-up menu is displayed 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 image of the tool path is displayed in the viewing area and the machine remains idle.

NOTE: The Draw Mode display remains on the Edit screen until cleared.

NOTE: If you set Draw Mode to display both compensated and uncompensated moves, Draw Mode runs the program twice: once with and once without compensated moves. The tool paths of both versions appear simultaneously on the screen for comparison.

Draw Mode Screen Description

The Draw Mode screen looks like the Edit screen with the addition of a viewing area in the upper right corner. Draw Mode soft keys activate and the **Draw** (**F2**) soft key highlights. Refer to **Figure 7-1**.
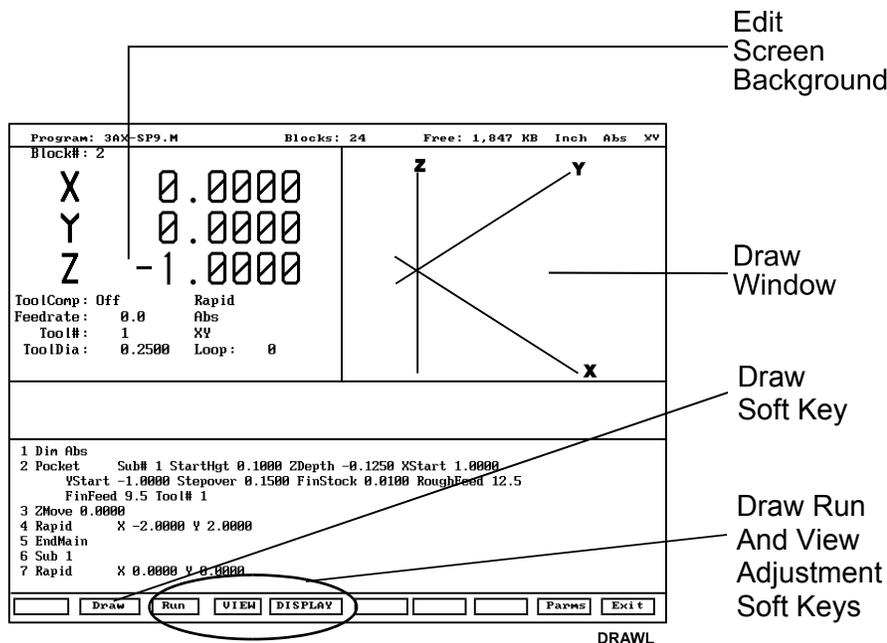
Figure 7-1, Draw Mode Simulation Mode

Putting Draw Mode in Hold

Press **Hold** (**F8**) or **HOLD** to pause a program running in Draw Mode. Press **Start** (**F7**) or **START** to resume.

Canceling Draw Mode

Press **Cancel** (**F9**) to stop program execution in Draw Simulation Mode.

Draw Mode Parameters

In Draw Mode, rapid moves appear as dotted lines; feed moves appear as solid lines; tools and drilled holes appear as cylinders.

Certain 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.
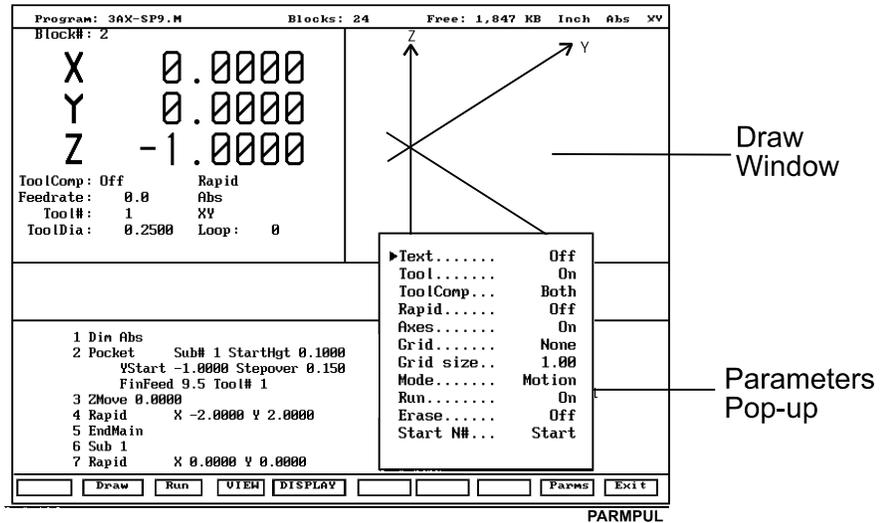
Figure 7-2, Draw Mode Parameters Pop-Up Menu

Text On or Off

Turn **Text On** to display program blocks as they run. This can help you to pinpoint a problem. Draw Mode runs slightly faster with **Text** turned **Off**. [Default: **Text On**]

To switch the **Text On**/**Off** setting:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **Text**, and press **ENTER**. **Text** switches **On** and **Off**.

3. Press **Parms** (**F9**). Pop-up closes.

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 program. Draw Mode displays only the active tool. The tool must have a diameter on the Tool Page to appear in Draw Mode. Draw Mode displays the tool as a cylinder scaled to the corresponding diameter.
[Default: **On**]

To switch **Tool On/Off** setting:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **Tool**, and press **ENTER**. **Tool** switches between **On** and **Off**.

3. Press **Parms** (**F9**). Pop-up closes.

> **SHORTCUT:** Press **Tool** (**F5**) to switch the Tool **On/Off**.

Draw Mode Compensated Moves

Use **ToolComp** to view compensated moves in Draw Mode.
[Default: **Both**]

There are three **ToolComp** options:

- ❑ **Ignore** a program's compensated moves.
- ❑ **Use** (display) a program's compensated moves.
- ❑ **Both** runs the program twice. (First without compensation, then with compensation.) Draw Mode displays both tool paths simultaneously for comparison.

To set the compensation parameters:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **ToolComp**, and press **ENTER**. Second pop-up is displayed.

3. Highlight the desired selection, and press **ENTER**. Smaller pop-up closes.

4. Press **Parms** (**F9**). Pop-up closes.

Showing Rapid Moves

Draw Mode displays Rapid moves as dotted lines. Switch the parameter **Off** to eliminate screen clutter. This parameter does not affect how the program runs, only how it is displayed. [Default: **On**]

To switch **Rapid** setting:

1. In Draw Mode, press **Parms** (**F9**). The CNC displays parameter pop-up menu.

2. Highlight **Rapid**, and press **ENTER**. **Rapid** switches **On** and **Off**.

3. Press **Parms** (**F9**). Pop-up closes.

> **SHORTCUT:** Press **Rapid** (**F6**) to switch the rapid move dotted lines **On** and **Off**.

Setting Grid Line Type

Draw Mode displays a 2-dimensional grid with dotted or solid lines. Grid lines are not displayed in the **Iso** view. [Default: **None**]

To set the **Grid** parameter:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **Grid**, and press **ENTER**. Grid pop-up menu is displayed.

3. Highlight the desired selection, and press **ENTER**. Grid pop-up closes.

4. Press **Parms** (**F9**). Pop-up closes.

Setting Grid Size

The adjustable grid size is measured in the active Inch/MM Mode. [Default: **1.00**]

To set the **Grid size**:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **Grid size**, and press **ENTER**. Number entry field is displayed.

3. Type desired size, and press **ENTER**. New value displays on pop-up menu.

4. Press **Parms** (**F9**). Pop-up closes.

Putting Draw Mode in Motion, Single-Step, or Auto Mode

The Draw Simulation Mode runs programs in one of three ways:

Automatic Mode   Fully automatic (**Auto**)

Single-Step Mode   One program block at a time (**Single-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 without motion commands (for example, **Dim** blocks), as in Single-Step Mode. This decreases program run time. Press **START** to execute each motion.

> **NOTE:** You can use the **spacebar** on an off-line keyboard 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**). Parameter pop-up menu is displayed.

2. Highlight **Mode**, and press **ENTER**. Pop-up menu is displayed.

3. Highlight the desired mode, and press **ENTER**. Mode pop-up closes.

4. Press **Parms** (**F9**). Pop-up closes.

Automatic Draw Mode Restart

The **Run** parameter determines whether Draw Mode automatically restarts after a **DISPLAY** or **VIEW** setting changes. This allows you to make more than one setting change before restarting Draw Mode.

When the **Run** parameter is turned **Off**, press **Run** (**F3**) to start or restart the Draw Mode. With **Run** turned **On**, Draw Mode automatically starts after each **DISPLAY** or **VIEW** change.
[Default: **On**]

To set the **Run** parameter:

1. In Draw Mode, press **Parms** (**F9**). Parms pop-up menu is displayed.

2. Highlight **Run**, and press **ENTER**. **Run** switches **On** or **Off**.

3. Press **Parms** (**F9**). Pop-up closes.

Setting the Erase Display Parameter

Turn **Erase On** to clear the display each time a new program runs in Draw Mode. Turn **Erase Off** to display the new program tool path without clearing the previous graphic. This allows you to compare two or more program paths.
[Default: **On**]

To set the **Erase** parameter:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **Erase**, and press **ENTER**. **Erase** switches **On** or **Off**.

Running Draw Mode for Selected Blocks

Use the **Start N#** and **End N#** commands to run a portion of a program or subprogram in Draw Mode. Specify the **Start N#** and **End N#** block settings on the **Parms** 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.

ANILAM

Starting Draw Mode at a Specific Block

To start Draw Mode at a specific block:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **Start N#**, and press **ENTER**. Start N# pop-up menu is displayed.

3. Highlight the desired selection, and press **ENTER**. If **Start Of Program** is selected, Draw Mode will start at Block 1. If **Other Block** is selected, type the block number, and press **ENTER**.

4. Press **Parms** (**F9**). Pop-up closes.

Ending Draw Mode at a Specific Block

To end Draw Mode at a specific block:

1. In Draw Mode, press **Parms** (**F9**). Parameter pop-up menu is displayed.

2. Highlight **End N#**, and press **ENTER**. Start N# pop-up menu is displayed.

3. Highlight the desired selection, and press **ENTER**. If **End Of Program**, Draw Mode will stop at the last program block. If **Other Block**, type the block number, and press **ENTER**.

4. Press **Parms** (**F9**). Pop-up closes.

Adjusting Draw Display

Draw Mode has several display settings for the moves shown in the viewing window. Refer to **Figure 7-3**. Activate these settings from the **DISPLAY** (**F5**) pop-up menu.
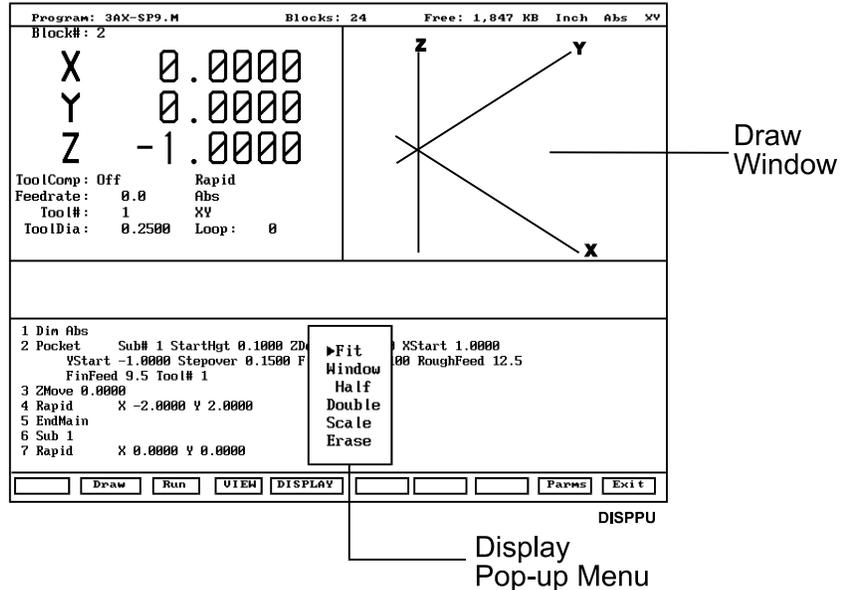
Figure 7-3, Display Pop-Up Menu

Fitting the Display to the Viewing Window

To fit the display in the viewing area:

1. In Draw Mode, press **Display** (**F5**). Pop-up menu is displayed.

2. Highlight **Fit**, and press **ENTER**. The CNC closes the pop-up menu, and the display adjusts to fit in the viewing window.

Halving the Display Size

To reduce the display size by half:

1. In Draw Mode, press **Display** (**F5**). Pop-up menu is displayed.

2. Highlight **Half**, and press **ENTER**. The display is scaled to half its original size.

Doubling the Display Size

To double the size of the display:

1. In Draw Mode, press **Display** (**F5**). Pop-up menu is displayed.

2. Highlight **Double**, and press **ENTER**. The display is scaled to twice its original size.

ANILAM

Scaling the Display by a Factor

To scale the Draw Mode display by a factor:

1. In Draw Mode, press **Display** (**F5**). Pop-up menu is displayed.

2. Highlight **Scale**, and press **ENTER**. Pop-up closes. The CNC prompts for the scale factor.

> **NOTE:** Enter a number preceded by a decimal point to reduce the display, or enter a whole number to enlarge the display.

3. Type the required factor, and press **ENTER**. The CNC scales the display to the specified factor.

Zooming In

To zoom in on a portion of the display:

1. In Draw Mode, press **Display** (**F5**). Pop-up menu is displayed.

2. Highlight **Window**, and press **ENTER**. A white window is displayed 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 window to original size.

5. With the window sized and positioned, press **Enter** (**F10**). The window closes. The next time Draw Mode runs, the CNC will zoom in on the selected portion of the display.

Erasing Display

To erase the display:

1. In Draw Mode, press **Display** (**F5**). Display pop-up menu is displayed.

2. Highlight **Erase**, and press **ENTER**. Display clears.

Changing Draw Views

Activate different view orientations from the **VIEW** (**F4**) pop-up menu. Refer to **Figure 7-4**.
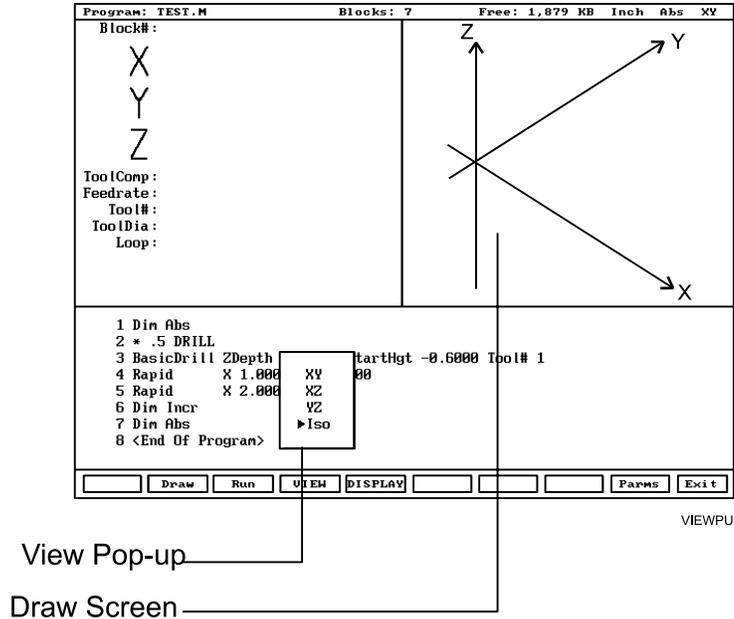
Figure 7-4, View Pop-up Menu

Selecting the View

To view Draw Mode from a different plane:

1. In Draw Mode, press **VIEW** (**F4**). VIEW pop-up menu is displayed.
2. Highlight a plane (XY, YZ, XZ, or Iso), and press **ENTER**. Draw Mode orients display to 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	Runs a program one block at a time.
Motion Mode	Only available in Single-Step Mode. Run a program from motion to motion.
Automatic Mode	Runs a program automatically.

The Automatic 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 will only run the currently selected program. Select programs from the Program Directory. Refer to "Section 9 - Program Management."

In the Manual Data Input Mode (MDI), it is not necessary to select MDI code blocks.

In MDI Mode you can program moves without creating and saving a program. MDI is only available in Manual Mode. Refer to "Section 3 - Manual Operation and Machine Setup."

All programming tools, moves and cycles are available in the MDI Mode.

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 enables you to step through the program and verify the moves before production.

The **S.STEP** screen differs from the Manual screen as follows:

❑ There are fewer active soft keys

❑ The **S.STEP** (**F5**) soft key highlights

❑ The **S.STEP** indicator appears in the status box (upper right corner of screen)

To run a program in Single-Step Mode:

1. From the Program Directory, select the required program and return to the Manual screen.

2. Press **S.STEP** (**F5**). Single-Step Mode activates.

3. Press **START**. 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**). Active soft keys highlight.

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**). Single-Step Mode activates.

> **NOTE:** Press **MOTION** (**F7**) to activate the Motion Mode.

3. Highlight the desired starting block.
4. Press **START**. The CNC executes 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. The CNC searches forward from the cursor and 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**). Single-Step Mode activates.
3. Press **SEARCH** (**F3**). The CNC prompts for search text or number.
4. Enter 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**. The CNC starts the program from the highlighted block.

ANILAM

Switching from Single-Step to Auto

To switch the CNC from Single-Step to Auto Mode:

1. In the Single-Step Mode, press **AUTO** (**F6**). The CNC completes the current move then holds.

2. Press **START**. The CNC restarts and runs the rest of the program in the Automatic Mode.

Automatic Program Execution

Auto Mode is the CNC's production mode. Execute any portion 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:

❑ There are fewer active soft keys

❑ **AUTO** (**F6**) highlights

❑ The **AUTO** indicator is displayed in the status box (upper right screen)

To run a program in Automatic Mode:

1. Select the required program and return to the Manual screen.

2. Press **AUTO** (**F6**). Automatic Mode activates.

3. Press **START**. The CNC runs the entire program.

Holding or Canceling an Auto Run

Press **HOLD** to pause program run. To restart a program after a hold, press **START**.

To cancel program run when a program is on hold, press **MANUAL** (**F4**). This cancels any active tool compensation and canned cycles. All other modal settings remain active.

Starting at a Specific Block

> **CAUTION: Choose 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**. The CNC automatically runs the program from the selected block.

To select the starting block before you run a program, use the **SEARCH** feature:

1. Select the required program and return to the Manual screen.

2. Press **AUTO** (**F6**). Auto Mode activates.

3. Press **SEARCH** (**F3**). The CNC prompts for search criteria.

4. Enter 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**. The CNC runs the program forward from the highlighted 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** only seeks forward in the program.

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 how to review 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 screen looks like the 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**). Upper right status box changes to a window.

3. Press **START**. Program starts and the CNC displays moves as they are executed.

Parts Counter and Program Timer


```
PROGRAM:

MANUAL                    HALTED
IN-POSN                   FEED
   ABS
   INCH

   LOOP: 0          JOG: RAPID
  DWELL: 0.0    SPINDLE: OFF
    RPM: 0       COOLANT: OFF

  PARTS: 0
  TIMER: 00:00:00
```

TIMER

Number of
Completed Parts
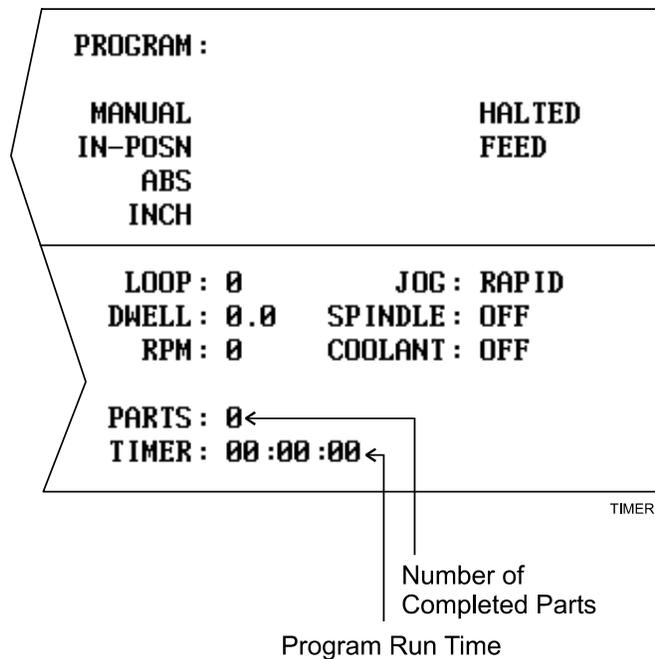
Program Run Time

Figure 8-1, Program Timer and Parts Counter

Refer to **Figure 8-1**. The CNC keeps track of program run time (**TIMER**) and the number of successfully completed parts (**PARTS**). Run time is displayed in hours, minutes and seconds. These two features are available in the Manual, Auto and S. Step Modes.

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.

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 the selected program is running. .

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 soft keys for prohibited functions do not appear in Background Mode. The following functions are not available:

❑   Draw Graphics
❑   DNC execution
❑   Selecting another program to run
❑   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 Program Directory activates.

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."

ANILAM

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 **PROGRAM** (**F2**).

The Program Directory's **USER** listing displays programs stored in the CNC. CNC program names are followed by an ".**M**" extension. Change the display mode to view programs in other formats. Refer to **Figure 9-1**.
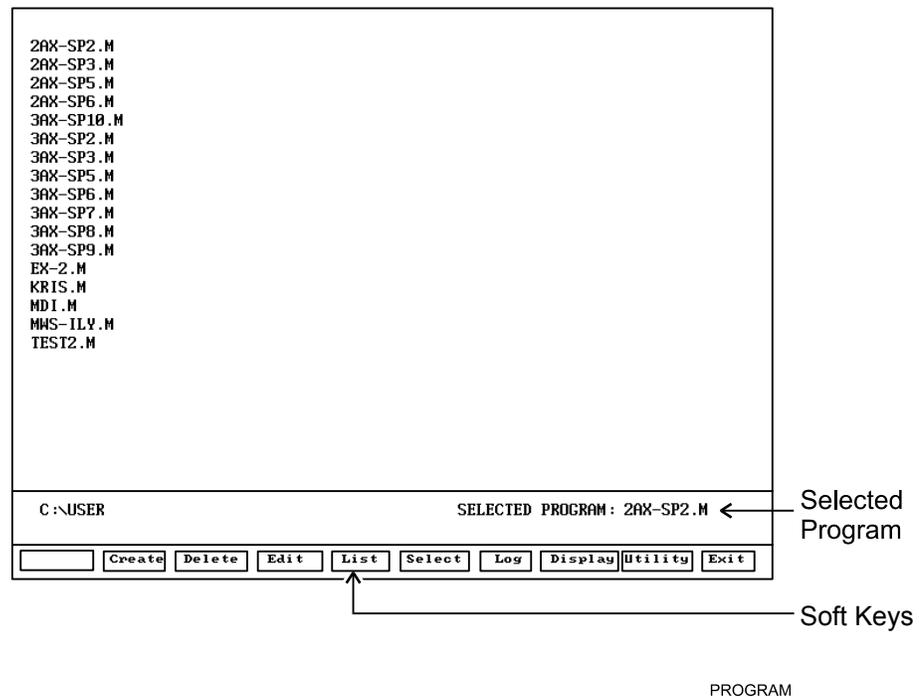
Figure 9-1, Program Directory

Changing the Program Directory

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 an "**.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 Manual Mode, press **PROGRAM** (**F2**). Program Directory activates. **USER** listing is displayed.

2. Press **Create** (**F2**). Message line displays prompt: "**NEW PROGRAM: _**"

3. Press **ASCII** (**F2**). ASCII Chart activates.

4. Use the ASCII Chart and the number keypad to enter the new program's name.

5. Press **ASCII** (**F2**). ASCII pop-up is displayed.

6. Press **ENTER** to place the new program in the **USER** listing in alphabetical order.

Choosing Program Names

Program names can contain up to eight characters. The CNC displays program names in capital letters. 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

The CNC will only run a program that you select. Only one program can be selected 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 program marked by the highlight.

NOTE: The selected program is the program selected for cutting parts.

To select a program for editing:

1. From the Program Directory, press **Display** (**F8**) to switch the display setting until only the CNC programs (*.M) are displayed in the Program Directory.

2. Highlight a program name.

3. Press **Edit** (**F4**). The Program Editor activates and the highlighted program is displayed on the Edit screen.

Maximizing Program Storage

The CNC has a finite amount of space available for programs. Use the System Information screen to check available space. Refer to .

Total Space Available for the System	Available program storage space
Total Free User Space	Remaining new programs storage space

During program execution, the control generates a second program given 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 a lot of old programs accumulate, the CNC could run out of memory. To make room, temporarily delete inactive ".S" files. The CNC will automatically replace the ".S" files the next time the programs are run.

By default, the Program Directory displays only part programs (programs followed by ".M"). To list *.S files on screen, press **Display** (**F8**) until the ".S" programs are displayed in the Program Directory.

Program File Utilities

You can access some utilities directly from the soft keys and others through the **Utility** (**F9**) pop-up menu. The Utility pop-up menu lists fourteen 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 view a program without making inadvertent changes. The **List** feature only displays CNC programs (programs with a ".**M**" extension).

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 programs 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 based on your response.

3. Press **Yes** (**F1**) to delete the selected program. Press **No** (**F2**) to cancel.

> **NOTE:** Deleting a program automatically deletes its associated ".S" file.

Reading Disks in Floppy Drives (Logging on to Other Drives)

Most utilities apply to programs in the **USER** listing by default. The **Log** feature makes it possible 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** listing.

ANILAM

Marking Programs

To mark a program:

1. From the Program Directory, highlight the program.

2. Press **ENTER**. The marked program highlights; the highlight bar advances to the next name.

3. Press **ENTER** to mark the next program.

> – *or* –

Highlight another program in the listing, and press **ENTER**.

4. Repeat these steps to mark all required programs.

Unmarking Marked Programs

To unmark a program:

1. Highlight any previously marked program, and press **ENTER**. The highlight clears.

Marking all Programs

To mark all programs:

1. From the Program Directory, press **Utility** (**F9**). Utility pop-up menu is displayed.

2. Highlight **More . . .**, and press **ENTER**. Second page of pop-up menu is displayed.

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 Utility pop-up menu is displayed.

2. Highlight **More . . .**, and press **ENTER**. The second page of pop-up menu is displayed.

3. Highlight **Unmark All**, and press **ENTER**. All programs are unmarked.

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 deletion. The soft keys change based on your response.

2. Press **Yes** (**F1**) to delete the selected programs. Press **No** (**F2**) to cancel command.

Restoring Programs

You can recover most deleted programs. Sometimes, only a portion of a deleted program can be recovered. Restore (undelete) a program as soon as possible after it was deleted.

To restore a program:

1. From the Program Directory, press **Utility** (**F9**). Utility pop-up is displayed.

2. Highlight **Restore**, and press **ENTER**. If the control finds programs to restore, it displays a pop-up menu listing recoverable programs. If the control cannot recover any deleted programs, it prompts: "**No programs available for restore.**"

> **NOTE:** When you delete a program, the CNC inserts a ? in place of the first letter of the program.

3. Highlight the desired program, and press **ENTER**. The CNC prompts for the first letter (automatically removed when a program is deleted) of the deleted program's name and ASCII soft keys activate.

4. Use the ASCII Chart to enter the first letter, and press **ENTER**. The CNC presents an information message.

5. Press **Cont** (**F10**) to transfer the program to the **USER** listing.

> **NOTE:** Restored programs might not contain all of the original information. Review the entire program.

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 Utility pop-up menu is displayed with **Copy** selection highlighted.

3. Press **ENTER**. **Copy to:** pop-up menu is displayed.

4. Highlight the target drive. (This is 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**). A Utility pop-up menu is displayed.

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**). A Utility pop-up is displayed.

3. Highlight **Print**, and press **ENTER**. The CNC prompts you to confirm the command. The soft keys change for your 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 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 to rewrite part programs written for other machines.

By default, converted programs are given the same name as the original, but with an ".M" extension added. If necessary, 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 prompted to confirm the new program name.

Ensure that the CNC has enough space available to store the converted program, which will be slightly larger than the original. The CNC can only convert 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 only displays program names that end with the .M extension. Press **Display** (**F8**) to switch the display setting and view programs in other formats.

To convert a G-Code program to conversational format:

1. To transfer the program to the control, copy it from a floppy disk or download it via the RS-232 link. Refer to "Section 11 - Communications and DNC" for downloading instructions.

2. From the Program Directory, mark the program being converted.

3. Press **Utility** (**F9**). A Utility pop-up menu is displayed.

4. Highlight **G-Code Converter**, and press **ENTER**. The CNC prompts you to confirm the conversion. The soft keys change for your response.

5. Press **Yes** (**F1**). The CNC prompts you to confirm the new program name. The soft keys change for your response. Press **No** (**F2**) to cancel.

6. To use the same name and location, press **Yes** (**F1**). The CNC converts the program and displays conversion statistics.

    – or –

    To enter a different name (or location), press **No** (**F2**). CNC prompts for new name. ASCII soft keys activate.

7. Enter 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 of the conversion.

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 to Right & Left of Decimal Place		Example of Properly Formatted Value	
	Mode		Mode	
	Inch	MM	Inch	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
XYZ			XYZ
G90/1			Dim Abs/Incr
G17/8/9			Plane XY/XZ/YZ
G75 Xn Yn Hn Zn An Bn In Jn Un Vn Wn Sn Kn Pn			Frame Pock
	X =	IslandLen	
	Y =	IslandWid	
	H =	StartHgt	
	Z =	ZDepth	
	A =	Stepover	
	B =	DepthCut	
	I =	RampFeed	
	J =	RoughFeed	
	U =	InsideRad	
	V =	OutsideRad	
	W =	FrameWidth	
	S =	FinStock	
	K =	FinFeed	
G76 Dn Jn Sn Kn			Hole Pocket
	D =	Diameter	
	J =	RoughFeed	
	S =	FinStock	
	K =	FinFeed	

Table 9-2, G-Code Equivalents (Continued)

G-Code Format			Conversational Equivalent
G77 Xn Yn Hn Zn Dn An Bn In Sn Kn Pn			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 Hn Zn Un An Bn In Jn Sn Kn Pn			Rectangular Pockets
	X =	XCenter	
	Y =	YCenter	
	H =	StartHgt	
	Z =	ZDepth	
	U =	CornerRad	
	A =	Stepover	
	B =	DepthCut	
	I =	RampFeed	
	J =	RoughFeed	
	S =	FinStock	
	K =	FinFeed	
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			BasicDrill
	Z =	ZDepth	
	R =	StartHgt	
	F =	Feed	
	P =	ReturnHgt	
G83 Zn Rn Fn In Pn			PeckDrill
	Z =	ZDepth	
	R =	StartHgt	
	F =	Feed	
	I =	Peck	
	P =	ReturnHgt	

Table 9-2, G-Code Equivalents (Continued)

G-Code Format			Conversational Equivalent
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	
G179 Xn Yn Cn Bn En Un Vn			Drill Pattern
	X =	X	
	Y =	Y	
	C =	Angle	
	B =	#XHoles	
	E =	#YHoles	
	U =	XIncr	
	V =	YIncr	
Tn			Tool# n
G4 Tn			Dwell n
On			Sub n
M99			EndSub
M98 Pn			Call n
M2			EndMain
Fn			Feed n
G00/1			Line with XYZ/XR/YR/RC format
G02/3			Arc with XYZIJK/XYZR/IJKABC format
G05			Ellipse with XYZIJABCL format
G06			Spiral with XYZIJKL format
G40/1/2			ToolComp Off/Left/Right

31-July-05

Checking Disks for Lost Data

Sometimes, a computer disk contains fragments of lost programs. This might happen if the 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**). A Utility pop-up is displayed.

2. Highlight **More . . .**, and press **ENTER**. The second page of the pop-up menu is displayed.

3. Highlight **Check Disk**, and press **ENTER**. The CNC prompts you to select a drive.

4. Highlight the drive, and press **ENTER**. The CNC checks the disk. If it finds lost clusters, it prompts you for recovery instructions and soft keys change for your response.

5. Press **Yes** (**F1**). The CNC attempts to recover lost clusters on the disk. Press **No** (**F2**) to cancel.

Display 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                │
├────────────────────────────────────────────────┤
│                                                │
│              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%)          │
│                                                │
├────────────────────────────────────────────────┤
│            Press F10 to continue.              │
└────────────────────────────────────────────────┘
```

SYSINFO

Figure 9-2, System Information Screen

![ANILAM]

To display the System Information screen:

1. From the Program Directory, press **Utility** (**F9**). The Utility pop-up menu is displayed.

2. Highlight **More . . .**, and press **ENTER**. The second page of the pop-up menu is displayed.

3. Highlight **System Info**, and press **ENTER**. The System Information screen is displayed.

> **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**). A Utility pop-up is displayed.

2. Highlight **More . . .**, and press **ENTER**. The second page of the pop-up menu is displayed.

3. Highlight **Copy ?**, and press **ENTER**. The CNC prompts for program source and ASCII soft key activates.

4. Use the ASCII Chart to enter the complete location (path) of the program, and press **ENTER**. A pop-up menu prompts for the destination drive or **Other . . .**

5. Highlight **Other**, and press **ENTER**. The CNC prompts for the destination drive and the ASCII soft keys activate.

6. Use the ASCII Chart to enter **C:\USER**[program name].**M**. Press **ENTER**. The CNC copies the program to the **USER** listing.

> **NOTE:** As an alternative, log on to the floppy disk drive that contains the program, mark the program and use the **Copy to:** utility.

Renaming Programs from/to Unspecified Locations

To rename a program located on any drive:

1. From the Program Directory, press **Utility** (**F9**). A Utility pop-up menu is displayed.

2. Highlight **More . . .**, and press **ENTER**. The second page of the pop-up menu is displayed.

3. Highlight **Rename?**, and press **ENTER**. The CNC prompts for the location of the source program and the ASCII soft keys activate.

4. Use the ASCII Chart to enter the location (complete path) of the program, and press **ENTER**. The CNC prompts for program destination and ASCII soft keys activate.

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.

> **TIP:** Use the **Rename?** feature to move a program to a different drive. Log on to the drive that contains the program and activate the **Rename?** feature. Enter the destination drive as part of the new program name. The CNC saves the program to the specified drive.

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**). A Utility pop-up is displayed.

2. Highlight **More . . .**, and press **ENTER**. The second page of the pop-up menu is displayed.

3. Highlight **Print ?**, and press **ENTER**. The CNC prompts for source program on message line and ASCII soft keys activate.

4. Use the ASCII Chart to enter the location (complete path) of the program, and press **ENTER**. The CNC prompts you to confirm the instruction. The soft keys change for your response.

5. Press **Yes** (**F1**) to send programs to the printer. Press **No** (**F2**) to cancel.

> **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 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 the information for up to 99 tools.

The Tool Page stores spindle speed, spindle direction and coolant information for machines equipped with these functions.

On machines equipped 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**). CNC displays Tool Page.

To enter the Tool Page from the Edit Mode:

1. In Edit Mode, press **Tool** (**F6**). CNC displays 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 display on the screen. Refer to **Figure 10-1**.

Position Display

Active Tool

```
Inch
  X+     0.0000        TOOL: 0      DIA: 0.0000
                       FEED: 0.0      %: 100
                       POSN: PROGRAM  RPM: 0
  Y+     0.0000
                       MACHINE       JOG: RAPID
  Z+     0.0000        X +   0.0000  SPINDLE: OFF
                       Y +   0.0000  COOLANT: OFF

  No.   Diameter   Length   Spindle RPM   Spindle Direction   Coolant
   1     0.0000    0.0000        0              NONE           NONE
   2     0.0000    0.0000        0              NONE           NONE
   3     0.0000    0.0000        0              NONE           NONE
   4     0.0000    0.0000        0              NONE           NONE
   5     0.0000    0.0000        0              NONE           NONE
   6     0.0000    0.0000        0              NONE           NONE
   7     0.0000    0.0000        0              NONE           NONE
   8     0.0000    0.0000        0              NONE           NONE
   9     0.0000    0.0000        0              NONE           NONE
  10     0.0000    0.0000        0              NONE           NONE
  11     0.0000    0.0000        0              NONE           NONE
  12     0.0000    0.0000        0              NONE           NONE
  13     0.0000    0.0000        0              NONE           NONE

   1     0.0000    0.0000        0              NONE           NONE

OFFSETS      ClrLine  Find   PgUp   PgDn         Calib        Exit
                                                            TOOLPAGE
```

Highlighted Row Display

Soft Key Labels

Figure 10-1, Tool Page

Row numbers correspond to tool numbers. When the CNC runs a program block that activates a tool number, it activates 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. The features of the Tool Page include:

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 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 with the tapping cycle only.
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 prompts: "**Enter Tool Number:_.**"

2. Enter a tool number, and press **ENTER**. The CNC displays the appropriate tool information.

Changing Tool Page Values

CAUTION:	**Set the CNC for the required Inch/MM Mode. The active Inch/MM Mode is displayed 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. Enter the new value.

3. Press **ENTER** or any **ARROW**. The CNC accepts the value.

Clearing a Tool (Whole Row)

To clear a Tool Page row:

1. Go to the Tool Page and position the highlight bar on the row to be cleared.

2. Press **ClrLine** (**F3**). CNC resets all values in the row to zero.

Clearing a Single Value

To clear a single value:

1. Go to the Tool Page and move the highlight bar to select the row.

2. Position the cursor at the required column.

3. Press **CLEAR** to change value to zero.

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, which is 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 how to set the location of Tool# 0, Z0.

NOTE: All Z moves are manual moves.

NOTE: All Jog features operate in the Tool Page.

Automatically Setting Tool-Length Offsets from the Tool Page

To set length offsets from the Tool Page automatically:

1. Activate the Tool Page and highlight a tool number.

2. Load the tool and manually position its tip at the part's Z0 position.

3. Press **Calib Z** (**F8**). The CNC inserts the Z position in the length-offset column.

Manually Setting Tool-Length Offsets from the Tool Page

To set length offsets from the Tool Page manually:

1. Retract the Z-axis to the Tool# 0, Z0 position.

2. Load tool and manually position its tip at the part's Z0 position.

3. Manually enter the $\pm Z$ position, as it is displayed in the position display. 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 you use 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**.
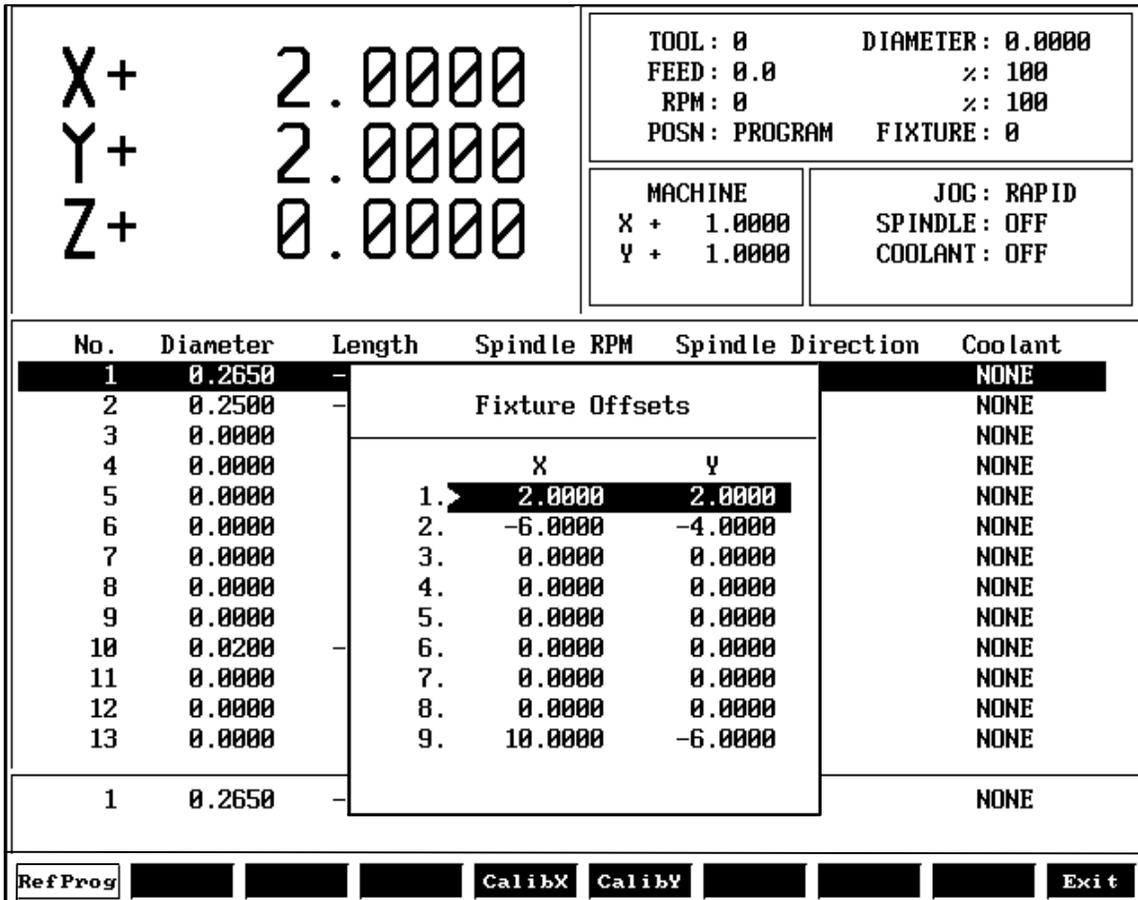
30002A-REFPROG

Figure 10-2, RefProg "Fixture Offsets" Pop-up

When **RefProg** is highlighted, use the calibration keys: **CalibX** (**F5**) or **CalibY** (**F6**) to calibrate an axis work coordinate offset, the calibration value is in reference to the current active offset verses 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.

31-July-05

![ANILAM]

Section 11 - Communications and DNC

Communications

The CNC can exchange data with any other RS-232 compatible devices. The baud, 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 Communications screen. Every time the CNC starts, it reloads the default settings.

CAUTION:	Only qualified personnel should alter the settings in the CNC's Setup Utility. Incorrect settings could disable the machine.

NOTE:	The CNC loads the updated setup file when setup parameters change, or when the CNC is turned on.

Installing the RS-232 Cable

RS-232 communication requires a cable connection between the sending and receiving machines. 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 builder determines the exact location of the RS-232 connector.

Use a cable designed for RS-232 communication. An appropriate cable has internally crossed wires between Pins 2 and 3. Refer to **Figure 11-1**.

RS-232 Connector
(Male 9-Pin)
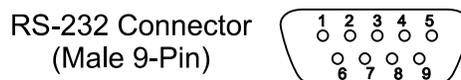
Pin 2 - Recieve Data
Pin 3 - Transmit Data

DB-9

Figure 11-1, RS-232 Communications Connector

Data sent from Pin 3 (transmit) of one machine must go to Pin 2 (receive) of the other.

Do not use computer extension cords for RS-232 communications (Pins 2 and 3 not crossed).

Accessing the Communication Package

To access the Communication screen:

1. In Manual Mode, press **Program** (**F2**). The Program Directory activates.

2. Press **Utility** (**F9**). Utility pop-up is displayed.

3. Highlight **Communications**, and press **ENTER**. Communication screen is displayed. Refer to **Figure 11-2**.

Figure 11-2, Communication Screen

PROGRAM   The program selected for transmission.

SERIAL COMMUNICATIONS PARAMETERS

Settings required for the two machines to send and receive programs.

ANILAM

Setting Communication Parameters

This manual does not attempt to discuss the merits of the different parameter choices. Refer to an appropriate computer communications reference for this information.

To change communication parameters:

1. From the Communication screen, press **Param** (**F9**). Parameter Pop-up Menu is displayed. 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

> **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. In the pop-up, 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 choices.

Selecting the Communications Port

Most CNC installations have at least one RS-232 connector. It is usually labeled and mounted on the back of the console. The CNC sends signals via 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 are equipped with the necessary hardware.

Setting the Baud

The CNC supports the following bauds: 110, 150, 300, 600, 1200, 2400, 4800, 9600, or 19,200. [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 "handshaking" in commercial communications packages.
[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 it is displayed 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

Refer to Figure 11-4, Test Link Screen. With the communications Screen active, press **TestLnk** (**F7**). The Test Link screen is displayed.

11-4

31-July-05
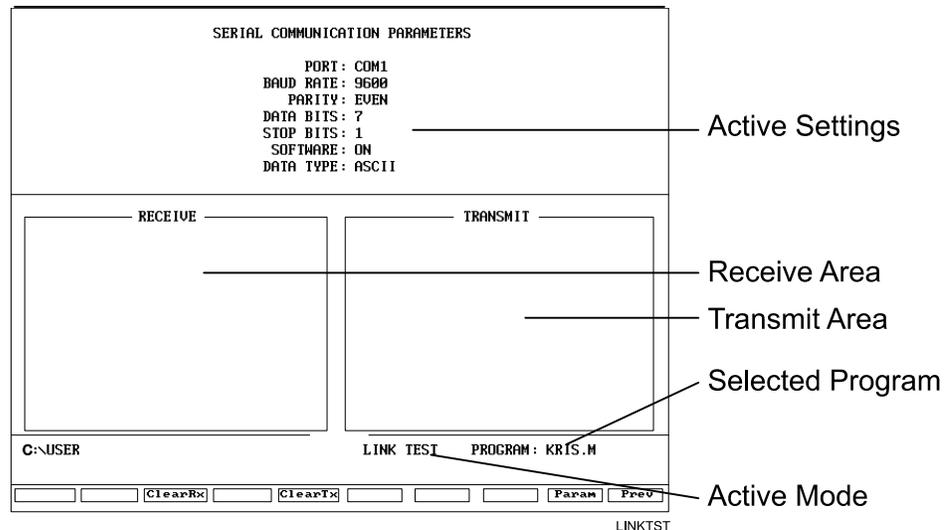
Figure 11-4, Test Link Screen

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 characters are displayed on the screen.

If the data type is set to ASCII, letters and numbers are displayed. If the data type is set to binary, the hexadecimal equivalents are displayed. Set both machines to use the same data display type.

| NOTE: | Hexadecimal characters are displayed as pairs of numbers or numbers and letters. |

To change the Link test screen data display:

From the Link test screen, press **Data** (**F1**) to switch between ASCII and binary display formats. The CNC displays the current mode in the settings area of the Link Test 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 Link Test screen active, use the CNC's keypad to enter any series of numbers. These characters are displayed in the transmission area.

4. Verify that the other machine (or computer) received the entire transmission correctly.

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 received the entire transmission correctly.

Clearing the Receive Area

Press **ClearRx** to clear the receive area.

Sending a Program

Highlight the appropriate program in the Program Directory before you activate the Communications screen. The active program name is displayed next to the **PROGRAM** label on the screen. The CNC sends the selected program. A prompt enables the operator to select another program to transmit during the sending process.

To send a program:

1. With the Communications screen active, the parameters set and the link tested, press **Send** (**F3**). The CNC prompts the operator to send the selected program or to change the program being sent.

2. Press **Select** (**F1**). The CNC sends the program.

      – *or* –

    Press **Change** (**F3**). ASCII soft keys are displayed. Enter the name of the desired program.

Receiving a Program

Enter a name for the received program before transmission.

To receive a program:

1. With the Communications screen active, the parameters set and the link tested, press **Receive** (**F1**). The CNC prompts the operator to type a name for the incoming program. ASCII soft keys are displayed.

2. Key in the desired name, and press **ENTER**. The CNC displays a "**Ready to Receive...**" message.

3. Begin transmission from the sending machine.

ANILAM

Setting the Transmission and Receiving Display

When transmitting or receiving with Text Mode on, the transmitted program is displayed on the screen. With Text Mode off, the display area remains blank. When Text Mode is on, **Text** (**F4**) highlights.

When data is exchanged, the "Transfer in Progress" symbol is displayed just above the soft key line.

Press **Text** (**F4**) while transmitting or receiving in order to switch the CNC in and out of Text Mode. The transmission or receiving process is faster in 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.

> **NOTE:**   You should 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.

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 or Automatic Mode. Real-Time Draw Mode is also available.

The CNC only runs programs written 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, in DNC Mode, do not run a program that contains subprogram calls.

The DNC screen is similar to other operating screens, but contains communications information. Refer to **Figure 11-5**.
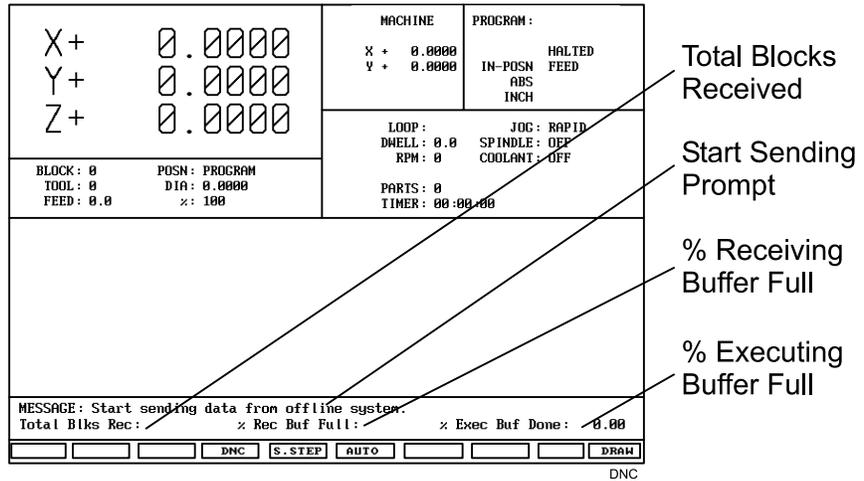
Figure 11-5, DNC Screen

Total Blks Rec:	The total number of program blocks the CNC has received at that point.
% Rec Buf Full	Percentage of receiving buffer used.
% Exec Buf Full	Percentage of executing buffer used.

An RS-232 link transmits program blocks faster than the receiving machine can run them. This makes it necessary to 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.

When using the software protocol (Xon/Xoff), the CNC can signal the sending machine to pause, if necessary, to avoid data loss.

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.

ANILAM

To put the CNC in Direct Numeric Control Mode:

1. With the Communications screen active, parameters set and the link tested, press **DNC** (**F4**). The DNC soft key highlights.

2. Press **Receive** (**F1**). The DNC screen activates and the CNC prompts the operator to start transferring from the other machine.

3. Press **START**. The CNC runs 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 DC Codes.

Table 11-1, DC 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" is 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 Communications 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, press **3** to stop the reader, if necessary. 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 prior to initiating 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, press **4** to stop the punch, if necessary. The CNC sends the DC4 code to the punch and the punch stops.

Section 12 - Calculators

The CNC features a powerful calculator package that contains three separate calculators:

- ❑ Math Calculator
- ❑ Right Triangle Calculator
- ❑ Geometry Calculator

The programmer 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 typically offered in a scientific calculator. These include basic math, trigonometry, unit conversion, logs, exponential operations, angle/radian conversions, and finding inverses.

Activating the Math Calculator

To activate the Math Calculator:

1. In Edit or MDI Mode, press **Calc** (**F7**). The calculator selection menu is displayed on the screen. Refer to **Figure 12-1**.
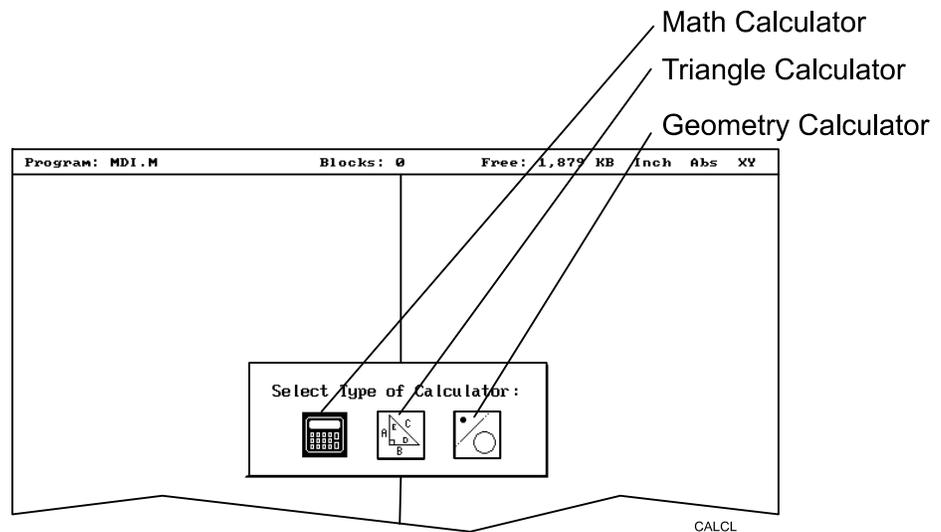
Figure 12-1, Calculator Selection Menu

2. Highlight the **MATH CALCULATOR** template, and press **ENTER**. The Math Calculator activates.
3. The Math Calculator is displayed in a pop-up box in the center of the screen. Refer to Figure 12-2, Math Calculator and Soft Keys.
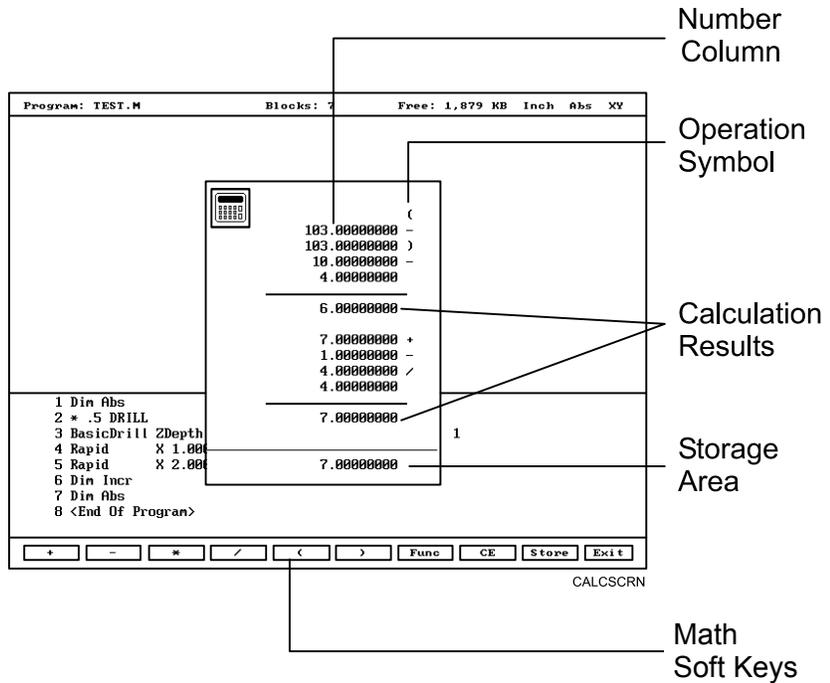
Figure 12-2, Math Calculator and Soft Keys

Math Calculator Basics

Numbers appear in the storage area, as entered. Select math operations using the appropriate soft key. Refer to Table 12-2, Function Selection Pop-Up Listing.

Refer to Table 12-1, Math Operation Soft Keys. Enter the first number of the calculation, and then press an operation soft key. The CNC places the number in the column. Next, select the operation performed with the next number. Enter the second number, and so forth.

After you enter the last number of a calculation, press **ENTER**. The CNC places the final number in the column and calculates the answer. The result is displayed in the storage area. Press **Store** (**F9**) to copy the result to the calculator's memory.

Press **CE** (**F8**) to clear previous calculations.

If the operator enters 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**)

Operations Involving Two Numbers

To add, subtract, multiply or divide two numbers:

1. With the Math Calculator active, enter the first number, followed by the appropriate function hot key: **+** (**F1**), **-** (**F2**), ***** (**F3**), or **/** (**F4**). The CNC adds number and operation symbol to the column.

2. Enter the second number, and press **ENTER**. CNC adds second number to the column and displays the answer in the storage area.

Math with a Column of Numbers

The CNC can manage a column of numbers requiring multiple operations, such as a continuous equation, in standard mathematical order (multiplication, division, addition, subtraction). Press **ENTER** to generate the result.

Using Parentheses

Use parentheses to indicate arithmetical operations in a non-standard order. (Example: find 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, if at all. Make sure the parenthetical expression contains a left parenthesis "(" at the beginning and a right parenthesis ")" at the end. Otherwise, the CNC will not calculate the result.

The CNC performs operations within parentheses top to bottom, as they appear in the column, and solves innermost expressions first.

For example, the following expression:

(7 + 4 + ((6 * 9) -1)) / 8

would generate 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 Box 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

ANILAM

To use an additional function:

1. With the Math Calculator active, enter the number and press **Func** (**F7**). The Function pop-up menu is displayed to the right of the calculator.

2. Highlight a function, and press **ENTER**. The result is displayed 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 holds 64 numbers. Numbers stored in memory can be recalled directly in a program.

The 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. In the Edit or MDI Mode, press **Calc** (**F7**). The calculator selection menu is displayed.

2. Highlight the **TRIANGLE CALCULATOR** template, and press **ENTER**. The Triangle Calculator activates. Refer to **Figure 12-3**.
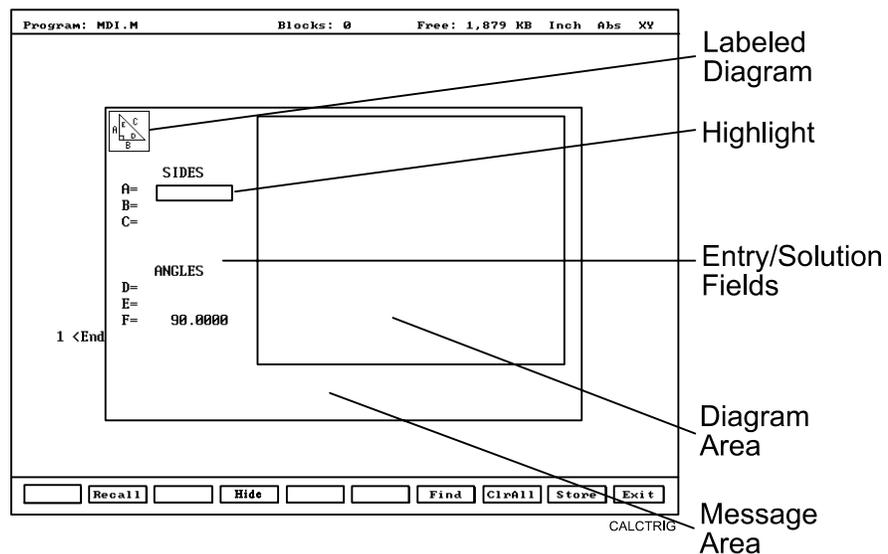
Figure 12-3, Right Triangle Calculator Screen

Using the Triangle Calculator

The Right Triangle Calculator only solves 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, the operator enters known element values: any two angles, any two sides or one angle and one side.

After the operator enters 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. A scaled drawing of the solved triangle is also displayed in the diagram area.

To clear a single incorrect value, highlight the value and press **CLEAR**. To clear all displayed values, press **ClrAll** (**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 in 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 enter the value.

3. Press **Find** (**F7**). Scaled drawing of triangle is displayed in the solution box. Calculated sides and angles are displayed in unused fields.

Storing Right Triangle Calculator Results

The CNC will save any side or angle value. Maximum: 64 values.

To store a value:

1. Solve the required triangle.

2. Highlight the value to be stored.

1. Press **ENTER** to store the selected value in memory.

Hiding the Right Triangle Calculator Screen

To hide the Right Triangle Calculator Screen temporarily:

1. Press **Hide** (**F4**) to hide the Right Triangle Calculator Screen temporarily.

2. Press **Find** (**F7**) to show the screen again.

The Geometry Calculator

The CNC uses Cartesian coordinates (X, Y-axis values) to define most positions. However, the operator 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

1. Refer to **Figure 12-4**. In Edit or MDI Mode, press **Calc** (**F7**). Calculator selection menu is displayed on the screen.
2. Highlight **GEOMETRY CALCULATOR** template, and press **ENTER**. Geometry Calculator activates.

Geometry Calculator Screen

The Geometry Calculator consists of a pop-up screen with a display area, a message area and eighteen geometry templates. The geometry construction is displayed in the display area. Refer to **Figure 12-4**.
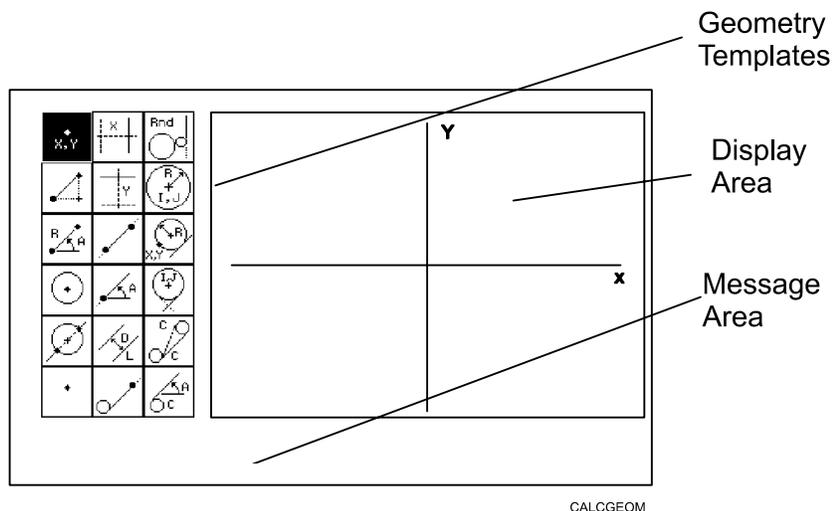
Figure 12-4, Geometry Calculator

Use the **DISPLAY** (**F5**) soft key selections to alter the view of 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. Pay close attention to 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 eighteen 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 will prompt the operator 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 the operator deletes an element from the sketch, the CNC deletes all stored information pertaining to 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 some feature of an existing element as a reference. Experiment to understand how each template operates.

ANILAM

NOTE:	After a series of deletions and additions, the display could appear incomplete. Press **Display** (**F5**) and select **Redraw** to refresh the screen.

Point Templates

Some point templates insert points at positions defined by the operator. Other point templates use other elements as references.

Many line and circle templates display a "**Select point definition . . .**," message when activated. This indicates that the selected tool requires the operator to select (or create) a reference point.

Templates that insert points at circle centers and element intersections will prompt the operator for the required element number(s). The CNC displays all possible intersections; the user selects one.

Table 12-3, Point Templates

Template	Purpose	Requirements
The following line templates are in the first column of the Calculator screen.		
X, Y	Defines a point in a sketch.	Must know absolute X and Y position of the point.
	Defines a point at an X & Y increment from existing point.	Must know incremental X and Y distances from existing point.
R A	Defines a point at radius and angle from existing point.	Must know distance from existing point. Must know number of degrees and direction ($\pm$) from the 3 o'clock position.
	Defines the center of an existing circle.	The circle must already be an element of the sketch.
	Defines the point of intersection between two existing elements. Prompts user to select an intersection when more than one 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 must already be an element of the sketch.
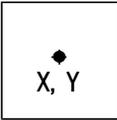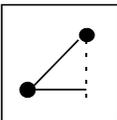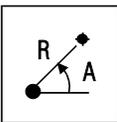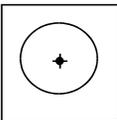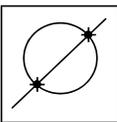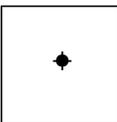
Line Templates

Line templates use other elements or axis positions as references. Templates that draw lines tangent to circles display all possible tangent lines and prompt the operator 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 Y-axis.	Must know absolute X-axis position of the line.
	Constructs a line parallel to X-axis.	Must know absolute Y-axis position of the line.
	Constructs a line between any two points. Prompts user to select a point tool to define each endpoint.	Use any point to locate the two endpoints.
	Constructs a line through a point, rotated specified number of degrees from the 3 o'clock position. Prompts user to select any point tool to define point of rotation.	Must know number of degrees and direction ($\pm$) from the 3 o'clock position. Use any point to locate point of rotation.
	Constructs a line parallel to existing line at specified (positive or negative) distance.	Existing line must already be an element of the sketch.
	Constructs a line through a selected point and tangent to an existing circle. Prompts user to select any point for point. Displays tangent lines on both sides of circle. User selects required tangent.	Circle must already 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 tangent lines possible. User selects required tangent.	Two circles must already 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 must already be an element of the sketch. Must know number of degrees of rotation from the 3 o'clock position.
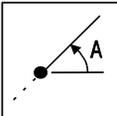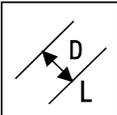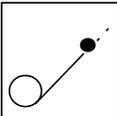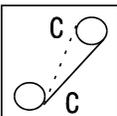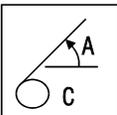
ANILAM

Circle Templates

Circle templates use other elements as positioning references.

Templates that draw circles tangent to other circles, lines or points display all possible tangent circles and prompt the operator 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 specified radius tangent to any two existing elements. All possible tangential circles are displayed. The user selects the required tangent.	Sketch must contain at least two elements.
	Constructs a circle of specified radius centered on a position. Prompts user to select a point to define the center point.	Use any point to locate the center. Must know radius.
	Constructs a circle of specified radius tangent to an existing line and through an existing point. Prompts user to select a point to define point.	Line must already 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 user to select any point to define center point.	Line must already be an element of the sketch. Use any point to locate the center.
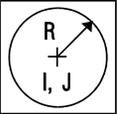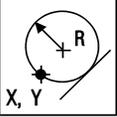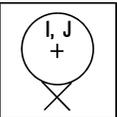
Deleting Selected Elements

To delete an element from the sketch:

1. With the Geometry Calculator active, press **GEOMETR** (**F7**). A pop-up menu is displayed.

2. Highlight **Delete Item**, and press **ENTER**. The screen prompts for the element number being deleted.

3. Enter the element number, and press **ENTER** to delete item.

Deleting All Elements

To clear all elements from the display area:

1. With the Geometry Calculator active, press **GEOMETR** (**F7**). A pop-up menu is displayed.

2. Highlight **Delete All**, and press **ENTER**. The screen prompts the user to confirm deletion.

3. Press **Yes** (**F1**) to clear the screen. Press **No** (**F2**) to cancel the command.

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 line, endpoint coordinates, and, sometimes, the angle in reference to the 3 o'clock, 0 degree position; for a circle, the circle center and radius.

To display the Geometry List:

1. With the Geometry Calculator open, press **GEOMETR** (**F7**). A pop-up menu is displayed.

2. Highlight **Geometry List**, and press **ENTER**. The Geometry List is displayed.

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**). A pop-up menu is displayed.

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. The CNC calculates and displays the shortest distance between the two elements.

Last Position Recall

Whenever the calculator prompts for an X position, Y position or circle, the operator can recall the last X position, Y position or circle used by the calculator.

To recall a position:

1. When the calculator 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 in 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. The operator can recall saved values only from the same calculator in which they were saved. Use the menu to select the type of recall. Refer to **Figure 12-5**.
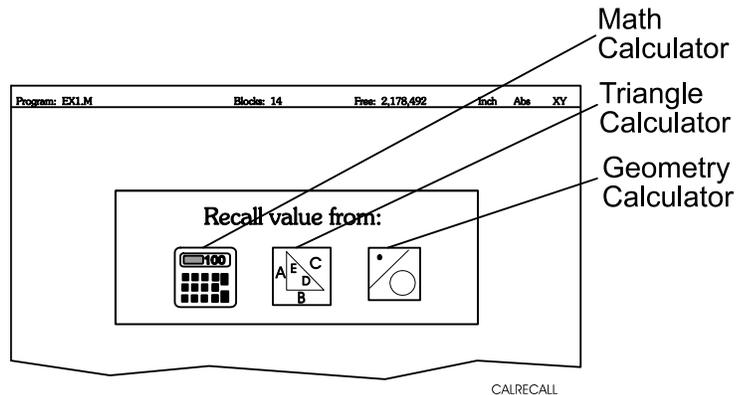
Figure 12-5, Recall Selection Pop-Up

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**). Calculator recall selection menu is displayed. Refer to **Figure 12-5**.

2. Highlight the **MATH CALCULATOR** template, and press **ENTER**. The **Select value:** pop-up is displayed with the most recently saved value displayed at the top. Refer to **Figure 12-6**.

3. Highlight the required value, and press **ENTER** to copy the stored value into the Graphic Menu.
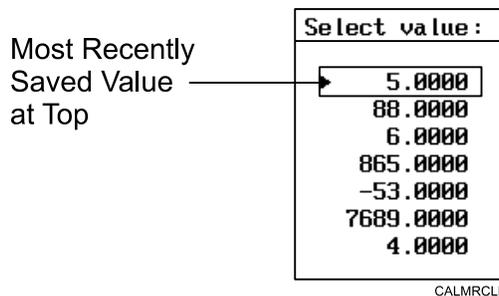
Figure 12-6, Math Calculator Select Value Pop-up

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**). The **Select value:** menu is displayed. Refer to Figure 12-5, Recall Selection Pop-Up.

3. Highlight the **TRIANGLE CALCULATO**R template, and press **ENTER**. The Triangle Calculator memory selection pop-up is displayed. Refer to **Figure 12-7**.

4. Highlight the required value, and press **ENTER** to copy stored value to the Graphic Menu.
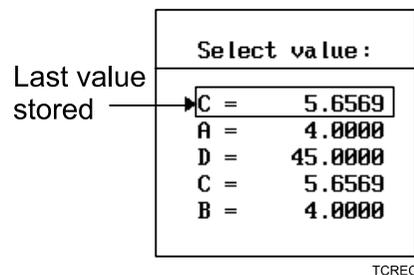
Figure 12-7, Triangle Calculator Recall Pop-Up

Recalling Values from the Geometry Calculator

Recall Geometry Calculator values from the calculator's **Select point:** pop-up menu. This menu is displayed next to a copy of the sketch that generated the points. The recall listing 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 or both of the position coordinates can be recalled.

ANILAM

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**). The calculator recall selection menu is displayed. Refer to Figure 12-5, Recall Selection Pop-Up.

3. Highlight the **GEOMETRY CALCULATOR** template, and press **ENTER**. The **Select point:** pop-up menu is displayed. Refer to **Figure 12-8**.

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**. The CNC copies selected values to the Graphic Menu.

NOTE:	Position coordinates can only be recalled to Graphic Menu entry fields that require X and/or Y coordinate values.

[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 12-8, Geometry Calculator Select point Pop-up Menu

Recalling Values from Calculator to Calculator

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 software requires an **Intel® based Personal Computer (PC) or 100% compatible. A minimum of 4MB of RAM is required. The hard disk drive must have a minimum of 4MB of space available.

The off-line software is also compatible with **Microsoft® **Windows® Operating Systems, such as **Windows NT®, Windows 95, Windows 98, or Windows 2000.

Passwords

Access to some setup parameters is restricted. Where required, the CNC prompts the user to enter a password. Refer to **Table 13-1**.

To access protected parameters:

1. Type the service password, and press **ENTER**. The CNC will enable you to change the protected settings.

2. There are four access levels protected by passwords. Operators are assigned limited passwords, which enable them to set parameters used in normal machine operations. Service passwords enable a higher level of access. The factory password is not used outside the manufacturer's plant.

Table 13-1, Off-line Passwords

Restriction Level	Password
Limited access	Q1
Service access	Q2
IPI access	Q3

Exiting the Software

Press **ESC** to exit the software from the Main Menu selection screen.

- - - - -

** Intel® is a registered trademark of Intel Corporation in the United States and/or other countries.

** Microsoft®, Windows®, and Windows NT® are registered trademarks of Microsoft Corporation in the United States and/or other countries.

Windows Off-line Software Installation

1. Insert the installation disk in the floppy drive.

2. Go to the task bar, and click on the **Start** button. Select **Run**. The Run window activates.

3. In the **Open** entry field, type "**A:setup**". Click on **OK**. The installation procedure will then begin.

4. Follow the on-screen prompts as they appear.

> **NOTE:** Substitute "**B:setup**" for "**A:setup**" if your 3.5 inch floppy is in the B:drive.

Running Off-line Software from Windows

1. If you selected "Desktop Icon" (recommended) during the installation, click the 3000M icon on your desktop.

2. If you selected Start Menu, start from there.

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 feature limits the amount of memory available to the software, preventing the CNC program from tapping into Windows' large virtual memory supply. Allocating too much memory to the control software will dramatically increase startup time and make the software run slower.

To change the **Maximum Memory Allocated** parameter:

1. In the CNC's startup screen, select **Setup Utility**, and press **ENTER**. The Setup Options menu is displayed.

2. Select **Operator Software**, and press **ENTER**. The General Software menu is displayed.

3. Select **Control Software**, and press **ENTER**. The Control Software menu is displayed.

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 editing a program that is larger than this value. In this case, the CNC will generate an error message indicating that there is not enough memory to edit the program. To correct this problem, change this parameter to be the size of program plus 1MB.

Disabled Features

The following software features, found in the Program Directory's **Utility** (**F9**) pop-up are not available under any Windows operating system:

- ❑ Check Disk
- ❑ Floppy Format
- ❑ Disk Optimization
- ❑ Program Restore

If you select one 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's (**F1** to **F10**) soft keys, located below the screen on the console. The **SHIFT** key activates secondary soft key functions, when applicable. To access secondary soft key labels on an off-line keyboard, press the **SHIFT** key and the appropriate function key (**F1** to **F10**) together.

Refer to Table 13-2, Keyboard Keystroke Equivalents. To make jog moves via 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** + **K** 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, Keyboard Keystroke Equivalents 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 the keyboard equivalents of the keypad keys.

Table 13-2, Keyboard Keystroke Equivalents

Key Name	Key Face	Keyboard Keystroke Equivalent
ABS/INCR	ABS INC	**ALT** + **E**
(**+/-**) key	+/-	**+** OR **-**
CLEAR	CLEAR	**ALT** + **C** – *or* – **DELETE**
ARROWS	(arrows)	**ARROWS**
ENTER	ENTER	**ENTER**
RAPID	(hand)	**ALT** + **F1**
FEED	(hand)	**ALT** + **F2**
JOG: 100	(hand)	**ALT** + **F3**
JOG: 10	(hand)	**ALT** + **F4**
JOG: 1	(hand)	**ALT** + **F5**
STOP CONTINUOUS JOG	none	**ALT** + **T**
X+	X+	**ALT** + **X**
X-	X-	**ALT** + **K**
Y+	Y+	**ALT** + **Y**
Y-	Y-	**ALT** + **L**
START	(start)	**ALT** + **S**
HOLD	(hold)	**ALT** + **H**
FEEDRATE OVERRIDE 0%	(dial)	**ALT** + **0**
FEEDRATE OVERRIDE 10%	(dial)	**ALT** + **1**
FEEDRATE OVERRIDE 20%	(dial)	**ALT** + **2**

(Continued…)

Table 13-2, Keyboard Keystroke Equivalents (Continued)

Key Name	Key Face	Keyboard Keystroke Equivalent
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**
FEEDRATE OVERRIDE 110%		**ALT + B**
FEEDRATE OVERRIDE 120%		**ALT + D**
SPINDLE FORWARD	SPDL FWD	**ALT + F**
SPINDLE OFF	SPDL OFF	**ALT + O**
SPINDLE REV	SPDL REV	**ALT + G**

Table 13-3, Off-line Keyboard, Tool Page Utilities

Function	Keystroke	Operation
Clear all entries on Tool Page.	**SHIFT + F3**	Clears all entries from Tool Page; prompts for confirmation before clearing page.
Exit Tool Page without saving changes.	**SHIFT + F10**	Exits Tool Page; unsaved edits 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 soft key labels.
Delete any program in specified path.	**SHIFT + F3**	Prompt for **PROGRAM TO DELETE: _** is displayed above soft key labels.
Display System Information screen.	**ALT + I**	Displays System Information screen.

Editing with a Text Editor

Refer to **Figure 13-1**. The operator can also edit programs with the ASCII text editor:
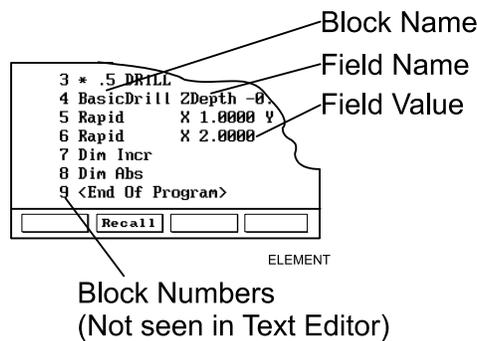
Figure 13-1, Program Block Elements

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 in 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 text editors are not case-sensitive.

❑ Spell block names as they appear in the Program Listing.

❑ A decimal point is not necessary after whole numbers.

❑ 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: The operator can set up machines with non-modal Arc centers.

❑ Arc direction (Cw/Ccw) is modal and remains in effect until canceled or changed.

Numbered values must conform to the specification listed for the G-code converter. (Refer to "Section 9 - Program Management.")

ANILAM

Section 14 - 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 an MS-DOS or 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 an 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
```

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 14-1, Mouse Operations and Table 14-2, 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 14-4, 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 14-1** and Table 14-2, DXF Hot Keys.

Table 14-1, 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.

ANILAM

DXF Hot Keys

Refer to **Table 14-2**.

Table 14-2, 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 14-3**.

Table 14-3, Soft Key Descriptions

Soft Key	Function	Description
F1	Toggle Select Mode	Select mode must be on when chaining shapes.
F3	Layers Menu	Pop-up menu has: ▪ All Layers on ▪ Invert Layers ▪ Toggle Layers Layers can be turned on or off as desired. **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.
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 14-4, Output Menu Descriptions and to Table 14-5, Display Menu Descriptions.

(Continued…)

Table 14-3, Soft Key Descriptions (Continued)

Soft Key	Function	Description
F10	Exit	**F10** exits the Setup menus, exits the DXF Converter, and returns to the Program page.
		Be sure to Save (**F8**) any work done before exiting. Anything not saved will be lost.
		If shapes have been created, a message, "Exit (Y/N)?" is displayed. This is a reminder to be sure you have saved your work.

Miscellaneous DXF Soft Key, F6

Press **Misc.** (**F6**) to display the miscellaneous DXF pop-up menu with the following options:

Entity Info	Displays entity information at the bottom of the screen. (See Table 14-1, Mouse Operations. This option is equivalent to pressing (**CTRL** + left click).
	On the off-line or using an auxiliary keyboard, the Entity Info is displayed by holding the **CTRL** key when you pick the entity.
Set Shift	Pastes the Entity Info onto the Shift field. (See Table 14-2, DXF Hot Keys. This option is equivalent to pressing (**ALT** + **T**).
Toggle Endpoints	Toggles entity endpoints. (See Table 14-2, DXF Hot Keys. This option is equivalent to pressing (**ALT** + **F**).
Link or New Shape	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**).

Output Menu Options

Refer to **Table 14-4**.

Table 14-4, 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 14-5**.

Table 14-5, 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 14-6** for the DXF entities supported.

Table 14-6, 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 14-6**. 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.

14-10

31-July-05

ANILAM

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 14-1**.

Figure 14-1, Example DXF File

Refer to **Figure 14-2**. All unneeded layers have been turned off. The Figure shows the drill locations and the contour selected.
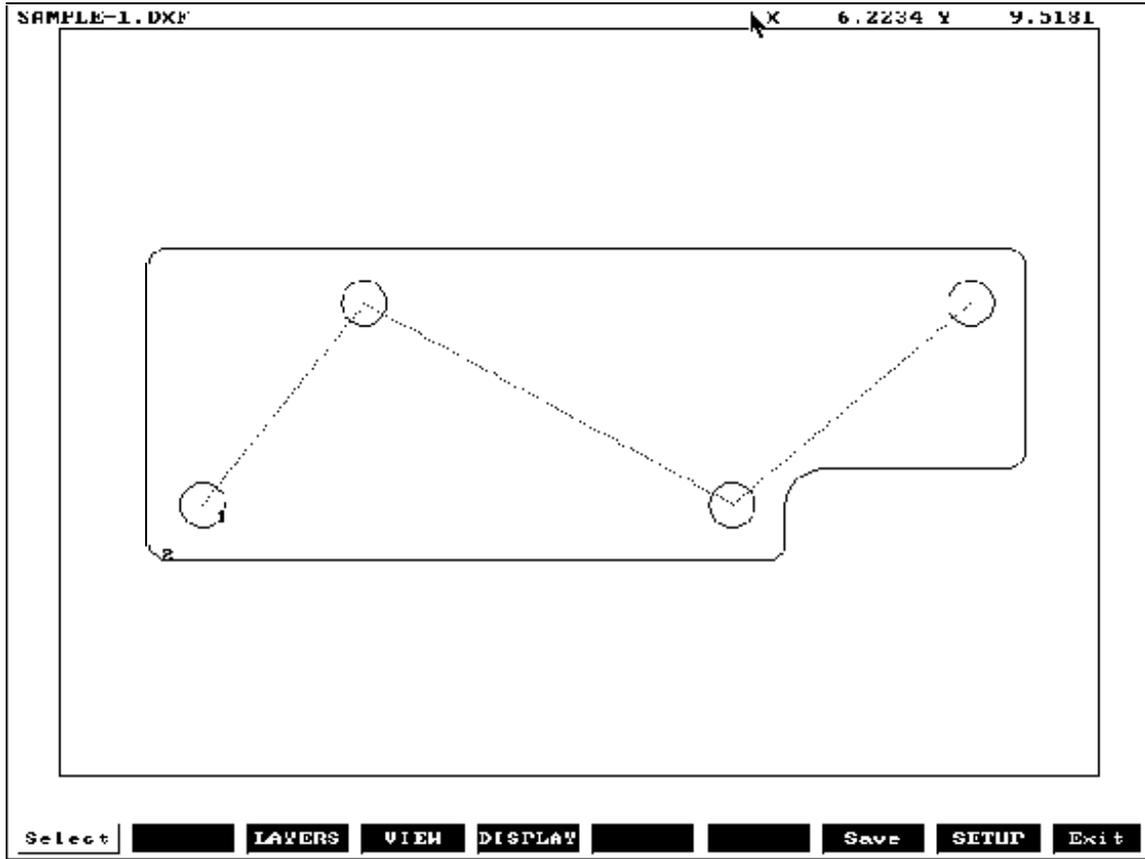
Figure 14-2, Zoomed Part with Unneeded Layers Turned Off

From **Figure 14-2**, the Output Menu (see Table 14-4, Output Menu Descriptions) would display as follows:

Table 14-7, Output Menu for Figure 14-2

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 14-2, Zoomed Part with Unneeded Layers Turned Off follows. See **Table 14-8**.

Table 14-8, 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 14-3, Edited Conversational Tool Path.

Edited Conversational Tool Path

The edited conversational tool path is illustrated in **Figure 14-3**.
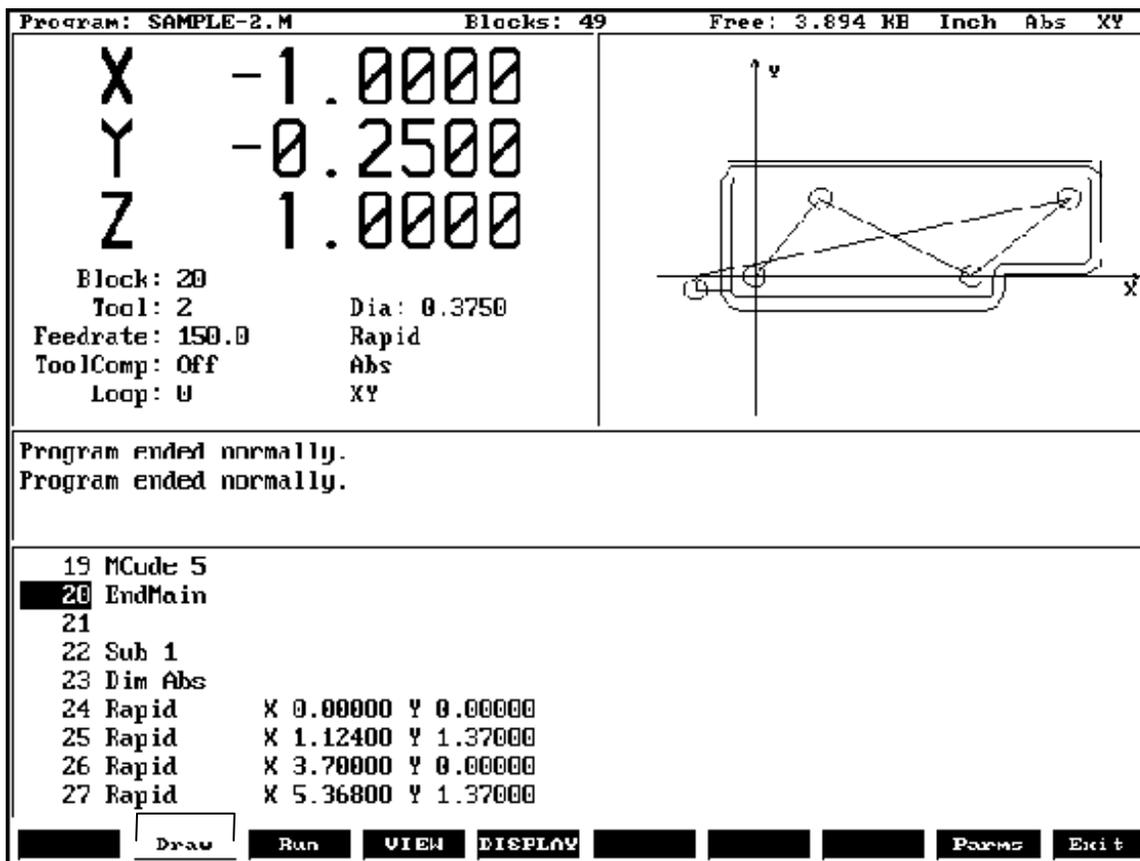
Figure 14-3, Edited Conversational Tool Path

Edited Conversational Program Listing

The edited conversational program used for the example above follows. Refer to **Table 14-9**.

Table 14-9, 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
```

ANILAM

Section 15 - CNC Software

Machine Software Installation

To install the machine software:

1. Insert the software disk in the floppy drive.

2. From the CNC's startup screen, select **Setup Utility**, and press **ENTER**. The **Setup Options** menu is displayed.

3. Select **Machine/Installation**, and press **ENTER**. The **Machine/Installation** menu is displayed.

4. Select **Software Update**, and press **ENTER**. The installation sequence begins. Follow the instructions on the screen.

Software Option Kit Installation

Software option kits contain new or optional features for the control.

Each kit typically contains the following items:

❑ Installation Procedure

❑ Installation Diskette

❑ Hardware Key

The hardware key enables purchased options.

Procedure

To install software option kits:

1. Use the shipping document to verify that the kit contains the ordered options.

2. Turn off the computer.

3. Gain access to the female DB-25 printer port connector. Install the hardware key.

NOTE:	Remove any previously installed hardware key before you install the new key.

3. Install the floppy disk in the floppy drive.

4. Install the software as described above.

NOTE:	In the Setup Utility, make any configuration changes necessary 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."

F

G

to program, 5-37
repetitive drilling cycle,
  subprograms, 5-27
repetitive operations, 5-5, 5-26
required, entry fields, 4-4
resetting
  absolute zero, 4-13
  after stop, 3-1
  the servos, 3-1, 3-2
restart
  Draw automatic, 7-8
  program, 8-2
  spindle, 3-2
restore, a program, 9-6
Resume (F2), 11-7
return, to machine home, 4-9
right triangle calculator. *See*
  triangle calculator
right-hand, tool compensation,
  illustration, 1-6
RMS
  (F6), 5-30
  description, 5-30
  SUB, graphic menu, illustration, 5-30
  subprogram, entry fields, 5-30
  subprogram, to call, 5-30
rotating, mirroring, and scaling.
  See RMS
rotating, mirroring, and scaling,
  subprograms, 5-30
rough and finish cycles,
  subprograms, 5-27
rounding
  arc-to-arc corner, 1-12
  corner, 1-11
  line-to-arc corner, 1-12
  line-to-line corner, 1-11
RPM, label, 3-4
RPM, spindle, 10-3
RS 274D, 9-8
RS-232
  cable, installing, 11-1
  communication connector, illustration,
    11-1
  communications, 8-6
Run, parameter, 7-8
running
  Draw, selected blocks, 7-8
  program, one step at a time, 8-1
  programs, 8-1
RUNNING, label, 3-4

S

S.STEP
  current operating mode, 3-4
S.Step mode, program listing,
  3-3
S.Step, Draw, 7-7
saving, a program block, 4-3
saving, edits, 6-3
scaling, subprograms, 5-30
scaling, the Draw display, 7-11
scientific, calculator, 12-1
screen saver, description, 2-5
screen, clutter, 7-6
screens
  communication, illustration, 11-2
  DNC, illustration, 11-8
  Draw (real-time mode), 8-5
  Draw, illustration, 7-3
  engraving cycle, illustration, 5-36
  geometry calculator, illustration, 12-7
  manual mode, illustration, 3-3
  math calculator, illustration, 12-2
  mill cycle, illustration, 5-39
  program editor, illustration, 6-1
  repeat cycle, illustration, 5-37
  system information, illustration, 9-12
  Test Link, illustration, 11-4
  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
searching, blocks, for words or
  numbers, 6-4
secondary display area
  ABS/INCR mode, 3-8
  illustration, 3-3
  labels, description, 3-4
secondary, status area, 3-9
seconds remaining in a dwell,
  3-4
selecting
  communication port, 11-3
  Draw view, 7-12
  program for editing, 9-3
  program for utilities, 9-3
send mode, data control codes,
  to use, 11-10

ANILAM

www.anilam.com