ANILAM

1100M CNC Three Axis Programming Manual

ANILAM
One Precision Way
Jamestown, NY 14701
☎ (716) 662-2899
FAX (716) 661-1884
anilaminc@anilamcom

P/N 70000289

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

—D—

—M—

—R—

—T—

Table Of Contents

Introduction

This manual supports the 1100M Three Axis CNC software package. Most of the topics in this manual are introduced by a short description followed by specific instructions for performing the required tasks. All example programs and training exercises are contained in Section 12 - Sample Programs and Practice Exercises.

There are two approaches to learning a software package, read the manual and later try to remember what it said, or go through the steps with the manual. This manual is written as both a training guide and a reference. We recommend all operators read the entire manual at least once to become familiar with its contents.

All topics are not listed in the Table Of Contents. Refer to the index in the back of the manual to find the topics not listed in the Table Of Contents.

In this manual, the 1100M Three Axis CNC is also referred to as the CNC or the control.

Notation

References to specific words, labels, and prompts displayed on the CNC are printed in bold text, the way they appear on the screen.

References to keypad and keyboard keys show the key name in parentheses accompanied by a graphic i.e. **(CAPS)** 

.

References to soft keys show the softkey label in bold text, adjacent to the function key in parentheses. i.e. **MANUAL** (F4).

Section 1 - 1100M Programming Concepts

Programs

A program is the set of instructions used by the CNC to direct the machine's movements. Each instruction is called a block. Each block executes independently allowing a program to be stepped along one block at a time.

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

Axis Descriptions

The machine moves along its axes of motion. All movements along an axis are in either a positive or negative direction. Not all machines use the same system for identifying axes. The descriptions here are commonly used for three axes mills.

NOTE: To keep the directions straight when trying to visualize machine movements, imagine tool motion rather than table motion.

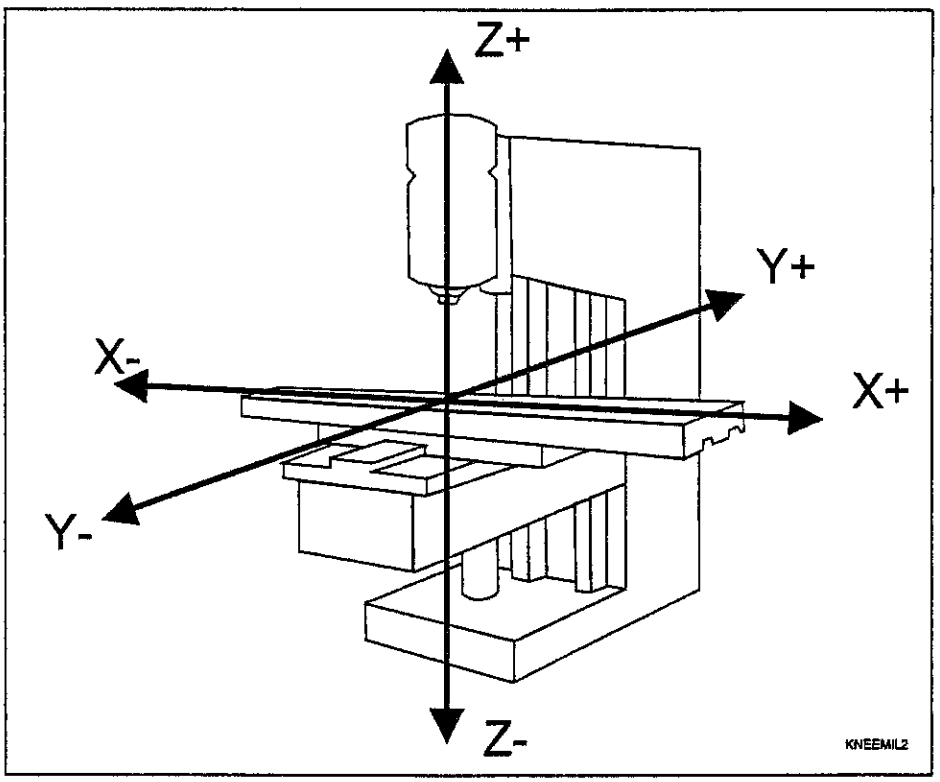


Figure 1 - 1, Mill Axes of motion

X Axis

Refer to **Figure 1 - 1, Mill Axes of motion**. Table movement along the X axis is left and right. Positive motion is table movement to the left, negative motion is table movement to the right.

Y Axis

Refer to **Figure 1 - 1, Mill Axes of motion**. Table movement along the Y axis is in and out. Positive motion is table movement out, negative motion is table movement in.

Z Axis

Refer to **Figure 1 - 1, Mill Axes of motion**. Spindle movement along the Z axis is up and down. Positive motion is tool movement up, negative motion is tool movements down (into the work).

Defining Positions

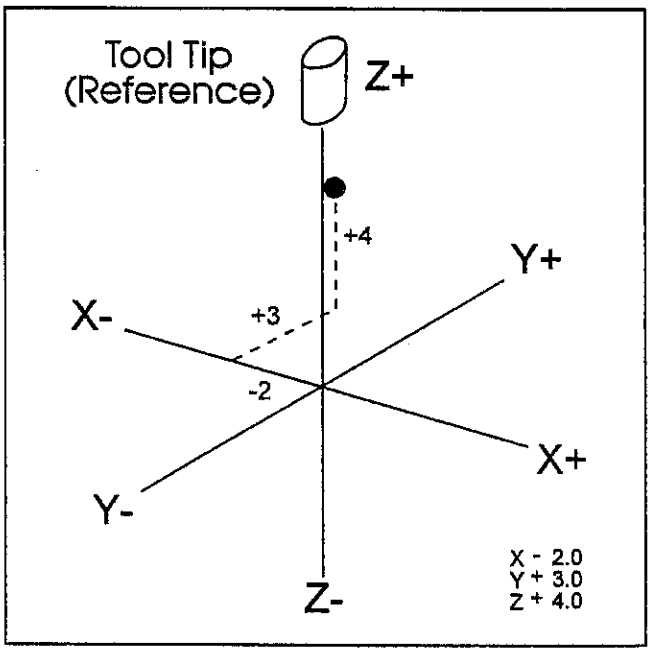


Figure 1 - 2, Locating Positions

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

Most positions are identified by X, Y, and Z coordinates. A position 2 inches left, 3 inches back, and 4 inches up has an X coordinate of X -2.0, a Y coordinate of Y3.0, and a Z coordinate of Z4.0.

Polar Coordinates

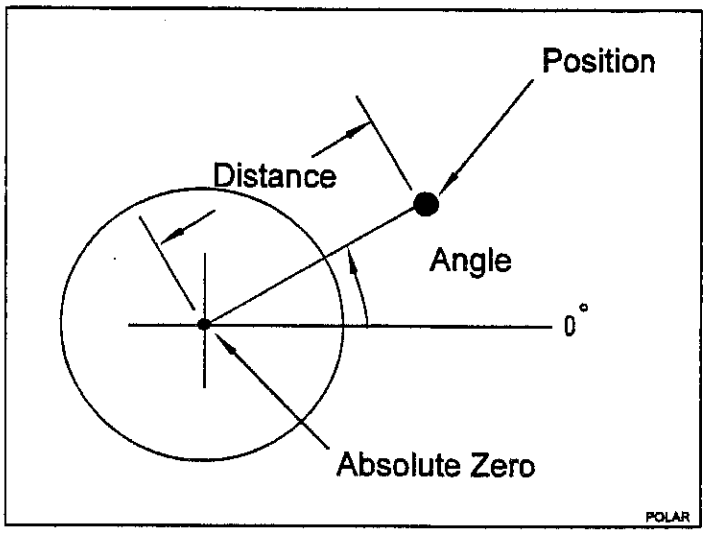


Figure 1 - 3, Polar Coordinate System

Refer to **Figure 1 - 3, Polar Coordinate System**. Sometimes when it is not convenient to use X, Y, Z coordinates, the Polar Coordinate System is used. Polar Coordinates can only define points that lie on a single plane. Polar coordinates use the distance from the origin, and an angle to locate points.

Absolute Positioning

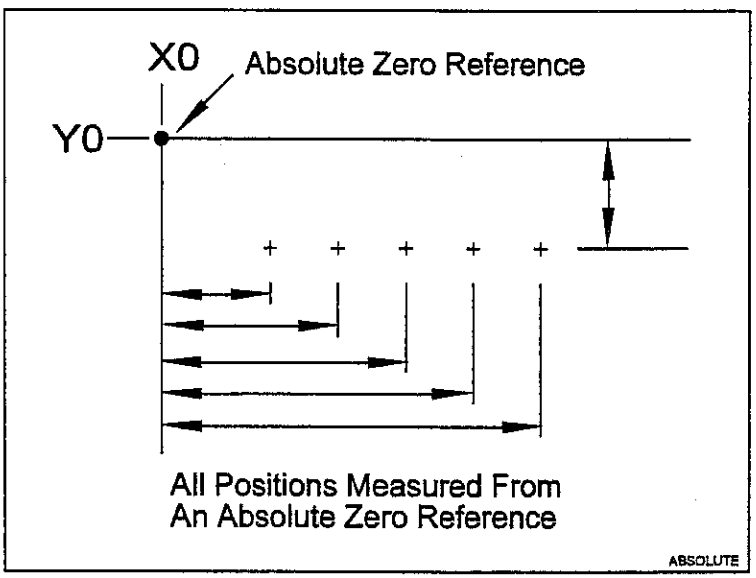


Figure 1 - 4, Absolute Positioning

Refer to **Figure 1 - 4, Absolute Positioning**. When the CNC is in the Absolute Mode, all positions are measured from the Absolute Zero Reference point. The Absolute Zero Reference is not a fixed position on

the machine. The Absolute Zero Reference is a point selected by the operator.

The operator can set the Absolute Zero Reference point (X0, Y0) anywhere. Usually the operator sets the Absolute Zero Reference at position that makes it easy to use the dimensions right off the blueprint. This is also called setting the "part zero".

The Absolute Zero Reference or part zero can be moved as often as necessary either manually or in a program.

Incremental Positioning

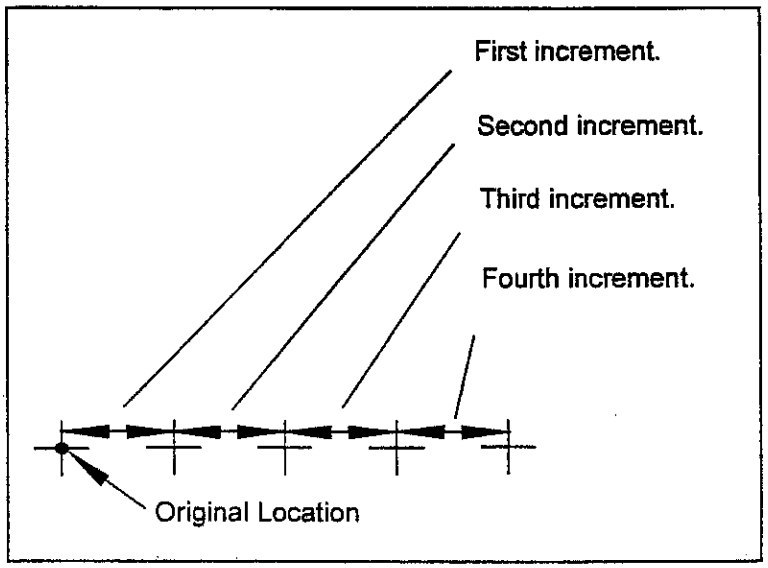


Figure 1 - 5, Incremental Positioning

Refer to **Figure 1 - 5, Incremental Positioning**. Incremental positions are measured from one point to another, or from the machines present position. This is convenient for performing an operation at regularly spaced intervals. Incremental positions are measured from the tools present position.

NOTE: An incremental 0 inch (0 mm) move will not make a position change.

Tool Length Offsets

The Z0 position of the quill with no tool length offset applied, is set by the operator. Usually it is the fully retracted position of the quill. A move to Z0 with tool #0 active is a move to Z0 with no tool length offset applied.

Because most tools have different lengths, the Z axis part zero reference is not set the same way it is set for the XY axes. Ideally the CNC shows the Z axis at the Z0 position when the tip of the tool is at the part Z0 position. This is done by entering a length offset for the tool, refer to **Section 8 - Tool Management**.

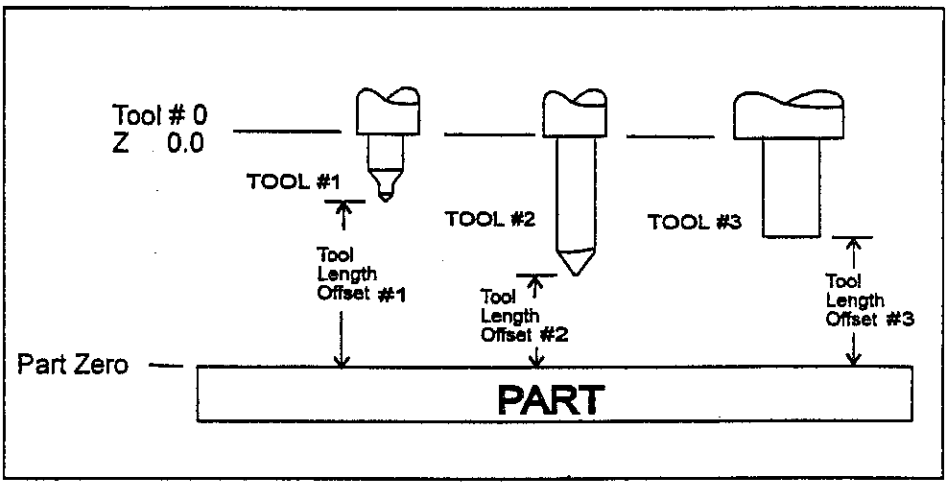


Figure 1 - 6, Tool Length Offset

Refer to **Figure 1 - 6, Tool Length Offset** . Tool length offset is the amount of quill travel required from Z0 tool #0 to put the tip of the tool at the part Z0.

When tool length offsets are used, the Z axis position display indicates Z0 when the active tool is moved to the Z axis part zero position. Tool length offsets simplify programming. To move to a position .5 inch into the work, the operator programs a move to a Z-.5 position, regardless of the tool's length.

Tool Diameter Compensation

NOTE: An operator should become familiar with the basic principles of CNC before attempting to write compensated moves.

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 ½ a diameter away from the path. This problem is overcome by using tool (diameter) compensation.

When tool compensation is active, the CNC offsets the tool by ½ a diameter to position the cutting edge of the tool on the programmed path.

This allows the operator to program the moves that cut the edge of a part by writing moves that follow the part's profile. The operator does not need to adjust the path to compensate for the tool's diameter.

Most moves can be compensated. The operator must specify right or left compensation. Right or left refers to the side of the path the tool offsets to, looking from behind the tool as it moves.

The endpoints of a compensated move are offset perpendicular to the programmed endpoint by ½ of the tool's diameter.

NOTE: The correct tool diameter must be active for tool compensation to be accurate. Refer to **Section 8 - Tool Management** and **Section 4 - Writing Programs** for specific information on activating tool information.

NOTE: A knowledge of tool compensation is not required to program any of the CNC's canned cycles. Tool compensation is built into canned cycles that require it.

NOTE: Simple tool compensation cannot be used with some shapes, for example, the ellipse because of its effect on the geometry.

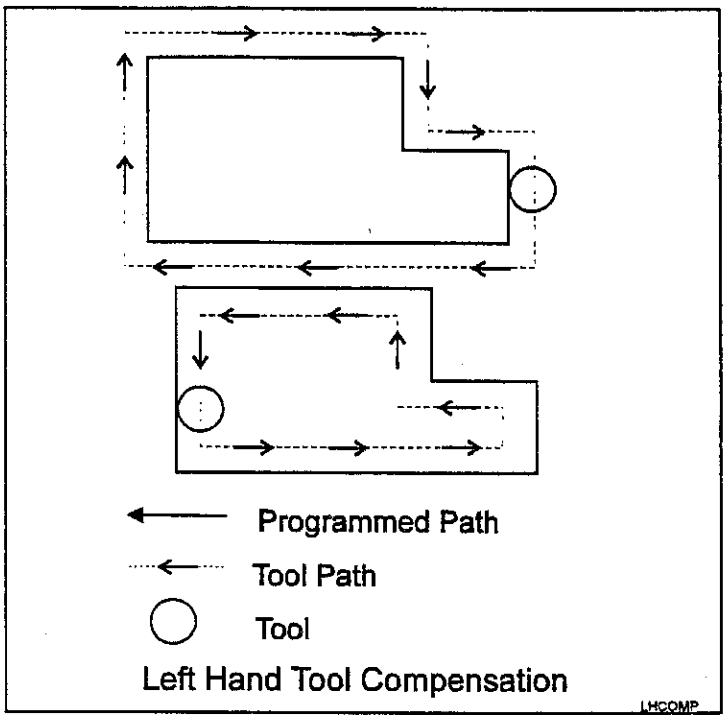


Figure 1 - 7, Left Hand Tool Compensation

Refer to **Figure 1 - 7, Left Hand Tool Compensation**. When left hand tool compensation is activated, the tool offsets to the left of the programmed path (looking from behind the tool as it moves).

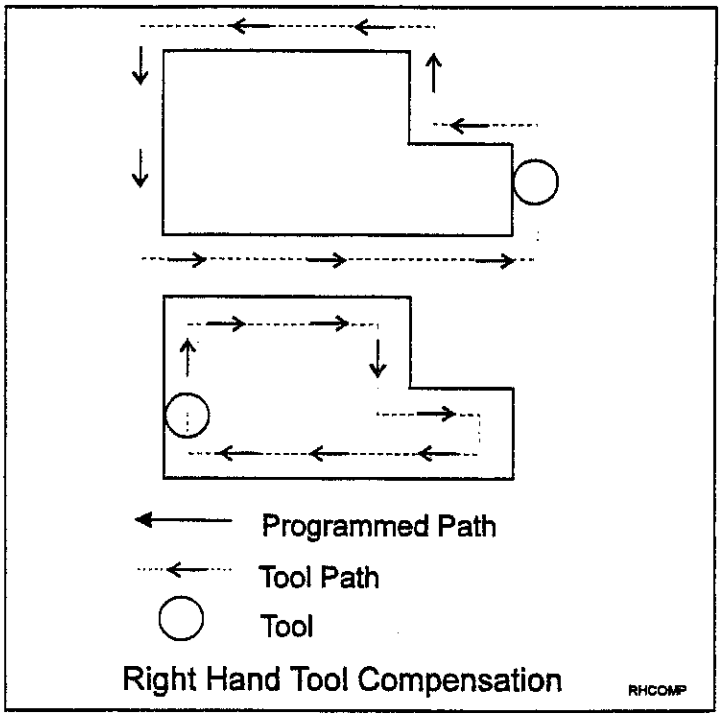


Figure 1 - 8, Right Hand Tool Compensation

Refer to **Figure 1 - 8, Right Hand Tool Compensation**. When right hand tool compensation is activated, the tool offsets to the right of the programmed path (looking from behind the tool as it moves).

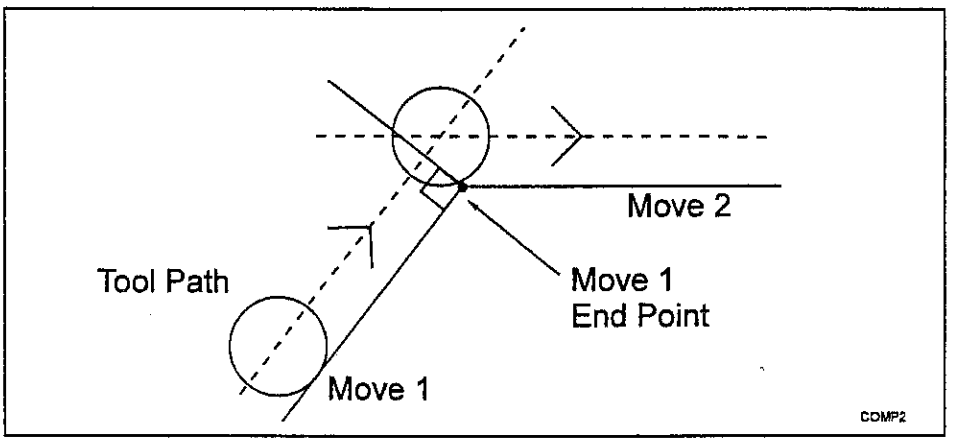


Figure 1 - 9, Consecutive Compensated Moves

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

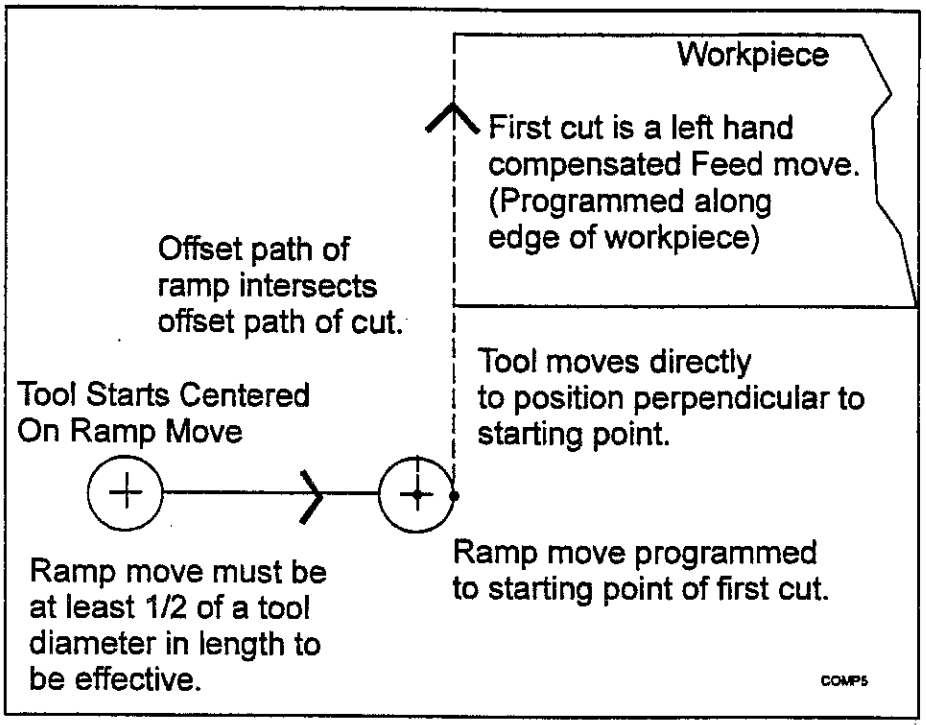


Figure 1 - 10, Ramping Into a Compensated Move

Refer to **Figure 1 - 10, Ramping Into a Compensated Move**. The moves to and from compensated moves are called ramp moves. The ramp move is required to give the CNC time to change the way it positions the tool. The ramp move must be at least ½ of the active tool's diameter in length.

At the start of a ramp move the tool is centered on the programmed path. At the end of the ramp move (starting point of the compensated move), the tool's center is perpendicular to the starting point, and offset by ½ of the tool's diameter.

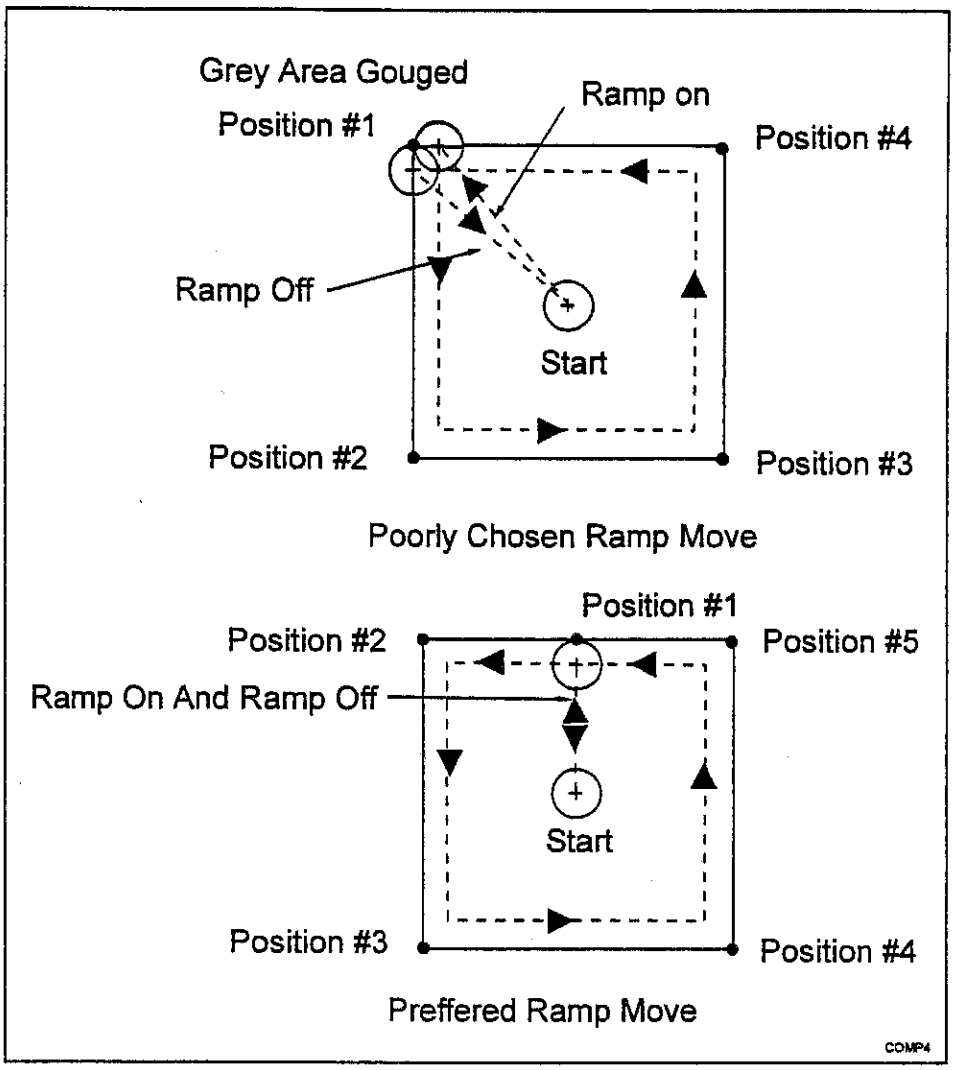


Figure 1 - 11, Ramp On/Off Choices for Milling Inside a Square

The operator should carefully consider how compensation will affect the position of the tool at the starting and end points of a move.

Refer to **Figure 1 - 11, Ramp On/Off Choices for Milling Inside a Square**. When the compensated moves start and stop in the corner, the tool gouges the work. This is because the tool offsets to a position perpendicular to the end points. If the compensated moves ramp on and off at the side, the work isn't gouged.

NOTE: When possible use a canned cycle to cut profiles and pockets. The CNC automatically selects ramp on/off positions in a canned cycle.

Using Tool Diameter Compensation and Length Offsets with Ball End Mills

When using a ball-end mill to cut contoured surfaces, if tool diameter compensation is used, it must be used with a tool length offset. Unlike an end cutter, the tool length offset for a ball - end mill is not set to the tip of the tool.

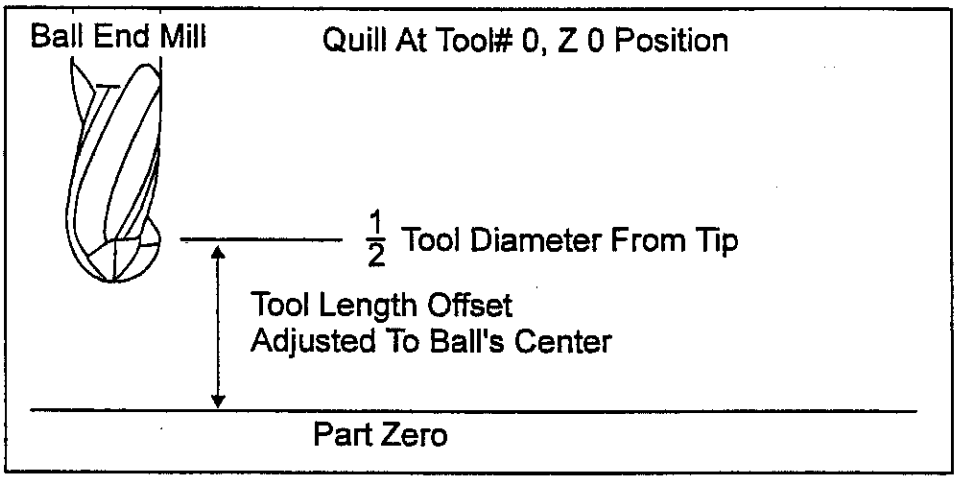


Figure 1 - 12, Setting Tool Length Offset for Ball End Mill

Refer to **Figure 1 - 12, Setting Tool Length Offset for Ball End Mill**. The tool length offset for a ball - end mill should be set ½ of the tool's diameter back from the tip. Refer to **Section 8 - Tool Management** for details on setting tool length offsets.

Angle Measurement

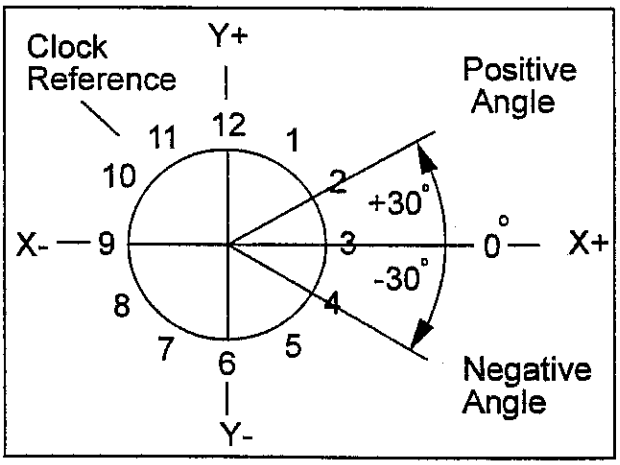


Figure 1 - 13, Absolute Angle Measurement

Refer to **Figure 1 - 13, Absolute Angle Measurement**. Angles are measured using the 3 o'clock position as the zero degree reference. Positive angles rotate in the counter clockwise direction, negative angles rotate to the clockwise direction.

Corner Rounding

Corner rounding permits the operator to radius the intersection between consecutive moves.

To activate corner rounding, the operator keys a radius value (positive) into the **CornerRad** field of the first move. When the program runs, it blends the end point of the first move into 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.

Corner rounding can be used between two Line moves, two Arc moves and between Lines and Arcs (that aren't already tangent).

Line to Line Corner Rounding

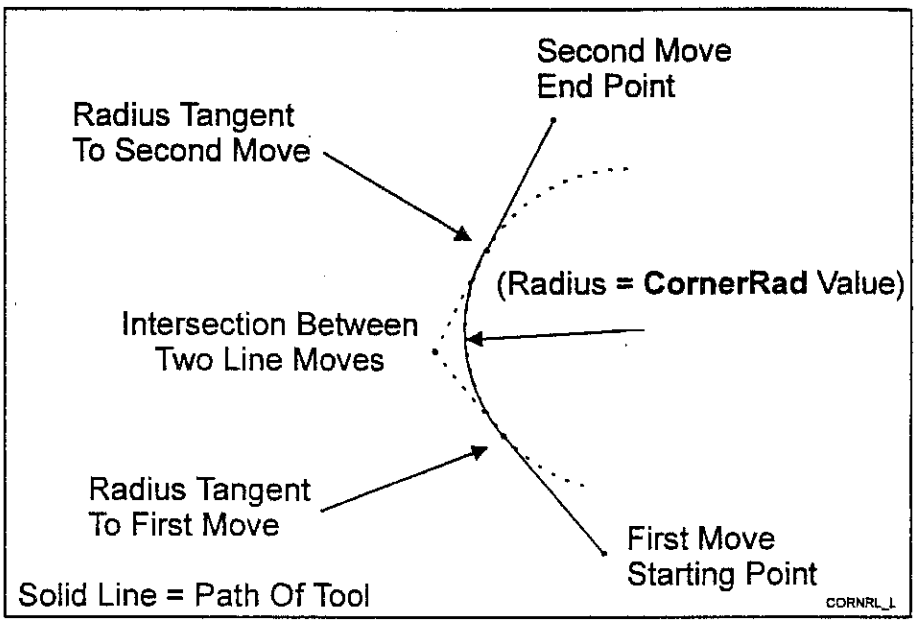


Figure 1 - 14, Line to Line Corner Rounding

Refer to **Figure 1 - 14, Line to Line 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.

Line to Arc Corner Rounding

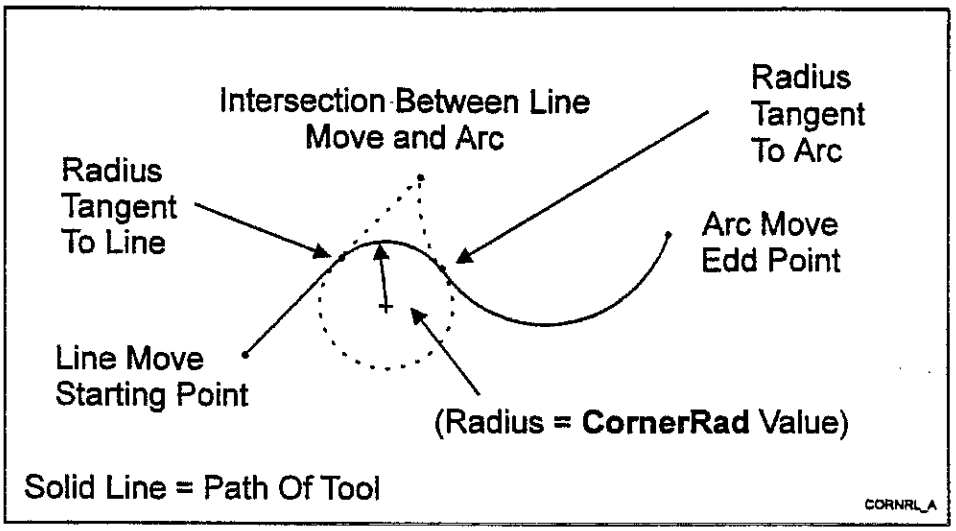


Figure 1 - 15, Line to Arc Corner Rounding

Refer to **Figure 1 - 15, Line 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.

NOTE: If the line move is tangent to the arc move, corner rounding will be ignored.

Arc to Arc Corner Rounding

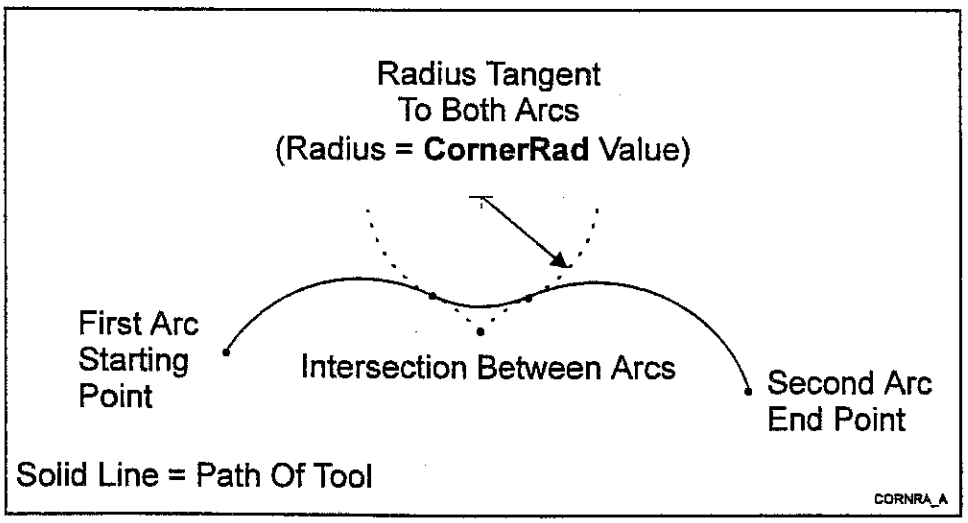


Figure 1 - 16, Arc to Arc Corner Rounding

Refer to **Figure 1 - 16, 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.

Chamfering

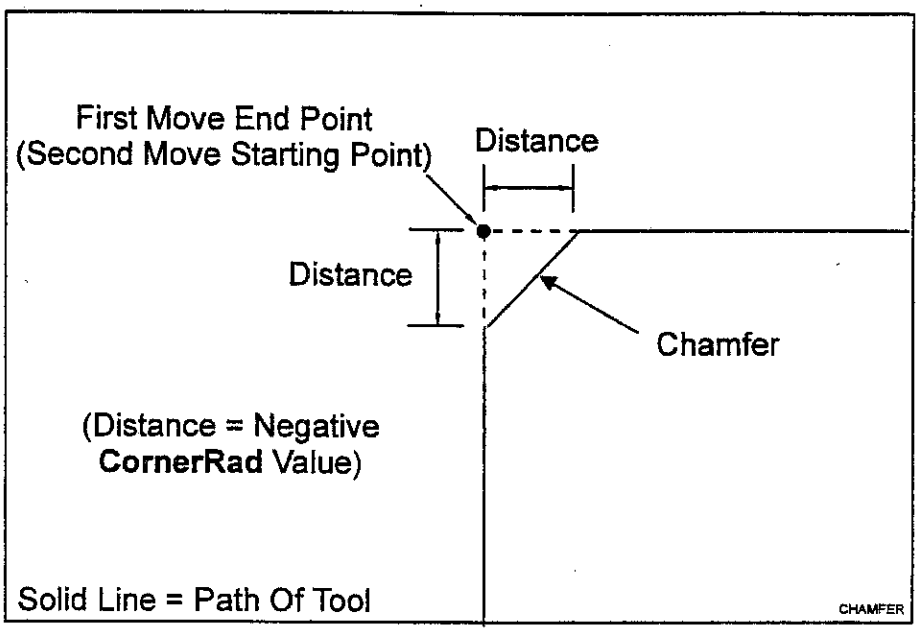


Figure 1 - 17, Chamfering

Refer to **Figure 1 - 17, Chamfering**. Chamfering is used between two consecutive line moves. A chamfer starts at a specified distance before the end point of the first move. A chamfer ends the same distance away from the starting point of the second move. Activate chamfering by keying a negative value into the **CornerRad** field of the first move. The value entered is the distance shown in the figure. The resulting tool path follows the solid line in the figure.

Plane Selection

Circular moves and tool diameter compensation are confined to the plane selected by the user. Three planes are available, the XY plane, the XZ plane, and the YZ plane. It is important to view a plane correctly when planning a circular move. Viewing a plane from the wrong side causes arc directions, angle references, and axis signs to appear reversed.

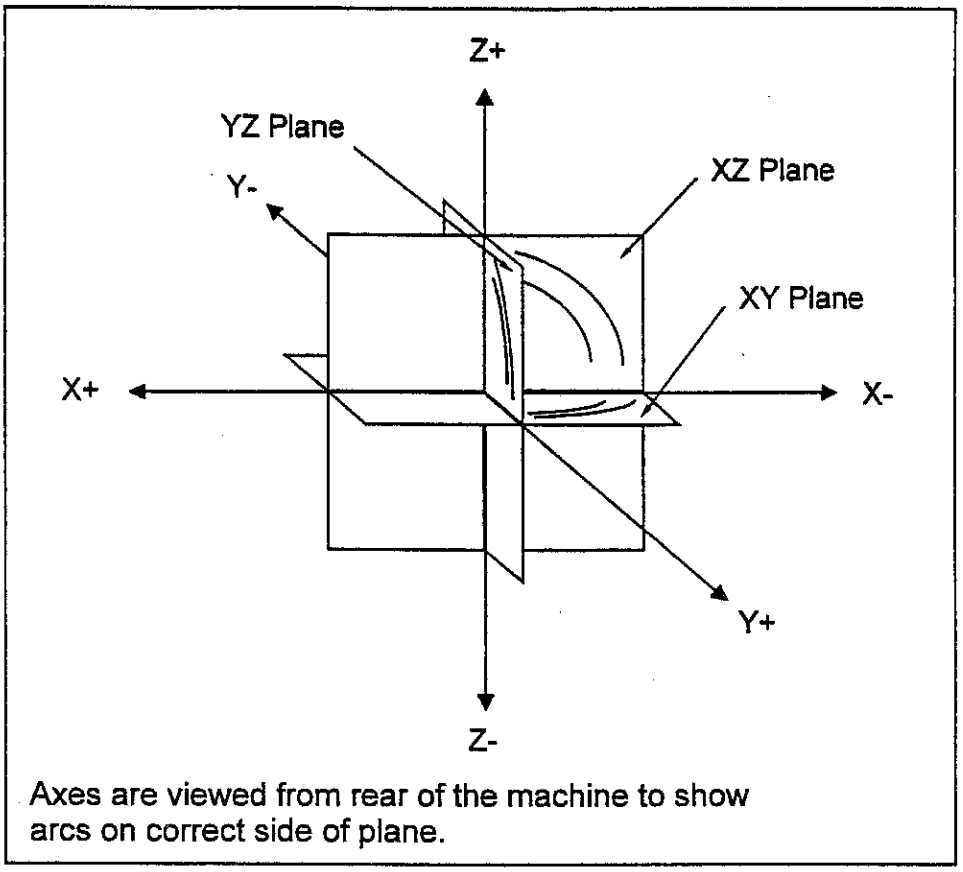


Figure 1 - 18, Plane Identification

Refer to **Figure 1 - 18, Plane Identification**. The standard rule is to view a plane looking in the negative direction along the unused axis.

Section 2 - CNC Console and Software Basics

The Console

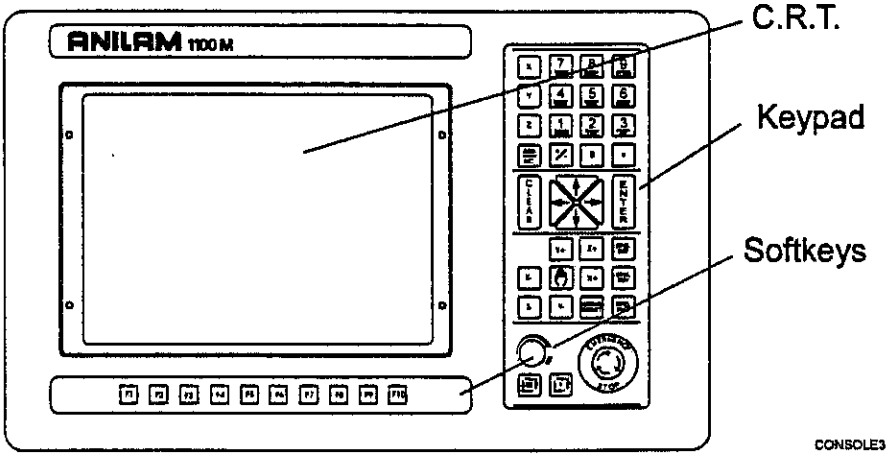


Figure 2 - 1, CNC Console

Refer to **Figure 2 - 1, CNC Console**. The CNC console consists of a C.R.T. (a 14 in. Monochrome VGA monitor) and two keypads, one to the right of the monitor, the other underneath.

Keypad

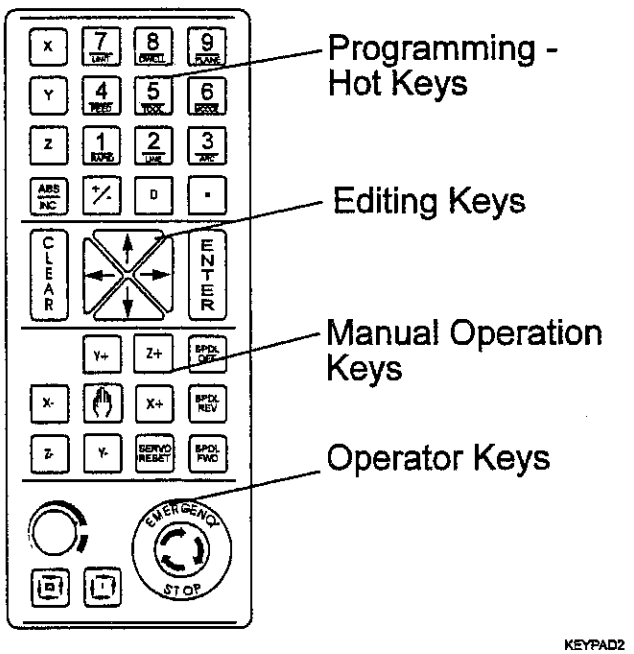


Figure 2 - 2, Keypad

Refer to **Figure 2 - 2, Keypad**. The keypad to the right of the monitor is laid out in four groups.

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

Programming - Hot Keys

Refer to **Table 2 - 1, Programming - Hot Keys**. Programming / Hot Keys are dual purpose keys. They let the operator key in position coordinates and provide quick access to functions that speed up programming. Hot key functions are active in the Edit and Manual Modes when the CNC is not specifically expecting a number entry.

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.
(ABS/INC)	ABS / INC	Toggles CNC between incremental and absolute modes.
(0)	0	Zero / Toggles comment asterisk in edit mode.
(1/RAPID)	1 RAPID	One / Hot key for programming a Rapid move.
(2/LINE)	2 LINE	Two / Hot key for programming a Line move. NOTE: A Line move is a straight line move in feed.
(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 / Hotkey for activating tool diameter compensation and length offset.
(6/MCODE)	6 MCODE	Six / Hotkey for programming an M-code.
(7/UNIT)	7 UNIT	Seven / Hotkey for toggling between inches (**Inch**) and millimeter (**MM**).
(8/DWELL)	8 DWELL	Eight / Hotkey for programming a Dwell.
(9/PLANE)	9 PLANE	Nine / Hotkey for selecting a plane.
(+/-)	+/-	Sign change / Toggle key.
(.)	.	Decimal point.

Editing Keys

Refer to **Table 2 - 2, Editing Keys**. The editing keys are used by the operator for editing programs.

Table 2 - 2, Editing Keys

Label or Name	Key Face	Purpose
(CLEAR)		Clears messages, values, commands and program blocks selected by the operator.
(ARROW)		Lets the operator move highlight bars and insertion points around the screen.
(ENTER)		Selects blocks for editing, activates menu selections, activates number entry.

Manual Operation Keys

Refer to **Table 2 - 3, Manual Operation Keys**. Manual Operation Keys let the operator manually control machine movements.

Table 2 - 3, Manual Operation Keys

Label or Name	Key Face	Purpose
(JOG)		Cycles the CNC through manual movement modes (**JOG: RAPID**, **JOG: FEED**, **JOG: 100**, **JOG: 10**, **JOG: 1**). NOTE: The actual speed of the machine during a move is determined by the machine builder.
(Z+)		Manually moves machine in positive Z direction.
(Z-)		Manually moves machine in negative Z direction.
(Y+)		Manually moves machine in positive Y direction.
(Y-)		Manually moves machine in negative Y direction.
(X+)		Manually moves machine in positive X direction.
(X-)		Manually moves machine in negative X direction.
(SERVO RESET)		Activates servo motors.
(SPINDLE FORWARD)		Starts spindle in a clockwise direction (viewed from the top of the motor) optional.
(SPINDLE REVERSE)		Starts spindle in a counter clockwise direction (viewed from the top of the motor) optional.
(SPINDLE OFF)		Stops the spindle.

Operator Keys

Refer to **Table 2 - 4, Operator Keys**. Operator Keys start, stop, override feedrate, and hold the machine. They are active in both the programmed and manual operating modes.

Table 2 - 4, Operator Keys

Label or Name	Key Face	Purpose
(FEEDRATE OVERIDE)		Overrides the feedrate from 0% to 120%. Overrides the rapid move rate from 0% to 100%.
(E-STOP)		The emergency stop button (E - STOP) (RED) disconnects the machine's servos stopping the spindle and all machine movement.
(START)		The start key (GREEN) starts all machine moves except jog.
(HOLD)		The hold key (RED) pauses any running program or programmed move. Press (START)  to continue.

Soft Keys (F1) - (F10)

The soft keys (also called function keys) are located just below the monitor and are labeled (F1) through (F10). Soft key functions are not hard wired, their functions change with changes in mode.

The current function of each soft-key appears in the label directly above each key. If a softkey label is blank, the softkey has no active function. A softkey function is only available while the label is displayed.

Optional / Off-Line Keyboard

The 1100M CNC supports an optional external keyboard. Most standard PC computer keyboards are compatible. Refer to **Section 11 - Software, Keyboard and Printer Installation**. All keypad inputs except the (E - STOP)  and (SERVO RESET)  are available on a keyboard.

WARNING: Emergency shutdown can not be made from a keyboard. Press the (E - STOP)  **button to perform emergency shutdowns.**

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

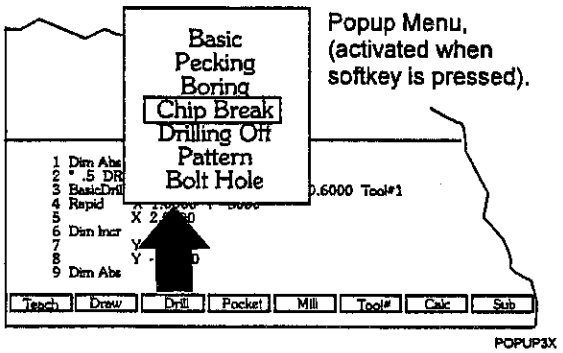


Figure 2 - 3, Pop-up Menu

Refer to **Figure 2 - 3, Pop-up Menu**. Pop up menus are temporary menus that let the operator make additional selections. Pop-up menus automatically appear where needed. Each pop-up menu contains a highlight bar. The (ARROW) keys move the highlight bar up and down the menu. The highlighted selection is activated by pressing (ENTER) [ENTER]. Close a pop-up menu by pressing the key that activated or by pressing (CLEAR) [CLEAR].

Screen Saver

After a set period of inactivity the CNC's screen dims to preserve the C.R.T. Pressing any key restores the CNC to a ready status. Keystrokes are not recognized until after the monitor is reactivated.

Toggling Selections

When the CNC prompts for a choice between one of several possible selections, the operator can toggle between the options by pressing the (+/-) [+/-] key. This key also produces the negative sign.

Highlight Bars

Menu screens are equipped with a highlight bar. The highlight bar lets the operator select one of the items on the menu. Move the highlight bar up and down (or left and right) by pressing the appropriate (ARROW) key. Pressing (ENTER) activates the highlighted selection.

Clearing Entries

Pressing (CLEAR) deletes all portions of an incomplete or mistaken entry.

Operator Prompts

The CNC displays a prompt when it requires specific information. When the CNC prompts for a text entry, an **ASCII** (F2) softkey appears. This softkey activates the ASCII Chart. The ASCII Chart lets the operator key letters in from the keypad.

ASCII Charts

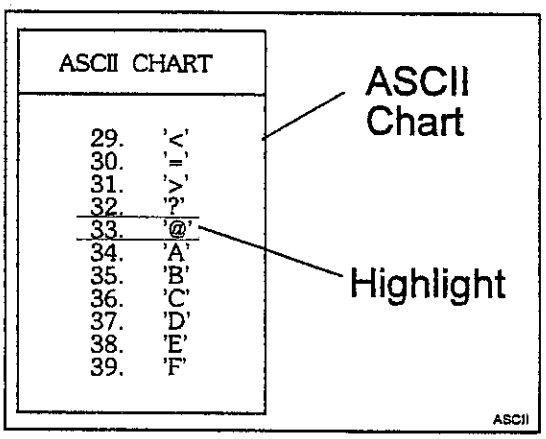


Figure 2 - 4, ASCII Chart Pop-up

Refer to **Figure 2 - 4, ASCII Chart Pop-up**. When the CNC prompts for a text entry, the **ASCII** (F2) softkey is displayed. Pressing the **ASCII** (F2) softkey toggles the ASCII Chart on and off. The ASCII Chart enables the operator to key text in from the keypad.

The Insertion Point

The insertion point appears when the ACSII Chart is active and when entering values on the Tool Page. The insertion point is a white underline that indicates where letters and numbers will be inserted.

Keying In Text

The operator must use the ASCII Chart (or a keyboard) to key in text. Enter text using the **ASCII** (F2) chart as follows:

1. Press **ASCII** (F2) (ASCII Chart activates).
2. Using the (ARROW)  keys, move the highlight to select the desired character.
3. Press (ENTER)  (selected character replaces the insertion point).

Typing Over and Inserting Letters and Numbers

The ASCII Chart has two text entry modes: "Typeover" and "Insert". By default the ASCII Chart runs in the Typeover Mode.

In the Typeover Mode, new characters replace characters marked by the insertion point.

In the Insert Mode, new characters appear at the insertion point and existing characters move to the right. The **Ins** (F3) softkey label is highlighted when the CNC is in the Insert Mode.

NOTE: Program names can only be changed when entered for the first time. Change an existing program name by renaming the program. Refer to **Section 7 - Program Management** for information on naming programs.

To put the CNC in the Insert Mode:

1. When the CNC prompts for a name, press **Ins** (F3) (**Ins**, softkey label highlights to indicate insert mode is on).

Deleting Letters

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

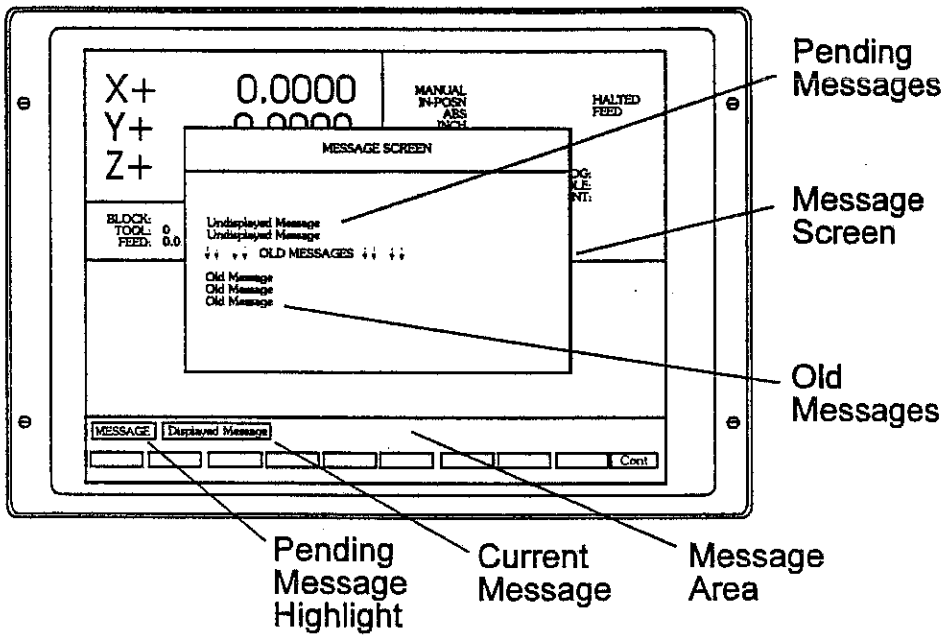
NOTE: If (CLEAR)  is pressed, the entire word is cleared.

3. After keying in the name, press the **ASCII** (F2) softkey (ASCII Chart closes).
4. Press (ENTER)  (name or text is saved to the CNC).

Messages / Error Messages

Messages generated by the CNC appear in the message area. The message area is present in all program running modes. When the CNC generates more than one message, the message with the highest priority appears in the message area. The Lower priority messages remain in memory. The **MESSAGE** label stays highlighted as long as pending messages remain in memory. There are two ways to review pending messages.

- Press (CLEAR) [CLEAR] (current message clears, next message appears).
- Go to the **MANUAL** screen and press **MESSAGE** (F1) (**MESSAGE** screen activates).



MESSAGE2

Figure 2 - 5, Messages Display

Refer to **Figure 2 - 5, Messages Display**. The **MESSAGE** screen lets the operator view the last eight messages (whether or not they have been displayed). As new messages are generated the oldest are cleared. Messages that haven't been displayed appear above the arrows, old messages appear below. Close the **MESSAGE** screen by pressing **Cont** (F10).

Clear stored messages by closing the **MESSAGE** screen and pressing (CLEAR) [CLEAR] until the **MESSAGE** highlight clears.

Some messages are advisory in nature, others put the CNC in hold.

If an error holds the CNC, the operator will need to put the CNC in the Manual Mode to correct the problem. Refer to **Section 6 - Running Programs** for information on clearing halted programs.

Section 3 - Manual Operation and Machine Setup

Powering On the CNC

The (E-STOP)  switch may be in or out when powering up. Start the CNC as follows:

1. Apply power to the CNC at the power switch located on the CNC cabinet (startup screen activates and prompts operator to **"Press F10 to continue"**).
2. Press the (F10) softkey (CNC displays the **Software Options** menu).
3. Using the (ARROW)  keys, position the highlight to select **"1. CNC Control"**, and press (ENTER)  (Manual Mode activates).

Shutting Down the CNC

1. Press the (E-STOP)  (servos disengage, and control goes to Manual Mode).
2. Press the **EXIT** (F10) softkey (CNC displays **Software Options** menu).
3. Remove power from the CNC at the power switch located on the CNC cabinet.

The Emergency Stop (E-STOP) Button

Pressing the (E-STOP)  takes all axes and spindle servos off line ending all machine movement.

Reset the (E-STOP)  by rotating the knob clockwise, in the direction of the arrows (switch makes clicking sound when reset).

Resetting the (E-STOP)  does not automatically reactivate the servos. The servos must be reset for the CNC to move the machine and to start the spindle. Press the (SERVO RESET)  to reset the servos.

Performing an Emergency Stop

1. Press the (E-STOP)  (servos disengage, and control goes to Manual Mode).

NOTE: An emergency stop cannot be performed using the keys of an off-line keyboard.

Activating/Resetting the Servos

For safety reasons, the control powers up with the servo motors disengaged. While the servos are off, the CNC cannot move the machine and the spindle will not start. A "**SERVO OFF !**" message is generated when the servos are disengaged. Refer to **Section 2 - CNC Console and Software Basics** for reviewing messages. The servos are also disengaged by pressing the (E-STOP)  or if the machine attempts to power past a limit switch.

Reset the servos as follows:

1. If the servos were disengaged by a limit switch, manually reposition the machine inside it's range of travel.
2. Press the (E-STOP)  .
3. Reset the (E-STOP)  by rotating it in the direction of the arrows (switch makes a clicking sound when reset)
4. Press (SERVO RESET) [SERVO RESET] (servos reset).

NOTE: The servos cannot be reset using the keys of an off-line keyboard.

Starting the Spindle

Pressing the (SPINDLE FORWARD) [SPDL FWD] and (SPINDLE REVERSE) [SPDL REV] keys will have no effect if the (E-STOP)  button is engaged or if the servos are off line.

To start the spindle:

1. Reset the (E-STOP)  if required.

2. Press (SERVO RESET) [SERVO RESET] if required.
3. Press either the (SPINDLE FORWARD) [SPDL FWD] or (SPINDLE REVERSE) [SPDL REV] key.

Manual Mode Screen

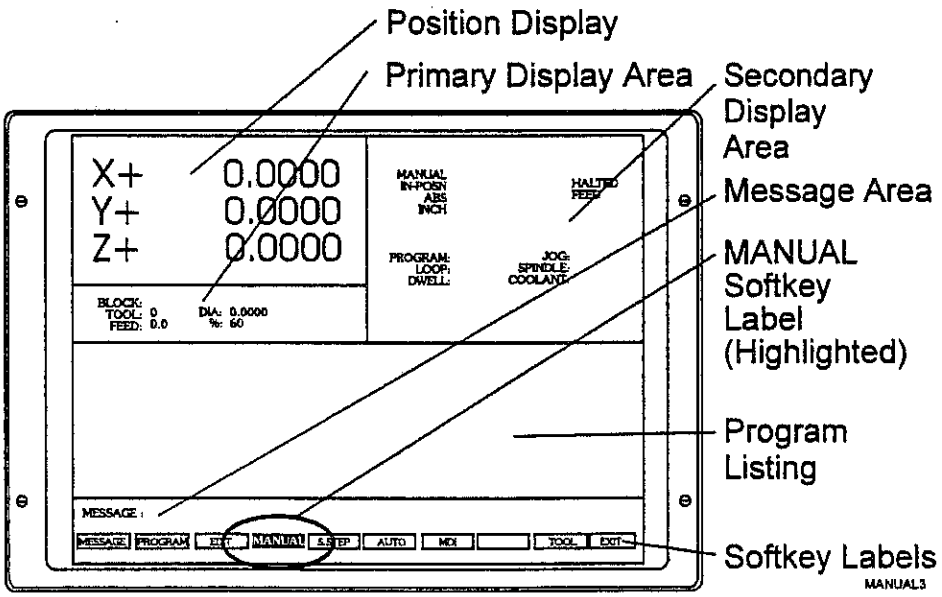


Figure 3 - 1, Manual Mode Screen

Refer to **Figure 3 - 1, Manual Mode Screen**. The **MANUAL** screen is the first control screen displayed. It is also the basic operating screen of the CNC. All other operating screens are similar in appearance and are selected from the **MANUAL** screen. To avoid confusion with the other screens, the **MANUAL** (F4) softkey label is highlighted while the CNC is in the Manual Mode.

The **MANUAL** screen is divided into five distinct areas.

Position Display Displays machines X, Y, and Z position coordinates.

Primary Display Area
Displays essential operating information.

Secondary Display Area
Displays non-essential operating information. (When Draw is activated the Secondary Display Area is replaced by the Part Graphics Display).

Message Line Messages, prompts, and reminders are displayed here.

Softkey Labels	These labels identify the function of the softkey directly underneath. Labels (and softkey functions) change from screen to screen, a highlighted label indicates an active mode.
Program Area	Displays program blocks as they are executed.

Primary Display Area Labels

BLOCK:	Current program block number.
TOOL:	Active tool.
FEED:	Current feed rate.
DIA:	Active tool diameter.
%:	Feedrate overide setting (0% to 120% for Feed moves, 0% to 100%) for Rapid moves.

Secondary Display Area Labels

MANUAL/AUTO/S.STEP:	Current operating mode.
ABS / INC	Current positioning mode.
INCH / MM	Current units mode.
PROGRAM:	Name of loaded program.
LOOP:	Number of loops remaining (when running a looped subprogram).
DWELL:	Seconds remaining in a dwell.
RPM:	Spindle RPM (not installed on all machines). May display programmed RPM or actual RPM, refer to builders documentation for details.
IN-POSN / RUN:	Tells operator whether machine has reached an end point or is still in transit.
HALTED/*HALTED	No asterisk, indicates machine is in a programmed hold, or has completed its program. With asterisk, indicates hold was activated by an event, or that (HOLD)  was pressed.
FEED/RAPID/ARC	Current move mode.
JOG:	Current jog mode.
SPINDLE:FWD/REV/OFF	Spindle status (Optional).
COOLANT:	Coolant status (Optional).

Manual Machine Operation

The two types of manual operation are available:

- Manual Operation - controlling the machine from the keypad letting the servos move the machine.
- DRO Mode (Digital Read Out) - Using the handles to move the machine with the CNC servos off.

Activating DRO Mode

In the DRO Mode the machine is moved with the handles and the CNC is used as a digital read out.

1. Press the (E-STOP) 

.
2. Gain access to the **SPINDLE MANUAL/AUTO** switch (the **SPINDLE MANUAL/AUTO** switch is usually located on the side of the servo cabinet).
3. Toggle the switch to the **MANUAL** position (servos and spindles disconnect).
4. Reset the (E-STOP) 

.
5. Press the (SERVO RESET) 

(spindle reactivates).

NOTE: The (E-STOP) 

will stop the spindle in the DRO Mode.

Deactivating DRO Mode

1. Press the (E-STOP) 

.
2. Gain access to the **SPINDLE MANUAL/AUTO** switch and move the switch to **AUTO**.
3. Reset the (E-STOP) 

.
4. Press the (SERVO RESET) 

(servos reactivate, CNC now operates machine).

NOTE: In this manual, all references to manual control assume the CNC is on, and the machine is being operated from the CNC keypads.

Manual Mode Settings

The operator can control every detail of the CNC's operation, not only what it does but how it does it. 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, for example, the move type (Rapid or Feed) the actual feedrate (IPM), the units (Inch or MM) etc.

There are always at least 3 to 4 modes active at any given time.

Before making a manual move the operator should make all of the mode adjustments required for the machine to move properly. Modes set from the **MANUAL** screen remain active if the CNC is put in a program operating mode or until the mode is changed by the program or operator.

The following modes can be set from the **MANUAL** screen.

- The position mode, whether the CNC uses Absolute or Incremental Positioning.
- The units mode, whether the CNC counts inches or millimeters.
- The move mode, whether the CNC makes a Rapid move or a move at a specific feedrate.
- The plane, whether the CNC is to execute an Arc move in some specific plane.
- The active tool, what tool diameter compensation and length offsets are applied.

The following settings can be set from the **MANUAL** screen.

- The location of the absolute zero reference (XY axis).
- The location of the tool change position (Z axis tool #0, Z0 position).

Refer to **Section 4 - Writing Programs** for information on changing modes and settings in a program,

There are four types of moves available when the CNC is in the Manual Mode.

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

Activating Manual Mode Rapid or Feed

Pressing the (JOG)  key cycles the CNC through all available jog modes. Two of the jog modes (Rapid and Feed) are CNC move modes. The active mode is displayed in the Secondary Display Area.

Setting a Feedrate

The basic feedrate is manually keyed in. The feedrate can be changed at any time manually or in a program. The CNC makes all Feed moves at the feedrate keyed in, if the (FEEDRATE OVERRIDE) is set to 100%.

The feedrate keyed can be adjusted using the (FEEDRATE OVERIDE). Each click of the (FEEDRATE OVERIDE) adjusts the feedrate by 10%. The overide knob has a range of 0% to 120%. Aligning the knob pointer with the white stripe sets the overide to 100%.

All moves are affected by the (FEEDRATE OVERIDE).

To key in a feedrate:

1. With the CNC in the Manual Mode, press the (4/FEED) hotkey (**FEED RATE** Graphic Menu prompts for value).
2. Key in desired feedrate and press **Save** (F10) or (ENTER) (CNC returns to **MANUAL** screen).
3. Press (START) to execute the feedrate change

- or -

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

CAUTION: If the CNC is shut down, when it first powers up again, the manufacturer's setup file will reload the CNC's default feedrate.

Adjusting Rapid Move Speed

The (FEEDRATE OVERIDE) knob also adjusts the Rapid move speed. Every click of the (FEEDRATE OVERIDE) adjusts the Rapid move speed by 10% of the default speed. The overide knob gives the operator a range of 0% to 100%. Aligning the knob pointer with the white stripe sets the overide to 100%.

Absolute/Incremental Modes

The CNC recognizes both absolute and incremental positions and may be switched between the two modes manually or from a program at anytime. In the absolute mode the CNC measures all points from the Absolute Zero Reference. In the Incremental Mode the CNC measures the next move from its present position. The active position mode is displayed in the Secondary Display Area.

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

NOTE: The CNC always displays absolute positions.

Activate the Absolute or Incremental Mode as follows:

1. Put the CNC in the **MANUAL** mode.
2. Press the (ABS/INC) [ABS/INC] key (mode changes, label changes in status box).

Inch/MM Modes

The CNC uses two units of measurement: inches and millimeters. If the Inch Mode is active, the CNC counts in inches. If the Millimeter Mode is active, the CNC counts in millimeters.

The units mode can be changed from the **MANUAL** screen, and by a program.

CAUTION: When the CNC first powers up, the control loads the default modes from the setup file.

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 the (7/UNIT) [7/UNIT] key to toggle the Units Mode selection (selected mode is displayed in the Secondary Status Area).

Setting the Absolute Zero Reference (X & Y Part Zero)

The Absolute Zero Reference is the point the CNC recognizes as X0, Y0 when the Absolute Positioning Mode is active. All absolute XY positions are measured from this point.

The CNC uses a floating Absolute Zero Reference. A floating Absolute Zero Reference can be located anywhere within or beyond the machines range of travel. This lets the operator apply dimensions referenced from a position outside the machines range of travel.

When the machine is powered up, it's present location is automatically set as the Absolute Zero Reference. When the machine is shut down, the position of the Absolute Zero Reference is lost. The operator can set or change the Absolute Zero Reference manually or from within a program.

Setting the Absolute Zero Reference to a location on the part, is also called setting Part Zero.

NOTE: The Z axis location of part zero is determined by setting tool length offsets for each tool.

NOTE: The location of the Absolute Zero Reference can be restored after a shut down if the machine has the Home Function installed.

CAUTION: If the part zero reference is not correctly located, the CNC will not position correctly in the Absolute Mode.

Method 1 - Locating Machine at the Absolute Zero Reference

1. Position the center of the spindle at the required absolute zero reference (part zero) position.
2. With the CNC in the **MANUAL** mode, press (X) [X] (CNC prompts for absolute zero reference X axis position, 0.00 displayed).
3. Press (Y) [Y] (CNC prompts for absolute zero reference Y axis position, 0.00 displayed).
4. Press (ENTER) [ENTER] (displayed values are set).

Method 2 - Presetting (Keying in an Absolute Zero Reference Position)

1. Adjust the machine's position to some known distance from the desired absolute zero reference (part zero).
2. Press (X) [X] (CNC prompts for absolute zero reference X axis position, 0.00 displayed).
3. Key in the desired X axis position (include sign) and press (Y) [Y] (CNC prompts for absolute zero reference Y axis position, 0.00 displayed).
4. Key in the desired Y axis position (include sign) and press (ENTER) [ENTER] (new values are set).

CAUTION: To enter positions, be sure to press (ENTER) [ENTER] and not (START) [START] pressing (START) [START] starts a move to the entered position.

NOTE: Absolute zero may be a point outside the machines actual range of travel.

Setting Tool Change Position (Tool #0, Z0)

Tool #0, Z0 is the Z axis zero position (Z0) with no tool length offset applied. Tool #0, Z0 is usually the furthest point of quill retraction. The Tool #0, Z0 position is often used for tool changes or as a safe place for making rapid XY moves.

The Z axis zero position (Z0) with any other tool active, may not be the same position as Tool #0, Z0.

When the machine is powered up, the machines present Z axis position is automatically set as the Tool #0, Z0 position. When the machine is shut down, the location of Tool #0, Z0 is lost.

Set the Tool #0, Z0 position before setting tool length offsets. Changing the location of the Tool #0, Z0 position will alter any existing tool length offset settings.

NOTE: Returning the Z axis to the Tool #0, Z0 position before powering down the machine, will preserve the Tool #0, Z0 setting (and tool length offsets) for use when the machine is powered back up.

Refer to **Section 1 - 1100M Programming Concepts** and **Section 8 - Tool Management** for more information on tool length offsets.

Method 1 - Locating Machine at Tool #0, Z0

1. With the CNC in the **MANUAL** mode, press the (5/TOOL) [5 TOOL] hotkey (CNC displays **TOOL MOUNT** Help Graphic).
2. Press **Save** (F10) (CNC prompts operator to "**Press START or execute MANUAL to Cancel.**").
3. Press (START) [START] (CNC activates tool #0, length offset is deactivated).
4. Adjust the machine to the desired tool change or tool #0, Z0 position.
5. Press (Z) [Z] (CNC prompts for absolute Z axis position, 0.00 is displayed).
6. Press (ENTER) [ENTER] (displayed value is set to Z axis).

Method 2 - Presetting (Keying in a Tool #0, Z0 Position)

1. With the CNC in the **MANUAL** mode, go to the tool #0, Z0 position.
2. Press (Z) [Z] (CNC prompts for the absolute Z axis position, 0.00 is displayed).
3. Key in the Z axis position (include sign) and press (ENTER) [ENTER] (new value is set).

Activating a Tool

Activating a tool enables the CNC to use the length offset and diameter value from the Tool Page. Refer to **Section 4 - Writing Programs** to activate a tool in a program.

To activate a tool manually:

1. With the CNC in the **MANUAL** mode, press the (5/TOOL) 5 TOOL hot key (CNC prompts for tool number).
2. Key in tool number, and press **Save** (F10) (CNC returns to Manual mode).
3. Press (START) (tool number displayed in the Primary Display Area is active).

   - or -

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

NOTE: If the Z axis is moved to the Tool #0, Z0 position while another tool is active, the Z axis position displayed on the CNC is the tool's length offset with the sign reversed.

Activating a Plane

By default the CNC's XY plane is active.

To change the active plane:

1. With the CNC in the Manual mode, press the (9/PLANE) 9 PLANE hot key (CNC prompts for a plane selection).
2. Press the (+/-) +/- to toggle to the desired plane selection.
3. Press **Save** (F10) (CNC returns to Manual mode).
4. Press (START) (selected plane is activated).

   - or -

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

Jog Moves

Jog moves are enabled at the following times:

- The CNC is in the Manual mode, the **Teach** mode, or the Tool Page is active, and
- The servos are on.

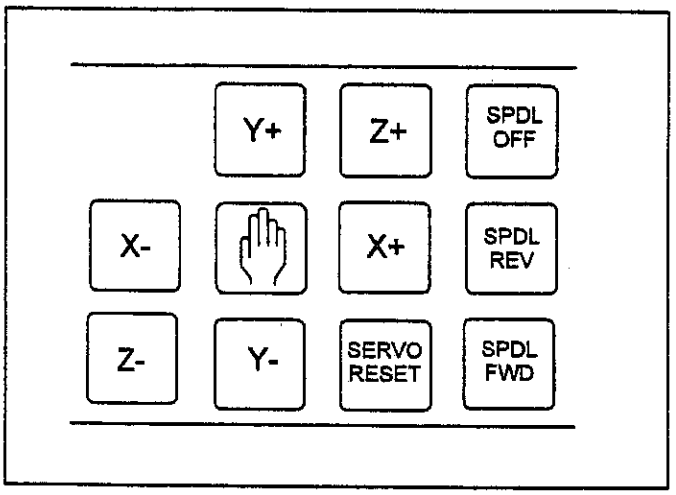


Figure 3 - 2, Manual Operation Keys

Refer to **Figure 3 - 2, Manual Operation Keys**. The Manual Operation Keys are used 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 **Table 3 - 1, Move Mode Selections**. There are five move modes available. The actual rate for each mode is determined when the machine is set up. Pressing the (JOG) key cycles the control through the available mode choices.

Table 3 - 1, Move Mode Selections

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

The operator can change the jog mode anytime the control is in the Manual mode.

Changing the Jog Mode

To change the jog mode:

1. With the CNC in the Manual mode, press (JOG) (mode selection toggles, selected mode is displayed in Secondary Display Area).

Repeatedly pressing the (JOG) key cycles the CNC through all of the jog mode selections.

NOTE: The operator can change the Jog Mode when the CNC is in Manual Mode, Teach Mode or from the Tool Page.

Jogging the Machine (Conventional)

1. With the CNC in the Manual Mode, the Teach Mode, or the Tool Page, use the (JOG) key to toggle through the move mode choices until **JOG: 1**, **JOG: 10**, or **JOG: 100** is visible in the Secondary Display Area.
2. Press (X+) X+ or (X-) X- or (Y+) Y+ or (Y-) Y- or (Z+) Z+ or (Z-) Z- for desired axis and direction (machine jogs along selected axis).

Jogging the Machine (Continuous)

1. With the CNC in the Manual Mode, the Teach Mode, or from the Tool Page, use the (JOG) key to toggle through the move mode choices until **JOG: Rapid** or **JOG: Feed** is displayed in the Secondary Display Area.
2. Press (X+) X+ or (X-) X- or (Y+) Y+ or (Y-) Y- or (Z+) Z+ or (Z-) Z- for desired axis and direction (movement occurs as long as key is pressed).

One Shot Moves

In the Manual Mode, the operator can position the machine using one shot moves. A one shot move can be either an Absolute or Incremental move at a **Rapid** or **Feed** rate. Hot Key functions are also available for one shot moves.

Perform a one shot move as follows:

1. Place the CNC in Manual Mode.
2. Using the (JOG) key, select a move mode.

NOTE: Any **Jog**: selection other than **Rapid** will cause the move to be done in **Feed**.

3. Using the (ABS/INC) ABS/INC key, select either the Absolute (**ABS**) or Incremental (**INC**) positioning mode.

4. Press (X) [X] and key in position.
5. Press (Y) [Y] and key in position.
6. Press (Z) [Z] and key in position.
7. Press (START) [▷].

NOTE: One shot moves can be paused by pressing (HOLD) [□] and continued by pressing (START) [▷]. The move can be canceled altogether by pressing **MANUAL** (F4).

CAUTION: To start a one shot move, be sure to press (START) [▷] and not (ENTER) [ENTER]. Pressing (ENTER) [ENTER] will move the Absolute Zero Reference to the position keyed in.

NOTE: One shot moves can also be done using the (1/RAPID) [1 RAPID] and (2/LINE) [2 LINE] hotkeys. Be sure to activate the required positioning mode (Absolute or Incremental) prior to selecting the move.

Manual Data Input (MDI)

MDI provides a quick way for the operator to program a single move, or a series of moves that may only be used only once. Although **MDI** requires a knowledge of programming, it is introduced in this section because its technically a manual operation. Refer to **Section 4 - Writing Programs** for instructions on writing **MDI** moves.

NOTE: The Manual Data Input Mode gives the operator access to all programming options and could be used to write a complete part program.

MDI blocks are stored in a permanent program named "**MDI.M**". **MDI** blocks are written the same way program blocks are written. The **MDI** editor has all the same features as the regular program editor. **Draw** can be used to view an **MDI** move before running.

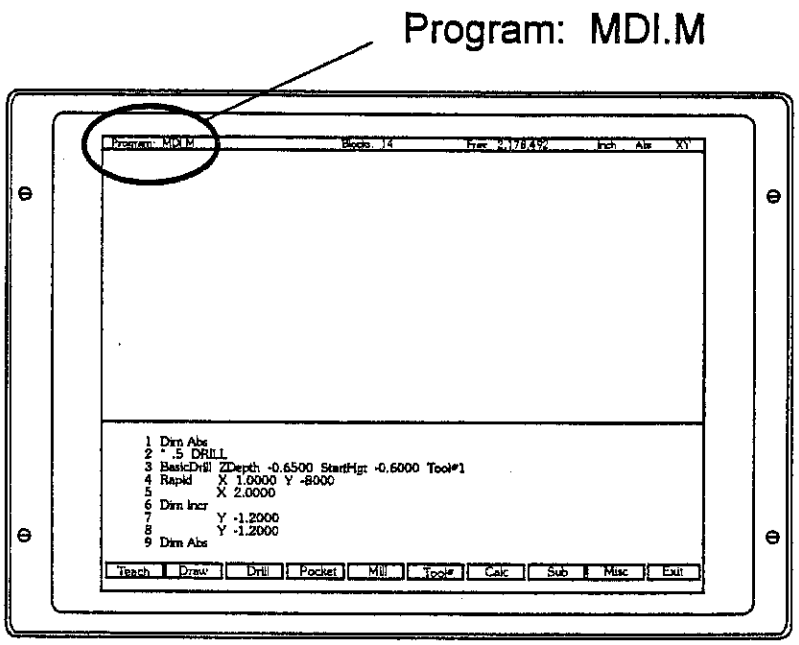


Figure 3 - 3, Manual Data Input (MDI) Screen

Refer to **Figure 3 - 3, Manual Data Input (MDI) Screen**. The moves in MDI remain in the MDI.M program until cleared or edited. An MDI can be run as often as necessary.

If the MDI.M program is accidentally deleted, the CNC will automatically create a new one the next time the MDI Mode is activated.

The **MDI** screen is very similar to the **Edit** screen. When the CNC is in the MDI Mode, the program name "**MDI.M**" appears in the upper left hand corner of the screen (next to the **Program** label).

The moves in an MDI can be saved as a program by renaming the MDI.M program. Refer to **Section 7 - Program Management**, for instructions on renaming programs.

To run an MDI:

1. From the **MANUAL** screen press **MDI** (F7) (CNC displays the Program Editor).

NOTE: Observe the default name "**MDI.M**" in the right hand corner Of the status line.

2. Key in required program blocks.
3. Run Draw to view the move if necessary.
4. Press **Exit** (F10) (CNC returns to **MANUAL** screen).
5. Press (START)  (**MDI** executes).

   or

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

Section 4 - Writing Programs

Program Basics

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

A program can control every detail of the CNC's operation, not only what it does but how it does it.

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, for example: the move type (Rapid or Feed); the actual feedrate (IPM); the units (Inch or MM); etc.

There are always at least 3 to 4 modes active at any given time.

The operator writes programs using combinations of moves, mode changes and canned cycles. The CNC has a built in library of canned cycles stored in its permanent memory.

Developing Part Programs

The most important part of writing a CNC program is the planning. The following guidelines are provided to help a first time programmer organize the process.

First, the operator needs to decide how to clamp the part and where to set Part Zero (the Absolute Zero Reference). Part Zero is best located at a point on the work positively positioned by the clamping fixture. This allows consistent machining of subsequent parts. Since absolute positions are defined from Part Zero, its location should be selected to permit direct application of dimensions off a print.

The operator also needs to decide what tools are required and to set the length offset for each tool. It may be necessary to adjust the position of the knee so that the longest tool will clear the top of the part in a Rapid, Tool #0 move to Z0. Treat tool information like part of the program.

Sitting down with the blueprint, choosing a Part Zero and jotting down the moves, positions and tools needed to cut the part, will speed up the programming process. The CNC has a calculator specifically designed for finding coordinates on complex geometric shapes, refer to **Section 10 - The Calculator**.

To develop a part program:

1. Enter the Program Directory (the **PROGRAM** screen) and create the program for the part (refer to **Section 7 - Program Management**).
2. Enter the Program Editor (the **Edit** screen) to open the new program and start writing blocks (refer to **Section 4 - Writing Programs**).

3. The first block in a program is usually an Absolute Mode block. Putting the CNC in the Absolute Mode at the start of a program enables absolute positioning right away. The Incremental Mode is usually only used when specifically needed.
4. The first move of the program is usually a Rapid move to Tool #0, Z0 to fully retract the quill for the next move.
6. The second move is a Rapid move to a convenient part change position (home).

NOTE: The same two moves, a Rapid to Tool #0, Z0 and a Rapid to the part change position, are used at the start and end of the program. This way, the program starts and ends at the part change position.

7. Moves towards a part should be done in two steps, a Rapid X, Y move at a clear height, followed by a Z move to 0.1 inch (2mm). above the surface of the cut (standard starting height). The first tool mount and cutter compensation can activated in this move (if needed).

NOTE: Programming tool changes in separate blocks makes them easier to find.

8. The remaining blocks in the program are the moves, cycles and tool changes required to machine the part.
9. The last two blocks of the program should be a Rapid move to Tool #0, Z0, followed by a Rapid X, Y move to the same part change position used at the start of the program.
10. Once written, the program can be checked by putting the CNC in the Draw Graphics Mode and running a Draw Simulation. Refer to **Section 5 - Viewing Programs With Draw**. Running a Draw simulation will let the operator view the program's moves for errors.
11. At this point, the operator can clamp the work on the table.
12 The operator now goes to the **MANUAL** screen and sets a Part Zero (the position of the Absolute Zero Reference) to a convenient point on the part, refer to **Section 3 - Manual Operation and Machine Setup**.
13. The operator then goes to the tool page and organizes the tooling. Each tool is assigned a number (probably in the order of use) and length offsets and tool diameters are entered, refer to **Section 8 - Tool Management**.
14. Before attempting to cut a part, the operator should do some type of dry run. There are a several ways to get a close look at the programmed moves. The program can be run from motion to motion, holding between each move. The program can be run from block to block, holding between each block. The program can be run with no tools installed, or with a tool installed, but the knee lowered so the tool never contacts the work.

15. After the program is tested, it is ready for production. Developing and testing a program represents an investment in time. Programs can be copied to a floppy disk for safe keeping, refer to **Section 7 - Program Management**.

Using the Edit Screen

Program blocks are written using the CNC's Graphic Menu Program Editor (the **EDIT** screen). Activating the Program Editor puts the CNC in the Edit Mode. Programs must be created before they can be edited or written. Refer to **Section 7 - Program Management** for instructions on creating new programs.

The Program Editor can be activated from the Program Directory or the **MANUAL** screen. When the Program Editor is activated from the Program Directory, the selected program is opened for editing. When the Program Editor is activated from the **MANUAL** screen, only a program loaded for execution will be opened for editing.

All programming procedures in this section of the manual are written assuming that the CNC is already in the Edit Mode.

Activating the Program Editor

Method #1, From the Program Directory

1. From the Program Directory, use the (ARROW)  keys to move the highlight to select the program for editing and press **Edit** (F4), (program editor opens selected program for editing).

Method #2, From the Manual Screen

1. From the **MANUAL** screen, press **EDIT** (F4). If a program is already loaded for execution, the editor will open it for editing. If no program is loaded, the CNC will display a "**No Program Selected !**" message. Press **PROGRAM** (F2) to activate the Program Directory and use Method #1.

The Program Editor Screen

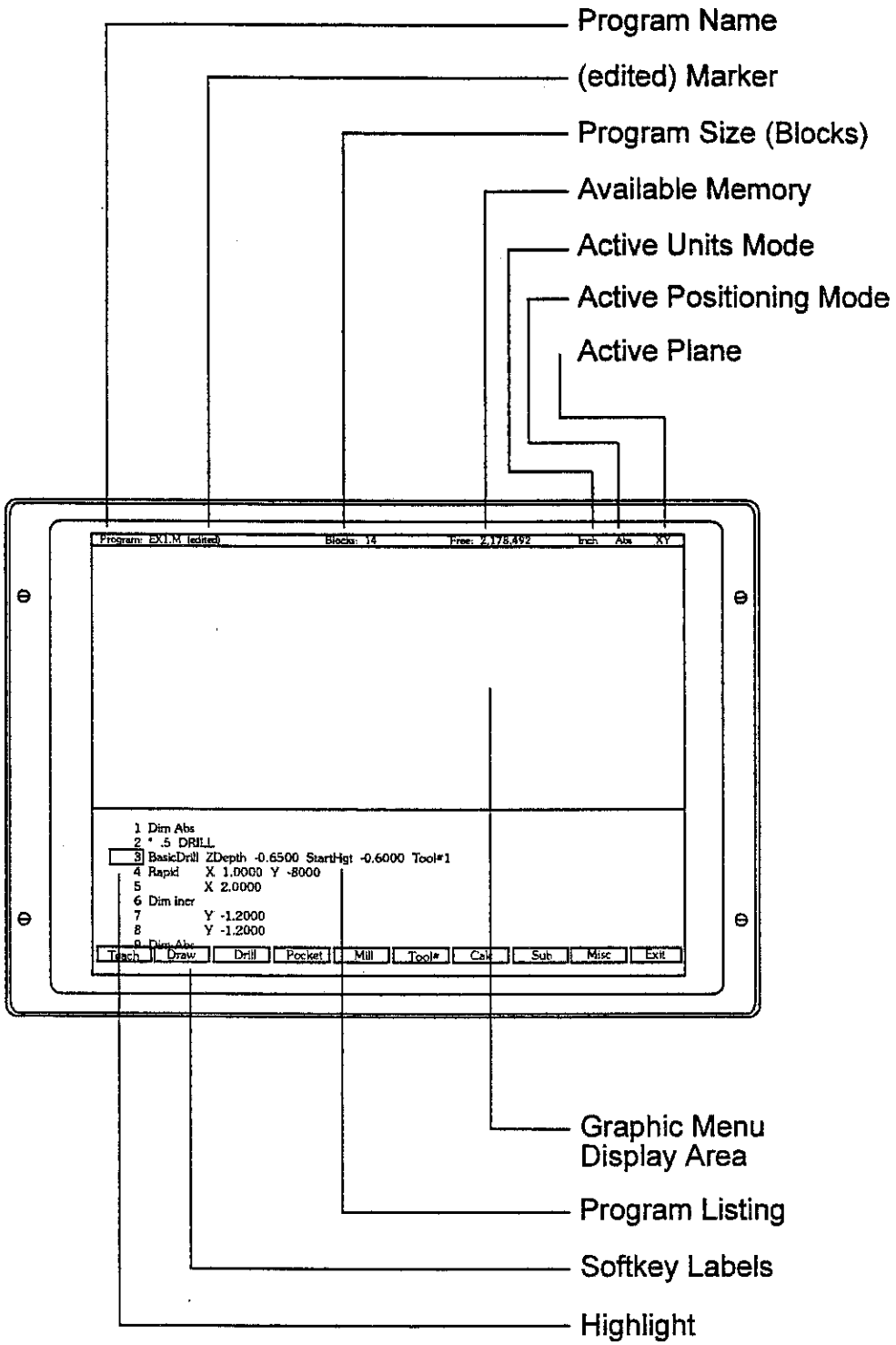


Figure 4 - 1. Program Editor

Refer to **Figure 4 - 1. Program Editor**. The Program Editor monitors mode changes as they are written into a program. The mode indicators displayed in the Program Editor indicate the active modes of the CNC when it has executed the program to the highlighted block.

Graphic Menus for writing and editing program blocks are customized for the CNC's active modes when required (this is useful when programming Arcs in different planes).

Program Editor's features are described as follows:

Program Name	Name of program opened for editing.
(edited) Marker	The **(edited)** marker indicates the program has been edited and the edits haven't been saved.
Program Size	Number of blocks in the program.
Available Memory	Amount of room remaining in the Editor's memory.
Active Plane	Current operating plane (**XY**, **XZ**, **YZ**).
Active Positioning Mode	Current measurement mode (**Abs**, **Inc**).
Active Units Mode	Current units mode (**Inch**, **MM**).
Graphic Menu Area	Area for displaying Graphic Menus.
Program Listing	Current listing of the blocks in the program, if program has been edited, listing reflects edits.
Highlight	Lets operator select a block for editing and acts as an insertion marker for adding new blocks. The CNC tracks program mode changes up to this point in the program listing.
Softkey Labels	These labels define the softkey functions. Three sets of softkeys are available. The default set (normally visible), the Misc softkeys (activated by pressing **Misc** (F9)) and the Sub softkeys (activated by pressing **Sub** (F8)).

Saving Edits

Although the program listing displays edits as soon as they are made, the edits are not saved until the editor is properly exited. If the **(edited)** marker is visible at the top of the Program Editor, the open program contains unsaved edits.

To save edits:

1. With the CNC in the Edit Mode, press **Exit** (F10) (CNC returns to the Program Directory, all edits are saved).

Canceling Unsaved Edits

If the edits have not been saved they can be canceled.

To cancel unsaved edits:

1. With the CNC in the Edit Mode, press **Misc** (F9) (Misc softkey labels are activated).
2. Press **Quit** (F8) (a **"WARNING: Program has been edited. Sure you want to QUIT?"** message appears, softkey labels change).
3. Press **Yes** (F1) (CNC returns to the Program Directory, all edits are lost).

- or -

Press **No** (F2) (no action is taken).

Deleting a Block

To delete a program block:

1. With the CNC in the Edit Mode, use the (ARROW)  keys to move the highlight to select the block being deleted.
2. Press (CLEAR)  (selected block is cleared from the listing, block numbering is adjusted).

Inserting a Block

1. With the CNC in the Edit Mode, use the (ARROW)  keys to move the highlight to select the block that will follow the inserted block.
2. Program the new block. When the new block is saved, it is inserted in front of the highlighted block. All of the blocks are automatically renumbered.

Editing (Re-editing) Blocks

1. With the CNC in the Edit Mode, use the (ARROW)  keys to move the highlight to select the block being edited.
2. Press (ENTER)  , if the existing block is a move or cycle, the original Graphic Menu is re-opened for editing. Use the (ARROW)  keys to move the highlight to the entry fields requiring changes. Press (CLEAR)  to erase existing values and key in new ones. When all changes have been made, press **Save** (F10) to close the block.

When the program block's Graphic Menu only offers a simple selection between two modes, highlighting the block and pressing (ENTER) toggles the block to the other selection.

Searching Blocks for Words or Numbers

Use Search to have the CNC search the program for a particular block number, or word. Search only searches forward in the program.

1. With the CNC in the Edit Mode, Press **Misc** (F9) (softkey secondary functions appear).
2. Press **Search** (F3) (CNC prompts for block number or word).
3. Key in block number, or word using the ASCII Chart.
4. Press (ENTER) (CNC searches and highlights the next block found containing word or block number).

NOTE: Refer to **Section - 2 CNC Console and Software Basics** for ASCII Chart instructions.

Scrolling the Program Listing

With the CNC in the Edit Mode, use the up and down (ARROW) keys to scroll the program listing up and down for viewing.

Paging Through the Program Listing

With long programs it is faster to advance the program listing up and down a page at time.

1. With the CNC in the Edit Mode, Press **Misc** (F9) (softkey secondary functions appear).
2. Page forward or backward by pressing the **PgUp** (F4) or **PgDn** (F5) keys.
3. Press **Prev** (F9) (Program Editor default softkeys return).

Jumping to First or Last Block in the Program

To jump to the first or last blocks of the program listing:

1. With the CNC in the Edit Mode, press **Misc.** (F9) (softkey secondary functions appear).
2. Press **Begin** (F6) (program listing displays the first block of the program).

- or -

Press **End** (F7) (program listing displays last block of program).

Using Comments

Comments containing setup and tooling notes can be added to a program. Comments are displayed on the screen as the program runs, but are not executed by the control.

Program blocks can be converted to comments. It is sometimes helpful to comment out specific program blocks when troubleshooting a program. When a program block is commented out, the CNC will ignore it.

Comment blocks are displayed in the program listing with an asterisk, * following the block number.

Writing a Comment Block

1. With the CNC in the Edit Mode, press **Misc.** (F9) (softkey secondary functions appear).
2. Press **Comment** (F2) (CNC prompts for a comment, **ASCII** softkey label appears).
3. Use the ASCII Chart to key in comments.

NOTE: The asterisk after the block number tells the CNC to ignore the comment.

NOTE: Refer to **Section - 2 CNC Console and Software Basics** for ASCII Chart instructions.

Commenting Out Existing Blocks

1. With the CNC in the Edit Mode, position the highlight to select the block being commented out.
2. Press the (ZERO) [0] key (asterisk * is placed after block number).

NOTE: Off line keyboard users must use the (ZERO) key, to toggle the asterisk on or off. A keyboard asterisk key will not work.

Uncommenting a Program Block

1. With the CNC in the Edit Mode, position the highlight to select the block being uncommented (having the comment asterisk removed).
2. Press the (ZERO) [0] key (asterisk, * that followed the block number is removed).

NOTE: Off line keyboard users must use the (ZERO) key, to toggle the asterisk on or off. The keyboard asterisk key will not work.

Writing Program Blocks

Selecting a move type, a mode, or a cycle from any of the CNC's hotkeys, softkeys, or a pop-up menus activates a Graphic Menu for writing the block. Once the Graphic Menu is completed and saved, the new block is added to the program listing.

When starting a new program, only the **"<End Of Program>"** is visible in the program listing. This is always the last line of the program. As new blocks are added, they are automatically numbered and inserted in front of the **"<End Of Program>"** line.

Using Graphic Menus

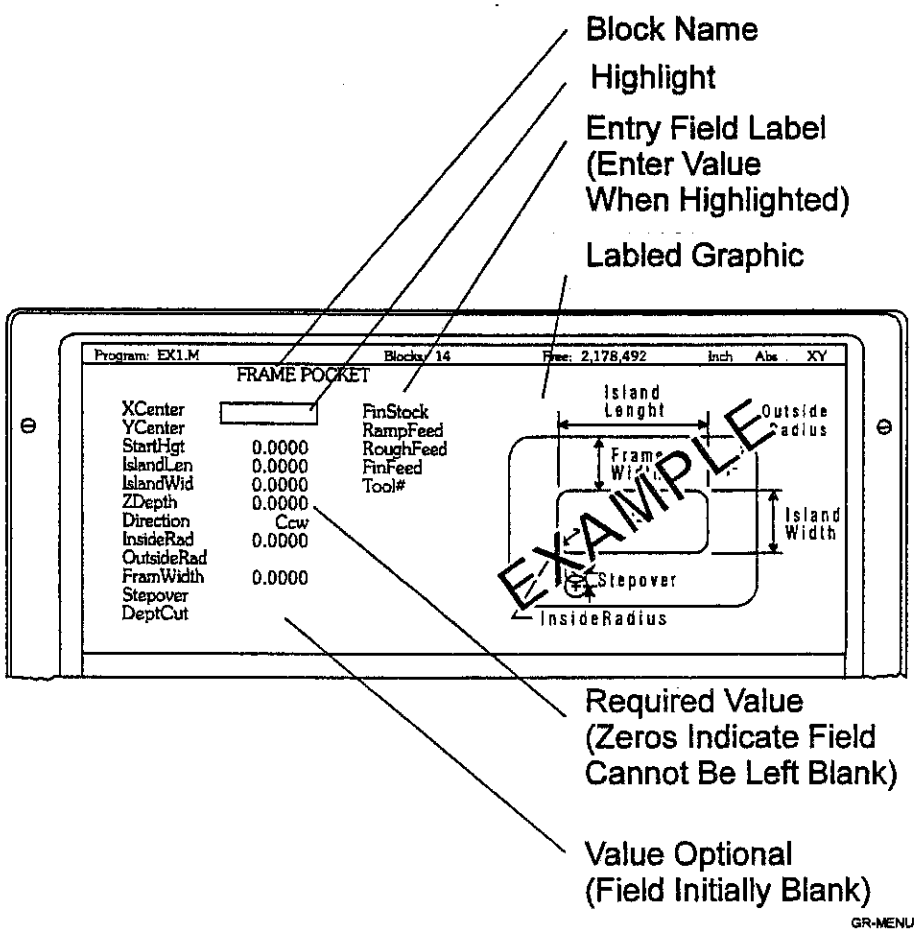


Figure 4 - 2, Sample Graphic Menu

Refer to **Figure 4 - 2, Sample Graphic Menu**. The Program Editor displays full screen Graphic Menus for writing and editing program blocks.

When a Graphic Menu is first displayed, the first entry field is highlighted. Once a field is highlighted, it will accept a value or selection made by the operator. Pressing (ENTER) [ENTER] advances the highlight to the next entry

field. Pressing (ENTER) [ENTER] from the last entry field closes the block and adds it to the program. Pressing **Save** (F10) from any entry field, also closes the block and adds it to the program. Move the highlight from field to field using the (ARROW)  keys. Entry fields can be filled in any order. Pressing the (CLEAR) [CLEAR] key clears the value in the highlighted field.

There are two types of entry fields in a Graphic Menu.

- Optional entry fields - these fields are completely blank when the Graphic Menu is first displayed.
- Required entry fields - these fields contain "**0.000**" when the Graphic Menu is first displayed.

Required entry fields cannot be left blank. The value can remain "**0.000**" (zero) if appropriate. If a required entry field is left blank, the CNC still writes the block, but when the block is executed, it holds the machine and generates an error message.

Optional entry fields don't require a value. When left blank, a default value or position is usually assumed. 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.

The operator must remember to key in decimal points and negative signs where needed. The CNC assumes a positive value if no negative sign is keyed in.

Use the (+/-) [+/-] key, to insert a negative sign. The (+/-) [+/-] key is also used to toggle the selection in entry fields that only offer specific choices (for example **Cw** or **Ccw**).

No Move Blocks

Blocks that do not initiate machine moves are called no move blocks. No move blocks set modes, activate tools, set feedrates etc.

Programming an Absolute / Incremental Mode Change

A **Dim** block puts the CNC in either the Absolute or Incremental Mode. The field entries for a **Dim** block are **Abs** or **Incr** respectively.

To Program a Dim block:

1. With the CNC in the Edit Mode, press (ABS/INC) [ABS/INC] (**SET INCR/ABS DIMENSION** Graphic Menu prompts operator to select **Abs** or **Incr**).

2. Toggle the choice using the (+/-) [+/-] key.

3. Press **Save** (F10) or (ENTER) (**Dim** block is added to the program listing).

Programming an Inch/MM Mode Change

A **Unit** block puts the CNC in either the Inches or Millimeter Mode. The field entries for a **Unit** block are **Inch** and **MM** respectively.

1. With the CNC in the Edit Mode, press (7/UNIT) (**SET INCH/MM UNIT** Graphic Menu prompts operator to select **Inch** or **MM**).
2. Toggle the choice using (+/-) key.
3. Press **Save** (F10) or (ENTER) (**Unit** block is added to the program listing).

Programming a Tool Change (Activating a Tool)

Tools are identified by tool number. Activating a tool activates the length offset and diameter compensation values listed on the like numbered row of the Tool Page. Refer to **Section 8 - Tool Management** for information on the Tool Page. Listed coolant and spindle operations are also activated (on machines that control these features).

Active tool values remain in effect until a different tool is activated. Tool values only need to be activated once per tool change.

Never activate a tool while tool compensation is active. Turn tool compensation off and ramp off first.

NOTE: Tool numbers can be programmed into most moves. Do not program tool information into more blocks than necessary. Each time a tool is activated the CNC will hold the program to permit installation of the new tool. Programming unnecessary tool changes will slow down the operation of the program.

Activating Tool #0 sets tool length offset and diameter compensation to 0.0.

The steps in a typical tool change are as follows:

1. Program a Rapid Z0.000, Tool #0 to cancel the length offsets and retract the quill to a safe position.
2. Program a Rapid move to the tool change X, Y position.
3. Program a block to activate the required tool. When executed, the spindle and coolant will stop so the operator can change the tool (on machines that control these features). When the tool change is complete, the operator presses (START). When the program is restarted, the new diameter, length offset, spindle RPM and direction (if applicable) and coolant (if applicable) are activated.

NOTE: A move to Tool #0 Z0 will only retract the Z axis if the CNC is in the Absolute Mode. An incremental 0 inch (mm) move will not make a position change.

NOTE: Programming tool changes in separate blocks makes them easier to find in the program.

To program a block to activate a tool:

1. With the CNC in the Edit Mode, press (5/TOOL) [5 TOOL] (**TOOL MOUNT** Graphic Menu prompts for **Tool #**).
2. Key in tool number and press (ENTER) [ENTER] (**Tool#** block is added to the program listing).

Activating Tool Diameter Compensation

Most move types can be compensated. Compensation is modal, it must be turned on before it is used and turned off when it is no longer needed.

Compensation can only be turned on/off in Rapid or Line move blocks. The move that turns compensation on/off should also be the ramp move required to adjust the tool's center. Refer to **Section 1 - 1100M Programming Concepts** for information on how tool compensation runs.

Tool compensation is activated by toggling the **ToolComp** selection in a Rapid or Line Graphic Menu. There are 3 **ToolComp** choices:

- **Left** (of the path)
- **Right** (of the path)
- **Off** (turn compensation off)

If the field is left blank (blank by default) the current compensation status is unaffected.

Once tool compensation is activated, all of the following moves are compensated until it is turned off in a Rapid or Line move. The correct tool diameter (Tool #) must be active when compensation is turned on (activating both in the same block is ok). Do not activate a new tool diameter while compensation is active. Turn off the compensation and ramp off first.

Tool compensation is built into many canned cycles. Cycles with built in compensation activate and deactivate compensation automatically. The correct tool diameter must be active for these cycles to be accurate. The required tool can be activated within the cycle. Refer to **Table 4 - 1, Move and Cycle Compensation Requirements** for a list of move and cycle compensation requirements.

Table 4 - 1, Move and Cycle Compensation Requirements

Move or Cycle	Program a Rapid or Line move to activate tool comp before. Correct tool diameter must be active.	Compensation is activated and deactivated automatically. Correct tool diameter must be active.
Rapid	X	---
Line	X	---
Modal	X	---
Arc	X	---
Ellipse (has special requirements refer to Programming an Ellipse)	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
Mold Rotation Cycle (has special requirements refer to Programming a Mold Rotation)	---	X
Elbow Mold Cycle (no compensation available)	---	---

Programming a Dwell

When the CNC executes a **Dwell** block, it pauses for the programmed length of time. To program a Dwell key in the time in seconds. Dwell resolution is one tenth (0.1) of a second. If the operator keys in 0.0 seconds, the CNC will Dwell until restarted. The control is restarted by pressing the (START)  key.

To program a Dwell:

Method 1, Using Hot Keys

1. With the CNC in the Edit Mode, press (8/DWELL) (**DWELL** Graphic Menu prompts for length of time in seconds).
2. Key in seconds and press (ENTER) (**Dwell** block is added to program listing).

Method 2, Using Soft Keys

1. With the CNC in the Edit Mode, press **Sub** (F8) (secondary softkey functions appear).
2. Press **Dwell** (F7) (**DWELL** Graphic Menu prompts for length of time in seconds).
3. Key in the seconds and press (ENTER) (**Dwell** block is added to program listing).

Programming a Return to Machine Home

A **Home** block returns the CNC to a permanent reference position and sets the Absolute Zero Reference at that point. Home could be set up to work any number of ways depending on what options the builder elected to install. Check with the machine builder for detailed information.

Changing/Setting the Absolute Zero Reference in a Program

NOTE: An operator should be familiar with basic programming principles before attempting to write programs that contain blocks to change the Absolute Zero Reference.

The Absolute Zero Reference is the X0, Y0 position that absolute dimensions are measured from. Refer to **Section 1 - 1100M Programming Concepts** for information on Absolute Positioning. A **SetZero** block sets the Absolute Zero Reference of one or more axes to a new position.

When the block is executed, the X and Y values in the Graphic Menu are used to define the present absolute position of the machine.

If non-zero X or Y values are entered, the machine's present position is set to those coordinates. This is sometimes called pre-setting. If X0 and Y0 values are entered, the machine's present position becomes the new Absolute Zero Reference. This is sometimes called re-setting.

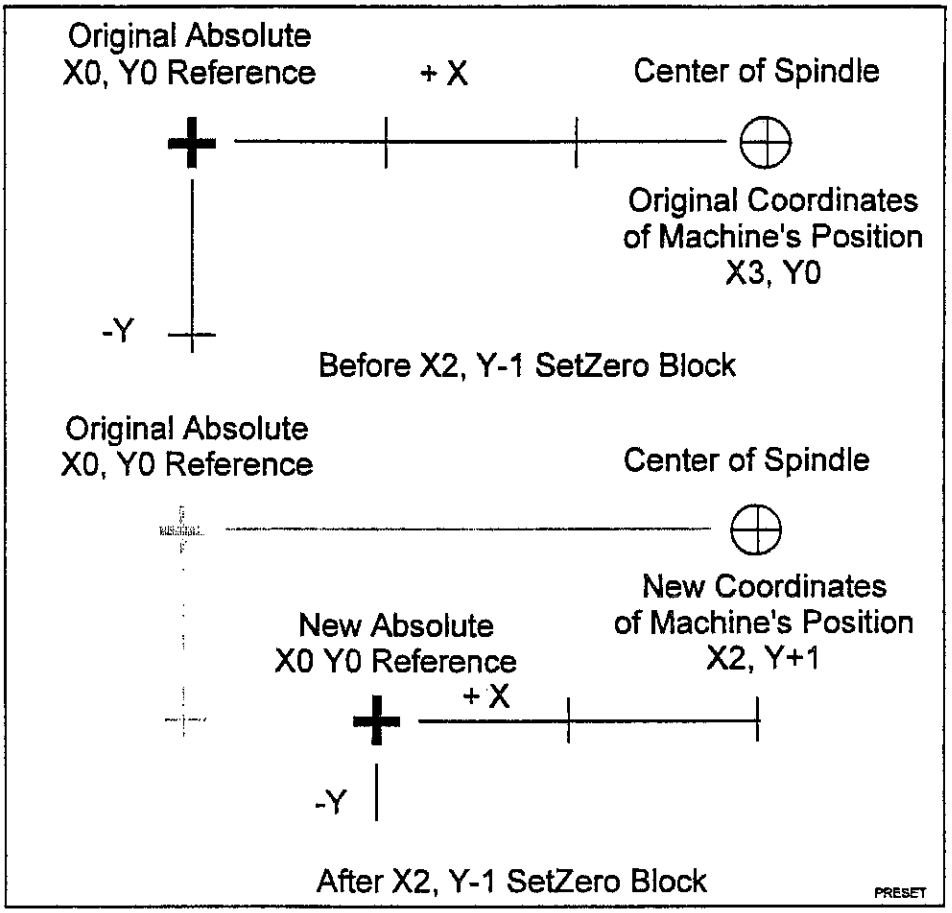


Figure 4 - 3, Executing a SetZero Block

Refer to **Figure 4 - 3, Executing a SetZero Block**. If X2, Y-1 values are entered, the machine's present position is set to X2, Y-1. It is important to recognize that changing the coordinates of the machine's present location also changes the position of the Absolute Zero Reference.

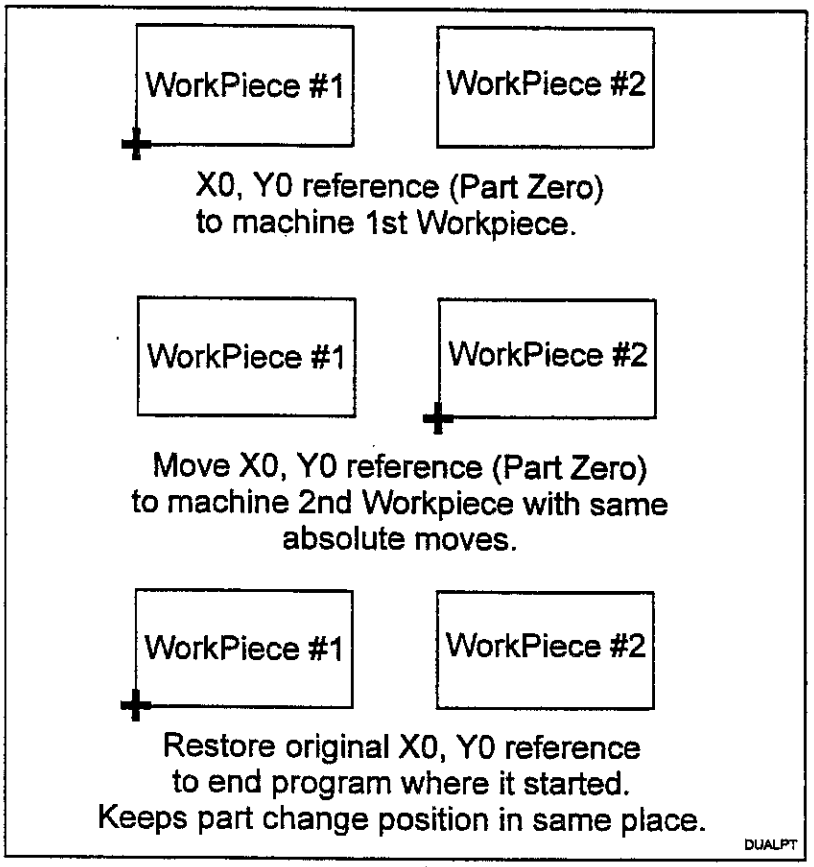


Figure 4 - 4, Using SetZero in a Program

Refer to **Figure 4 - 4, Using SetZero in a Program**. Changing the location of the Absolute Zero Reference lets a program cut more than one part using the same moves. It is important to restore the location of the original X0, Y0 reference at the end of the program. This way programmed part change positions do not move each time the program is run.

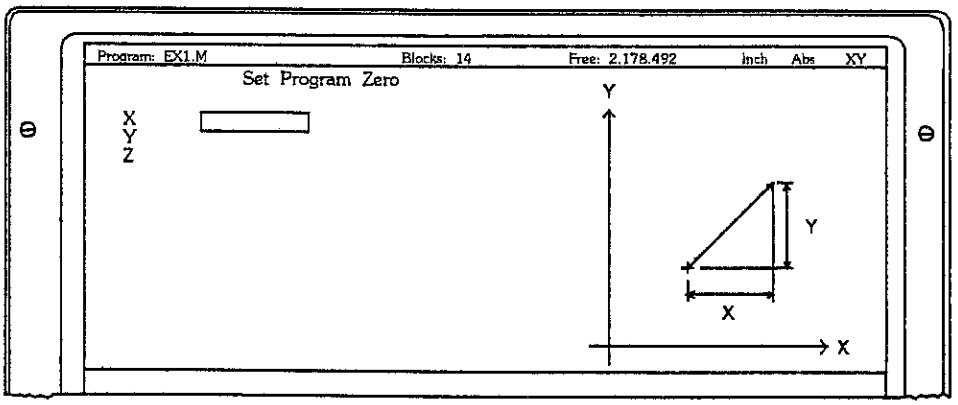


Figure 4 - 5, Set Zero Graphic Menu

Refer to **Figure 4 - 5, Set Zero Graphic Menu**. When the **X**, **Y**, or **Z** values are left blank no change is made to the position on that axis.

In most programs the position of the Z axis should not be changed. Changing the position of the Z axis is the same as changing Tool #0, Z0. Changing the location of the Z0 position alters all existing tool length offset settings.

To program a **SetZero** block:

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill softkey functions appear).
2. Press **More** (F7) (pop-up menu appears).
3. Position the highlight to select **SetZero** and press (ENTER) (**Set Program Zero** Graphic Menu prompts for the absolute coordinates of the machine's present position).
4. Key in **X** position (optional) and press (ENTER) (highlight moves to the **Y** entry field).
5. Key in **Y** position (optional) and press the (ENTER) (highlight moves to **Z** entry field).

NOTE: Changing the location of the Z0 position will alter any existing tool length offset settings.

6. Key in **Z** position (optional) and press the (ENTER) (**SetZero** block is added to the program listing).

Programming a Plane Change

The CNC executes Arc moves and compensates for tool diameters in three different planes, the XY plane, the XZ plane and the YZ plane. By default, the CNC operates in the XY plane. The CNC's active plane must be changed to execute moves in the XZ or YZ planes. A **Plane** block is used to change the CNC's active plane. When moves in the XZ or YZ plane are complete, a second **Plane** block must be written to restore the original XY plane. Refer to **Section 1 - 1100M Programming Concepts** for basic information on planes.

Programming a **Plane** block also changes the active plane of the Program Editor. The Program Editor customizes Arc move Graphic Menus when different planes are active.

The Program Editor only responds to Plane Mode changes made by adding **Plane** blocks. Deleting a **Plane** block from the program listing does not change the active plane of the editor. Deleting (or inserting) a **Plane** block does not change the way existing blocks are programmed.

To program a **Plane** block:

Method #1, Using the Hot Keys

1. With the CNC in the Edit Mode press the (9/PLANE) 9 PLANE hot key (**SET PLANE** Graphic Menu prompts for plane selection).
2. Press the (+/-) +/- to toggle the selection to the desired plane and press (ENTER) ENTER (Plane block is added to the program listing).

Method #2, Using the Softkeys

1. With the CNC in the Edit Mode press **Mill** (F5) (Mill softkeys are displayed).
2. Press **More** (F7) (More pop-up menu appears).
3. Using the (ARROW) keys, position the highlight to select **Plane** and press (ENTER) ENTER. (**SET PLANE** Graphic Menu prompts for plane selection).
4. Press the (+/-) +/- to toggle the selection to the desired plane and press (ENTER) ENTER (Plane block is added to the program listing).

Programming a Feedrate Change

Programming a **Feed** block sets the feedrate for Line moves, Arcs and cycles that do not contain specifically programmed feedrates. **Feed** blocks also set the feedrate for Modal moves. **Feed** blocks can be added to the program as often as necessary.

NOTE: A **Feed** block does not activate the feed mode.

To program a **Feed** block:

Method #1 from the Hotkeys

1. With the CNC in the Edit Mode, press the (4/FEED) 4 FEED key (**FEEDRATE** Graphic Menu prompts for new feedrate).
2. Key in required feedrate and press (ENTER) ENTER (**Feed** block is added to the program listing).

Method #2 from the Softkeys

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill softkey labels appear).
2. Press **More** (F7) (More pop-up menu appears).

3. Using the (ARROW) keys, position the highlight to select **Feed** and press (ENTER) (**FEED RATE** Graphic Menu prompts for feedrate).
4. Press **Save** (F10) or (ENTER) (**Feed** block is added to the program listing).

Straight Moves

Programming a Rapid Move

Rapid moves are moves executed at the CNC's Rapid rate. Rapid moves are used to save time when positioning for a cut or a canned cycle.

Rapid moves can also be used to activate and deactivate tool diameter compensation (cutter compensation).

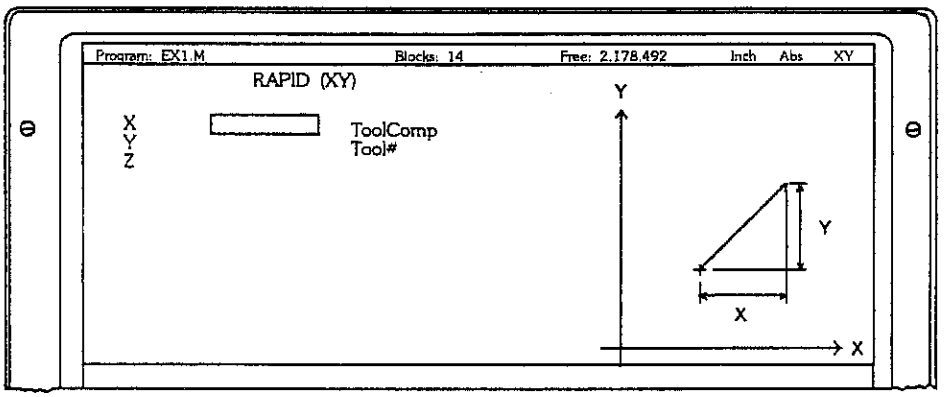


Figure 4 - 6, Rapid Move Graphic Menu

Refer to **Figure 4 - 6, Rapid Move Graphic Menu**.

To program a Rapid move:

Method 1, Using Hot Keys

1. With the CNC in the Edit Mode, press the (1/RAPID) key (**RAPID (XY)** Graphic Menu prompts for the **X** position).
2. Key in the **X** position (optional) and press (ENTER) (highlight moves to the **Y** field).
3. Key in the **Y** position (optional) and press (ENTER) (highlight moves to the **Z** field).

4. Key in the **Z** position (optional) and press (ENTER) (highlight moves to **ToolComp** field).
5. Using (+/-) set **ToolComp** (optional) and press (ENTER) (highlight moves to the **Tool#** field).
6. Key in a tool number (optional) and press (ENTER) (**Rapid** block is inserted into the program listing).

Method 2, Using Soft Keys

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill soft key labels appear).
2. Press **Rapid** (F2) (**RAPID (XY)** Graphic Menu prompts for **X** position).
3. Key in the **X** position (optional) and press (ENTER) (highlight moves to the **Y** field).
4. Key in the **Y** position (optional) and press (ENTER) (highlight moves to the **Z** field).
5. Key in the **Z** position (optional) and press (ENTER) (highlight moves to **ToolComp** field).
6. Using (+/-) set **ToolComp** (optional) and press (ENTER) (highlight moves to the **Tool#** field).
7. Key in a tool number (optional) and press (ENTER) (**Rapid** block is inserted into the program listing, CNC returns to the Mill softkey line).
8. Press (F9) **Prev** (Program Editor default softkeys return).

Programming a Line Move

Line moves are straight line moves executed in Feed.

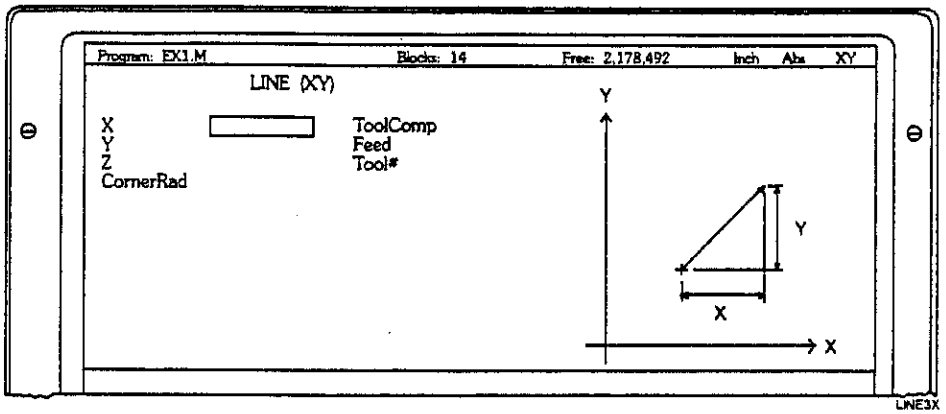


Figure 4 - 7, Line Move Graphic Menu

Refer to **Figure 4 - 7, Line Move Graphic Menu**.

To program Line move:

Method 1, Using Hot Keys

1. With the CNC in the Edit Mode, press the (2/LINE) [2 LINE] key (**LINE (XY)** Graphic Menu prompts operator for **X** position).
2. Key in **X** position (optional) and press (ENTER) [ENTER] (highlight moves to the **Y** entry field).
3. Key in **Y** position (optional) and press (ENTER) [ENTER] (highlight moves to **Z** entry field).
4. Key in **Z** position (optional) and press (ENTER) [ENTER] (highlight moves to **CornerRad** entry field).
5. Key in a **CornerRad** value (optional) and press (ENTER) [ENTER] (highlight moves to **ToolComp** field).
6. Using (+/-) [+/-] set **ToolComp** (optional) and press (ENTER) [ENTER] (highlight moves to **Feed**).
7. Key in a feedrate change (optional) and press (ENTER) [ENTER] (highlight moves to **Tool#**).
8. Key in a tool number (optional) and press (ENTER) [ENTER] (**Line** block is added to the program listing).

Method 2, Using Softkeys

1. With the CNC in the Edit Mode, press **Mill** (F5) (soft key labels change).
2. Press **Line** (F3) (**LINE (XY)** Graphic Menu prompts operator for **X** position).
3. Key in **X** position (optional) and press (ENTER) [ENTER] (highlight moves to the **Y** entry field).
4. Key in **Y** position (optional) and press (ENTER) [ENTER] (highlight moves to **Z** entry field).
5. Key in **Z** position (optional) and press (ENTER) [ENTER] (highlight moves to **CornerRad** entry field).

6. Key in a **CornerRad** value (optional) and press (ENTER) (highlight moves to **ToolComp** field).
7. Using (+/-) set **ToolComp** (optional) and press (ENTER) (highlight moves to **Feed**).
8. Key in a feedrate change (optional) and press (ENTER) (highlight moves to **Tool#**).
9. Key in a tool number (optional) and press (ENTER) (**Line** block is added to the program listing, CNC returns to the **Mill** softkey line).
10. Press (F9) **Prev** (Program Editor default softkeys return).

Programming a Modal Move

Modal moves have no specific move rate. If the CNC is in the Feed Mode the move is executed at the active feedrate. If the CNC is in the Rapid Mode the move is executed in Rapid.

To program a Modal move:

1. With the CNC in the Edit Mode, press (X) (Y) or (Z) (**MODAL MOVE** Graphic Menu prompts operator for the **X** , **Y** and **Z** positions).
2. Key in required positions using the (ARROW) keys to move the highlight to the required entry fields.
3. From the last field on the Graphic Menu press (ENTER) or **Save** (F10) (modal move block is added to program).

NOTE: When using modal moves be sure the CNC is in the correct move mode.

Teach Mode (Programming From the Part)

In the Teach Mode, the CNC writes program blocks that duplicate manually executed moves. This lets the operator program a move without knowing the coordinates of the endpoint. Programming in Teach Mode lets the operator write a program by picking endpoints off a part.

Teach Mode programming can also be used with the CNC in DRO Mode.

The Teach Mode is activated from the **Edit** screen by pressing the **Teach** (F1) softkey. The **Teach** (F1) softkey label is highlighted when the Teach Mode is active.

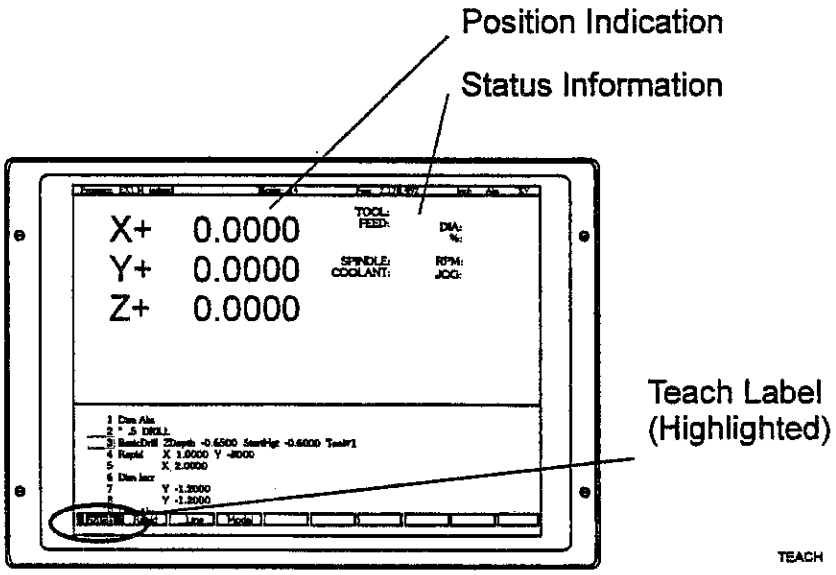


Figure 4 - 8, Program Editor With Teach Mode Active

Refer to **Figure 4 - 8, Program Editor With Teach Mode Active**.
To program a move in the Teach Mode:

Teaching a Rapid Move

1. With the CNC in the Edit Mode, press **Teach** (F1) (Teach Mode is activated, position display shows current location).
2. Manually position the machine to the endpoint and press **Rapid** (F2) (CNC inserts a **Rapid** block).
3. When done programming moves in the Teach Mode, press **Teach** (F1) (Teach Mode deactivates).

Teaching a Line Move

1. With the CNC in the Edit Mode, press **Teach** (F1) (Teach Mode is activated, position display shows current location).
2. Manually position the machine to the required endpoint and press **Line** (F3) (CNC inserts a **Line** block).
3. When done programming moves in the Teach Mode, press **Teach** (F1) (Teach mode deactivates).

NOTE: A Line move is a straight line move done in Feed.

Teaching a Modal Move

1. With the CNC in the Edit Mode, press **Teach** (F1) (Teach Mode is activated, position display shows current location).

2. Manually position the machine to the required endpoint and press **Modal** (F4) (CNC inserts a **Modal** block).
3. When done programming in the Teach Mode, press **Teach** (F1) (Teach mode deactivates).

NOTE: Modal moves are moves executed in the active mode of the CNC. When programming Modal moves ensure the CNC is in the correct Rapid or Line mode.

Straight Moves With Trigonometry

Refer to **Figure 4 - 9, Move Orientation**. The CNC can write Rapid and Line moves when one or both end point axis coordinates are unknown. The CNC does this by using trigonometric functions to calculate the missing endpoint. Because trigonometry is used, the move must be defined as part of a right triangle with the components identified as shown in the figure.

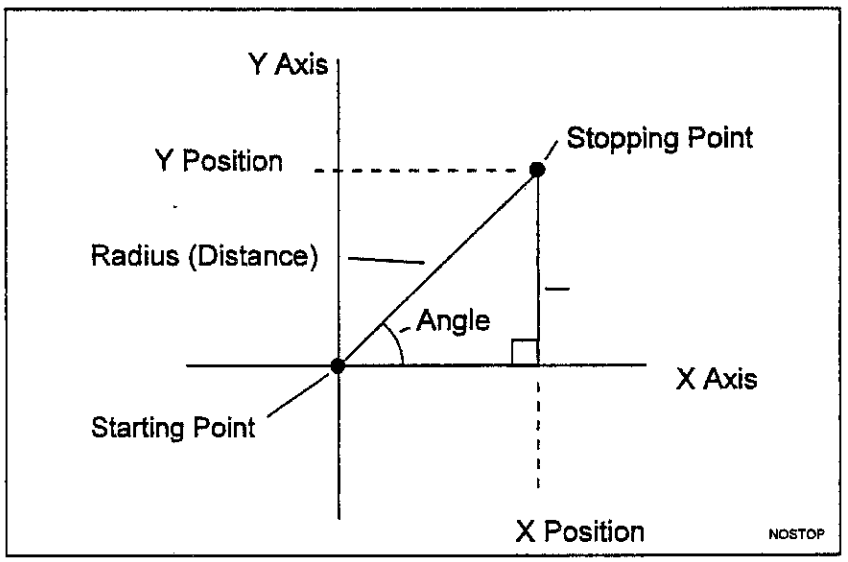


Figure 4 - 9, Move Orientation

The CNC can calculate move end points using two positions, or a position and angle paired as follows:

- The angle and radius.
- The X position and the angle.
- The Y position and the angle.
- The X position and the radius.
- The Y position and the radius.

The Rapid and Line Graphic Menus for programming with trigonometry are the same except the Rapid move Graphic Menus do not contain **CornerRad** and **Feed** entry fields.

The trigonometry can be used in either the Absolute or Incremental Modes. Programming these tools in the Incremental Mode is preferred because it is easier to understand.

NOTE: When programming moves with trigonometry, it is important to consider the effects of being in Absolute or Incremental Mode.

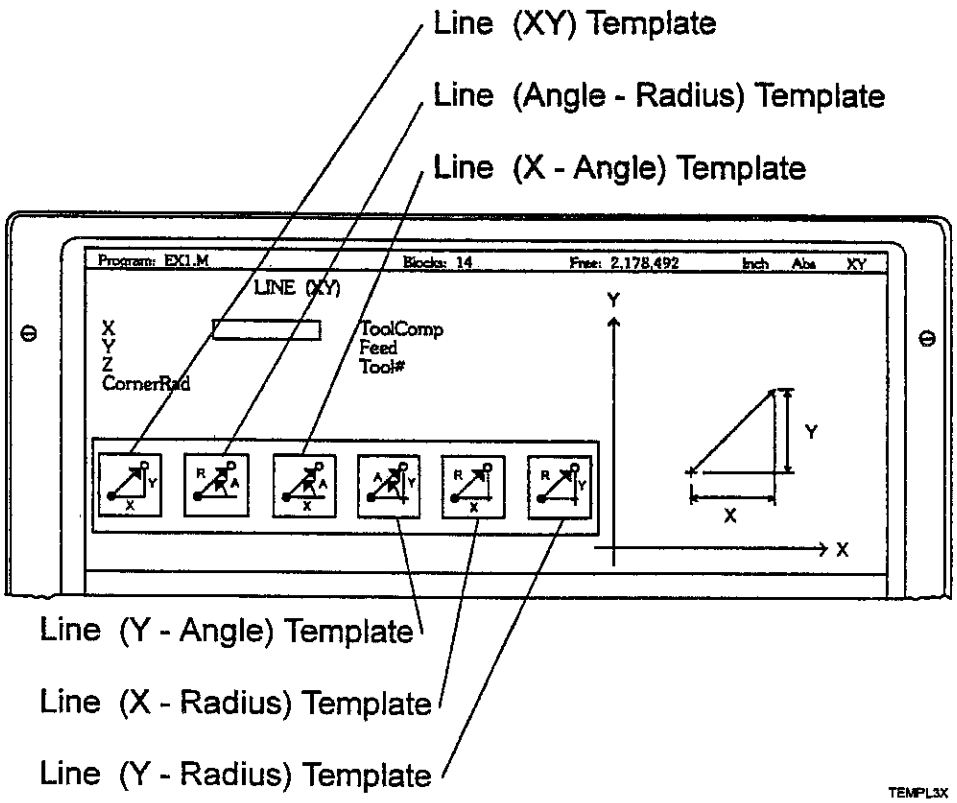


Figure 4 - 10, Template Pop-up Menu

Programming a Move Using the Angle and Radius

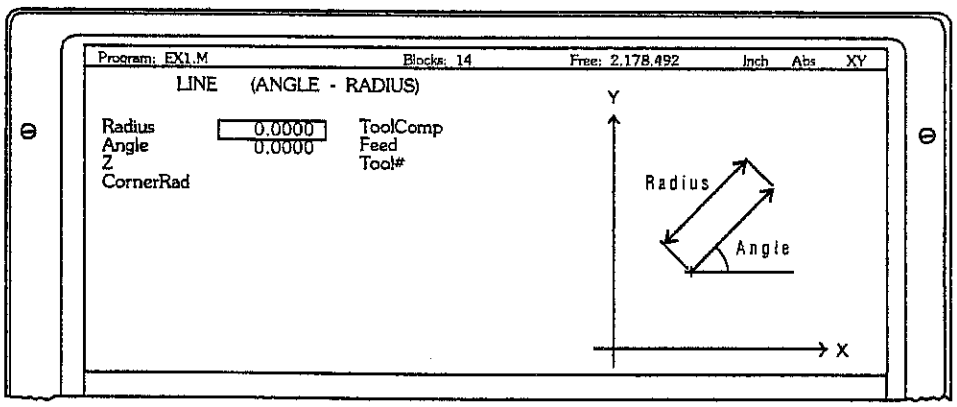


Figure 4 - 11, Line (Angle Radius) Graphic Menu

Refer to **Figure 4 - 11, Line (Angle Radius) Graphic Menu.**

To program a move using the angle and radius:

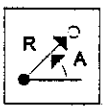
1. With the CNC in the Edit Mode, press **Mill** (F5) and select either **Rapid** (F2) or **Line** (F3).

   - or -

   With the CNC in the Edit Mode, press (1/RAPID) [1 RAPID] or (2/LINE) [2 LINE] (**RAPID (XY)** or **LINE (XY)** Graphic Menu appears).

NOTE: A Line move is a straight line move done in Feed.

2. Press **More. . .** (F4) (template pop-up menu appears, refer to **Figure 4 - 10, Template Pop-up Menu**).

3. Using (ARROW) keys, move the highlight to select the (ANGLE - RADIUS) 

template and press (ENTER) [ENTER] (Angle Radius) Graphic Menu prompts for labeled values).

4. Key in **Radius** (value required) and press (ENTER) [ENTER] (highlight moves to **Angle** entry field).

5. Key in **Angle** (value required) and press (ENTER) [ENTER] (highlight moves to **Z** entry field).

6. Key in **Z** position (optional) and press (ENTER) [ENTER] (highlight moves to **CornerRad** entry field).

7. Key in a **CornerRad** value (optional) and press (ENTER) [ENTER] (highlight moves to **ToolComp** field).

8. Using (+/-) [+/-] set **ToolComp** (optional) and press (ENTER) [ENTER] (highlight moves to **Feed**).

9. Key in a feedrate (optional) and press (ENTER) [ENTER] (highlight moves to **Tool#**).

10. Key in a tool number (optional) and press (ENTER) [ENTER] (**Tool#** block is inserted in the Program listing).

NOTE: The **RAPID (ANGLE RADIUS)** Graphic Menu does not have **CornerRad** and **Feed** entry fields.

Programming a Move Using the X Position and the Angle

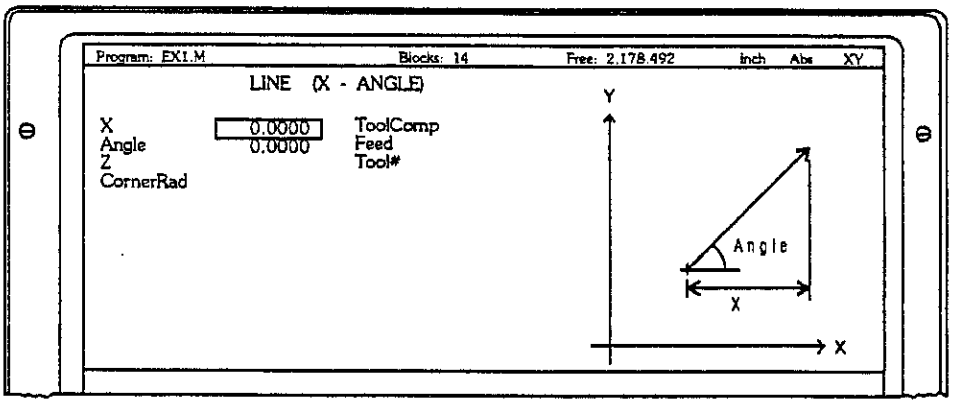


Figure 4 - 12, Line (X - Angle) Graphic Menu

Refer to **Figure 4 - 12, Line (X - Angle) Graphic Menu**.

To program a move using the X position and the angle:

1. With the CNC in the Edit Mode, press **Mill** (F5) and select either **Rapid** (F2) or **Line** (F3).

   - or -

   From the Program Editor, press (1/RAPID) [1 RAPID] or (2/LINE) [2 LINE] (**RAPID (XY)** or **LINE (XY)** Graphic Menu appears).

NOTE: A Line move is a straight line move done in Feed.

2. Press **More. . .** (F4) (template pop-up menu appears, refer to **Figure 4 - 10, Template Pop-up Menu**).

3. Using (ARROW) keys move the highlight to select the (X - ANGLE) template and press (ENTER) [ENTER] (**X - ANGLE** Graphic Menu prompts for labeled values).

4. Key in the **X** position (value required) and press (ENTER) [ENTER] (highlight moves to **Angle** entry field).

5. Key in the **Angle** (value required) and press (ENTER) [ENTER] (highlight moves to **Z** entry field).

6. Key in a **Z** position (optional) and press (ENTER) [ENTER] (highlight moves to **CornerRad** entry field).

7. Key in a **CornerRad** value (optional) and press (ENTER) [ENTER] (highlight moves to **ToolComp** entry field).

8. Using (+/-) [+/-] set **ToolComp** (optional) and press (ENTER) [ENTER] (highlight moves to **Feed** entry field).

9. Key in a feedrate (optional) and press (ENTER) [ENTER] (highlight moves to **Tool#** entry field).

10. Key in a tool number (optional) and press (ENTER) [ENTER] (block is inserted into the Program listing).

NOTE: The **RAPID (X - ANGLE)** Graphic Menu does not have **CornerRad** and **Feed** fields.

Programming a Move Using the Y Position and the Angle

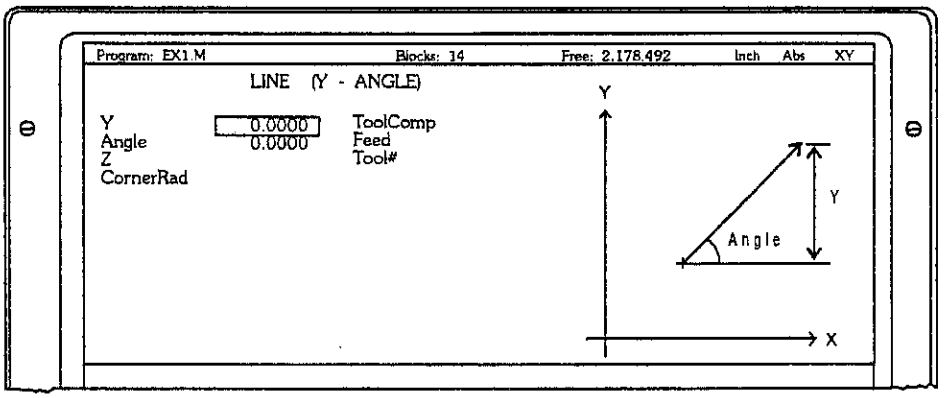


Figure 4 - 13, Line (Y - Angle) Graphic Menu

Refer to **Figure 4 - 13, Line (Y - Angle) Graphic Menu**.

To program a move using the Y position and the angle:

1. With the CNC in the Edit Mode, press **Mill** (F5) and select either **Rapid** (F2) or **Line** (F3).

- or -

With the CNC in the Edit Mode, press (1/RAPID) [1 RAPID] or (2/LINE) [2 LINE] (**RAPID (XY)** or **LINE (XY)** Graphic Menu appears).

NOTE: A Line move is a straight line move done in Feed.

2. Press **More. . .** (F4) (template pop-up menu appears, refer to **Figure 4 - 10, Template Pop-up Menu**).

3. Using (ARROW) keys, move the highlight to select the (Y - ANGLE) template and press (ENTER) (**Y - ANGLE** Graphic Menu prompts for labeled values).

4. Key in the **Y** position (value required) and press (ENTER) (highlight moves to **Angle** entry field).

5. Key in the **Angle** (value required) and press (ENTER) (highlight moves to **Z** entry field).

6. Key in the **Z** position (optional) and press (ENTER) (highlight moves to **CornerRad** entry field).

7. Key in a **CornerRad** value (optional) and press (ENTER) (highlight moves to **ToolComp** entry field).

8. Using (+/-) set **ToolComp** (optional) and press (ENTER) (highlight moves to **Feed** entry field).

9. Key in a feedrate (optional) and press (ENTER) (highlight moves to **Tool#** entry field).

10. Key in a tool number (optional) and press (ENTER) (block is inserted into the program listing).

NOTE: The **RAPID (Y - ANGLE)** Graphic Menu does not have **CornerRad** and **Feed** entry fields.

Programming a Move Using the X Position and the Radius

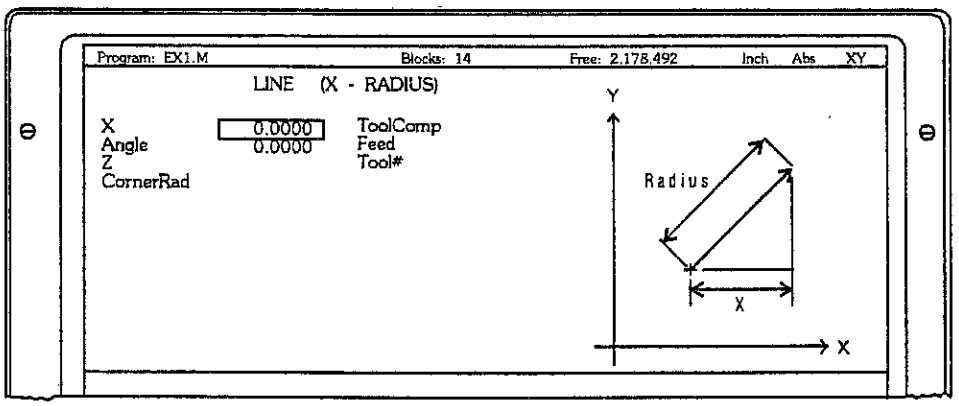


Figure 4 - 14, Line (X - Radius) Graphic Menu

Refer to **Figure 4 - 14, Line (X - Radius) Graphic Menu**.

To program a move using the X position and the radius:

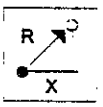
1. With the CNC in the Edit Mode, press **Mill** (F5) and select either **Rapid** (F2) or **Line** (F3).

   - or -

   With the CNC in the Edit Mode, press (1/RAPID) [1 RAPID] or (2/LINE) [2 LINE] (**RAPID (XY)** or **LINE (XY)** Graphic Menu appears).

NOTE: A Line move is a straight line feed move.

2. Press **More. . .** (F4) (template pop-up menu appears, Refer to **Figure 4 - 10, Template Pop-up Menu**).

3. Using (ARROW) keys, move the highlight to select the (X - RADIUS) 

template and press (ENTER) [ENTER] **(X - RADIUS)** Graphic Menu prompts for labeled values).

4. Key in the **X** position (value required) and press (ENTER) [ENTER] (highlight moves to **Radius** entry field).

5. Key in the **Radius** (value required) and press (ENTER) [ENTER] (highlight moves to **Z** entry field).

6. Key in a **Z** position (optional) and press (ENTER) [ENTER] (highlight moves to **CornerRad** entry field).

7. Key in a **CornerRad** value (optional) and press (ENTER) [ENTER] (highlight moves to **ToolComp** entry field).

8. Using (+/-) [+/-] set **ToolComp** (optional) and press (ENTER) [ENTER] (highlight moves to **Feed** entry field).

9. Key in a feedrate (optional) and press (ENTER) [ENTER] (highlight moves to **Tool#** entry field).

10. Key in a tool number (optional) and press (ENTER) [ENTER] (block is inserted into the program listing).

NOTE: The **Rapid (X - RADIUS)** Graphic Menu does not have **CornerRad** and **Feed** entry fields.

Programming a Move Using the Y Position and the Radius

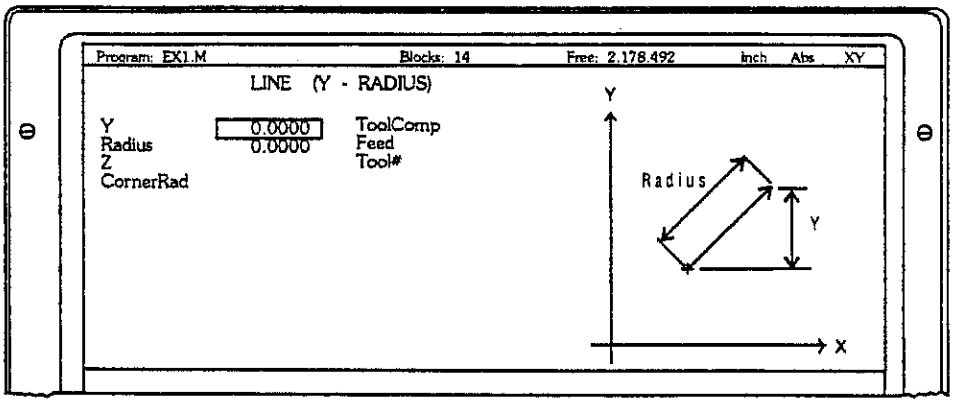


Figure 4 - 15, Line (Y - Radius) Graphic Menu

Refer to **Figure 4 - 15, Line (Y - Radius) Graphic Menu**.

To program a move using the Y position and the radius:

1. With the CNC in the Edit Mode, press **Mill** (F5) and select either **Rapid** (F2) or **line** (F3).

   - or -

   With the CNC in the Edit Mode, press (1/RAPID) [1 RAPID] or (2/LINE) [2 LINE] (**RAPID (XY)** or **LINE (XY)** Graphic Menu appears).

NOTE: A Line move is a straight line move done in Feed.

2. Press **More. . .** (F4) (template pop-up menu appears, refer to **Figure 4 - 10, Template Pop-up Menu**).

3. Using (ARROW) keys, move the highlight to select the (ANGLE - RADIUS) template and press (ENTER) [ENTER] (**Y - RADIUS)** Graphic Menu prompts for labeled values).

4. Key in the **Y** position (value required) and press (ENTER) [ENTER] (highlight moves to the **Radius** entry field).

5. Key in the **Radius** (value required) and press (ENTER) [ENTER] (highlight moves to the **Z** entry field).

6. Key in the **Z** position (optional) and press (ENTER) [ENTER] (highlight moves to the **CornerRad** entry field).

7. Key in a **CornerRad** value (optional) and press (ENTER) (highlight moves to **ToolComp** entry field).
8. Using (+/-) set **ToolComp** (optional) and press (ENTER) (highlight moves to **Feed** entry field).
9. Key in a feedrate (optional) and press (ENTER) (highlight moves to **Tool#** entry field).
10. Key in a tool number (optional) and press (ENTER) (block is inserted into the program listing).

NOTE: The **RAPID (Y - RADIUS)** Graphic Menu does not offer **CornerRad** and **Feed** values.

Arcs, Ellipses and Spirals

NOTE: All blocks are programmed by filling in the entry fields of a Graphic Menu. The process of entering values and advancing through entry fields is explained at the beginning of this section. This process is also illustrated in the descriptions for programming Rapid and Line moves. Step by step programming exercises are also included in **Section 12 - Sample Programs and Practice Exercises**.

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Selecting the Plane for an Arc, Ellipse or Spiral Move

Refer to **Section 1 - 1100M Programming Concepts** for basic information on planes.

Arcs, Ellipses and Spirals are programmed to execute in the XY plane by default. If an Arc, Ellipse or Spiral move in the XZ or YZ plane is required, program the plane change before programming the move.

Programming the plane puts the Program Editor in a mode to customize the Arc, Ellipse and Spiral Graphic Menus for the new plane.

The Graphic Menus for moves in the XY, XZ and YZ planes all contain the same entry fields. The only difference is which fields require a value and which fields are optional. Entry fields for "in plane" positions will always require a value. Entry fields for "off plane" positions will always be optional.

Always program a plane change to return the editor to the XY plane when moves in other planes are complete.

NOTE: The only way to activate a new plane in the Program Editor is to program a new plane change block. Deleting a **Plane** block does not change the editors plane. Deleting (or inserting) a **Plane** block does not change the way existing blocks are programmed.

The following paragraphs describe Arc, Ellipse and Spiral moves in the XY plane.

Arc moves can be programmed three different ways.

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

Programming an Arc Using an Endpoint and Radius

With the End Point - Radius Arc the operator defines the Arc by keying in the direction of the Arc, the end point and the radius. The CNC cuts an Arc of the proper radius, from the present position to the end point. The end point coordinates are modal values and must be correctly defined for the Absolute/Incremental Mode of the CNC.

In the XY plane, if Z axis starting and end positions are not the same, the Arc move will be helical in nature.

Refer to **Figure 4 - 16, Endpoint Radius Arc Types**. There are two Arcs that can intersect any two points, an Arc with an included angle less than 180 degrees and an Arc with an included angle greater than 180 degrees.

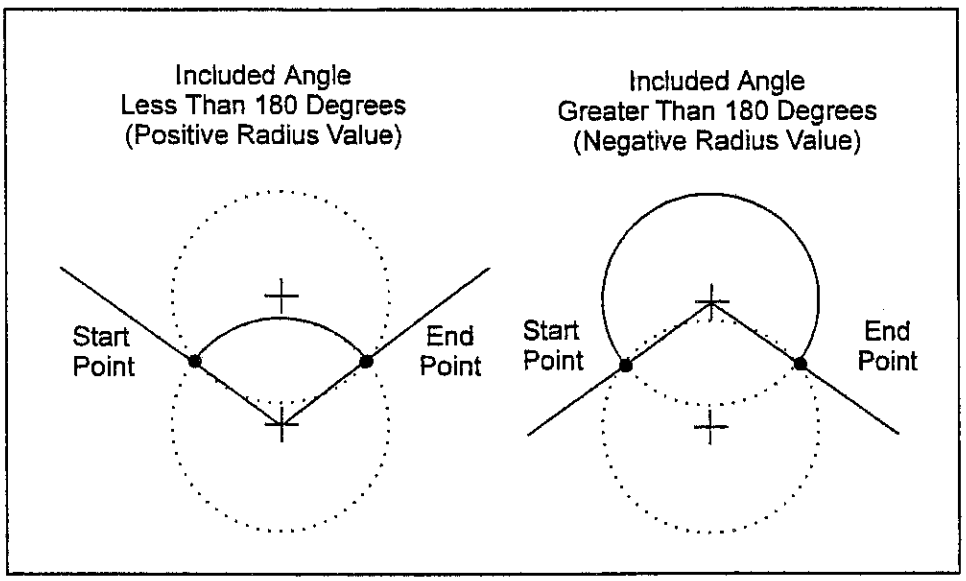


Figure 4 - 16, Endpoint Radius Arc Types

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

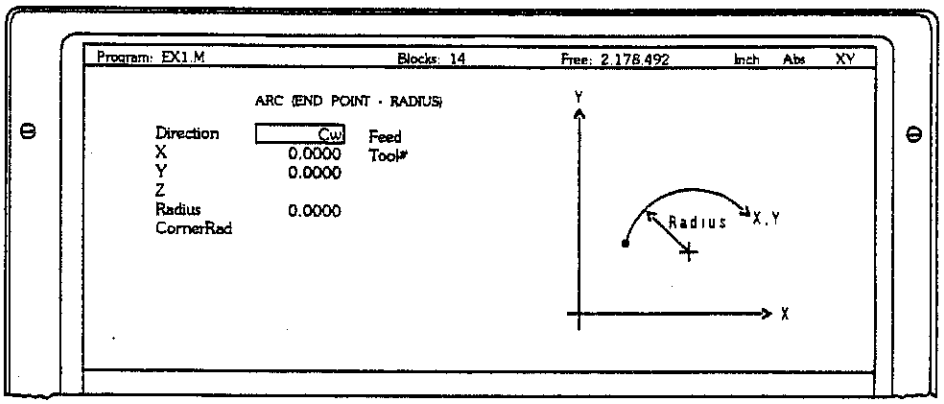


Figure 4 - 17, ARC (Endpoint - Radius) Graphic Menu

Refer to **Figure 4 - 17, ARC (Endpoint - Radius) Graphic Menu**.
To program an Arc using an endpoint and radius:

Method #1, Using the Programming Hot keys

1. With the CNC in the Edit Mode, press (3/ARC) [3 / ARC] (**ARC (END POINT - RADIUS)** Graphic Menu prompts for labeled values).
2. Fill in the entry fields as labeled.

Method #2, Using the Softkeys

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill secondary softkeys are displayed).
2. Press **Arc** (F4) (**ARC (END POINT - RADIUS)** Graphic Menu prompts for labeled values).
3. Refer to **Figure 4 - 17, ARC (Endpoint - Radius) Graphic Menu**.
4. Fill in the **ARC (END POINT - RADIUS)** entry fields labeled as follows:

Direction — This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-) [+/-] key (selection required).

X — The X coordinate of the Arc end point (value required).

Y — The Y coordinate of the Arc end point (value required).

Z — The Z coordinate of the end point (optional).

Radius — The radius of the Arc (value required, can be positive or negative).

CornerRad	Corner radius setting (optional), refer to **Section 1 - 1100M Programming Concepts** for details.
Feed	Feedrate (optional).
Tool#	Active tool (optional).

Programming an Arc Using the Center and Endpoint

NOTE: The Center and Endpoint Arc is the recommended tool for cutting helical threads.

With the Center - End Point Arc, the operator defines the Arc by keying in the end point, the center point and the direction. The CNC will cut the Arc from the present position to the end point.

The Arc's radius is the distance from the present position to the center position keyed in. The end point is a modal value and must be correctly defined for the current Absolute/Incremental Mode of the CNC.

In the XY plane, if Z axis starting and end positions are not the same, the Arc move will be helical in nature. The **Revs** value will determine the number of turns in the helix.

Because the radius is calculated from the starting point and the center point, the end point must lie on the Arc's path. If the end point is not exactly on the Arc's path, the control will attempt to compensate by adjusting the center point. The amount of adjustment allowed is determined by the builder's setup tolerances. The CNC generates an error message if the error is greater than the setup tolerance.

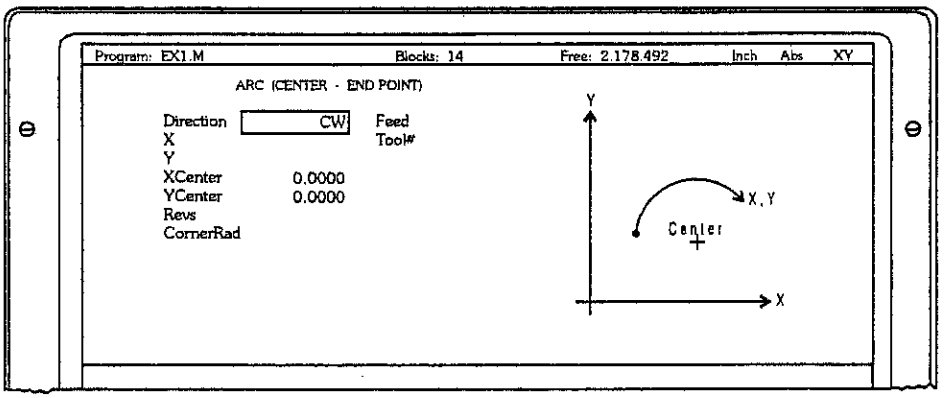


Figure 4 - 18, Arc (Center - End Point) Graphic Menu

Refer to **Figure 4 - 18, Arc (Center - End Point) Graphic Menu**.

To program a Center - End Point Arc:

Method #1, Using the Hot Keys

1. With the CNC in the Edit Mode, press (3/ARC) [3 / ARC] (**ARC (END POINT - RADIUS)** Graphic Menu appears).
2. Press **More...** (F4) (Arc template pop-up menu appears, refer to **Figure 4 - 19, Arc Template** Pop-up Menu).

3. Using the (ARROW) keys, position the highlight to select the Center - End Point template and press (ENTER) (**ARC (CENTER - END POINT)** Graphic Menu prompts for labeled values).
4. Fill in the entry fields as labeled

Method #2, Using the Softkeys

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill secondary softkeys are displayed).
2. Press **Arc** (F4) (**ARC (END POINT - RADIUS)** Graphic Menu appears).
3. Press **More...** (F4) (Arc template pop-up menu appears, refer to **Figure 4 - 19, Arc Template Pop-up Menu**).

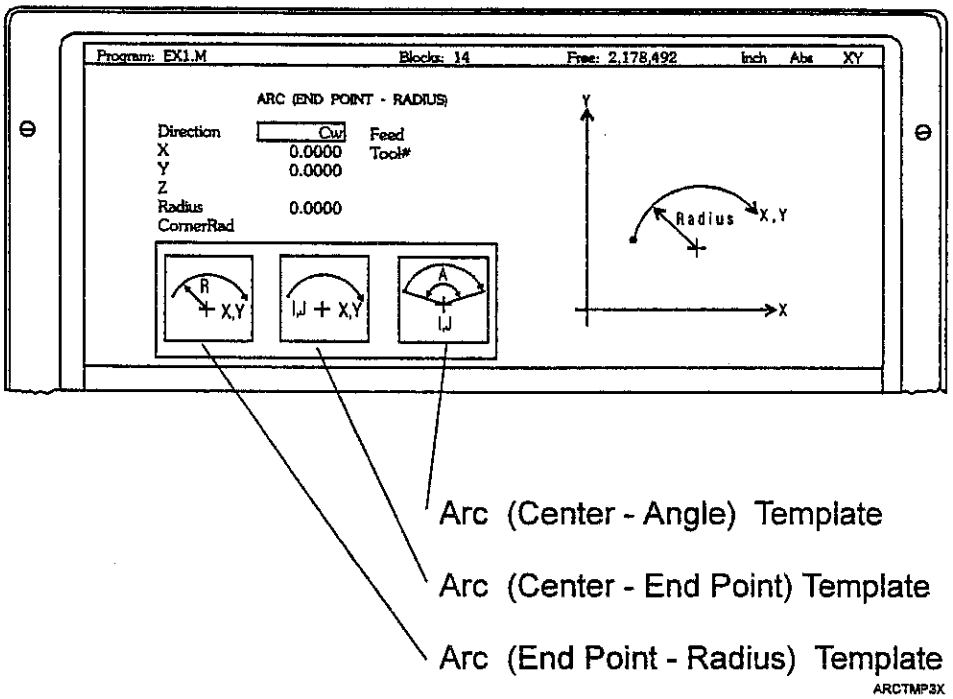


Figure 4 - 19, Arc Template Pop-up Menu

4. Using the (ARROW) keys, position the highlight to select the (CENTER - END POINT) template and press (ENTER) (**ARC (CENTER - END POINT)** Graphic Menu prompts for labeled values).

5. Fill in the **ARC (CENTER - END POINT)** entry fields labeled as follows:

Direction	This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-)  key (selection required).
X	The X coordinate of the Arc end point (optional).
Y	The Y coordinate of the Arc end point (optional).
Z	The Z coordinate of the Arc end point (optional).
XCenter	The X coordinate of the Arc's center (value required).
YCenter	The Y coordinate of the Arc's center (value required).
Revs	Number of turns from starting position to endpoint Z position (optional).
CornerRad	Corner radius setting (optional), refer **Section 1 - 1100M Programming Concepts** for details.
Feed	Feedrate (optional).
Tool#	Active tool (optional).

Programming an Arc Using the Center and the Included Angle

With the Center - Angle Arc, the operator defines the Arc by keying in the center point 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 radius is the distance between the starting position and the center point. It is calculated by the CNC.

Refer to **Figure 4 - 20, Absolute Mode, Center - Angle Arc** and **Figure 4 - 21, Incremental Mode, Center - Angle Arc**. The angle and centerpoint 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.

In the XY plane, if Z axis starting and end positions are not the same, the Arc move will be helical in nature.

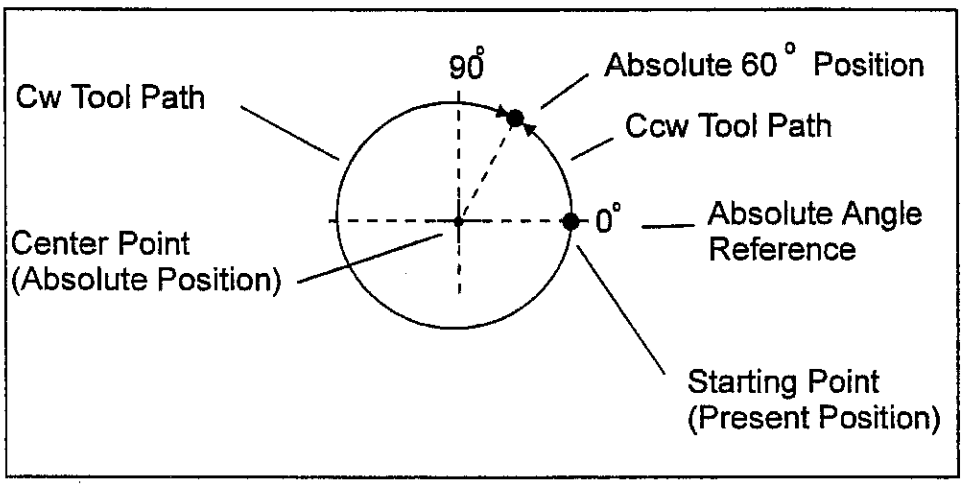


Figure 4 - 20, Absolute Mode, Center - Angle Arc

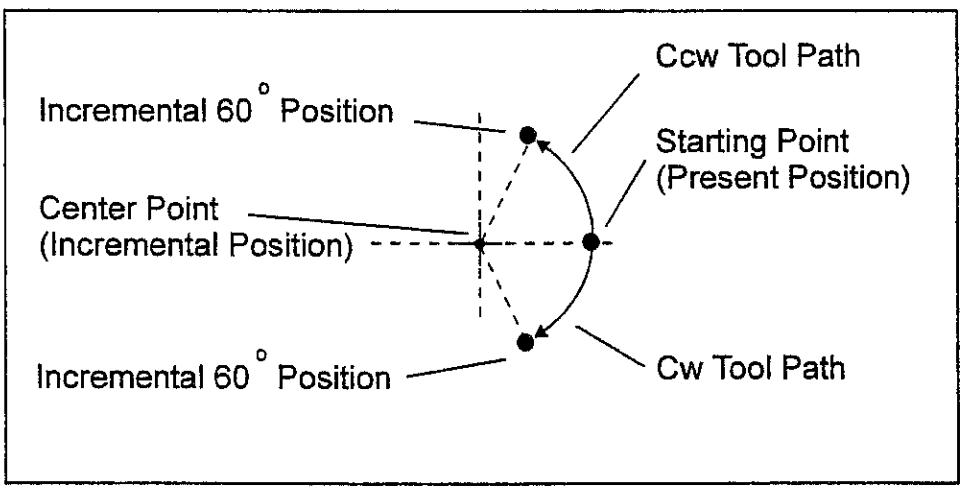


Figure 4 - 21, Incremental Mode, Center - Angle Arc

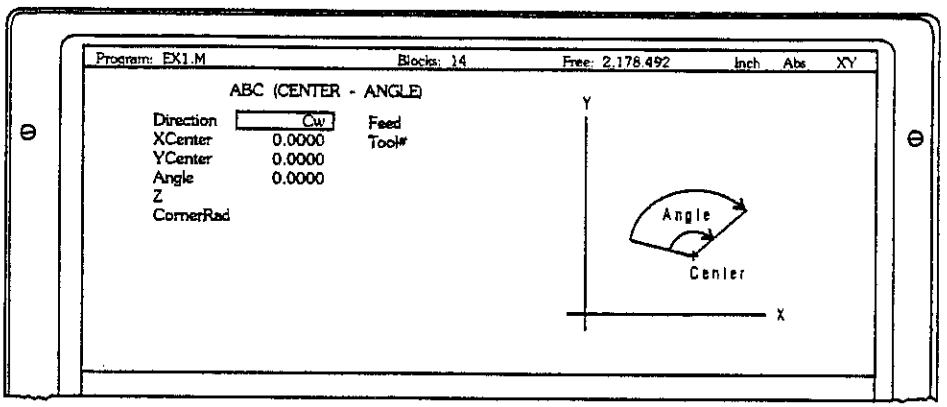


Figure 4 - 22, Arc (Center - Angle) Graphic Menu

Refer to **Figure 4 - 22, Arc (Center - Angle) Graphic Menu**.
To program an Arc using the center and the included angle:

Method #1, Using the Programming Hot Keys

1. With the CNC in the Edit Mode, press (3/ARC) 3 ARC (**ARC (END POINT - RADIUS)** Graphic Menu appears).
2. Press **More...** (F4) (Arc template pop-up menu appears, refer to **Figure 4 - 19, Arc Template Pop-up Menu**).
3. Using the (ARROW) keys, position the highlight to select the (CENTER - ANGLE) template and press (ENTER) ENTER (**ARC (CENTER - ANGLE)** Graphic Menu prompts for labeled values).
4. Fill in the entry fields as labeled.

Method #2, Using the Softkeys

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill secondary softkeys are displayed).
2. Press **Arc** (F4) (**ARC (END POINT - RADIUS)** Graphic Menu appears).
3. Press **More...** (F4) (Arc template pop-up menu appears, refer to **Figure 4 - 19, Arc Template Pop-up Menu**).
4. Using the (ARROW) keys, position the highlight to select the (CENTER - ANGLE) template and press (ENTER) ENTER (**ARC (CENTER - ANGLE)** Graphic Menu prompts for labeled values).
5. Fill in the **ARC (CENTER - ANGLE)** entry fields labeled as follows:

Direction	This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-)  key (selection required).
XCenter	The X coordinate of the Arc's center (value required).
YCenter	The Y coordinate of the Arc's center (value required).
Angle	Included angle of the Arc (value required).
Z	The Z coordinate of the Arc end point (optional).
CornerRad	Corner radius setting (optional), refer to **Section 1 - 1100M Programming Concepts** for details.
Feed	Feedrate (optional).
Tool#	Active tool (optional).

Programming an Ellipse

The Ellipse Cycle simplifies the programming required to cut an ellipse. When the Ellipse Cycle runs, the CNC feeds from its present location, around the defined ellipse, to the end point. The present location (starting point) and the endpoint must lie on the ellipse. An ellipse must be programmed using incremental values.

Standard tool compensation cannot be used with the ellipse cycle (it would not produce an ellipse). Ellipse cutter compensation can be activated using the M1040, M-code. The cutter must be in place when M1040 is executed (no ramp move).

In the XY plane, if Z axis starting and end positions are not the same, the Arc move will be helical in nature.

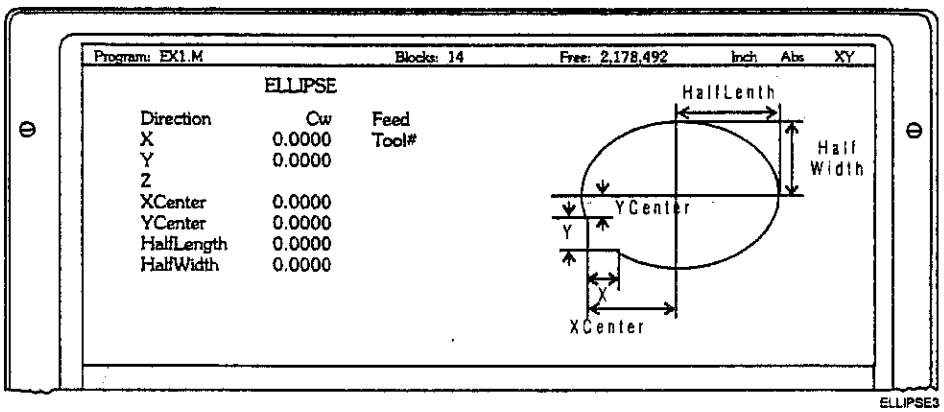


Figure 4 - 23, Ellipse Graphic Menu

Refer to **Figure 4 - 23, Ellipse Graphic Menu**.

To program an Ellipse:

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill softkey labels appear).
2. Press **More** (F7) (More pop-up menu appears).
3. Using the (ARROW) keys, position the highlight to select **Ellipse** and press (ENTER) (**ELLIPSE** Graphic Menu prompts for labeled values).
4. Fill in the **ELLIPSE** entry fields labeled as follows:

Direction — This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-) key (selection required).

X — X coordinate of endpoint (value required, incremental).

Y	Y coordinate of endpoint (value required, incremental).
Z	Z coordinate of endpoint (value required, incremental).
XCenter	X coordinate of ellipse centerline (value required).
YCenter	Y coordinate of ellipse centerline (value required).
HalfLength	One half of the overall X dimension of the ellipse (value required, incremental).
HalfWidth	One half of the overall Y dimension of the ellipse (value required, incremental).
Feed	Feedrate (optional).
Tool#	Active tool (optional).

Programming a Spiral

A spiral is an Arc with a continuously changing radius. To program a Spiral, the operator keys in the **Direction** of the cut, the **X** and **Y** coordinates of the end point, the coordinates of the center (**XCenter**, **YCenter**) and the number of revolutions (**Revs**). A spiral must be programmed using incremental values.

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

- Present position is closer to the center than the end point (cycle spirals outward).
- End point is closer to the center than the machine's present position (cycle spirals inward).

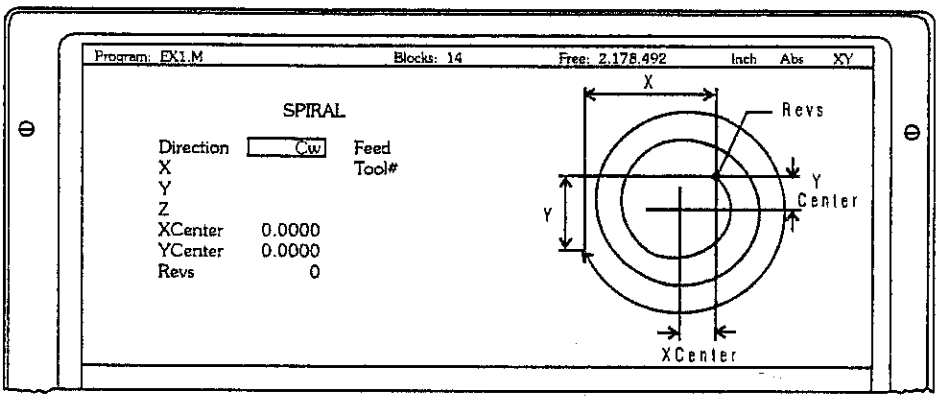


Figure 4 - 24, Spiral Graphic Menu

Refer to **Figure 4 - 24, Spiral Graphic Menu**.

To program a Spiral cycle:

1. With the CNC in the Edit Mode, press **Mill** (F5) (Mill softkey labels appear).
2. Press **More** (F7) (More pop-up menu appears).
3. Using the (ARROW) keys, position the highlight to select **Spiral** and press (ENTER) (**SPIRAL** Graphic Menu prompts for labeled values).
4. Fill in the **SPIRAL** entry fields labeled as follows:

Direction	This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-)  key (selection required).
X	X coordinate of endpoint (optional, incremental).
Y	Y coordinate of endpoint (optional, incremental).
Z	Z coordinate of endpoint (optional, incremental).
XCenter	X coordinate of center (value required, incremental).
YCenter	Y coordinate of center (value required, incremental).
Revs	Number of complete revolutions made in the cycle (value required).
Feed	Feedrate (optional).
Tool#	Active tool (optional).

Drilling Cycles

NOTE: All blocks are programmed by filling in the entry fields of a Graphic Menu. The process of entering values and advancing through entry fields is explained at the beginning of this section. This process is also illustrated in the descriptions for programming Rapid and Line moves. Step by step programming exercises are also included in **Section 12 - Sample Programs and Practice Exercises**.

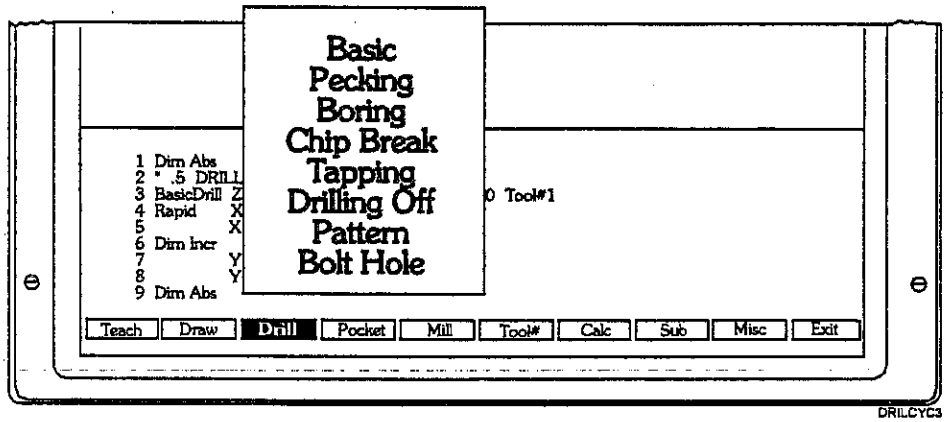


Figure 4 - 25, Drill Cycle Pop-up Menu

Refer to **Figure 4 - 25, Drill Cycle Pop-up Menu**. Drill cycles simplify the programming required for repetitive drilling, boring and tapping operations. Specific Drill cycles are selected from the Program Editor's **Drill** (F3) pop up menu.

Drill cycles are modal. Once the CNC encounters a block for any type of Drill cycle, it executes that cycle at the end point 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.

Programming a Basic Drill Cycle

The Basic Drill Cycle starts once the CNC is in position. The tool rapids to the **StartHgt** then Z feeds to **ZDepth**. At the end of the cycle the tool rapids to **ReturnHgt** to provide clearance for the next move.

A **DrillOff** block must be programmed to turn off the cycle. The operator can program any number of patterns and moves before turning off the cycle.

Refer to **Figure 4 - 26, Basic Drilling Graphic Menu**.

To program a **BasicDrill** block:

1. With the CNC in the Edit mode, press **Drill** (F3) (drill cycle pop-up menu appears).

2. Using the (ARROW) keys, position the highlight to select **Basic** and press (ENTER) (**BASIC DRILLING** Graphic Menu prompts for labeled values).

3. Key in labeled values and press (ENTER) (display clears, **BasicDrill** block is added to the program listing).

NOTE: After programming a **BasicDrill** cycle, moves must be programmed to tell the CNC where to drill. The CNC will execute the **BasicDrill** cycle at the end point of all subsequent moves until it sees a **Drilling Off** block.

4. Program subsequent moves to position the work to where the drilling is to occur, the CNC will drill a hole at the end point of every move.

5. After programming the last drill move, press **Drill** (F3) (drill cycle pop-up menu appears).

6. Highlight **Drilling Off** and press (ENTER) (**DrillOff** block is included in program to take CNC out of the drilling mode).

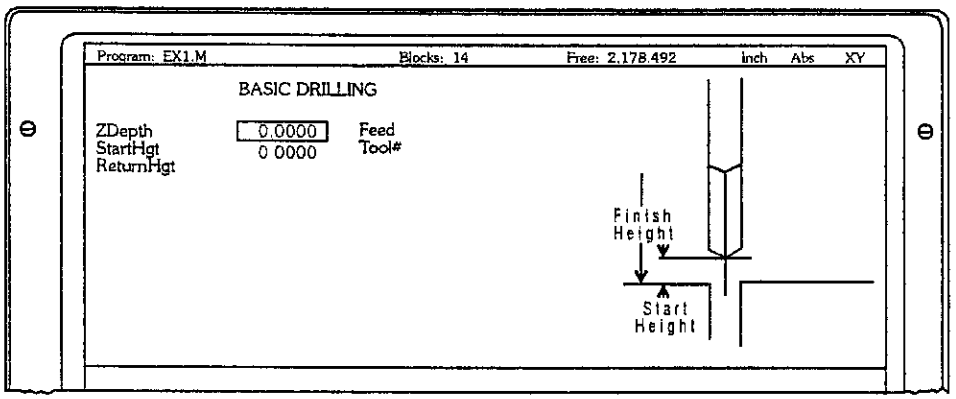


Figure 4 - 26, Basic Drilling Graphic Menu

BASIC DRILLING entry fields are labeled as follows:

ZDepth — The absolute depth of the finished hole (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be drilled.

ReturnHgt — The absolute position the tool returns to at the end of the cycle (optional).

Feed — Feedrate (optional).

Tool# Active tool (optional).

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Programming Peck Drilling Cycle

Peck drilling is a modal operation. Once the CNC encounters a **PeckDrill** block, it peck drills at the end point of every subsequent block, until it sees a **Drilling Off** block. **PeckDrill** dimensions cannot be changed between moves, it must be shut off and a new one programmed in.

The cycle starts once the CNC is in position. The tool rapids to the **StartHgt**, Z feeds for the **Peck** then rapids back to the **StartHgt**. This cycle is repeated until the tool reaches **ZDepth**. At the end of the cycle, the tool rapids to **ReturnHgt** to provide clearance for the next move.

Refer to **Figure 4 - 27, Peck Drilling Cycle Graphic Menu**.

To program a Peck Drilling cycle:

1. With the CNC in the Edit mode, press **Drill** (F3) (drill cycle pop-up menu appears).
2. Highlight **Pecking** and press (ENTER) (**PECK DRILLING** Graphic Menu prompts for labeled values).
3. Key in labeled values and press (ENTER) (display clears, **PeckDrill** block is added to the program listing).

NOTE: After programming a **Pecking** cycle, moves must be programmed to tell the CNC where to drill. The CNC will execute the **Pecking** cycle at the end point of all subsequent blocks until it executes a **Drilling Off** block.

4. Program subsequent moves to position the work to where the drilling is to occur, the CNC will drill a hole at the end point of every move.
5. After programming the last drill move, press **Drill** (F3) (drill cycle pop-up menu appears).
6. Highlight **Drilling Off** and press (ENTER) (**DrillOff** block is included in program to take CNC out of the drilling mode).

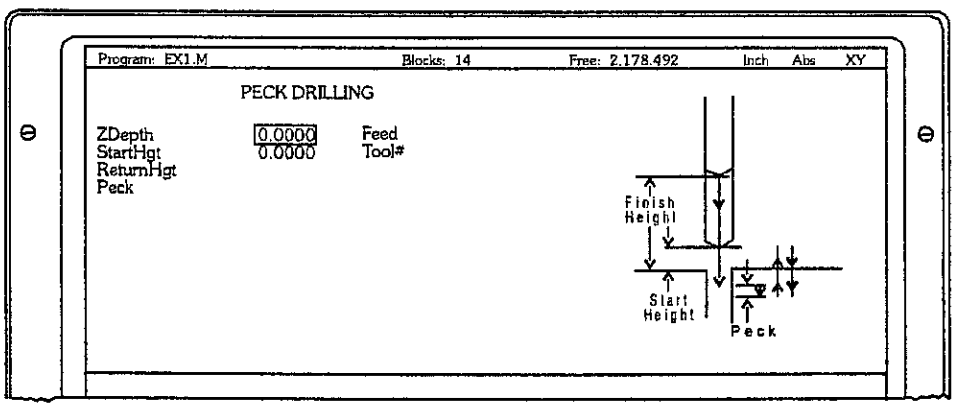


Figure 4 - 27, Peck Drilling Cycle Graphic Menu

PECK DRILLING entry fields are labled as follows:

ZDepth	The absolute depth of the finished hole (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be drilled.

ReturnHgt	The absolute position the tool returns to at the end of the cycle (optional).
Peck	The depth drilled in each peck (value required).
Tool#	Active tool (optional).

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Programming a Boring Cycle

Boring is a modal operation. Once the CNC encounters a **Boring** block, it executes a Boring Cycle at the end point of every subsequent move, until it sees a **Drilling Off** block. The Boring Cycle's dimensions cannot be changed between moves. The cycle must be shut off and a new one programmed in.

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

When a **Dwell** is entered, the CNC pauses at **ZDepth** for the time (number of seconds) keyed in. **Dwell** resolution is one tenth (0.1) of a second. If the operator keys in 0.0 seconds, the CNC will Dwell until manually restarted.

Refer to **Figure 4 - 28, Boring Cycle Graphic Menu**.

To program a Boring cycle:

1. With the CNC in the Edit mode, press **Drill** (F3) (drill cycle pop-up menu appears).
2. Highlight **Boring** and press (ENTER) (**BORING CYCLE** Graphic Menu prompts for labeled values).
3. Key in labeled values and press (ENTER) (display clears, **Boring** block is added to the program listing).

NOTE: After programming a **Boring** block, moves must be programmed to tell the CNC where to bore. The CNC will execute the **Boring** block at the end point of all subsequent blocks until it sees a **Drilling Off** block.

4. Program subsequent moves to position the work to where the holes are to be bored, the CNC will execute the Boring Cycle at the end point of every move.
5. After programming the last drill move, press **Drill** (F3) (drill cycle pop-up menu appears).
6. Highlight **Drilling Off** and press (ENTER) (**DrillOff** block is included in program to take CNC out of the drilling mode).

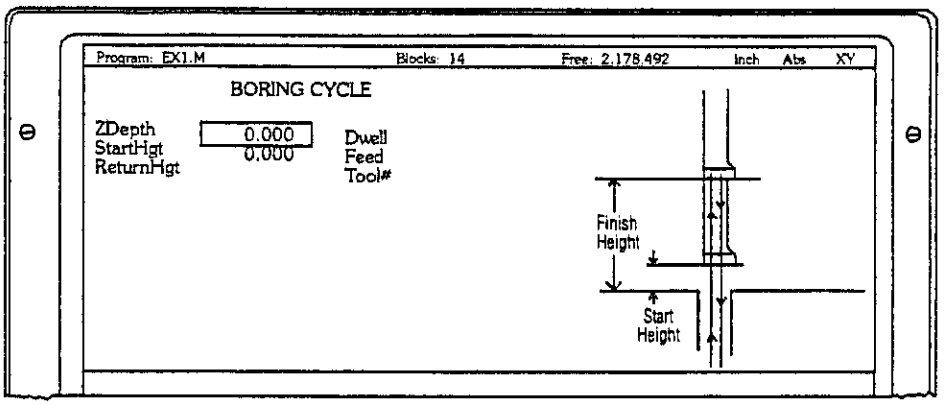


Figure 4 - 28, Boring Cycle Graphic Menu

BORING CYCLE entry fields are labled as follows:

ZDepth The absolute depth of the finished hole (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be bored.

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

Dwell Length of time for pause at **ZDepth**.

Feed Feedrate (optional).

Tool# Active tool (optional).

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Programming a Chip Break Cycle

The Chip Break Cycle is a modal operation. Once the CNC encounters a **ChipBreak** block, it executes the Chip Break Cycle at the end point of each block, until it sees a **Drilling Off** block. Chip Break values cannot be changed between moves. The cycle must be shut off and a new one programmed in.

The cycle starts once the CNC is in position. The tool rapids to the **StartHgt**, feeds to the **FirstPeck**, retracts .02in. (.4 mm) (default value) then feeds to the next peck. Retract moves occur at the end of each peck to break the chip. This cycle is repeated until the tool reaches **ZDepth**. At the end of the cycle the tool moves to **ReturnHgt** to provide clearance for the next move.

When a **PeckDecr** is entered, the CNC decrements the depth of each peck by that amount. The **MinPeck** sets the minimum peck the cycle can decrement to. A **ChipBrkInc** is the size of the retract move that breaks the chip.

When a retract depth (**RetractDep**) is used, the CNC pecks to the **RetracDep**, retracts to the **StartHgt**, then continues pecking to the next **RetracDep** increment. The first full retract occurs one **RetractDep** increment after the first peck.

Refer to **Figure 4 - 29, Chip Break Graphic Menu**.

To program a Chip Break cycle:

1. With the CNC in the Edit mode, press **Drill** (F3) (drill cycle pop-up menu appears).
2. Highlight **Chip Break** and press (ENTER) (**CHIP BREAKING CYCLE** Graphic Menu prompts for labeled values).
3. Key in labeled values and press (ENTER) (display clears, **Chip Break** block is added to the program listing).

NOTE: After programming a **ChipBreak** block, moves must be programmed to tell the CNC where to drill. The CNC will execute the **ChipBreak** block at the end point of all subsequent blocks until it sees a **Drilling Off** block.

4. Program subsequent moves to position the work to where the drilling is to occur the CNC will drill a hole at the end point of every move.
5. After programming the last drill move, press **Drill** (F3) (drill cycle pop-up menu appears).
6. Highlight **Drilling Off** and press (ENTER) (**DrillOff** block is included in program to take CNC out of the drilling mode).

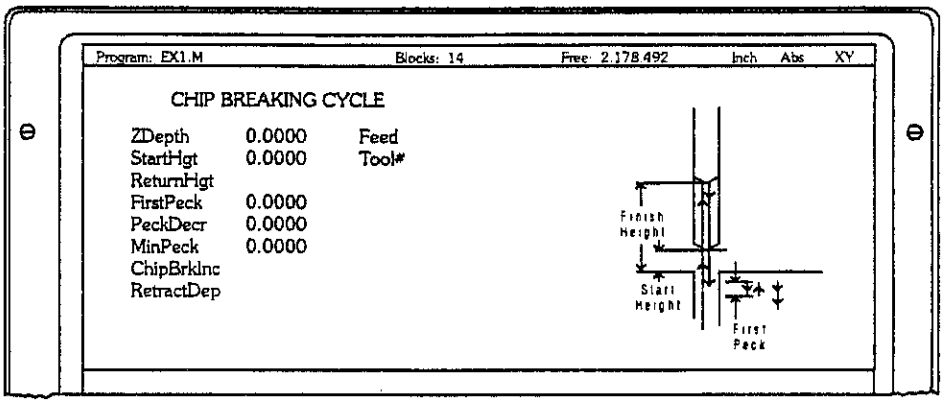


Figure 4 - 29, Chip Break Graphic Menu

CHIP BREAKING CYCLE entry fields are labled as follows:

ZDepth The absolute depth of the finished hole (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be drilled.

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

FirstPeck The absolute depth drilled in each peck (value required).

MinPeck Smallest peck allowed (value required).

ChipBrkInc Size of chip break retract (optional).

RetractDep Z increment before full retract (optional).

Tool# Active tool (optional).

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Programming a Tapping Cycle

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

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

When the cycle runs, the CNC rapids to the **StartHgt**, feeds to the **ZDepth**. The spindle then stops and reverses out to the **ReturnHgt**. At **ReturnHgt** the spindle stops and changes back to the original direction.

The Tapping Cycle can be used with all available patterns. A **Drilling Off** must be programmed to turn off the cycle when it is no longer needed.

NOTE: The 1100M CNC can support spindle FWD, REV, OFF (M3, M4, M5) and spindle RPM control. The machine builder determines which M-code options to install when the machine is setup.

Refer to **Figure 4 - 30, Tapping Cycle Graphic Menu**.

To program a **Tapping** block:

1. With the CNC in the Edit mode, press **Drill** (F3) (drill cycle pop-up menu appears).
2. Using the (ARROW) keys, position the highlight to select **Tapping** and press (ENTER) (CNC displays **TAPPING CYCLE** Graphic Menu).
3. Key in labeled values and press (ENTER) (display clears, new tapping cycle block is added).

NOTE: After programming a **Tapping** block, moves must be programmed to tell the CNC which holes to tap. The CNC will execute the **Tapping** block at the end point of all subsequent blocks until it sees a **Drilling Off** block.

4. Program subsequent moves to position the work to where the tapping is to occur, the CNC will tap a hole at the end point of every move.
5. After programming the last tapping move press **Drill** (F3) (drill cycle pop-up appears).
6. Highlight **Drilling Off** and press (ENTER) (**Drill Off** block is included in program to take CNC out of the tapping mode).

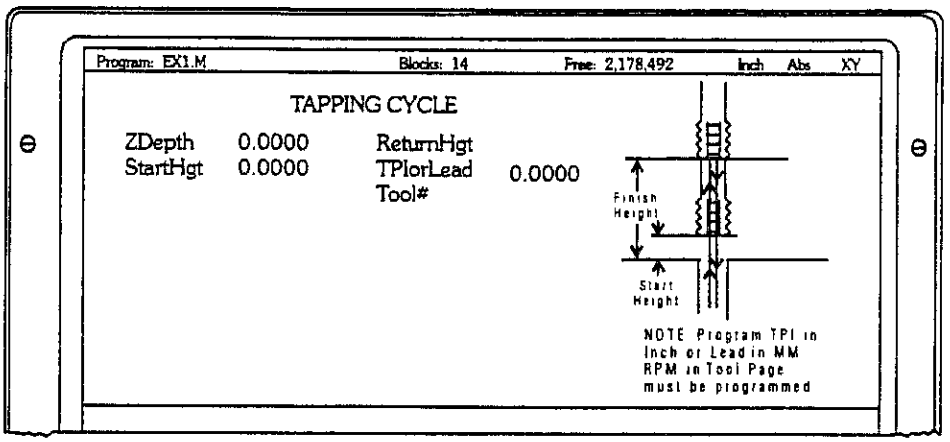


Figure 4 - 30, Tapping Cycle Graphic Menu

TAPPING CYCLE entry fields are labled as follows:

ZDepth — The absolute depth of the tapped threads (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be tapped.

ReturnHgt — The absolute position the tool returns to at the end of the cycle (optional).

TPIorLead — TPI in Inch or Lead in MM (value required).

Tool# — Active tool (optional).

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Programming a Drill Pattern

The Pattern Cycle instructs the CNC to execute a pattern of regularly spaced moves. A Pattern Cycle should be located between a Drill Cycle and its **DrillOff** block. When run, the Drill Cycle is executed at every end point in the pattern.

In a Pattern Cycle the operator keys in size, location, spacing, Tool # and rotation of the pattern.

Refer to **Figure 4 - 31, Pattern Drill Graphic Menu**.

To program a Drill Pattern cycle:

1. With the CNC in the Edit Mode, press **Drill** (F3) (drill cycle pop-up menu appears).
2. Select **Pattern** from the **Drill** pop up and press (ENTER) (pattern drill Graphic Menu prompts for labeled values).
3. Fill in the entry fields as labeled.

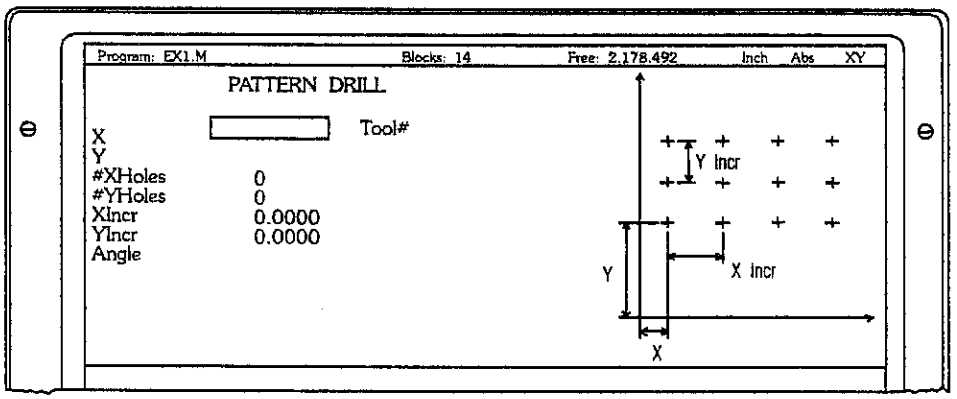


Figure 4 - 31, Pattern Drill Graphic Menu

PATTERN DRILL entry fields are labled as follows:

X — X coordinate of corner hole, no entry puts corner hole at present location (optional).

Y — Y coordinate of corner hole, no entry puts corner hole at present location (optional).

NOTE: It is recommended the desired absolute coordinates be entered when possible.

#XHoles — Number of rows that lie along the X axis, must enter value greater than 0 (value required).

#YHoles — Number of rows that lie along the Y axis, must enter value greater than 0 (value required).

NOTE: Entering a one in either the **#XHoles** or the **#YHoles** field 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, rotation angle is number of degrees counter clockwise from X axis or 3 o'clock position.
Tool#	Active tool (optional).

NOTE: A **Drilling Off** block should be programmed after this block if the drilling is complete.

Programming a Bolt Hole Pattern

The Bolt Hole Cycle instructs the CNC to execute a series of moves in a circular pattern. A Bolt Hole Cycle should be located between a Drill Cycle and its **DrillOff** block. When run, the Drill Cycle is executed at every endpoint in the pattern.

The size, location, spacing, tool # and rotation of the pattern are programmed by the operator.

Refer to **Figure 4 - 32, Bolt Hole Pattern Graphic Menu**.

To program a Bolt Hole cycle:

1. With the CNC in the Edit Mode, press **Drill** (F3) (drill cycle pop-up menu appears).
2. Select **Bolt Hole** from the pop up and press (ENTER) (bolt hole Graphic Menu prompts for labeled values).
3. Fill in the entry fields as labeled.

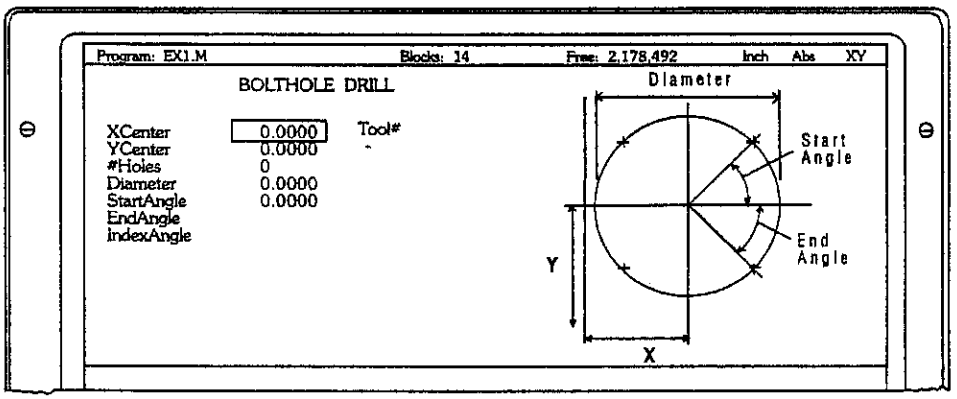


Figure 4 - 32, Bolt Hole Pattern Graphic Menu

BOLTHOLE DRILL entry fields are labeled as follows:

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).
#Holes	Number of equally spaced holes 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 last hole (optional).

IndexAngle	The number of degrees the 3 o'clock reference position is rotated around the center (rotates entire pattern).
Tool#	Active tool (optional).

NOTE: A **Drilling Off** block should be programmed after this block if the drilling is complete.

Canned Pocket Cycles

NOTE: All blocks are programmed by filling in the entry fields of a Graphic Menu. The process of entering values and advancing through entry fields is explained at the beginning of this section. This process is also illustrated in the descriptions for programming Rapid and Line moves. Step by step programming exercises are also included in **Section 12 - Sample Programs and Practice Exercises**.

Refer to **Figure 4 - 33, Pocket Cycle Pop-up Menu**. Pocket canned cycles simplify programming of the repetitive moves required to mill out pockets. Select the pocket canned cycles from the Program Editor's **Pocket** (F4) pop up menu.

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

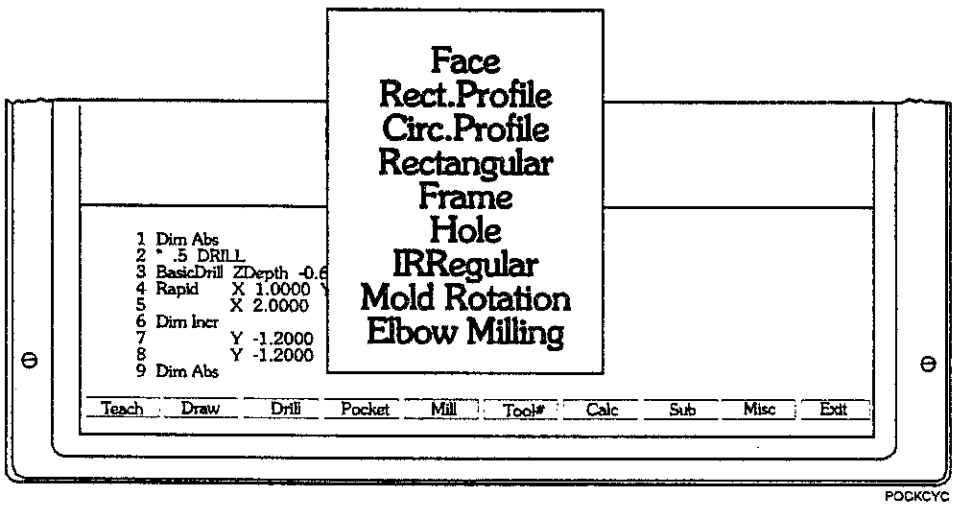


Figure 4 - 33, Pocket Cycle Pop-up Menu

Programming a Facing Cycle

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

Refer to **Figure 4 - 34, Face Cycle Tool Approach**. Execution begins one tool radius from the start point. The selected stepover determines the approach axes.

Face cycles can start in any corner of the surface and cut in any direction depending on the sign (+/-) of the **Length** and **Width** values. A slightly oversize **Length** and **Width** should be programmed to ensure complete facing of the surface.

At the end of the cycle, the tool rapids to **StartHgt**, then rapids back to the start position.

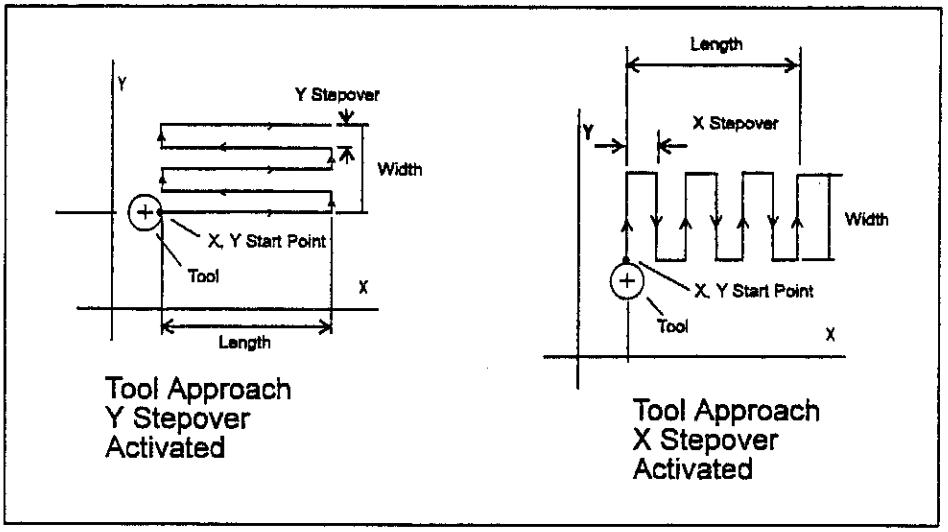


Figure 4 - 34, Face Cycle Tool Approach

Refer to **Figure 4 - 35, Face Pocket Cycle Graphic Menu**.

To program a Facing cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu appears).
2. Move the highlight to select **Face** and press (ENTER) (Graphic Menu prompts for labeled values).
3. Fill in the entry fields as labeled.

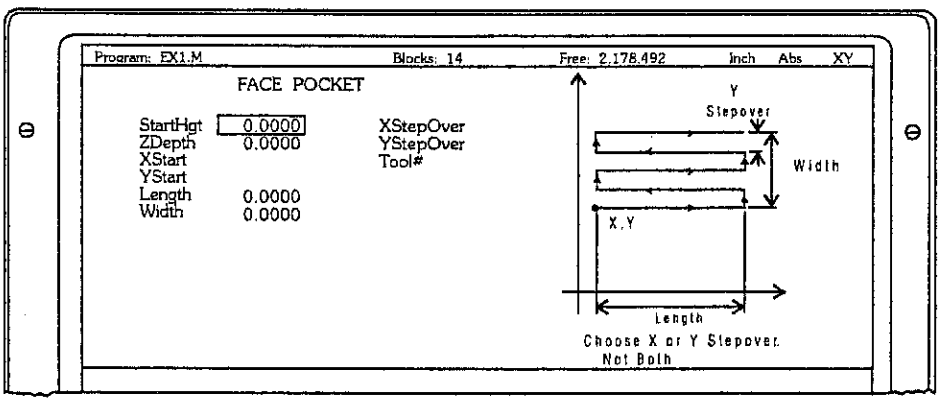


Figure 4 - 35, Face Pocket Cycle Graphic Menu

FACE POCKET entry fields are labled as follows:

ZDepth The absolute depth of the finished surface (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

XStart	X coordinate of the starting point (optional).
YStart	Y coordinate of the starting point (optional).

NOTE: It is recommended the desired starting point absolute coordinates be entered when possible.

Length	The X axis length to be faced (value required).
Width	The Y axis length to be faced (value required).
XStepover	Width of cut in the X axis direction. If no value is entered CNC defaults to 70% of the active tool radius. Maximum stepover permitted is 70% of the active tool radius.
YStepover	Width of cut in the Y axis direction. If no value is entered CNC defaults to 70% of the active tool radius. Maximum stepover permitted is 70% of the active tool radius.

NOTE: Enter either X or Y stepover only. Do not enter both.

NOTE: The program editor will allow the operator to write a block containing a **Stepover** value greater than 70% of the active tool radius. Testing a program in the Draw Graphics Mode will reveal this type of error.

Tool#	Active tool (optional).

Programming a Rectangular Profile Cycle

The Rectangular Profile Cycle is provided for cleaning up the inside or outside profile of a rectangle. When executed, the CNC rapids to the Ramp #1 starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.

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

Refer to **Figure 4 - 36, Rectangular Profile Graphic Menu**.

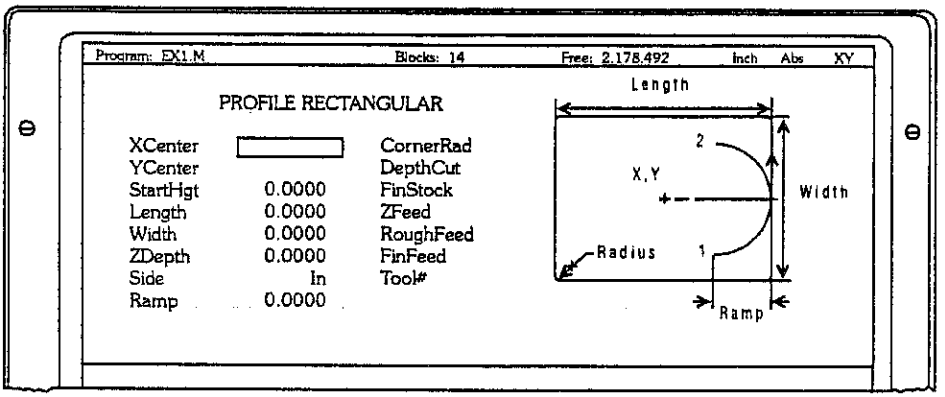


Figure 4 - 36, Rectangular Profile Graphic Menu

Refer to **Figure 4 - 37, Outside Profile Ramp Moves**. When cutting an inside profile the ramps are as shown on the Graphic Menu. When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2 as shown in the figure.

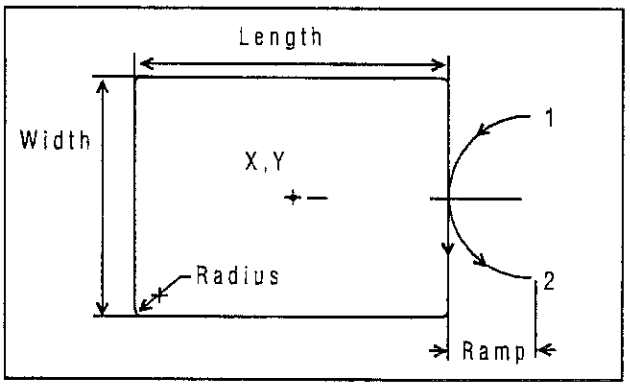


Figure 4 - 37, Outside Profile Ramp Moves

The Rectangular Profile Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated before or within the **ProfRect** block.

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

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified stock on the profile and depth for a finish pass. The rectangle is cut to the **Length**, **Width** and **ZDepth** dimensions on the finish pass. If a negative **FinStock** value is keyed in, the CNC will leave the finish stock and not make a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

To program a Rectangular Profile cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (**Pocket** pop-up menu appears).
2. Using the (ARROW) keys, position the highlight to select **Rect.Profile** and press (ENTER) (**PROFILE RECTANGULAR** Graphic Menu prompts for labeled values).
3. Fill in the **PROFILE RECTANGULAR** entry fields labeled as follows:

XCenter — X coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

YCenter — Y coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

Length — Finished length of rectangle (value required).

Width — Finished width of rectangle (value required).

ZDepth — The absolute depth of the finished profile (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

Side — Setting for cutting on the inside of the profile (In) or the outside (Out). This setting is toggled between selections using the (+/-) key (selection required).

Ramp — Radius of the ramping moves (value required).

CornerRad	Corner radius setting. If a negative value is used, both the direction of cut and the starting and end points are reversed (optional).
DepthCut	Z axis increment used for each pass (optional).
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass (optional).
ZFeed	Z axis feedrate (optional).
RoughFeed	Rough pass feedrate (optional).
FinFeed	Finish pass feedrate (optional).
Tool#	Active tool (optional).

Programming a Circular Profile Cycle

The Circular Profile Cycle can be used for cleaning up the inside or outside profile of an existing circle.

Refer to **Figure 4 - 38, Circular Profile Graphic Menu**. When executed the CNC rapids to Ramp#1 starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.

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

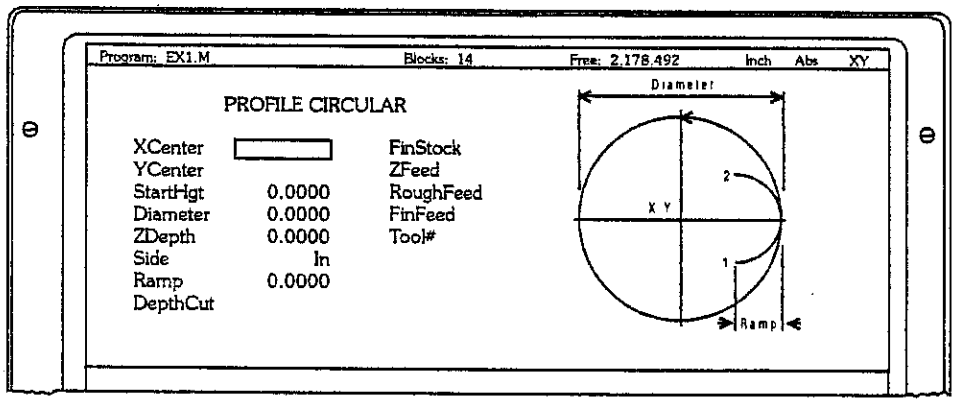


Figure 4 - 38, Circular Profile Graphic Menu

Refer to **Figure 4 - 39, Ramp Position for Outside Profile**. When cutting an outside profile, the tool ramps into the work along Ramp #1 and away from the work along Ramp #2 as shown.

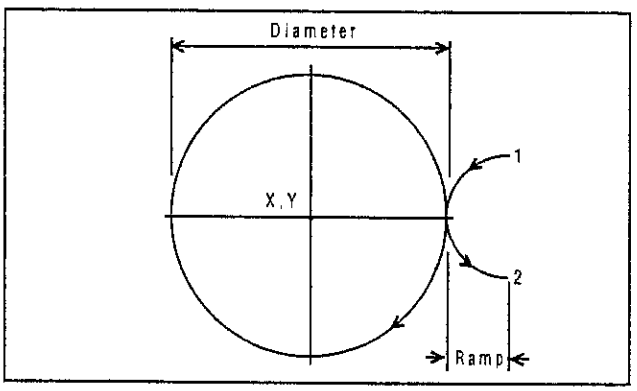


Figure 4 - 39, Ramp Position for Outside Profile

The Circular Profile Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated before or within the **ProfCirc** block.

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

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified stock on the profile and depth for a finish pass. The circle is cut to the **Diameter** dimension on the finish pass. If a negative **FinStock** value is keyed in, the CNC will leave the finish stock and not make a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

To program a Circular Profile cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (**Pocket** pop-up menu appears).
2. Refer to **Figure 4 - 38, Circular Profile Graphic Menu**. Using the (ARROW) keys, position the highlight to select **Circ.Profile** and press (ENTER) (**PROFILE CIRCULAR** Graphic Menu prompts for labeled values).
3. Fill in the **PROFILE CIRCULAR** entry fields labeled as follows:

XCenter — X coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

YCenter — Y coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

Diameter — Finished diameter of circle. If a negative value is used, both the direction of cut and the starting and end points are reversed (value required).

ZDepth — The absolute depth of the finished profile (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

Side — Setting for cutting on the inside of the profile (In) or the outside (Out). This setting is toggled between selections using the (+/-) key (selection required).

Ramp	Radius of the ramp into and away from the cut, may be set to zero (value required).
DepthCut	Z axis increment used for each pass (optional).

FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass (optional).
ZFeed	Z axis feedrate (optional).
RoughFeed	Rough pass feedrate (optional).
FinFeed	Finish pass feedrate (optional).
Tool#	Active tool (optional).

Programming a Rectangular Pocket Cycle

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

The Rectangular Pocket Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated before or within the **RectPock** block.

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

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified stock on the profile and depth for a finish pass. The rectangle is cut to the **Length**, **Width** and **ZDepth** dimensions on the finish pass. If a negative **FinStock** value is keyed in, CNC will leave the finish stock and not add a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

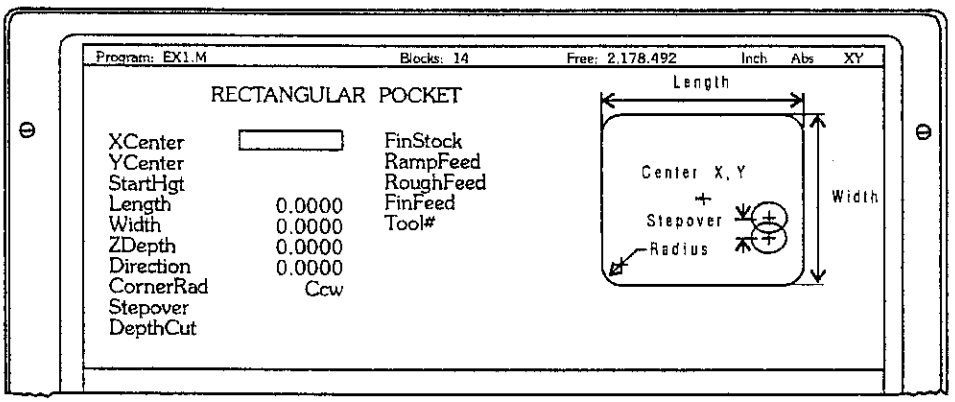


Figure 4 - 40, Rectangular Pocket Graphic Menu

Refer to **Figure 4 - 40, Rectangular Pocket Graphic Menu**.

To program a Rectangular Pocket cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu appears).
2. Move the highlight to select **Rectangular** and press (ENTER) (**RECTANGULAR POCKET** Graphic Menu prompts for labeled values)
3. Fill in the **RECTANGULAR POCKET** entry fields, labeled as follows:

XCenter — X coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

YCenter — Y coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

NOTE: It is recommended the desired absolute center coordinates be entered when possible.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

Length — The inside Y length, of the finished pocket (value required).

Width — The inside X length, of the finished pocket (value required).

ZDepth — The absolute depth of the finished pocket (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

Direction — This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-) key (selection required).

Stepover — Width of cut, if no value is entered CNC defaults to 70% of the active tool radius. Maximum stepover permitted is 70% of the active tool diameter (optional).

NOTE: The program editor will allow the operator to write a block containing a **Stepover** value greater than 70% of the active tool radius. Testing a program in the Draw Graphics Mode will reveal this type of error.

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 keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass (optional).
ZFeed	Z axis feedrate (optional).
RoughFeed	Rough pass feedrate (optional).
FinFeed	Finish pass feedrate (optional).
Tool#	Active tool (optional).

Programming a Circular Pocket Cycle

Circular Pocket cycles simplify the programming required to mill out a circular pocket. When executed, the CNC rapids to the center, rapids to the **StartHgt**, then ramps into the work while circling outward. Once the tool reaches the pocket **Diameter**, the tool circles back toward the center until the pass is complete.

The Circular Pocket Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated before or within the **CircPock** block.

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

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified stock on the profile and depth for a finish pass. The profile is cut to the **Diameter** and **ZDepth** dimensions on the finish pass. If a negative **FinStock** value is keyed in, CNC will leave the finish stock and not add a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

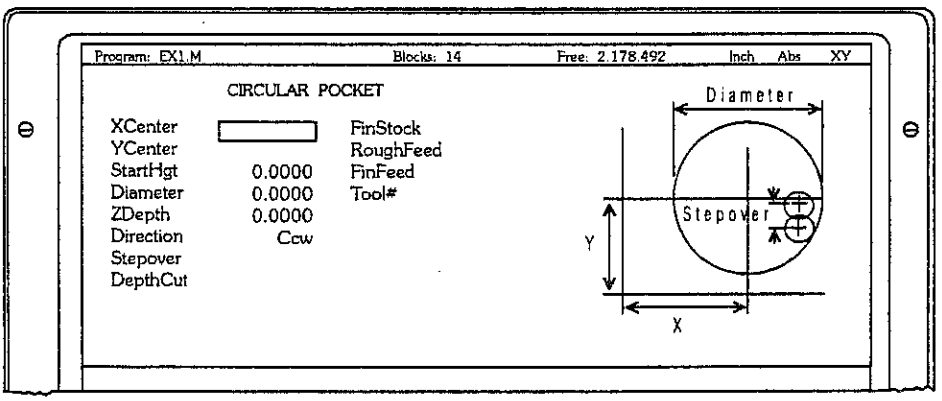


Figure 4 - 41, Circular Pocket Cycle Graphic Menu

Refer to **Figure 4 - 41, Circular Pocket Cycle**.

To program a Circular Pocket cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu appears).
2. Move the highlight to select **Circular** and press (ENTER) (**CIRCULAR POCKET** Graphic Menu prompts for labeled values).
3. Fill in the **CIRCULAR POCKET** entry fields, labeled as follows:

XCenter — X coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

YCenter — Y coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

NOTE: It is recommended the desired absolute center coordinates be entered when possible.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

ZDepth — The absolute depth of the finished hole (value required).

NOTE: **ZDepth** must be lower than **StartHgt**.

Direction — This setting allows the operator to choose between a clockwise (**Cw**) or counter clockwise (**Ccw**) direction. The setting is toggled by using the (+/-) key (selection required).

Stepover — Width of cut, if no value is entered, CNC defaults to 70% of the active tool radius. Maximum stepover permitted is 70% of the active tool diameter (optional).

NOTE: The program editor will allow the operator to write a block containing a **Stepover** value greater than 70% of the active tool radius. Testing a program in the Draw Graphics Mode will reveal this type of error.

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 keyed in. If a negative value is keyed in, 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).

Programming a Frame Pocket Cycle

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

The Frame Pocket Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated before or within the **FramePock** block.

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

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified stock on the profiles and depth for a finish pass. The frame is cut to the **IslandLen**, **IslandWid** and **FrameWidth** dimensions on the finish pass. If a negative **FinStock** value is keyed in, CNC will leave the finish stock and not add a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

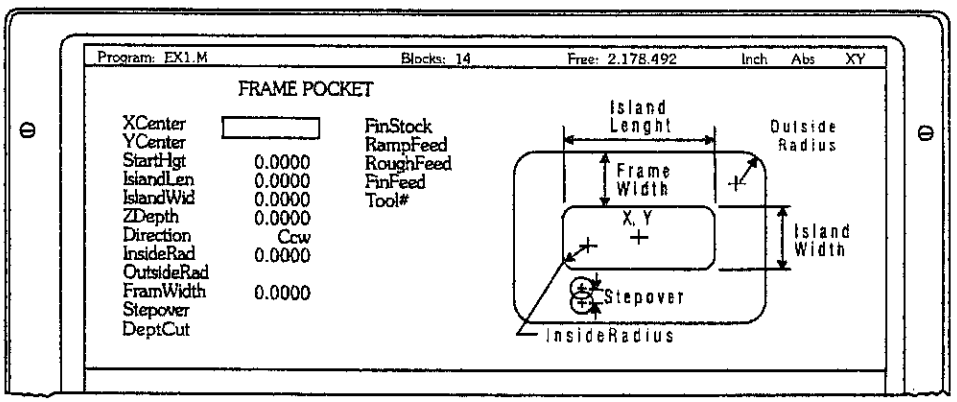


Figure 4 - 42, Frame Pocket Cycle Graphic Menu

Refer to **Figure 4 - 42, Frame Pocket Cycle Graphic Menu**.

To program a Frame Pocket cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu appears).
2. Move highlight to select **Frame** and press (ENTER) (**FRAME POCKET** Graphic Menu prompts for labeled values).
3. Fill in the **FRAME POCKET** entry fields, labeled as follows:

XCenter — X coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

YCenter — Y coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

NOTE: It is recommended desired absolute center coordinates be entered when possible.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

IslandLen — The outside length, X axis, of finished island (value required).

IslandWid — The outside width, Y axis, of finished island (value required).

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

NOTE: **ZDepth** must be lower than **StartHgt**.

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

Direction — This setting allows the operator to choose between conventional and climb milling the pocket The choices are clockwise (**Cw**) and counter clockwise (**Ccw**). The setting is toggled between the two using the (+/-) key (selection required).

InsideRad — Radius of the island corners (value required).

OutsideRad — Outside radius of the frame corners (value required).

FrameWidth	Width of finished frame (value required).
Stepover	Width of cut, if no value is entered, CNC defaults to 70% of the active tool radius. Maximum stepover permitted is 70% of the active tool diameter (optional).

NOTE: The program editor will allow the operator to write a block containing a **Stepover** value greater than 70% of the active tool radius. Testing a program in the Draw Graphics Mode will reveal this type of error.

DepthCut	Z axis increment used for each pass (optional).
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass (optional).
RampFeed	Z axis feedrate (optional).
RoughFeed	Rough pass feedrate (optional).
FinFeed	Finish pass feedrate (optional).
Tool#	Active Tool (optional).

Programming a Hole - Mill Cycle

Hole Mill cycles can be used to cut through holes, clean up the inside diameter of existing holes or to counter bore existing holes. When executed, the CNC rapids to the ramp, feeds into the circumference along the ramp and cuts the **Diameter**. When the hole is finished, the CNC ramps away from the circumference and rapids back to the center.

The Hole Mill Pocket Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated before or within the **HolePock** block.

StartHgt and **ZDepth** are optional, but must be used together if used. If **DepthCut** is keyed in, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified amount on the profile and makes an additional pass cutting to the **Diameter**. If a negative **FinStock** value is keyed in, CNC will leave the finish stock, but not add a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

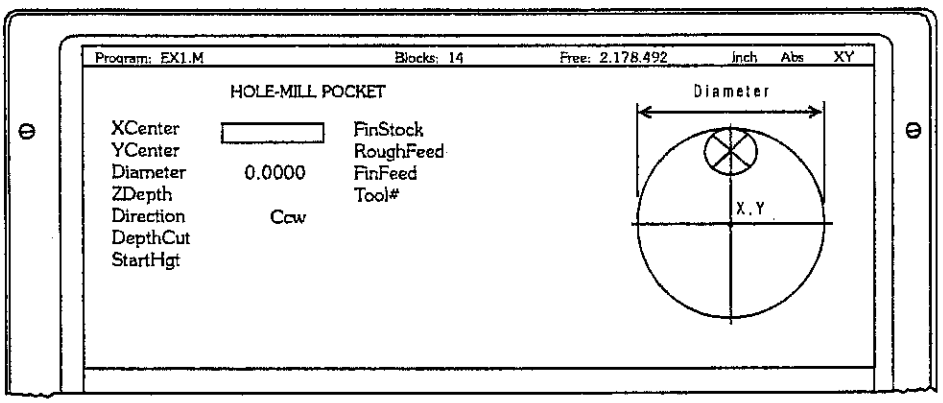


Figure 4 - 43, Hole - Mill Pocket Graphic Menu

Refer to **Figure 4 - 43, Hole - Mill Pocket Graphic Menu**.

To program a Hole Mill Pocket cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop-up menu Graphic Menus).
2. Move highlight to select **Hole** and press (ENTER) (**HOLE-MILL POCKET** Graphic Menu prompts for labeled values).
3. Fill in the **HOLE-MILL POCKET** entry fields labeled as follows:

XCenter — X coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

YCenter — Y coordinate of the center. If no coordinate is entered, pocket is centered at present position (optional).

NOTE: It is recommended that the desired absolute center coordinates be entered when possible.

Diameter — Diameter of the pocket (value required).

ZDepth — The Absolute depth the finished pocket (optional).

NOTE: **ZDepth** and **StartHgt** must be used together if used.

NOTE: **ZDepth** must be lower than **StartHgt**.

Direction — This setting allows the operator to choose between a clockwise (Cw) or counter clockwise (Ccw) direction. The setting is toggled by using the (+/-) key (selection required).

DepthCut — Z axis increment used for each pass (optional).

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

FinStock — Amount of stock left by the machine before the finish pass. Zero is assumed if no value is keyed in. If a negative value is keyed in, 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).

Programming an Irregular Pocket Cycle

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

The subprogram should begin with a Rapid move to an absolute starting position on the profile, followed by the moves and Arcs necessary to define the outline and return to the starting position.

NOTE: The subprogram to define the outline can be tested by temporarily programming a subprogram call in the beginning of the main program and running the program in the Simulation Draw Mode. Remember to delete or comment out the subprogram call before actually using the program.

Refer to **Figure 4 - 44, Stepover Move Orientation**. The pocket's interior is milled out by consecutive side to side moves that stepover across the pocket. The machine reverses direction when it reaches any portion of the pocket outline. The moves must be oriented so an irregularity in the outline does not prevent the entire pocket from being milled out.

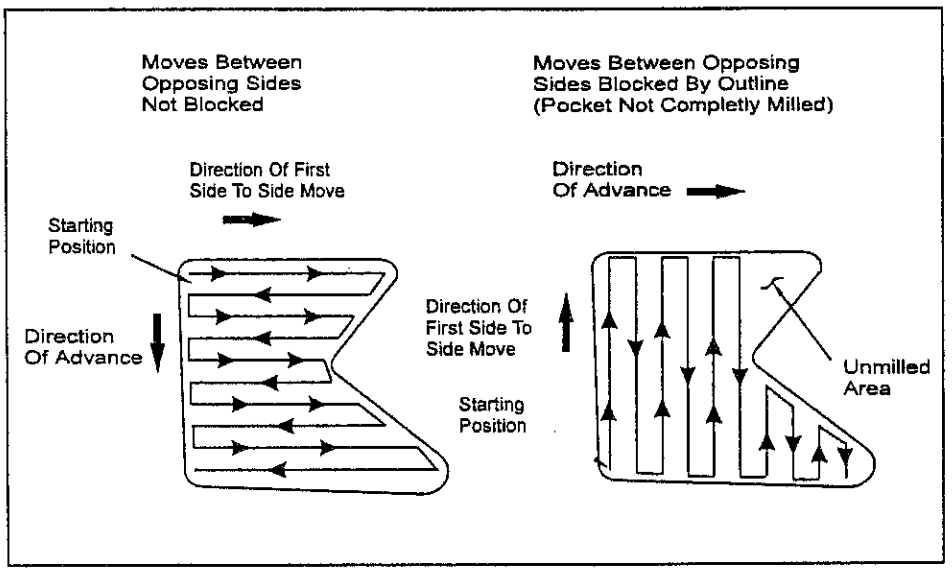


Figure 4 - 44, Stepover Move Orientation

Determining Move Direction

Two factors determine how moves stepover 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 direction of the first move.

When the **Angle**, **X** and **Y** fields are left blank and the first move of the subprogram is a Line move, the default starting position is the intersection of the first and last feed moves in the subprogram (compensated for tool diameter and **Finstock**). 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 default starting position is 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 line drawn through the Arc's endpoint.

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

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

Refer to **Figure 4 - 45, Orienting Stepover From the Subprogram**. Many irregular pockets can be milled by planning the subprogram and leaving the **Angle**, **XStart** and **YStart** fields blank.

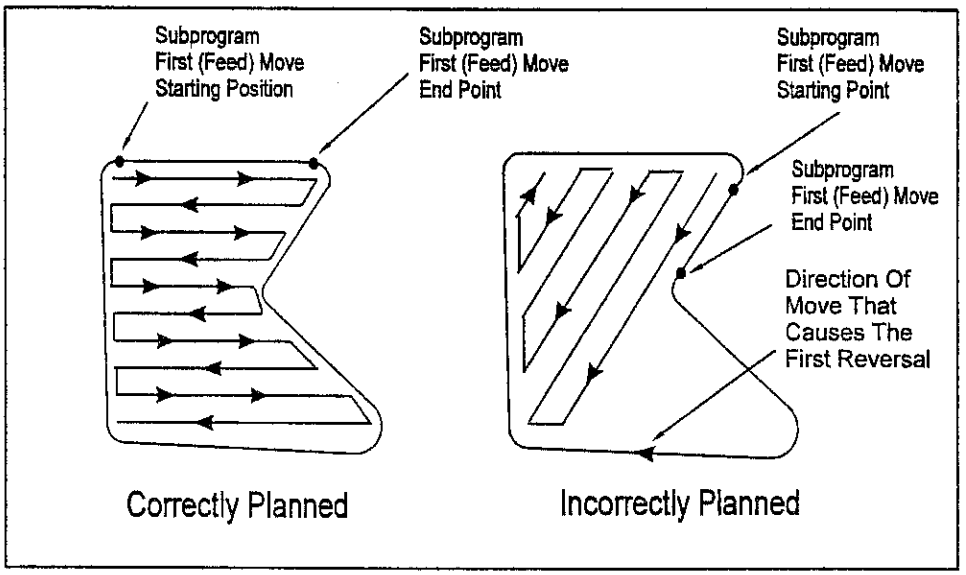


Figure 4 - 45, Orienting Stepover From the Subprogram

Refer to **Figure 4 - 46, Specifying Move Angle**. Keying in an **Angle** forces the direction of the first move to the absolute angle keyed in. The angle must point to a direction inside the pocket for the cycle to run.

NOTE: When the starting position of the cycle is defined by an Arc in the subprogram and an **Angle** value is specified, be sure the angle points to a direction inside the pocket. An angle tangent to the starting point of an Arc does not point to the inside of the Arc. Adjusting the angle inside the pocket by only 0.01 degrees is sufficient to run the cycle.

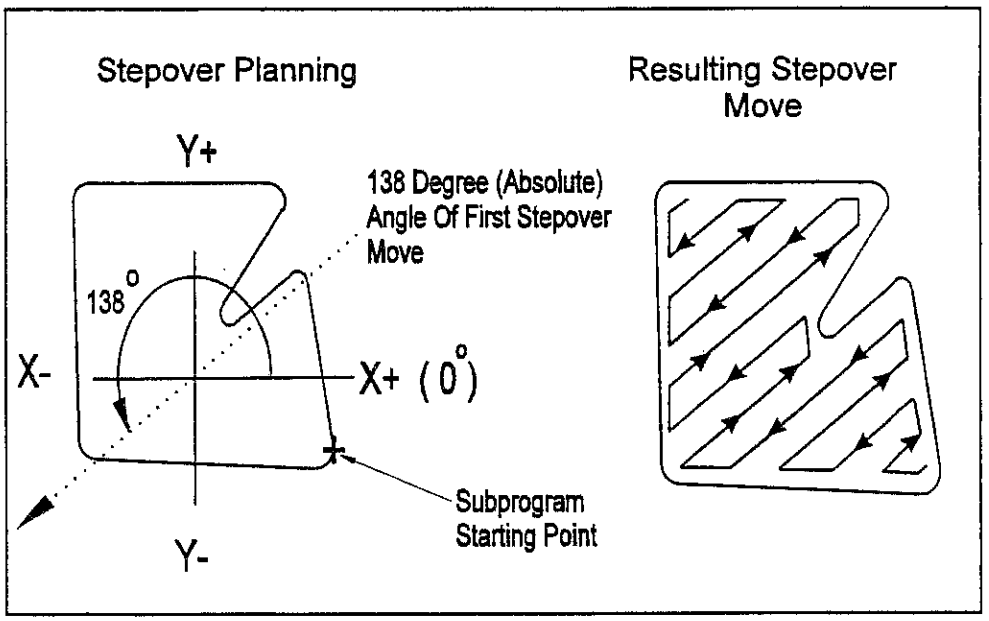


Figure 4 - 46, Specifying Move Angle

Keying in **X** and **Y** values forces the starting position of the first move to the position keyed in. This position should be compensated for tool diameter and **FinStock**.

IRRegular Pocket Execution

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

The Irregular Profile Cycle automatically compensates the tool's diameter. The correct tool diameter must be activated 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 stepover move starting position.

The CNC mills out the pocket with side to side moves that stepover across the pocket.

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

NOTE: The program editor will allow the operator to write a block even if the positions keyed in are illogical. The control will stop a program and generate a message at an illogical block for safety. Test programs with Draw to find any illogical blocks.

When **FinStock** is used, the CNC leaves the specified stock on the profile and depth for a finish pass. The pocket is cut to the finished dimensions on the finish pass. If a negative **FinStock** value is keyed in, CNC will leave the finish stock and not add a finish pass.

If **RoughFeed** and **FinFeed** are left blank, the CNC executes feed moves at the current feedrate. If a **RoughFeed** is keyed in, only the rough feed is affected. When a **FinFeed** rate is keyed in, only the feedrate of the finish pass is affected.

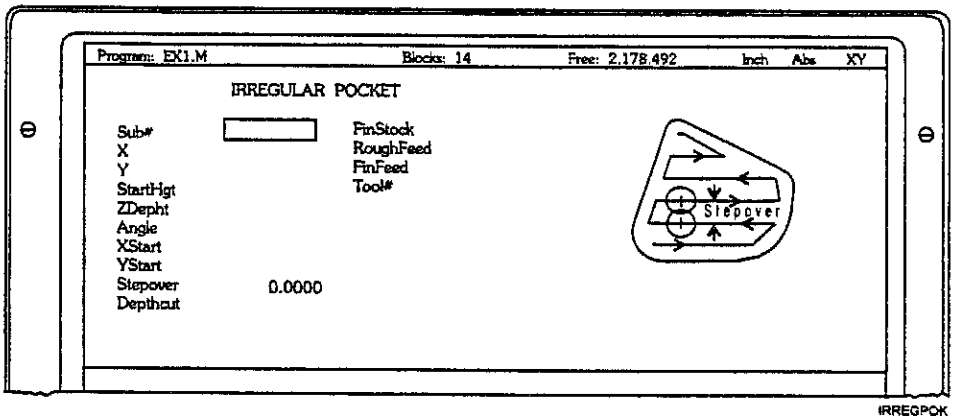


Figure 4 - 47 , Irregular Pocket Graphic Menu

Refer to **Figure 4 - 47 , Irregular Pocket Graphic Menu**.

To program an Irregular Pocket cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu Graphic Menus).
2. Using the (ARROW) keys move highlight to select **IRRegular** and press (ENTER) (**IRREGULAR POCKET** Graphic Menu prompts for labeled values).
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 stepover 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 stepover 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 used.

NOTE: It is recommended the desired absolute center coordinates be entered when possible.

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

NOTE: **StartHgt** is 0.100 inch (2.0 mm) above the surface to be cut.

ZDepth	The Absolute depth of the finished pocket (optional).
NOTE: **ZDepth** must be lower than **StartHgt**.	
Angle	Absolute angle of the first move, if left blank first move of stepover 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: **XStart** and **YStart** values must be used together, if used.	
Stepover	Width of cut (value required).
DepthCut	Z axis increment used for each pass (optional).
FinStock	Amount of stock left by the machine before the finish pass. Zero is assumed if no value is keyed in. If a negative value is keyed in, the CNC will leave the stock, but not make a finish pass (optional).
RampFeed	Z axis feedrate (optional).
RoughFeed	Rough pass feedrate (optional).
FinFeed	Finish pass feedrate (optional).
Tool#	Active tool (optional).

Mold Cycles

NOTE: All blocks are programmed by filling in the entry fields of a Graphic Menu. The process of entering values and advancing through entry fields is explained at the beginning of this section. This process is also illustrated in the descriptions for programming Rapid and Line moves. Step by step programming exercises are also included in **Section 12 - Sample Programs and Practice Exercises**.

Programming a Mold Rotation

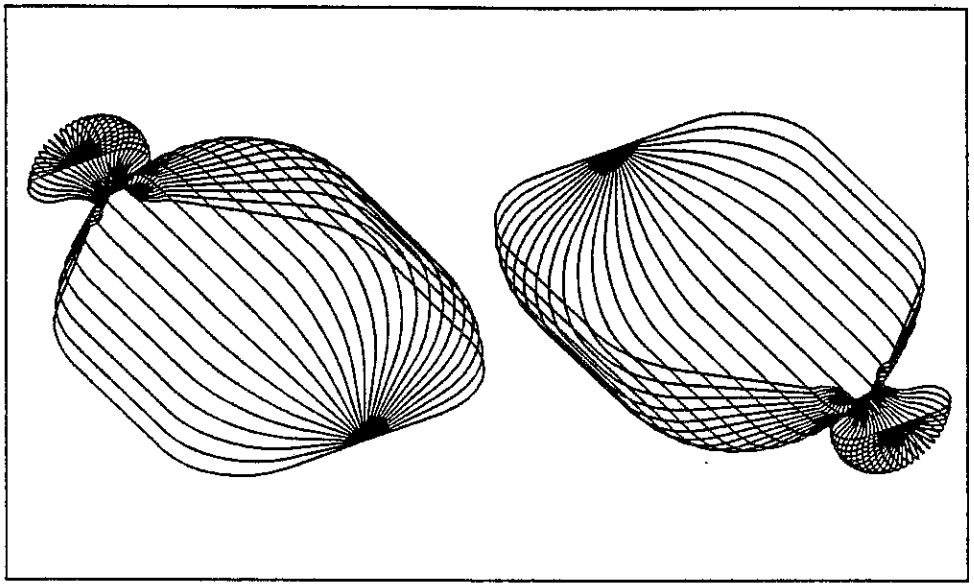


Figure 4 - 48, XY Axis Mold Rotations

Refer to **Figure 4 - 48, XY Axis Mold Rotations**. A Mold Rotation is used to mill cylindrically symmetric cavities and cores. A cylindrically symmetric shape is a shape defined by rotating a profile around an axis. There are two categories of mold rotation:

- Rotation around X or Y axes
- Rotations around the Z axis

Rotations Around X and Y Axes (Small Radius)

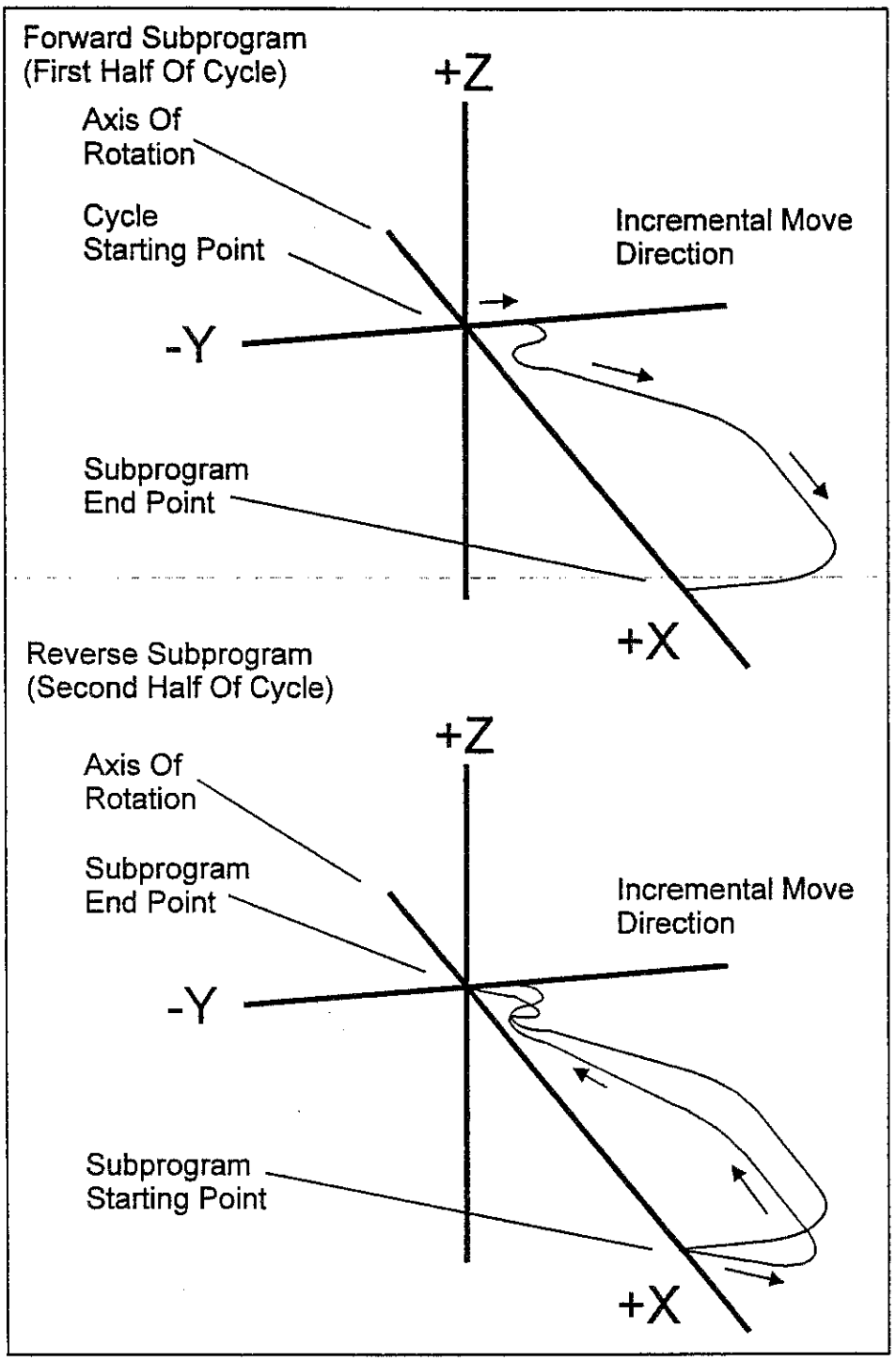


Figure 4 - 49, Subprogram Orientation

Refer to **Figure 4 - 49, Subprogram Orientation**. Each Mold Rotation block requires two subprograms: a forward subprogram (**FwdSub**) to define the profile moving away from the starting point and a reverse subprogram (**RevSub**) to define the profile moving back to the starting point.

In one cycle, the CNC executes the forward subprogram to the profile endpoint, then executes the reverse subprogram back to the starting point. Each cycle is incremented around the axis of rotation from the **StartAngle** to the **EndAngle**. The amount of rotation in each increment is determined from the number of **Cycles** and the size of the start and end angle values programmed into the block.

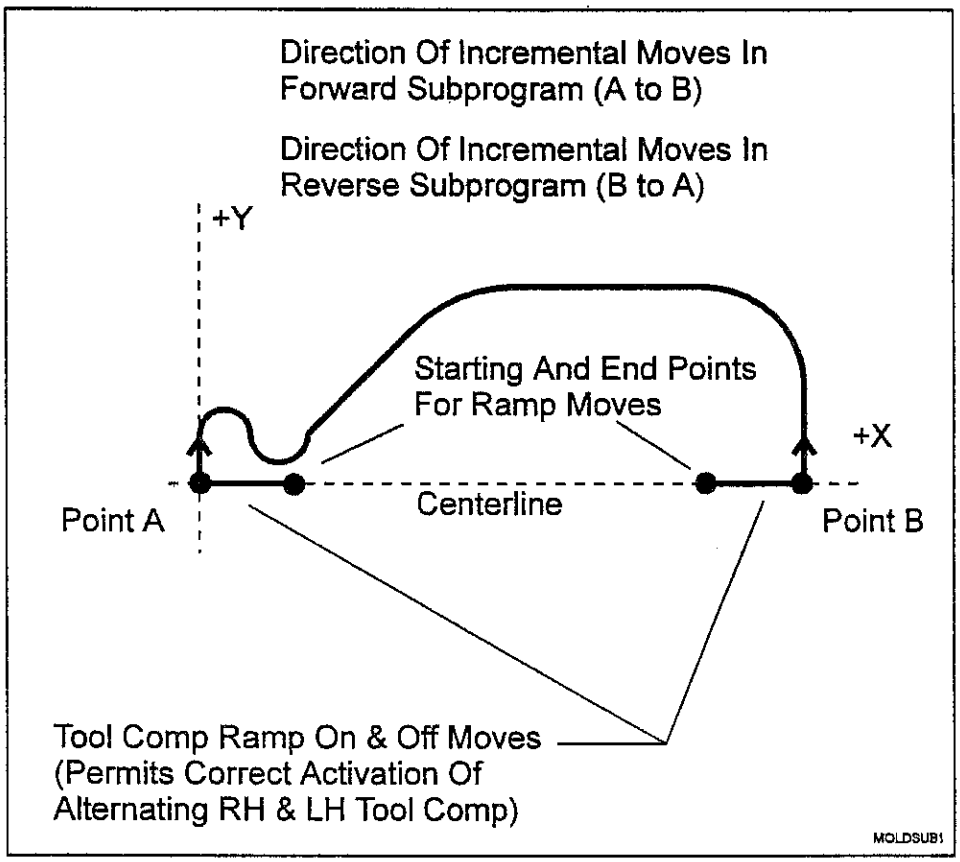


Figure 4 - 50, Subprogram Specifics

Refer to **Figure 4 - 50, Subprogram Specifics**. The cycle starts cutting the first subprogram (profile) from the machine's present position.

In small radius rotations, subprogram start and endpoints can lie on the centerline of rotation. All subprogram moves must be incremental. Both subprograms must produce profiles that are exactly the same shape, but that execute in opposite directions.

When the rotation is around the X axis, all of the moves in the subprogram must be contained in the +Y half of the XY plane. When the

rotation is around the Y axis, all of the moves in the subprogram must be contained in the +X half of the XY plane.

If tool compensation is not used, the path in the subprogram will have to be adjusted for the radius of the tool. If cutting a core, the path will have to be increased by one tool radius. If cutting a cavity, the path will have to be reduced by one tool radius.

Tool compensation can be used by writing compensated moves in the subprograms. Tool compensation for each subprogram must be on opposite sides.

Each activation of tool compensation requires a ramp on/off move for position adjustment. Ramp moves can be added to the subprograms as shown.

Refer to **Section 1 - 1100M Programming Concepts** for information on using tool length offset and diameter compensation with ball-end mills.

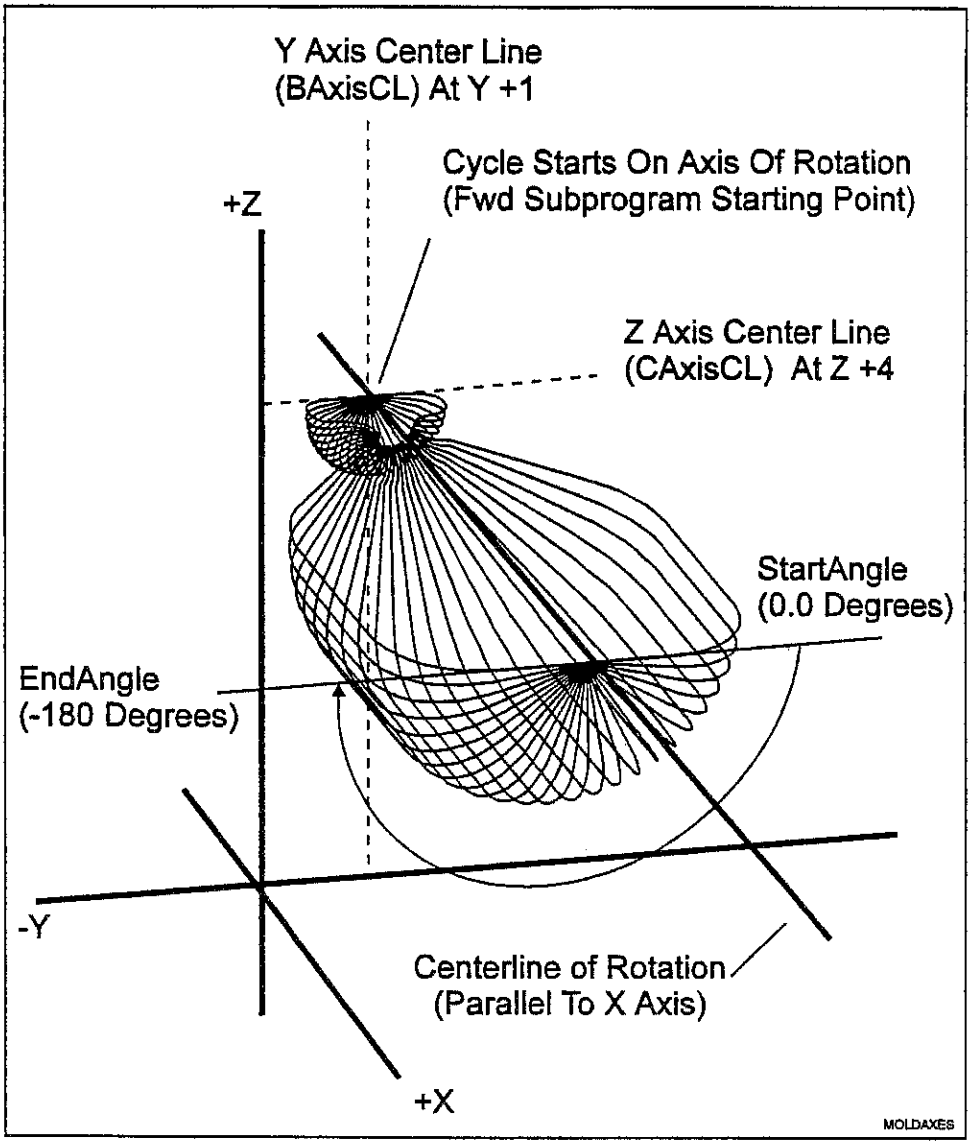


Figure 4 - 51, Axes Of Rotation

Refer to **Figure 4 - 51, Axes Of Rotation**. The rotation occurs around the axis selected in the **AxisRot** field (centerline of rotation). Use **CAxisCL** and **BAxisCL** values to define the position of the centerline when it is not located at the axis zero (X0, Y0, Z0).

When the rotation is around the X axis, the centerline is defined by the Y axis position (in the **BAxisCL** field) and the Z axis position (in the **CAxisCL** field)

When the rotation is around the Y axis, the centerline is defined by the X axis position (in the **BAxisCL** field) and the Z axis position (in the **CAxisCL** field).

The **StartAngle** and the **EndAngle** are absolute start and stop angles of the rotation. Use negative angle values to cut cavities and positive angle values to cut cores.

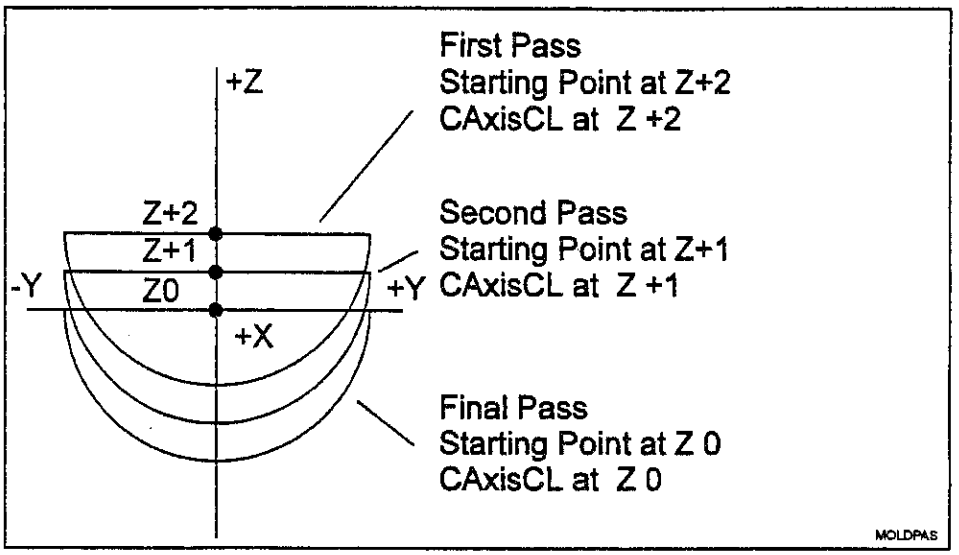


Figure 4 - 52, Cutting a Cavity Using More Than One Pass

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

Refer to **Figure 4 - 52, Cutting a Cavity Using More Than One Pass**. The starting position and **CAxisCL** value of each block should be varied to bring each pass closer finished shape. Reverse the sequence shown in the figure to cut a core.

Refer to **Figure 4 - 53, Rotating XY Mold Rotations Around Z**. X and Y axis rotations can be rotated around the Z axis by entering a **ZAngle**.

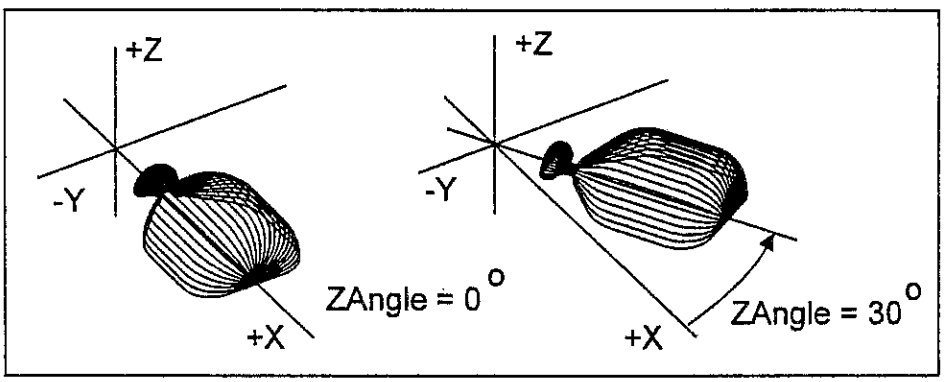


Figure 4 - 53, Rotating XY Mold Rotations Around Z

Rotations Around X and Y Axes (Large Radius)

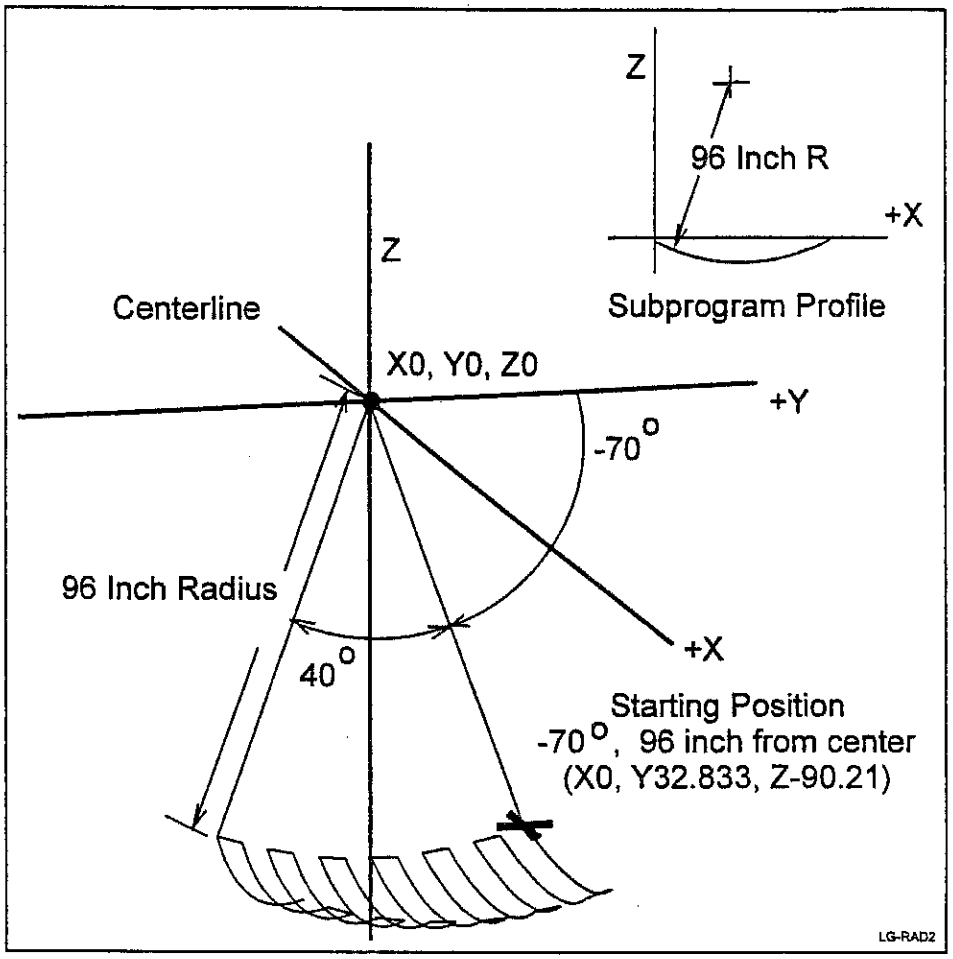


Figure 4 - 54, Large Radius Mold Rotation

Refer to **Figure 4 - 54, Large Radius Mold Rotation**. The mold rotation cycle starts executing the subprograms at the machine's present position. To cut a large radius rotation, start the cycle at a the required distance from the centerline. The additional distance is automatically added to the radius of the rotation.

If the rotation does not swing a full 180°, the starting position will be a point on the starting angle the required distance from the centerline. This positioned can be easily determined using the Geometry Calculator, refer to **Section 10 - The Calculator**.

Rotation Around the Z Axis

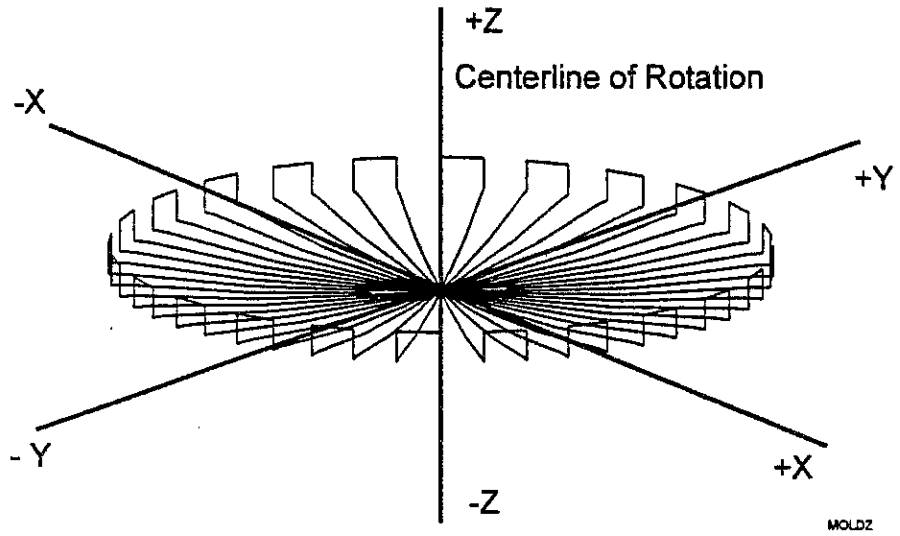


Figure 4 - 55, Z Axis Mold Rotation

Refer to **Figure 4 - 55, Z Axis Mold Rotation**. The centerline of rotation is parallel to the Z axis (**AxisRot**). The **BAxisCL** and **CAxisCL** values are the X and Y coordinates of the centerline. Enter the X coordinate in the **BAxisCL** field and the Y coordinate in the **CAxisCL** field.

The machine must be positioned at the center of the rotation when the cycle starts. The finished shape will be centered on the starting position. The Z axis position of the starting point determines the Z axis position of the finished shape.

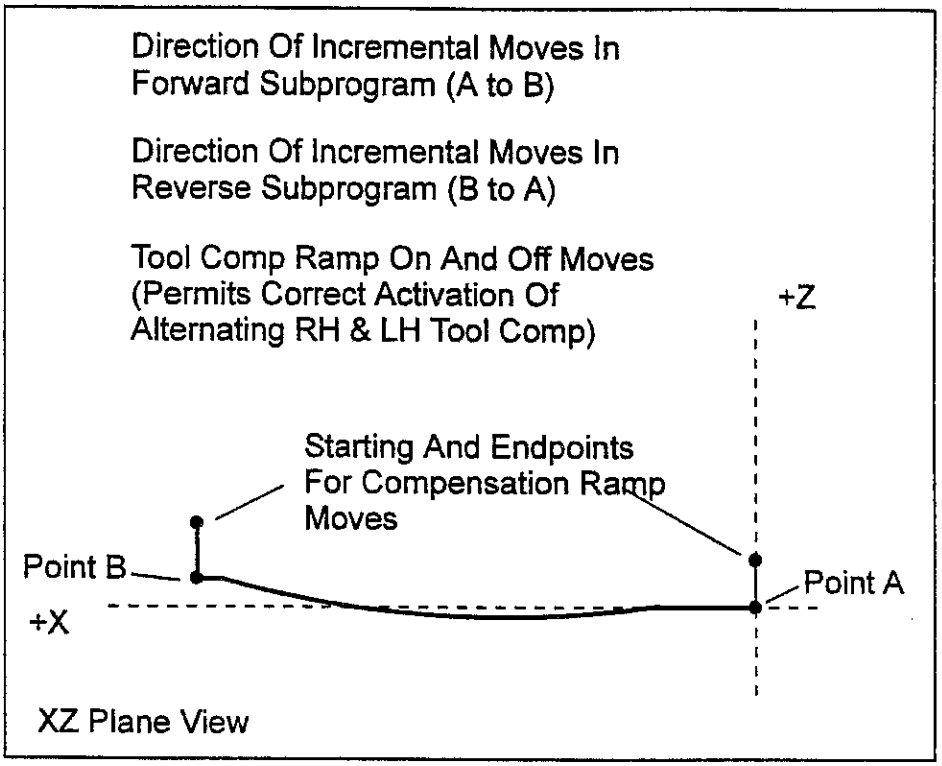


Figure 4 - 56, Z Axis Rotation Subprogram Details

Refer to **Figure 4 - 56, Z Axis Rotation Subprogram Details**. All of the moves in subprograms for Z axis rotations must be contained in the +X half of the XZ plane. Rules for using tool compensation are the same as for Y and X axis rotations.

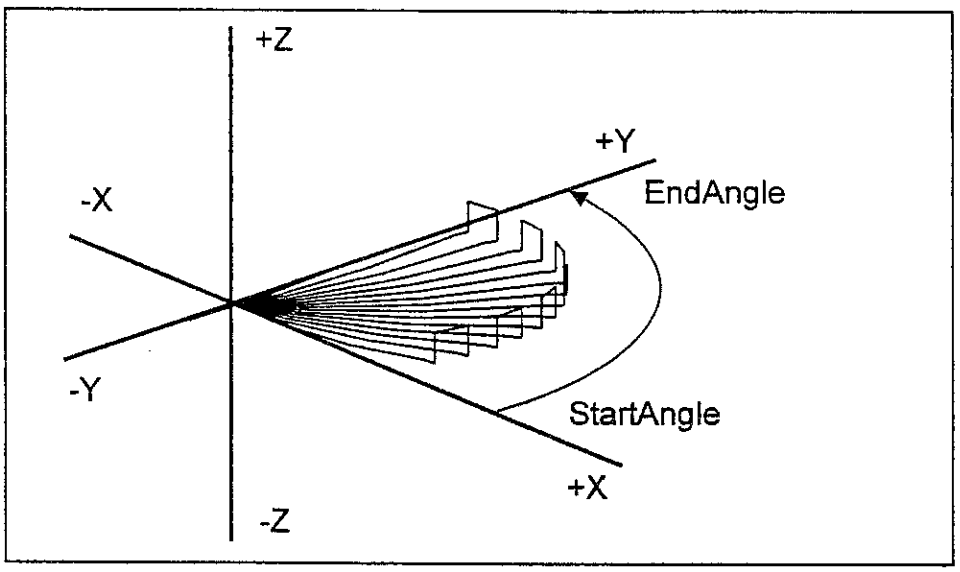


Figure 4 - 57, Z Rotation Start and End Angles

Refer to **Figure 4 - 57, Z Rotation Start and End Angles**. Z axis rotation start and end angles are as defined in the figure.

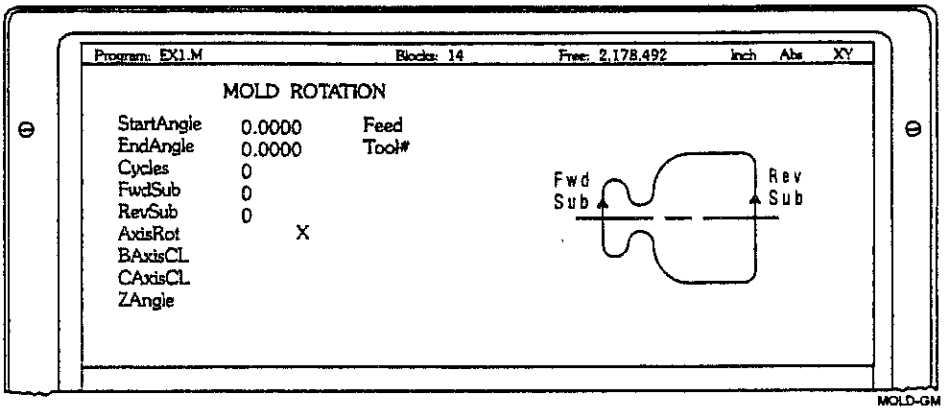


Figure 4 - 58, Mold Rotation Graphic Menu

Refer to **Figure 4 - 58, Mold Rotation Graphic Menu**.

To program a **MoldRot** block:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu Graphic Menus).
2. Using the (ARROW) keys move highlight to select **Mold Rotation** and press (ENTER) (**MOLD ROTATION** Graphic Menu prompts for labeled values).
3. Fill in the **MOLD ROTATION** entry fields labeled as follows:

StartAngle	Starting angle of the rotation (value required, absolute).
EndAngle	Final angle of the rotation (value required, absolute).
Cycles	Number of subprogram cycles executed between start and stop angles (value required).
FwdSub	Forward subprogram call (value required).
RevSub	Reverse subprogram call (value required).
AxisRot	Axis of rotation select, toggle using (+/-) key (selection required).
BAxisCL	First position coordinate of the rotated axis (optional).
CAxisCL	Second position coordinate of the rotated axis (optional).
ZAngle	Rotated position of XY axis mold (optional).
Feed	Feedrate (optional).

Tool# Active tool (optional).

NOTE: After programming a mold rotation, be sure to activate the plane required for proper execution of the moves that follow it.

Programming an Elbow Milling Cycle

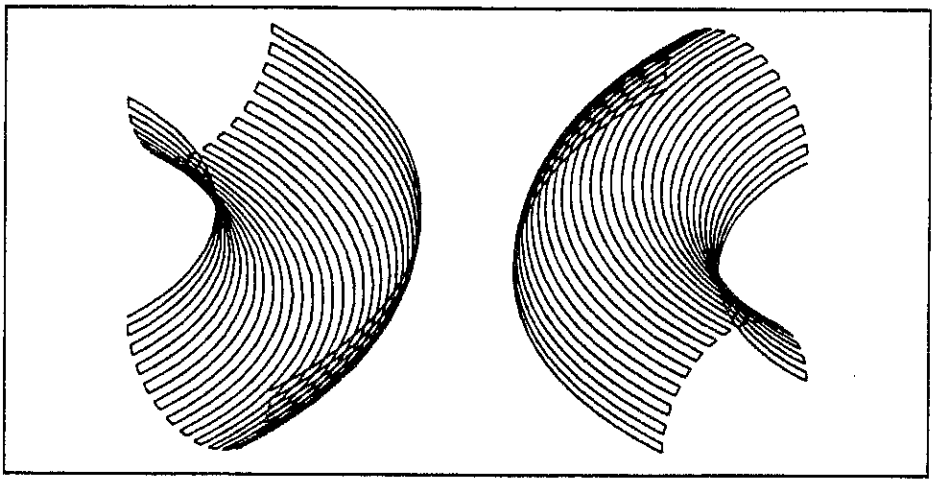


Figure 4 - 59, Elbow Cavity and Core

Elbow Milling cycles simplify the programming required for milling elbow shaped cavities and cores, refer to **Figure 4 - 59, Elbow Cavity and Core**. Finished elbows can have the same radius at each end, (bent cylindrical shape) or a different radius at each end (bent conical shape).

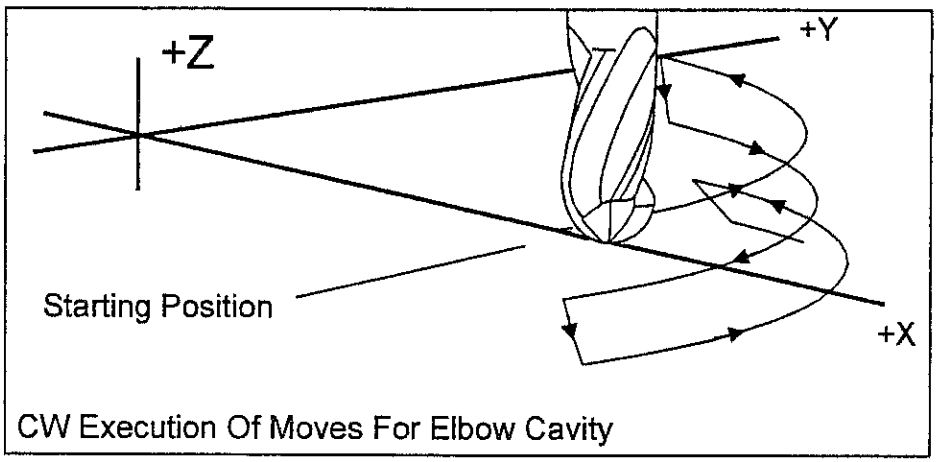


Figure 4 - 60, Execution Of Elbow Milling Cycle Moves

Refer to **Figure 4 - 60, Execution Of Elbow Milling Cycle Moves**. The Elbow Milling Cycle starts at the machine's present position. The CNC executes passes back and forth around the elbow inner radius from the start radius to the end radius. The tool is incremented further into the cavity (or around the core) at the end of each pass until the elbow is complete. The cycle stops at the opposite side of the elbow at the same Z axis position it started from.

Tool compensation cannot be used with the Elbow Milling Cycle. When cutting a particularly deep elbow, it may be necessary to cut the shape in several passes. The easiest way to do this is to program the elbow in a

subprogram. The main program should consist of moves to position the machine at consecutively lower Z axis starting positions. The subprogram is called at each successive starting position.

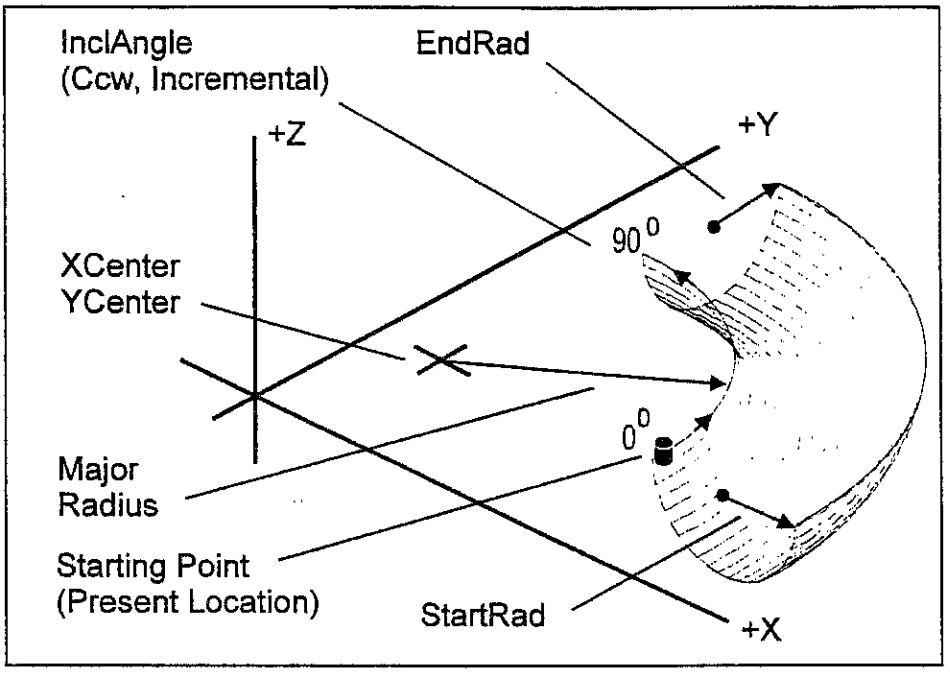


Figure 4 - 61, Elbow Milling Cycle Details

Refer to **Figure 4 - 61, Elbow Milling Cycle Details**. The starting position of the machine must be carefully considered when programming an Elbow Milling Cycle. The distance between the starting point and the **XCenter**, **YCenter** determines the elbow's inner radius. The line between the starting point and the **XCenter**, **YCenter** is the zero degree reference for the included angle. When **XCenter**, **YCenter** values are blank the CNC uses X0, Y0 (default) values.

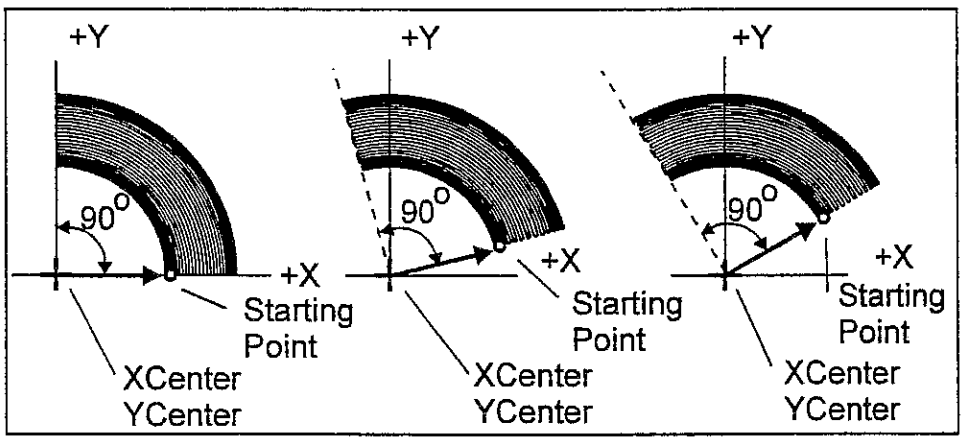


Figure 4 - 62, Starting Points Effect On Orientation

Refer to **Figure 4 - 62, Starting Points Effect On Orientation**. When the line between the starting point and the **XCenter, YCenter** does not lie along an X or Y axis, the orientation of the finished elbow will shift around the XY center accordingly.

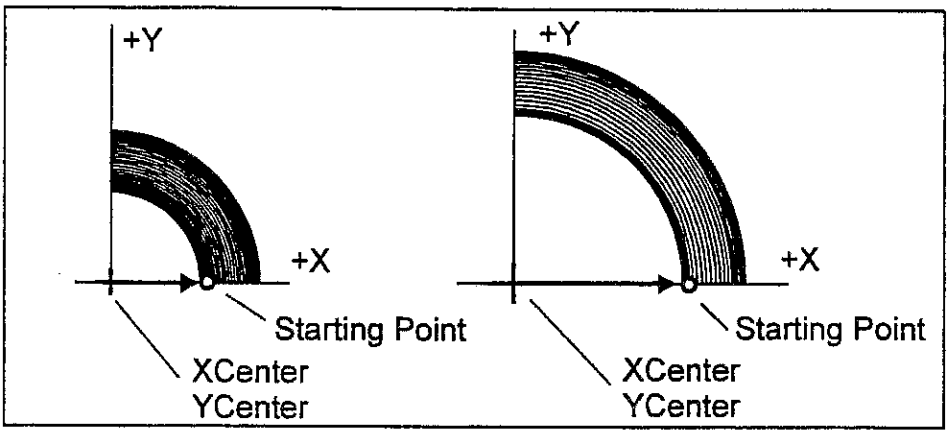


Figure 4 - 63, Starting Points Effect On Size

Refer to **Figure 4 - 63, Starting Points Effect On Size**. The elbow inner radius is determined by the distance between the starting point and the XY center. Moving the starting point away from the XY center increases the overall size of the finished elbow as shown.

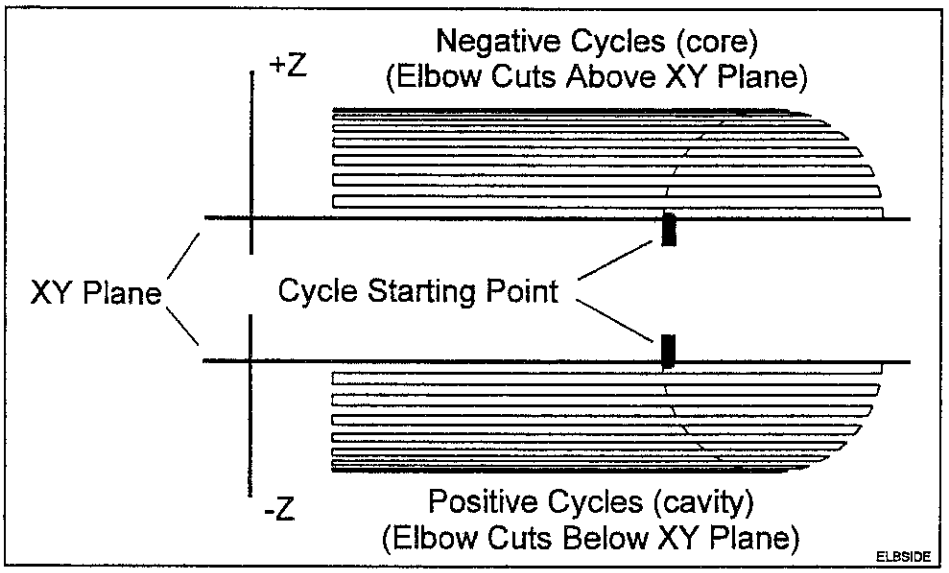


Figure 4 - 64 , Core and Cavity Detail

Refer to **Figure 4 - 64 , Core and Cavity Detail**. The **Cycles** value determines the number of passes used to cut the elbow. A negative cycle value cuts a core while a positive cycle value cuts a cavity.

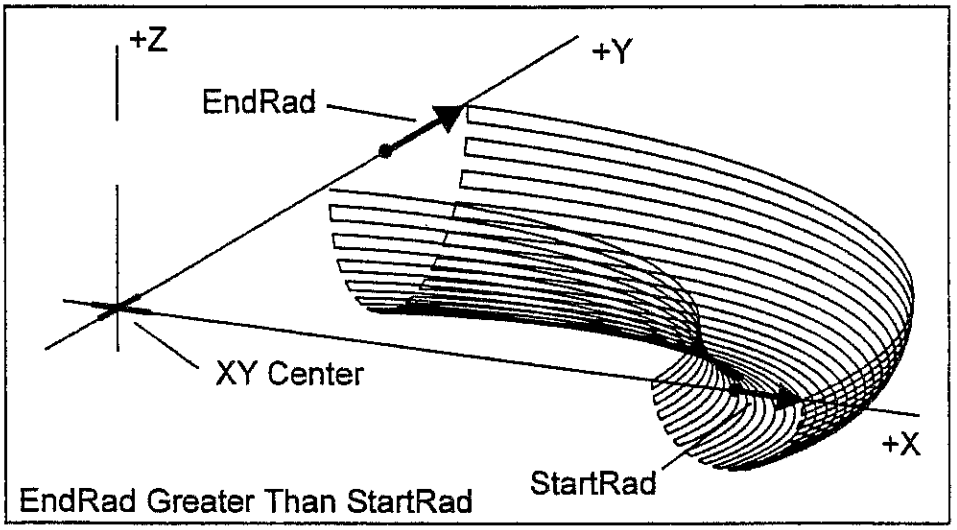


Figure 4 - 65, Conical Elbow Details

Refer to **Figure 4 - 65, Conical Elbow Details**. Programming an Elbow Milling Cycle with unequal **StartRad** and **EndRad** values produces a conical elbow.

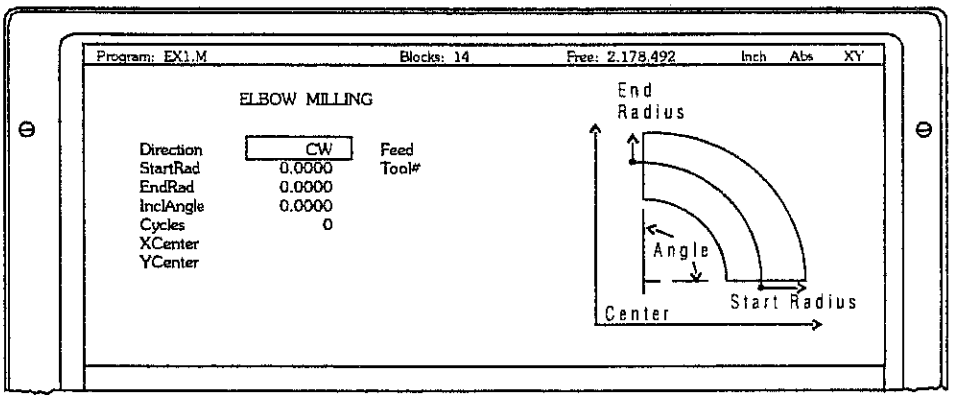


Figure 4 - 66, Elbow Milling Graphic Menu

Refer to **Figure 4 - 66, Elbow Milling Graphic Menu**.

To program an Elbow Milling cycle:

1. With the CNC in the Edit Mode, press **Pocket** (F4) (pop up menu Graphic Menus).
2. Using the (ARROW) keys, move highlight to select **Elbow Milling** and press (ENTER) (**ELBOW MILLING** Graphic Menu prompts for labeled values).
3. Key in **ELBOW MILLING** entry fields labeled as follows:

Direction	Direction, clockwise (**Cw**) or counter clockwise (**Ccw**) of first pass away from starting point, toggle selection with (+/-) key (selection required).
StartRad	Radius of the cavity/core at the starting end of the elbow (value required).
EndRad	Radius of the cavity/core at the far end of the elbow (value required).
InclAngle	The angle (centered at XY center) between the **StartRad** end and the **EndRad** end of the elbow (value required).
Cycles	The number of passes used to cut the elbow (value required).
XCenter	X coordinate of the elbow inner radius centerpoint (optional).
YCenter	Y coordinate of the elbow inner radius centerpoint (optional).
Feed	Feedrate (optional).
Tool#	Active tool (optional).

Subprograms

Repetitive operations can be programmed in a subprogram and called from the main program as often as needed. Subprograms also give the operator some useful options.

- Subprograms can be called or "nested" from other subprograms. The CNC supports up to 10 levels of nesting.
- Subprograms can repeat or loop, incrementally moving along any axis each time the loop is run.
- Subprograms can rotate as they loop.
- Subprograms can be scaled.
- Subprograms can be mirrored.

The following examples describe two situations where using subprograms saves time.

Situation: 1

A workpiece must be center-drilled, drilled, then counterbored. Each of the 3 tools must go to the same hole positions. If there are 10 hole positions, that would mean programming 30 hole locations (10 for each tool). The 10 hole locations can be programmed into the subprogram and the subprogram called 3 separate times from the main program, once for each tool.

Situation: 2

The outside of a workpiece must be roughed out with a roughing mill, then finished with a finishing mill, the profile can be programmed in a subprogram. The subprogram is called from the main program twice, once for each tool. The tool diameter can be set to .5300 inch for the 1/2 inch roughing mill and set to .5000 inch for the 1/2 inch finishing mill. Tool #1 will leave .0150 inch per side, for tool #2 to clean-up to size.

Subprogram Structure

When using subprograms, the operator must 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 appears 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            Z -0.0625
11  Dim Incr
12  Line      X 0.375
13  Line      Y 0.375
14  Line      X -0.375
15  Line      Y -0.375
16  Dim Abs
17            Z 0.1000
18  EndSub
19  <End Of Program>
```

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, executes the subprogram through block 18, then returns to the main program, block 4.

Organizing Programs Containing Subprograms

To organize a program that calls a subprogram:

1. Write the main program like you would any other. Include subprogram Call blocks as they are needed. At this point, the operator only needs a rough idea of the subprogram's purpose, but he does 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.

NOTE: The type of subprogram call determines what features the subprogram has. The operator must choose between Call, Loop and RMS subprograms.

3. When the main program is finished, insert an EndMain block.
4. Start writing the subprogram by adding a Sub block. Each Sub block is followed buy the subprogram Call number. No two subprograms can have the same number. Subprogram numbers can be any number from 1 to 9999.
5. Write the subprogram blocks.
6. Finish the subprogram with an EndSub block.
7. Repeat steps 4 - 6 until all subprograms are written.

Calling Subprograms From the Main Program

1. With the CNC in the Edit Mode, press **Sub** (F8) (subprogram softkey labels appear).
2. Press **Call** (F3) (Graphic Menu prompts for subprogram number being called).
3. Key in subprogram number and press (ENTER) (CNC adds a **Call** block to program).

Ending Main Programs

1. With the CNC in the Edit Mode, press **Sub** (F8) (softkey subprogram labels appear).
2. Press **EndMain** (F4) (**EndMain** is added to program).

Starting Subprograms

1. With the CNC in the Edit Mode, press **Sub** (F8) (**Sub** softkeys appear).
2. Press **Sub** (F2) (CNC prompts for subprogram number).
3. Key in **Sub** number and press (ENTER) (**Sub** block is added to the program). The **Sub** number must agree with the matching **Call** number.

Ending Subprograms

1. With the CNC in the Edit Mode, press **Sub** (F8) (**Sub** softkeys appear).
2. Press **EndSub** (F2) (**EndSub** block is added to program).

Looping Subprograms

Looping subprograms repeat themselves a set number of times before returning to the main program. Each time the loop is run it can also increment the machine.

A looping subprogram is called by a Loop block.

To call a Loop subprogram:

NOTE: Only subprograms can loop .

1. With the CNC in the Edit Mode, press **Sub** (F8) (**Sub** softkeys appear).
2. Press **Loop** (F5) (**LOOP SUB** Graphic Menu prompts for labeled entries).
3. Fill in the **LOOP SUB** entry fields labeled as follows:

Sub#	Subprogram identification number (value required).
#Loops	Number of time loop is to run before returning to main program (value required).
Xincr	Distance X axis increments every cycle, zero if no entry is made (optional).
Yincr	Distance Y axis increments every cycle, zero if no entry is made (optional).
Tool#	Active tool (optional).

Rotating, Mirroring and Scaling Subprograms (RMS).

RMS subprograms can be scaled, and/or rotated, and/or or mirrored. **RMS** subprograms are called from the main program by inserting an **RMS** block. Rotation, scaling and mirroring functions are turned off when the subprogram ends.

NOTE: Only subprograms can rotate, mirror, scale, or loop.

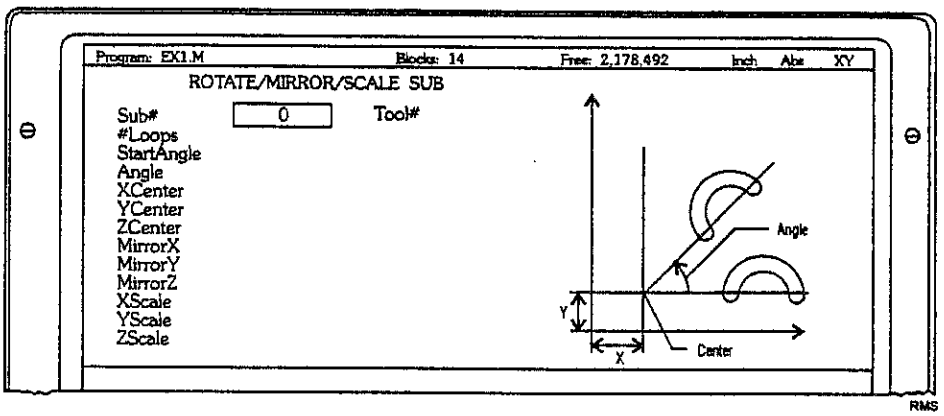


Figure 4 - 67, Rotate / Mirror / Scale Graphic Menu

Refer to **Figure 4 - 67, Rotate / Mirror / Scale Graphic Menu**.

To call an RMS subprogram:

1. With the CNC in the Edit Mode, press **Sub** (F8) (softkey labels display subprogram functions).
2. Press **RMS** (F6) (**ROTATE/MIRROR/SCALE SUB** Graphic Menu prompts for labeled values).
3. Fill in the **ROTATE/MIRROR/SCALE SUB** entry fields labeled as follows:

Sub# Subprogram number (value required).

#Loops Number of times subprogram will loop before returning to main program, 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 from 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	Toggle Yes or No by pressing (+/-)  mirrors the X axis values if yes (optional).
MirrorY	Toggle Yes or No by pressing (+/-)  mirrors the Y axis values if yes (optional).
XScale	X axis scale factor, multiplies all X axis positions by number keyed in, one if no entry is made (optional).
YScale	Y axis scale factor, multiplies all Y axis positions by number keyed in, one if no entry is made (optional).
Tool#	Active tool (optional).

Programming M Code Blocks

The 1100M control has the ability to support a broad range of M-code functions. It is the machine builder who determines which M-codes are set up. Refer to the machine builder's technical data to find out exactly which M-codes (and S-codes) are available.

Some programmed events activate the same functions that are activated by programming M-codes (and S-codes). Refer to **Table 4 - 2, Commonly Available M-codes and Their Use** for a listing of the most commonly used M-code (and S-code) functions.

Table 4 - 2, Commonly Available M-codes and Their Use

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" listing.	Stop spindle in normal manner.
M03, M05, M04	Execution of the 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.
S-function	Activation of a tool that lists a specific Spindle RPM on the tool page.	Not specified, used in conjunction with automatic spindle option.

To program an **MCode** block:

1. With the CNC in the Edit Mode, press **MCode** (F8) (Graphic Menu prompts for **MCode** number and X, Y, Z, Values as may be required).
2. Key in values and press **Save** (F10) or (ENTER) (**MCode** block is added to program).

Editing CNC Programs with a Text Editor

Refer to **Figure 4 - 68, Program Block Elements**. Under normal circumstances, the CNC's Program Editor is the only program editing tool the operator will ever need. Sometimes situations arise where it is convenient to edit CNC programs using an ASCII Text Editor. The Individual parts of a program block, as seen in the CNC's Program Editor, are labeled as shown.

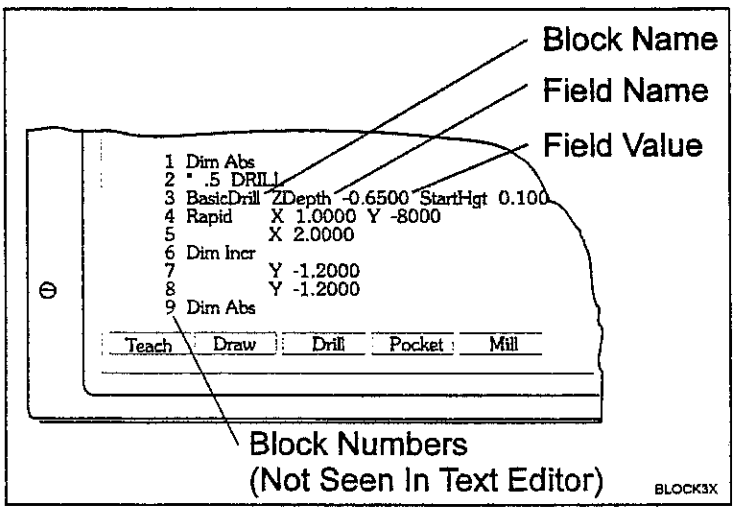


Figure 4 - 68, Program Block Elements

If a CNC is available, use the Program Directory's List utility or the Program Editor to view examples of the format. If no CNC is available, refer to the portions of this section that cover the requirements for each block.

Many blocks contain fields that must have values entered (even if it's 0.000). In this manual, entry fields that cannot be left blank are identified by the words "value required" in parenthesis. These same fields are shown in Graphic Menus initially containing "0.000".

The following rules apply when creating or editing CNC blocks with a text editor.

- Every program block must start with the block name, the name identifies the move, cycle or mode change.
- All entry fields names and values required for the block to execute must appear following the block name, each separated by at least one space.
- In blocks that make Absolute/Incremental or Inch/Millimeter mode changes, the block names are "Dim" and "Unit" respectively. The field value choices are "**Abs/Incr**" or "**Inch/MM**" respectively.
- All block names, field names and field values must be separated by at least one blank space.
- Block numbers are not displayed on ASCII text editors and should not be keyed in. The CNC automatically numbers the blocks when it loads the program.
- The text in program blocks is not case sensitive and does not have to be capitalized the way it appears on the CNC. The Program Editor only uses capitals to make the program listing easier to read.
- All block names must be spelled the way they appear in the program listing (and in this manual).
- A decimal point is not necessary for whole numbers.
- One CR/LF (Carriage Return / Line Feed combination) is required at the end of every block. Carriage Return / Line Feeds are usually inserted by pressing the enter key on most computer keyboards. A CR/LF is not visible on most ASCII text editors. Do not use more than one consecutive CR/LFs.
- If an asterisk (*) is the first character of a block, the CNC will not execute the block, the block will appear as a program comment.
- Any one, two or three axes may be included on a Rapid, Line, Arc or Modal move.
- **Feed** (feedrate) is optional in a Line block, but should be stated at least once in the first feed move of the program.
- ToolComp (Left, Right or Off) must be accompanied by a Line or Rapid move of at least the cutter radius in length.
- Arc center points are modal (defined in the active mode).

NOTE: Some machines may be set up with non-modal Arc center points.

- Arcs are modal, if two or more Arcs in the same direction are subsequent to each other, only the first Arc must state **Arc Ccw**.
- All of the rules regarding active modes, cycle operation etc. that apply to programming in the Program Editor must be applied.
- Numbered values must conform to the specification listed for the G code converter, refer to **Section 7 - Program Management**.

Section 5 - Viewing Programs With Draw

The 1100M has two Draw Modes, Draw Simulation Mode and Real Time Draw Mode.

- In Draw Simulation Mode, the CNC runs programs to simulate machine movements in the viewing area. The machine itself does not move.
- In the Real Time Draw Mode, the CNC displays the machine moves in the viewing area as it makes them.

In this manual, **Draw** (with a capital D) refers to the CNC's Draw Simulation Mode, **DRAW** (all capitalized) refers to the CNC's Real Time Draw Mode.

The same spelling convention is used on the softkey labels.

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

The viewing settings for both modes are changed from the Draw Simulation Mode softkeys. Refer to **Section 6 - Running Programs** for instructions on running DRAW while cutting a part.

Draw lets the operator view a program's operation before cutting any material. In Draw, the operator can change views, zoom in or out, and change scales, to better inspect the program.

NOTE: Changes to the display settings can be made at any time in the Draw Simulation Mode. The display settings cannot be changed while running in the Real Time Draw Mode.

The mode of CNC when Draw is started determines which Mode is activated.

When **Draw** is started from the Edit Mode or from the Manual Data Input Mode (MDI), Draw Simulation Mode is activated.

When **DRAW** is started from a Programmed Operation Mode, Real Time Draw Mode is activated.

Starting Draw

Draw Simulation Mode is started from the Edit Mode or the MDI Mode. The **DISPLAY** (F5) and **Parms** (F9) settings determine how Draw looks and runs. It is better to adjust viewing settings before starting the simulation. The softkeys will permit the operator to make some changes while a simulation is running.

In the Draw Simulation Mode, the CNC does not hold the operation of the program for Dwells and tool mounts

To activate the Draw Simulation Mode:

1. With the CNC in the Edit Mode and a program loaded, press **Draw** (F2) (viewing area appears in upper right hand corner of the screen, Draw softkeys are active).
2. Press **DISPLAY** (F5) (pop-up appears, **Fit** is already highlighted).
3. Press (ENTER) ENTER (**Fit** scales image to fit in viewing area).
4. Press **Run** (F3) (Draw runs the loaded program, path of tool is displayed in viewing area and the machine remains idle).

NOTE: The Draw display remains on the editing screen until it is cleared.

NOTE: If Draw is set to display both compensated and uncompensated moves, the moves in the part program will be drawn twice.

Draw Screen Description

Refer to **Figure 5 - 1 Draw Simulation Mode**. With Draw active, the CNC screen looks like an edit screen with a viewing area in the upper right hand corner. The **Draw** (F2) soft key is highlighted and the Draw softkeys are active.

NOTE: Both Draw screens have a viewing area in the upper right corner of the screen, but the operation of each is very different. Real time Draw looks like a programmed operation screen with a viewing area in the upper right corner. The **DRAW** (F10) softkey is highlighted, and the program running softkeys active. Simulation Draw looks like an **Edit** screen with a viewing area in the upper right corner, and the Draw softkeys active.

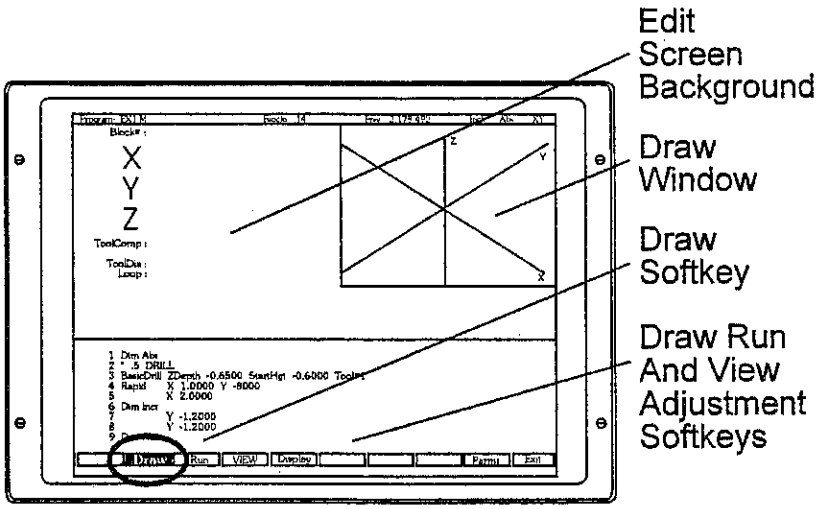


Figure 5 - 1 Draw Simulation Mode

Putting Draw in Hold

Draw can be paused anytime by pressing the **Hold** (F8) softkey or the (HOLD) key.

Restarting Draw When in Hold

When Draw is in hold, press **Start** (F7) or the (START) key to continue.

Canceling Draw

The execution of a program in Draw Simulation Mode can be canceled by pressing the **Cancel** (F9) softkey.

Draw Parameters

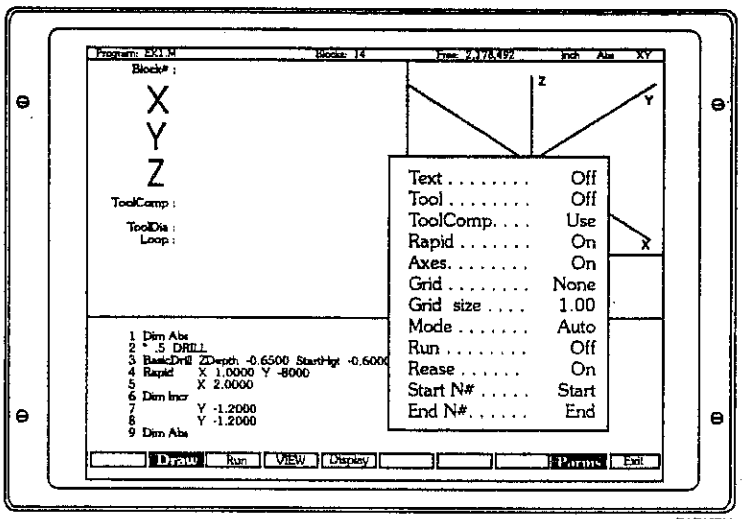


Figure 5 - 2, Draw Params Pop-up Menu

In Draw, features are distinguished using the following convention.

- Rapid moves (Dotted Lines - when turned on)
- Feed moves (Solid Lines)
- Tool & Drilled Holes (Cylinder)

Viewing parameters are set two ways. Refer to **Figure 5 - 2, Draw Params Pop-up Menu**. Before the program is run, set the parameters from the **Parms** (F9) pop-up menu. While the program is running, change the parameters from the softkey menu. Some options appear in both places, some do not. Softkeys toggle the labeled feature on or off, active softkey features are highlighted.

Text On or Off

Running Draw with **Text On** displays program blocks as they are executed. This can help the operator pinpoint a problem. Running Draw with **Text Off** helps Draw run faster. The default setting is **Text On**.

To toggle the **Text** setting **On** or **Off**:

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move the highlight bar to select **Text** and press (ENTER) (**Text** toggles between **On** and **Off**)
3. Press **Parms** (F9) (**Parms** pop-up closes).

NOTE: The **Text** parameter can also be changed, while Draw is running by using the **Text** (F4) softkey.

Tool On or Off

Running Draw with **Tool On**, displays a drawing of the tool as it moves through the part. A tool must be active and have a diameter entered on the tool page in order to display. The tool is drawn as a cylinder. The size of the cylinder is scaled to the tool's diameter.

The default setting is **On**.

To toggle **Tool On** or **Off**:

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move the highlight bar to select **Tool** and press (ENTER) (**Tool** toggles between **On** and **Off**)
3. Press **Parms** (F9) (**Parms** pop-up closes).

NOTE: The **Tool** parameter can also be changed while Draw is running by using the **Tool** (F5) softkey.

Drawing Compensated Moves

The **ToolComp** setting determines if and how Draw displays compensated moves. This lets the operator see the effects of compensation on the moves in a program. The Draw **ToolComp** setting does not affect how a program runs when it is used to cut a part.

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.) The moves from both are displayed simultaneously.

Both is the only selection that allows a direct comparison between the two options, it is the default setting.

To set the compensation parameter:

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move highlight bar to select **ToolComp** and press (ENTER) (second pop-up appears).
3. Move the highlight to the desired selection and press (ENTER) (smaller pop-up closes).
4. Press **Parms** (F9) (**Parms** pop-up closes).

Showing Rapid Moves

Draw displays Rapid moves as dotted lines. The dotted lines can be toggled off to eliminate screen clutter. This parameter has no effect on how the program runs, only on how the display looks. The default setting is **On**.

To toggle **Rapid On** or **Off**:

1. With the Draw Mode active, press **Parms** (F9) (CNC displays parameter pop-up menu).
2. Move the highlight bar to select **Rapid** and press the (ENTER) (**Rapid** toggles between **On** and **Off**)
3. Press **Parms** (F9) (**Parms** pop-up closes).

NOTE: The **Rapid** parameter can also be changed while Draw is running by using the **Rapid** (F6) softkey.

Setting Grid Line Type

Draw can display a two dimensional grid with dotted or solid lines. Grid lines are not visible in the **Iso** view. The default **Grid** setting is **None**.

To set the **Grid** parameter:

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move the highlight bar to select **Grid** and press (ENTER) (**Grid** pop-up menu is displayed).
3. Move the highlight bar to the desired selection and press (ENTER) (**Grid** pop-up closes).
4. Press **Parms** (F9) (**Parms** pop-up closes).

Setting Grid Size

The grid size is adjustable. The units are determined by the CNC's current mode. The default setting is **1.00**.

To set the **Grid size**:

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move highlight bar to select **Grid size** and press (ENTER) (number entry field is displayed).
3. Key in desired size and press (ENTER) (new value displays on pop-up menu).
4. Press **Parms** (F9) (**Parms** pop-up closes).

Putting Draw in Motion, S.Step, or Auto Mode

The Draw Simulation Mode executes programs three ways:

- In the Automatic Mode, fully automatic (**Auto**)
- In the Single Step Mode, one program block at a time (**S.Step**)
- In the Motion Mode, one move at a time (**Motion**)

In the Automatic Mode (**Auto**), blocks are executed sequentially until the program is finished, the CNC is put in a hold, or an error stops the program.

NOTE: The CNC can be toggled into the Automatic Mode (**Auto**) from another mode by pressing the **Auto** (F1) softkey.

In the Single Step mode (**S.Step**), one block of program is executed each time the (START) key is pressed. This allows the operator to run the program one block at a time.

NOTE: The CNC can be toggled into the Single Step Mode (**S.Step**) from other modes by pressing the **S.Step** (F2) softkey.

In the Motion Mode (**Motion**), a program is executed from one motion to the next, executing one motion each time the (START) key is pressed.

NOTE: The CNC can be toggled into the Motion Mode (**Motion**) from other modes by pressing the **Motion** (F3) softkey.

Select the default mode as follows:

1. With the Draw Mode active press **Parms** (F9) (parameter pop-up menu appears).

2. Move the highlight bar to select **Mode** and press (ENTER) (**Mode** pop-up menu is displayed).
3. Move the highlight to select the desired mode and press (ENTER) (**Mode** pop-up closes).
4. Press **Parms** (F9) (**Parms** pop-up closes).

NOTE: Off-line keyboard users running Draw in the Single Step or Motion Modes can use the space bar to continue the program after each hold.

Automatic Draw Restart

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

When **Run** is **Off**, Draw is started and restarted using the **Run** (F3) softkey.

When **Run** is **On**, Draw automatically starts after each **DISPLAY** or **VIEW** change.

The default setting is **On**.

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move the highlight bar to select **Run** and press (ENTER) (**Run** toggles between **On** and **Off**)
3. Press **Parms** (F9) (**Parms** pop-up closes).

Erasing the Draw Display

The **Erase** parameter sets Draw to clear the display when it starts a program. When **Erase** is **Off**, the old drawing remains in the viewing area and new moves are drawn over it. The default setting is **On**. Set the **Erase** parameter as follows:

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu).
2. Move the highlight bar to select **Erase** and press (ENTER) (**Erase** toggles between **On** and **Off**)
3. Press **Parms** (F9) (**Parms** pop-up closes).

Running Draw for Selected Blocks

Draw can run any portion of a program or a subprogram. Run a portion of a program by specifying the **Start N#** and **End N#** block settings on the **Parms** menu. The default settings are the **Start** and **End**.

To run a portion of a subprogram, both the starting and ending blocks must be within the subprogram. To run a subprogram select starting and ending blocks, that include the **Call**, **Loop** or **RMS** block, from the main program.

If a starting block is inside a subprogram and the ending block is in the main program, the CNC will stop and generate an error message at the **EndSub** block because it could not find the subprogram starting block.

NOTE: There is no way to see if the **Start N#** and **End N#** settings have been changed without checking the **Parms** pop-up menu. It is a good idea to restore the default settings after using this feature.

Starting Draw at a Specific Block

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move the highlight bar to select **Start N#** and press (ENTER) (**Start N#** pop-up menu appears).
3. Move the highlight to mark the desired selection and press (ENTER) (if **Start Of Program** is selected, Draw will start at block 1, if **Other Block** was selected, key in the block number and press (ENTER)).
4. Press **Parms** (F9) (**Parms** pop-up closes).

Ending Draw at a Specific Block

1. With the Draw Mode active, press **Parms** (F9) (parameter pop-up menu appears).
2. Move highlight bar to select **End N#** and press (ENTER) (**Start N#** pop-up menu appears).
3. Move the highlight to mark the desired selection and press (ENTER) (if **End Of Program** is selected, Draw will stop at the last program block, if **Other Block** was selected, key in the block number and press (ENTER)).
4. Press **Parms** (F9) (**Parms** pop-up closes).

Adjusting Draw Display

Draw has several display settings to allow the operator the best view of the moves in the viewing window. Refer to **Figure 5 - 3, Display Pop-up Menu**. Activate these settings from the **DISPLAY** (F5) pop-up menu.

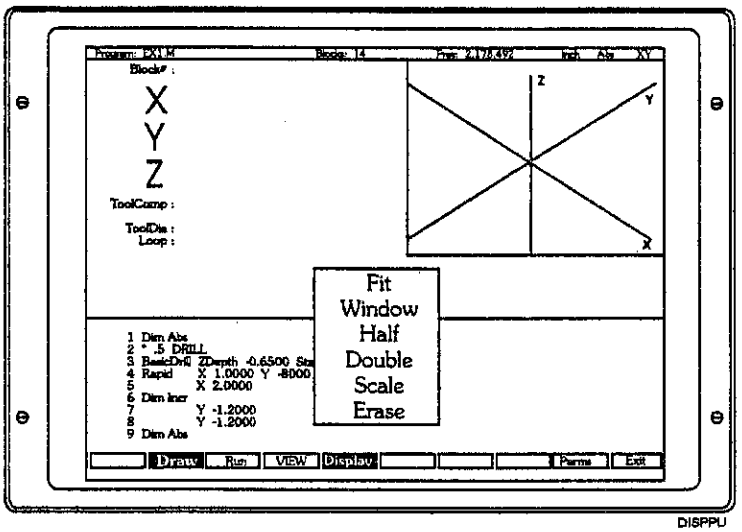


Figure 5 - 3, Display Pop-up Menu

Fitting the Display to the Viewing Window

Draw can automatically scale the display to fit in the viewing area.

To fit the display to the viewing area:

1. With the CNC in the Draw Mode, press **Display** (F5) (pop-up menu appears).

2. Move highlight to select **Fit** and press (ENTER) (pop-up closes, the display adjusts to fit in the viewing window).

Halving Display Size

Draw can reduce the size of the display to one-half existing size.

To reduce the display size by one half:

1. With the CNC in the Draw Mode, press **Display** (F5) (pop-up menu appears).

2. Move highlight to select **Half** and press (ENTER) (pop-up closes, the next time Draw is run, the display will be one half its present size).

Doubling Display Size

Draw can double the size of the display.

To double the size of the display:

1. With the CNC in the Draw Mode, press **Display** (F5) (pop-up menu appears).
2. Move highlight to select **Double** and press (ENTER) (pop-up closes, the next time Draw is run, the display will be twice its present size).

Scaling the Display by a Factor

Draw can enlarge or reduce the display size by a factor.

To scale the Draw display:

1. With the CNC in the Draw Mode, press **Display** (F5) (pop-up menu appears).
2. Move highlight to select **Scale** and press (ENTER) (pop-up closes, CNC prompts for scale factor to be keyed in).

NOTE: Keying in a decimal number reduces the size, keying in a whole number enlarges the size.

3. Key in the desired factor and press (ENTER) (prompt disappears, the next time Draw is run, the display will be scaled by the factor keyed in).

Windowing (Zooming) In

Draw lets the operator zoom in on any portion of the display.

To zoom in on a portion of the display:

1. With the CNC in the Draw mode, press **Display** (F5) (pop-up menu appears).
2. Move the highlight to select **Window** and press (ENTER) (a white window appears inside of viewing window).
3. Using the (ARROW), keys, center the window over the area of interest.
4. Enlarge or reduce the window by pressing the **Expand** (F5) or **Compres** (F6) softkeys.

NOTE: Use **Reset** (F7) to restore window to its original size.

5. Once the window is sized and positioned, press **Enter** (F10) (window is closed, the next time **Draw** is run, the portion of the display framed by the window will fill the viewing window.

Erasing Display

To erase the display:

1. With the CNC in the Draw Mode, press **Display** (F5) (**Display** pop-up menu appears).

2. Move highlight to select **Erase**, and press (ENTER) (display is erased).

Changing Draw Views

Draw lets the operator choose between four different views of the machine's moves. Refer to **Figure 5 - 4, View Pop-up Menu**. Activate different views from the **VIEW** (F4) pop-up menu.

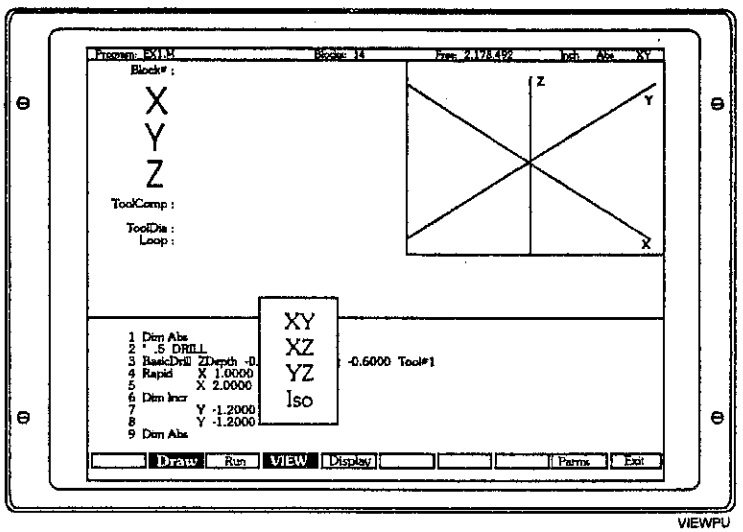


Figure 5 - 4, View Pop-up Menu

Selecting the Top View (XY)

1. From the Draw Mode, press **VIEW** (F4) (**VIEW** pop-up menu appears).

2. Move the highlight to select **XY** and press (ENTER) (Draw is now set to show top view).

Selecting the Front View (XZ)

1. With the Draw Mode active, press **VIEW** (F4) (**VIEW** pop-up menu appears).

2. Move the highlight to select **XZ** and press (ENTER)  (Draw is now set to show front view).

Selecting the Side (End) View (YZ)

1. With the Draw Mode active, press **VIEW** (F4) (**VIEW** pop-up menu appears).

2. Move the highlight to select **YZ** and press (ENTER)  (Draw is now set to show side view).

Selecting the Isometric (3D) View

The isometric view is an elevated three dimensional view of the machine's moves. In the **Iso** mode, grid lines are not shown.

1. With the Draw Mode active, press **VIEW** (F4) (**VIEW** pop-up menu appears).

2. Move the highlight to select **Iso** and press (ENTER)  (Draw is now set to show an isometric view from the lower right quadrant).

Section 6 - Running Programs

There are three modes of programmed operation.

Single Step Mode Runs a program one block at a time.

Motion Mode Runs a program from motion to motion.

Automatic Mode Runs a program automatically.

All screens for running programs look very similar to the Manual Mode screen. Use the softkey labels to distinguish between modes. The label for the active mode will be highlighted.

A program must be loaded before it can be run to cut a part. Programs are loaded from the Program Directory, refer to **Section 7 - Program Management** for information on loading programs.

In the Manual Data Input Mode it is not necessary to load MDI code blocks.

NOTE: The Manual Data Input Mode (MDI) allows the operator to program a few quick moves without creating and saving a program. MDI is usually associated with manual operation and it is only available in the Manual Mode. Refer to **Section 3 - Manual Operation and Machine Setup** for information on MDI.

NOTE: All programming tools, moves, cycles etc. are available in the Manual Data Input Mode (MDI).

Loading Programs for Running

The first step in running a program is to load the program into the CNC's memory. Programs are loaded from the Program Directory. Refer to **Section 7 - Program Management** for instructions on loading a program.

Running a Program One Step at a Time

The Single Step screen provides access to two different execution modes: the Single Step Mode (**S.STEP**) and the Motion Mode (**MOTION**). Both of these modes allow the operator to step through the program and verify the moves before using it for production.

Refer to **Figure 6 - 1, Single Step Screen**. The **S.STEP** screen looks like a **MANUAL** screen with fewer softkeys and the **S.STEP** (F5) softkey highlighted.

To run a program in Single Step Mode:

1. Load the required program and return to the **MANUAL** screen, refer to **Section 7 - Program Management**.
2. Press **S.STEP** (F5) (Single Step Mode activates).

3. Press (START) (CNC executes a single block or motion).
4. Repeat step 3. as required.

NOTE: When the CNC is in the Auto Mode, pressing **S.STEP** (F5) activates Single Step Mode.

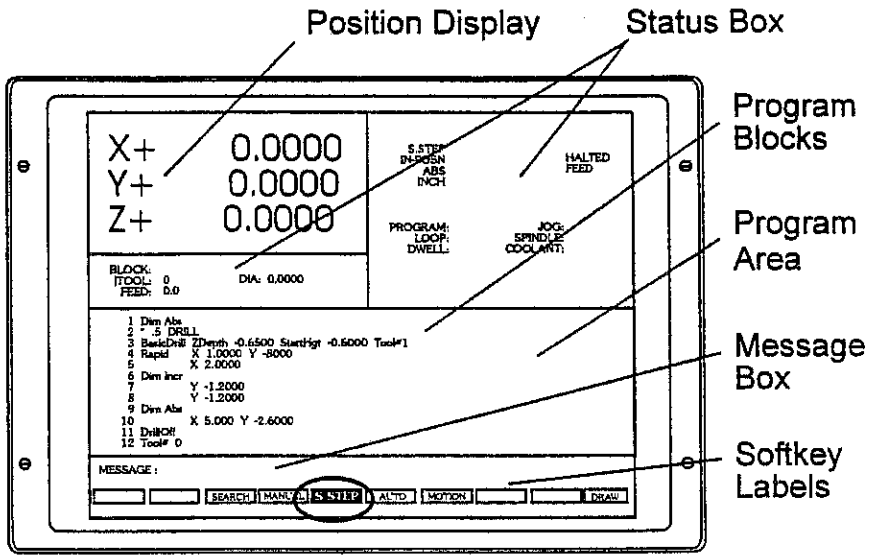


Figure 6 - 1, Single Step Screen

Toggling Between Motion and Single Step Mode

Toggle the CNC between the Single Step (**S.STEP**) and the Motion (**MOTION**) Mode using the **MOTION** (F7) softkey. When the Motion Mode is active, the **MOTION** (F7) softkey label is highlighted.

- In the Single Step Mode, the CNC holds before the execution of each block. The operator must press (START) to execute each block.
- In the Motion Mode, the CNC holds before the execution of each machine move. The operator must press (START) to execute each machine move.

Holding or Canceling a Single Step Run

Press (HOLD) to hold the execution of the program. To restart a program that is on hold, press (START). To cancel a program that is on hold, press **MANUAL** (F4). When a program is canceled, tool compensation is canceled (if it was active) and canned cycles are terminated. All other modal settings remain active.

Single Step Execution of Selected Program Blocks

Method 1, Using Arrow Keys to Select Starting Block

Select the starting block before starting the program.

1. Load the desired program and return to the Manual screen.
2. Press **S.STEP** (F5) (Single Step Mode is activated).

NOTE: If the Motion Mode is required, press the **MOTION** (F7) softkey at this time.

3. Using the (ARROW)  keys, position the highlight to mark the desired starting block.
4. Press (START)  (CNC executes the next block or motion).

Method 2, Using Search to Select Starting 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 first block found containing the specified information is highlighted and displayed. Search only searches forward in the program. Perform the Search before starting the program.

1. Load the desired program and return to the Manual screen.
2. Press the **S.STEP** (F5) softkey (Single Step Mode activates).
3. Press **SEARCH** (F3) (CNC prompts for number or text to search for).
4. Key in the required number or text and press (ENTER)  (The CNC runs the search. The first block containing the number or text will be marked by the highlight).
5. Press (START)  (CNC starts executing the program from the highlighted block).

Switching from Single Step to Auto

To change the CNC from Single Step to Auto Mode:

1. While running in the Single Step Mode, press **AUTO** (F6) (CNC completes current move then holds).
2. Press (START)  (CNC restarts and runs the rest of the program in the Automatic Mode).

Automatic Program Execution

The Auto Mode is the CNC's production mode. All or any portion of a program can be executed in the Auto Mode. The CNC can be put in the Auto Mode from either the Manual or Single Step Modes.

Refer to **Figure 6 - 2, Auto Screen**. The **AUTO** screen is similar to a **MANUAL** screen, but has fewer softkeys. The **AUTO** (F6) softkey label is highlighted when the Auto Mode is active.

To run a program in Automatic Mode:

1. Load the required program and return to the Manual screen.
2. Press **AUTO** (F6) (Automatic Mode activates).
3. Press (START) (The CNC begins executing program blocks).

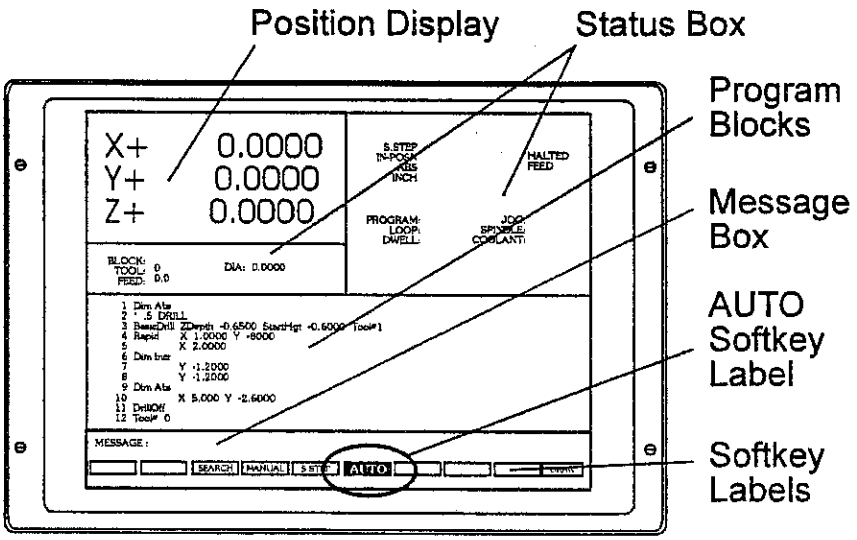


Figure 6 - 2, Auto Screen

Holding or Canceling an Auto Run

Press (HOLD) to hold the execution of the program. To restart a program that is on hold, press (START). To cancel a program that is on hold, press **MANUAL** (F4). When program execution is canceled, tool compensation is canceled (if it was active) and canned cycles are terminated. All other modal settings remain active.

Starting at a Specific Block

Method 1, Using Arrow Keys to Select Starting Block

Select the starting block before starting the program.

1. Load the required program and return to the Manual Mode.
2. Using the (ARROW)  keys, position the highlight to mark the required starting block.
3. Press (START)  (CNC begins automatic program execution from the selected block).

Method 2, Using Search to Select Starting 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 first block found containing the specified information is highlighted and displayed. Search only searches forward in the program. Perform the Search before starting the program.

1. Load the required program and return to the Manual screen.
2. Press the **AUTO** (F6) softkey (Auto Mode activates).
3. Press **SEARCH** (F3) (CNC prompts for number or text to search for).
4. Key in number or text and press (ENTER)  (CNC runs the search. The first block containing the number or text will be marked by the highlight).
5. Press (START)  (CNC starts executing the program from the highlighted block).

Clearing a Halted Program

When the CNC encounters a program block that generates an error, a Warning message is displayed and the CNC stops execution of the program. The operator must press **MANUAL** (F4) to reactivate the keypad.

A program error could generate more than one message. Refer to **Section 2 - CNC Console and Software Basics** for instructions on reviewing undisplayed error messages.

After the program is corrected, it can be loaded and restarted at the appropriate block.

Using Draw While Running Programs

Refer to **Figure 6 - 3, Draw (Real Time Mode)**. When Real Time Draw is used, the CNC displays the moves as they are executed. Activating

Draw changes the secondary display area to a DRAW viewing area and highlights the **DRAW** (F10) softkey label.

All of the display options available in Draw Simulation Mode are available in the Real Time Draw Mode, but the settings must be made from the Draw Simulation Mode screen, refer to **Section 5 - Viewing Programs With Draw**. Press (CLEAR) [CLEAR] at any time to clear the Draw display.

NOTE: Both Draw Modes have a window in the upper right corner of the screen, but the operation of each is very different. Draw Real Time Mode looks like a PROGRAM screen that has a viewing window in the upper right corner, the **DRAW** (F10) softkey label highlighted, and the program execution softkeys active. Draw Simulation Mode looks like an Edit screen that has a window in the upper right corner and the Draw softkeys active.

To activate **Draw** while running a program.

1. Load the required program and put the CNC in required execution mode (**S.STEP** or **AUTO**).
2. Press **DRAW** (F10) (upper right status box changes to a window).
3. Press (START) [START] (program execution starts, moves are displayed as they are executed by the CNC).

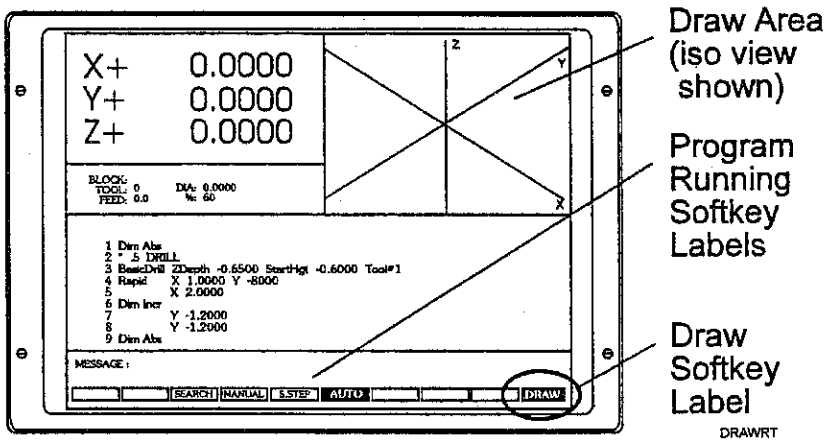


Figure 6 - 3, Draw (Real Time Mode)

Section 7 - Program Management

The Program Directory provides access to all of the program management and disk utilities. These functions include creating, loading, deleting, undeleting and copying programs. The Program Directory also provides access to the floppy disk drives and the communications utilities.

Activate the Program Directory from the **MANUAL** screen by pressing the **PROGRAM** (F2) softkey.

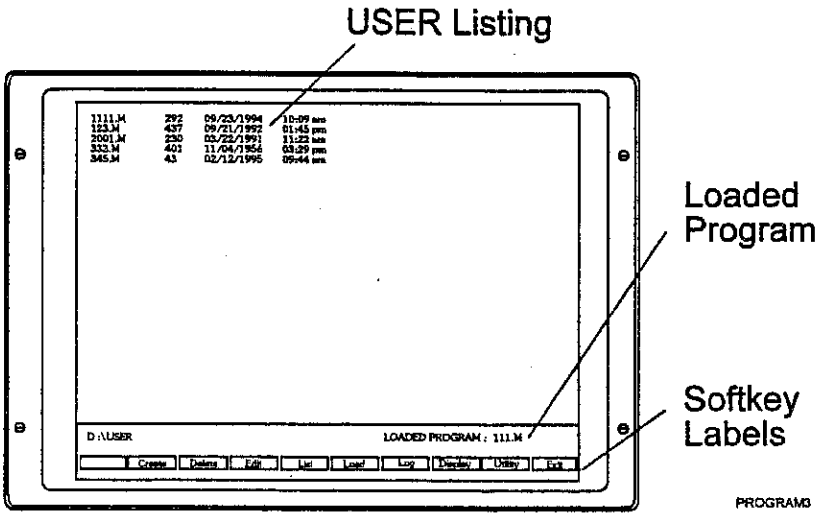


Figure 7 - 1, Program Directory

Refer to **Figure 7 - 1, Program Directory**. The Program Directory's **USER** listing displays a list of the programs stored in the CNC. All CNC programs have "**.M**" extensions after their name. Change the display mode to view programs with other formats.

Changing the Program Directory's Display

The Program Directory has four display modes.

- Display only CNC programs (names with "**.M**" extensions)
- Display CNC programs (names with "**.M**" extensions) with size, date and time of last edit
- Display all programs in all formats
- Display all programs in all formats with size, date and time of last edit

Cycle the Program Directory through different display modes by pressing the **Display** (F8) softkey.

The display mode that only shows "**.M**" (CNC part programs) is the one used most often.

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. With the CNC in the Manual Mode, press **PROGRAM** (F2) (Program Directory activates, **USER** listing is displayed).
2. Press **Create** (F2) (message line displays **"NEW PROGRAM: _"** prompt).
3. Press **ASCII** (F2) (ASCII Chart activates).

NOTE: The ASCII Chart is not required if the new program name contains no letters.

4. Using the ASCII Chart and the number keypad, key in the new program's name.
5. When the name is complete, press **ASCII** (F2) (ASCII pop-up disappears).
6. Press (ENTER) (new program's name is alphabetically placed in the **USER** listing).

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

Choosing Program Names

A name cannot be longer than eight characters. If more than eight characters are keyed in, only the first eight will be used. The CNC only displays program names in capital letters. Names should only contain letters, numbers, and dashes. No two programs can have the same name. The CNC automatically places the ".M" extension after the first eight characters.

Loading a Program for Running

A program must be loaded before it can be executed by the CNC. Loading a new program unloads the previous one.

To Load a program:

1. Using the (ARROW) keys, position the highlight to select a program and press **Load** (F6) (CNC loads program, program name appears next to the **"LOADED PROGRAM"** label at the bottom of the screen).

Selecting a Program for Editing and Utilities

The program marked by the highlight when the **Edit** (F4) softkey is pressed will be displayed in the editor, not the loaded program. **Delete** (F3), **List** (F5) and most other utilities will also carry out their functions on the program marked by the highlight when activated.

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

Select a program for editing as follows:

1. From the Program Directory, use the **Display** (F8) softkey to set the display for CNC programs only.
2. Using the (ARROW)  keys, position the highlight to select the name of the program being edited or written.
3. Press **Edit** (F4) (program editor activates, highlighted program is displayed in the editor).

Maximizing Program Storage Space

The CNC has a fixed amount of space available for programs. Check the space available with the System Information utility.

Refer to **Figure 7 - 4, System Information Screen**.

Total Space Available For The System

the amount of program storage space built into the CNC.

Total Free User Space

the space remaining for new programs.

When a program is executed on the machine (or in Draw), the control generates a second program of the same name followed by ".S". The ".S" programs contain information required by the CNC. Normally ".S" programs are slightly larger than the part program that generated it. When a part program is deleted, the associated ".S" file is also deleted.

The CNC is not the best place for long term storage of part programs. Part programs should be periodically copied to floppy disks (or other computers) for backup and storage.

If a lot of old programs are allowed to accumulate, the CNC could run out of memory. One way to temporarily make room is to delete the ".S" files of programs that aren't currently being used. The CNC will automatically replace the ".S" files the next time the programs are run.

The Program Directory displays only part programs (programs followed by ".M") by default. Use the **Display** (F8) key to set the Program Directory to display programs with ".S" extensions.

NOTE: Under normal conditions, the CNC will have more than enough memory to operate. If running out of memory does become a problem, Anilam does offer memory upgrade options for most of it's products.

Program File Utilities

Some utilities are accessed directly from the softkeys, and some are accessed through the **Utility** (F9) pop-up menu. The Utility pop-up menu lists 14 utilities on two pages. Highlighting **MORE ...**and pressing (ENTER) activates the second page of the menu.

Displaying Program Blocks (Listing a Program)

List (F6) displays the blocks in a program. This lets an operator look over a program without the chance of an accidental change. The List utility only displays CNC programs (programs with a "**.M**" extension on their name).

The softkeys available on the List screen are the same softkeys activated by pressing **Misc** (F9) from the Edit screen. The **Draw** (F2), **Search** (F3), **PgUp** (F4), **PgDn** (F5), **Begin** (F6), and **End** (F10) softkeys all function as they do in the Program Editor (refer to **Section 4 - Writing Programs**).

To list a program:

1. Using the (ARROW) keys, move the highlight bar to select the program being listed.
2. Press **List** (F5) (CNC displays the programs blocks). No editing is possible in the List Mode.

Deleting a Program

1. From the Program Directory, position the highlight bar to select the program to be deleted.
2. Press **Delete** (F3) (CNC prompts the operator to confirm the deletion, softkeys change for the operators response).
3. Press **Yes** (F1) (CNC deletes the selected program).

- or -

Press **No** (F2) (deletion is cancel).

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

Reading Disks in Floppy Drives (Logging to Other Drives)

Most utilities operate on the programs in the USER listing by default. The Log utility makes it possible for the CNC to use the utilities on floppy disks in the machines floppy disk drive.

To log the CNC to a floppy disk drive:

1. From the Program Directory, press **Log** (F7) (CNC prompts operator to select a drive).

2. Using the (ARROW)  keys, move highlight to select a drive and press (ENTER)  (Program Directory displays selected drive's contents).

Marking and Unmarking Programs

Many of the utilities can be used on more than one program at a time. The Program Directory allows the operator to select one, some or all of the programs in the **USER** listing.

Marking Programs

To mark a program:

1. From the Program Directory, use the (ARROW)  keys to position the highlight bar over the program's name.
2. Press (ENTER)  (marked program is highlighted, highlight bar advances to the next name)
3. Press (ENTER)  to mark the next program.

   - or -

   Use the (ARROW)  keys to reposition the highlight over any other program in the listing and press (ENTER)  .
4. Repeat these steps to mark as many program as required.

Unmarking Marked Programs

1. Use the (ARROW)  keys to position the highlight over any previously marked program and press (ENTER)  (the mark clears).

Marking all Programs

1. From the Program Directory, press **Utility** (F9) (Utility pop-up menu appears).
2. Using the (ARROW)  keys, position the highlight to select **More . . .** and press (ENTER)  (second page of pop-up menu appears).

3. Using the (ARROW)  keys, position the highlight to select **Mark All** and press (ENTER)  (all programs are marked for pending operation).

Unmarking all Marked Programs

1. From the Program Directory, press **Utility** (F9) (Utility pop-up menu appears).
2. Using the (ARROW)  keys, position the highlight to select **More . . .** and press (ENTER)  (second page of pop-up menu appears).
3. Using the (ARROW)  keys, position the highlight to select **Unmark All** and press (ENTER)  (all programs are unmarked).

Deleting Groups of Programs

1. From the Program Directory, mark all of the programs to be deleted and press the **Delete** (F3) softkey, (CNC prompts the operator to confirm the deletion, softkeys change for operators response).
2. Press **Yes** (F1) (CNC deletes the selected programs).

- or -

Press **No** (F2) (deletion is cancel).

Undeleting Programs

Deleted programs can be recovered if another program has not already used the deleted program's memory. Sometimes only a portion of a deleted program is recoverable. For the best results, undelete a program as soon as possible after deletion.

To undelete a program:

1. From the Program Directory, press **Utility** (F9) (Utility pop-up appears).
2. Using the (ARROW)  keys, position the highlight to select **UnDelete** and press (ENTER)  (If the control finds programs to undelete, it displays a pop-up menu. If the control doesn't find deleted programs, a "**No programs available for undelete**" prompt appears).

3. When deleted programs are found, use the (ARROW) keys to position the highlight to select the program and press (ENTER) (CNC prompts for the first letter of the deleted program's name, ASCII softkeys activate).

4. Use the ASCII Chart to key in the letter and press (ENTER) (CNC presents information message).

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

NOTE: Undeleted programs may not contain all of the original information. Review any undeleted programs for accuracy before attempting to use them.

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

Copying Programs to Floppy Disks

Copying programs to floppy disks provides an easy way to transfer programs to other machines. Programs can also be copied to floppy disks for storage.

1. From the Program Directory, mark all of the programs to be copied.

2. Press **Utility** (F9) (Utility pop-up menu appears, **Copy** selection is highlighted).

3. Refer to **Figure 7 - 2, "Copy To" Pop-up Menu**. Press (ENTER) (**Copy to:** pop-up menu appears).

4. Using the (ARROW) keys, position the highlight to select the target drive and press (ENTER) (marked programs are copied to target drive).

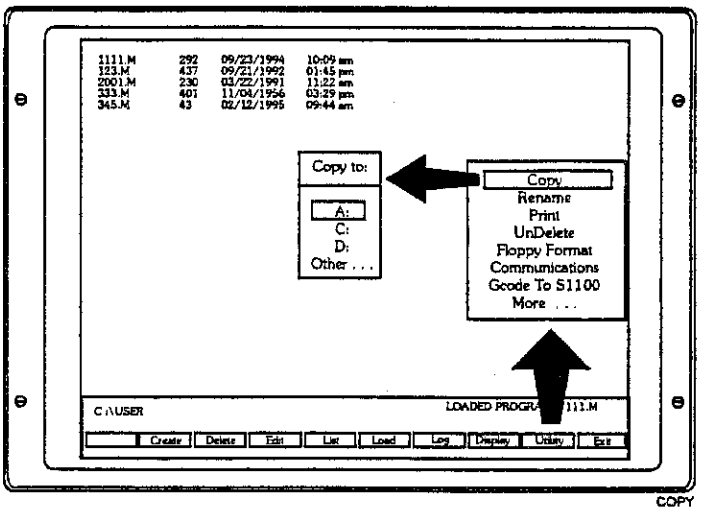


Figure 7 - 2, "Copy To" Pop-up Menu

Renaming Programs

1. From the Program Directory, use the (ARROW) keys to position the highlight on a program.
2. Press **Utility** (F9) (Utility pop-up menu appears).
3. Use the (ARROW) keys to position the highlight to select "**Rename**", and press (ENTER) (CNC prompts for a new program name, ASCII Chart softkeys display).
4. Using the ASCII chart and the number keypad, key in the new name and press (ENTER) (the old name is replaced by the new name).

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

Printing Programs

The CNC can print to any a standard IBM PC compatible printer. The print utility can be used to make paper copies of part programs. When more than one program is marked all will be sent to the printer. Refer to **Section 11 - Software, Keyboard and Printer Installation** for printer installation instructions.

To print one or more programs:

1. From the Program Directory, mark all the programs to be printed.

- or -

Using the (ARROW)  keys position the highlight to select a program for printing.

2. Press **Utility** (F9) (utility pop-up appears).
3. Using the (ARROW)  keys, position the highlight to select **"Print"**, and press (ENTER)  (CNC prompts the operator to confirm the command, softkeys change for operators response).
4. Press **Yes** (F1) (programs are sent to printer).

- or -

Press **No** (F2) (print canceled).

Formatting Floppy Disks

Before programs can be transferred to floppy disks, the disk must be formatted. Most disks come pre-formatted, but the CNC can format a disk when necessary.

Format a floppy diskette as follows.

1. From the Program Directory. press **Utility** (F9) (Utility pop-up appears).
2. Using the (ARROW)  keys, position the highlight to select **Floppy Format** and press (ENTER)  (warning message is displayed, CNC prompts the operator to confirm the format, softkeys change for operators response).

CAUTION: Formatting a floppy disk erases all of the data on the disk.

3. Press **Yes** (F1) (CNC prompts for disk insertion).

- or -

Press **No** (F2) (format is canceled).

4. Insert the floppy disk in the CNC's disk drive and press **Cont** (F10) (CNC displays disk statistics as it formats).

Converting G-code Programs to CNC Conversational Format

The CNC only runs programs that are written in the Anilam Conversational Language Format. The G-code conversion utility translates G-code programs to the Anilam Conversational Language Format. This saves the operator from rewriting part programs written for other machines.

By default, converted programs are given the same name as the original, but with a ".M" extension added. If necessary, the operator can assign any valid name to a converted program. The conversion utility does not alter the original G-code program in any way. Programs that have **".M"** or **".S"** extensions on their name cannot be converted.

By default, the conversion utility writes the translated program to the CNC's current drive. If the original program is located in the **USER** listing, the converted program will be written to the **USER** listing. If the CNC is logged to a floppy drive, it will read the original program and write the new program to the floppy drive. The operator can force the location of the new program to any valid location by selecting **No** (F2) when prompted to confirm the new program's name.

The operator should ensure enough space is available for the converted program. Converted programs are slightly larger than the original. Only one program can be converted at a time.

The G-code utility supports an enhanced version of the RS 274D specification for G-code. Some G-code blocks may not convert directly. G-code blocks that do not convert directly are inserted into the converted program as comment blocks. Commented blocks are marked by an asterisk.

Carefully review and test all converted programs with Draw before running them. Refer to **Section 5 - Simulating and Viewing Programs With Draw** for details.

NOTE: The Program Directory's default display setting only displays program names that end with **".M'**. It may be necessary to change the display setting to view programs with other formats. Toggle the settings with the **Display** (F8) softkey.

To convert a G-code program to conversational format:

1. Transfer the program to the control either by copying it from a floppy disk or by downloading it via the RS-232 link. Refer to **Section 9 - Communications and DNC** for downloading instructions.
2. From the Program Directory, mark the program being converted.
3. Press **Utility** (F9) (Utility pop-up menu appears).
4. Using the (ARROW)  keys, position the highlight to select **G-code To 1100M**, and press (ENTER)  (CNC prompts the operator to confirm the conversion, softkeys change for operators response).
5. Press **Yes** (F1) (CNC prompts the operator to confirm the new program's name, softkeys change for operator's response).

   - or -

   Press **No** (F2) (conversion is canceled).

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

   - or -

   To key in a different name (or location) and press **No** (F2) (CNC prompts for new name, ASCII softkeys activate).

7. Key in the desired name (or path), close the ASCII pop-up and press (ENTER) (CNC writes the conversational program, status box displays progress).

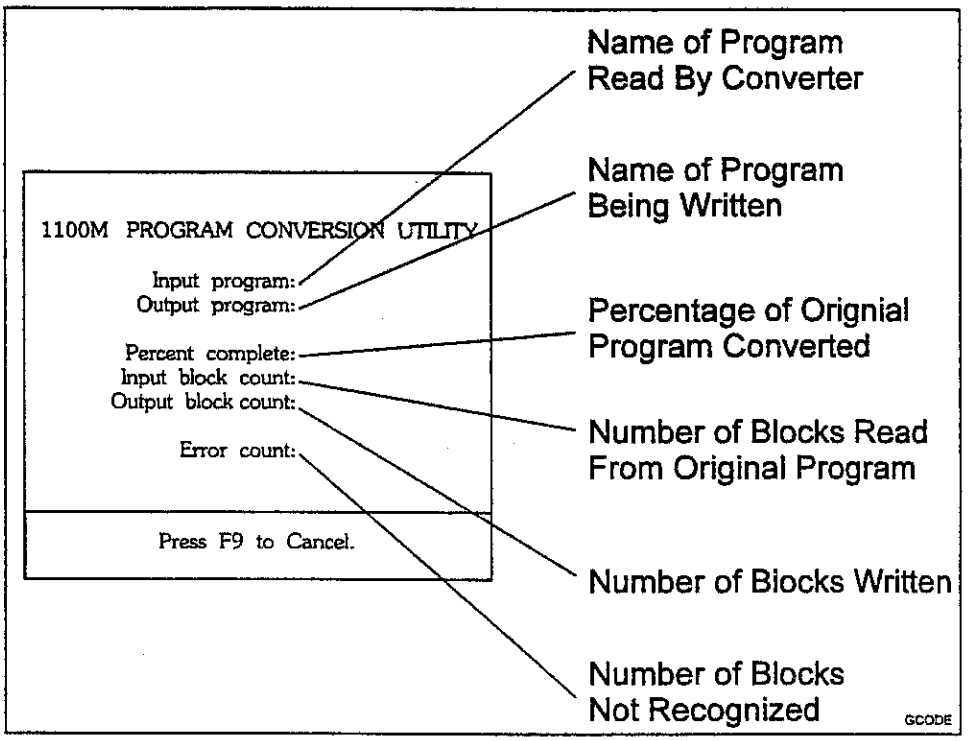


Figure 7 - 3, G-code Converter Status Display

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

Refer to **Table 7 - 1, G-code Conversion Number Formats** and **Table 7 - 2, G-code Equivalents** for G-code compatibility specifications. For best results, the G-code in the program being converted should conform to these specifications.

Table 7 - 1, G-code Conversion Number Formats

	Maximum Number of Digits to Right & Left of Decimal Place		Example of Properly Formatted Value	
	Mode		Mode	
	Inches	MM	Inches	MM
Axes	4.4	5.3	X9999.9999	X99999.999

Feed	3.1	4.1	F999.9	F9999.9

Table 7 - 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	
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	

Table 7 - 2, G-code Equivalents (Continued)

G-code Format			Conversational Equivalent
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 =	RoughFeed	
	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 7 - 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

Checking Disks for Lost Data

Sometimes computer disks contain program fragments that have been lost. This might happen if a computer loses power while it is saving a program. Lost program fragments are invisible in the **USER** listing, but do use up disk space. Check a disk for lost program fragments as follows:

1. From the Program Directory, press **Utility** (F9) (Utility pop-up appears).
2. Using the (ARROW)  keys, position the highlight to select **More . .** ., and press (ENTER)  (second page of pop-up menu appears).
3. Using the (ARROW)  keys, position the highlight to select **Check Disk**, and press (ENTER)  (CNC prompts operator to select the drive to be checked).
4. Using the (ARROW)  keys, position the highlight to select the desired drive, and press (ENTER)  (CNC checks the disk, if lost clusters are found the CNC prompts the user for recovery instructions, softkeys change for operators response).
5. Press **Yes** (F1) (CNC attempts to recover lost clusters on disk).

   - or -

   Press **No** (F2) (utility deactivates).

Display System Information

Refer to **Figure 7 - 4, System Information Screen**. The System Information screen displays specific details about the CNC and its software. Most of the information on this screen is only required during machine setup or troubleshooting.

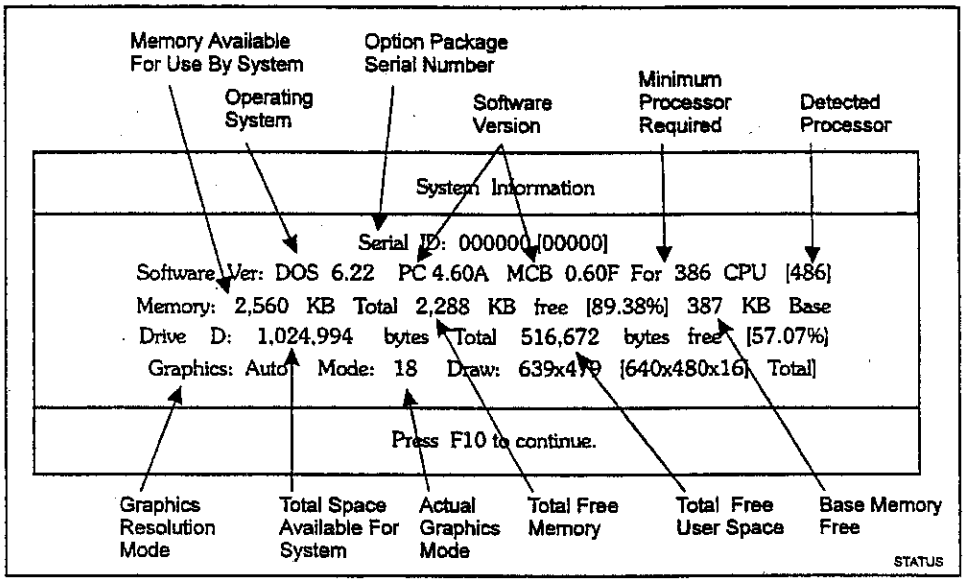


Figure 7 - 4, System Information Screen

Display the System Information screen as follows:

1. From the Program Directory, press **Utility** (F9) (Utility pop-up menu appears).
2. Using the (ARROW) keys, position the highlight to select **More . .** . and press (ENTER) (second page of pop-up menu appears).
3. Using the (ARROW) keys, position the highlight to select **System Info**, and press (ENTER) (System Information screen displays).

Copying Programs from/to Unspecified Locations

Copy programs from floppy disk drives into the **USER** listing as follows.

1. From the Program Directory, press **Utility** (F9) (Utility pop-up appears).
2. Using the (ARROW) keys, position the highlight to select **More .** . . and press (ENTER) (second page of pop-up menu appears).
3. Using the (ARROW) keys, position the highlight to select **Copy ?** and press (ENTER) (CNC prompts for program source, ASCII softkey is active).

4. Using ASCII Chart, key in the complete location (path) of the program being copied, and press (ENTER) (pop-up prompts for destination drive or **Other . . .**).
5. Using the (ARROW) keys, position the highlight to select **Other** and press (ENTER) (CNC prompts for destination, ASCII softkey is active).
6. Using ASCII Chart, key in **D:\USER**[program name]**.M,** and press (ENTER) (program is copied into **USER** listing).

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

NOTE: An alternative procedure is to Log onto the floppy disk drive containing the program, mark the program and use the **Copy to:** utility.

Renaming Programs from/to Unspecified Locations

Rename a program located on any drive as follows:

1. From the Program Directory, press **Utility** (F9) (Utility pop-up menu appears).
2. Using the (ARROW) keys, position the highlight to select **More . .** .,and press (ENTER) (second page of pop-up menu appears).
3. Using the (ARROW) keys, position the highlight to select **Rename ?** and press (ENTER) (CNC prompts for the location of the source program, ASCII softkeys are active).
4. Using the ASCII Chart, key in the location (complete path) of the program being renamed and press (ENTER) (CNC prompts for programs destination, ASCII softkeys are active).
5. Using ASCII Chart, key in the location and new name (complete path) of the program and press (ENTER) (program is renamed).

NOTE: By keying in a different program destination the **Rename ?** utility can be used to move a program to a different drive.

NOTE: An alternative procedure is to Log onto the floppy drive containing the program, mark the program and use the Rename utility.

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

Printing from Floppy Drives

The CNC can print to any a standard IBM PC compatible printer. Refer to **Section 11 - Software, Keyboard and Printer Installation** for printer installation instructions.

To print programs from floppy drives:

1. From the Program Directory, screen press **Utility** (F9) (Utility pop-up appears).
2. Using the (ARROW)  keys, position the highlight to select **More . .** . and press (ENTER)  (second page of pop-up menu appears).
3. Using the (ARROW)  keys, position the highlight to select **Print ?** and press (ENTER)  (CNC prompts for source program on message line, ASCII softkey is active).
4. Using the ASCII Chart, key in the location, (complete path) of program being renamed, and press (ENTER)  (CNC prompts the operator to confirm the instruction, softkeys change for operators response).
5. Press **Yes** (F1) (programs are sent to printer).

   - or -

   Press **No** (F2) (print is canceled).

NOTE: Refer to **Section 2 - CNC Console and Software Basics** for ASCII Chart instructions.

NOTE: An alternative procedure is to Log onto the desired drive, mark the desired program and use the Print utility.

Section 8 - Tool Management

The Tool Page gives the CNC easy access to the tool length offset and tool diameter values for each tool.

Activation of a tool automatically activates the length offset and diameter values recorded on the Tool Page. When a tool is activated, the length offset is applied immediately to provide an accurate Z axis position display.

The active diameter value is important when programming compensated moves and when using cycles that have tool compensation built in. The correct diameter must be active for compensated moves and cycles to function accurately.

Tool length offset and tool diameter values are entered on the numbered lines of the Tool Page. The numbered lines on the Tool Page identify the tool number (**Tool #**) that activates those values.

A tool activation can be programmed as a separate block or included within the block for most moves and cycles. Tool activations programmed as separate blocks are easier to find in a program listing.

The Tool Page can hold the information for up to 99 tools. Tool information is always available for use by the CNC.

Machines equipped with spindle RPM and direction control can also activate spindle speed and direction information. Machines equipped with coolant functions, can also be controlled from the Tool Page.

On machines equipped with collet type tool holders it is impractical to use the Tool Page to store tool length offsets. Tool length offset can be set at tool change (on the fly). Tool Page diameters are still required for compensated moves and when using cycles that have compensation built in.

Entering the Tool Page

Method 1, From the Manual Mode

1. Put the CNC in Manual Mode.
2. Press **TOOL** (F9) (screen displays Tool Page).

Method 2, From the Edit Mode

1. With the CNC in the Edit Mode, press **Tool** (F6) (screen displays Tool Page).

Tool Page Description

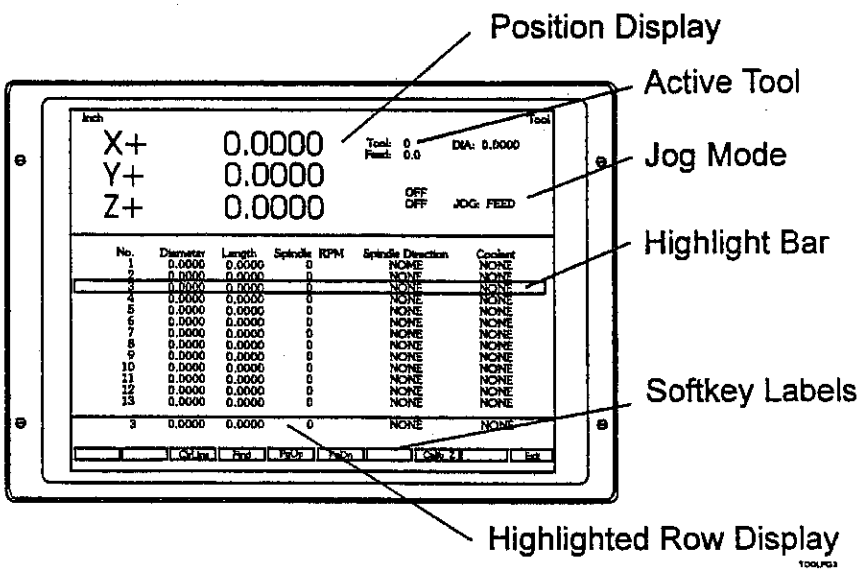


Figure 8- 1, Tool Page

Refer to **Figure 8- 1, Tool Page**. Tool information can only be entered or changed in the row marked by the highlight. Use the (ARROW) keys to move the highlight to the desired row for keying in values. The information on the highlighted row is re-displayed at the bottom of the screen for easier reading. The insertion point (a white underline) marks the location of information to be typed in.

The rows are numbered. Row numbers correspond to tool numbers. When the CNC executes a program block that activates a tool number, the values on that row of the Tool Page are activated.

Activation of Tool# 0 cancels the active diameter and length offset of the CNC. The Tool# 0, Z0 position is usually set as the fully retracted position of the quill.

All of the CNC's Jog features can be run from the Tool Page.

No. Row numbers link the values on a row of the Tool Page to a tool number. A program block that activates a tool number activates the values and settings on that row of the Tool Page. It is the operator's responsibility to keep track of the row number that contains the values for each tool.

Diameter Tool diameter allows the CNC to perform diameter compensation.

Length Tool lengths are Z axis length offsets. Length offsets adjust the Z axis position so the CNC positions the tool's tip.

Spindle RPM	On machines setup for spindle RPM control, these values program the spindle RPM. On machines not set up for spindle functions, this value is only required when the tapping cycle is used.
Spindle Direction	On machines setup for spindle direction control (machines built with M functions), these settings control the direction of the spindle. On machines not setup for spindle control, these settings can be used as a reminder for the operator.
Coolant	On machines setup for coolant control (machines built with M functions), these settings will control the coolant pumps. On machines not setup for coolant control these settings can be used as a reminder for the operator.

Using the Tool Page

Use the up and down (ARROW)  keys to highlight and select tool numbers (row numbers). Use the left and right (ARROW)  keys to move from column to column. Tool page values are automatically converted to their inch or millimeter equivalent with changes in the CNC's Unit Mode. The operator must use values that match the CNC's active Unit Mode.

NOTE: The Tool Page is the only place the CNC converts values between inch and MM equivalents. Programmed positions are not converted when the Unit Mode is changed.

Press **PgUp** (F5) or **PgDn** (F6) to advance the tool table forward or backward by a page.

Finding Tools by Number

1. From the Tool Page, press **Find** (F7) (CNC prompts operator to, "**Enter Tool Number:_.**").
2. Key in a tool number and press (ENTER)  (selected tool is displayed).

Changing Tool Page Values

1. Go to the Tool Page and highlight the desired row. Position the underline to the desired column.

CAUTION: Be sure the CNC is set for the same units, metric or inches, as the value being entered. The units setting can be verified in the top left corner of the Tool Page.

2. Key in the new value with all appropriate decimal values.
3. Press the (ENTER) key (value is entered).

- or -

Press any (ARROW) key (value is entered).

Clearing a Tool (Whole Row)

1. Go to the Tool Page and position the highlight bar to select the row being cleared.
2. Press **ClrLine** (F3) (all values in the row turn to zero).

Clearing a Single Value

1. Go to the Tool Page and move the highlight to select the required row.
2. Position the underline to select the desired column.
3. Press the (CLEAR) key (value changes to zero).

Setting Tool Length Offset

Setting tool length offsets is actually setting a Z axis part zero position for each tool used in a program. Because all tool length offsets are relative to the machine Tool #0, Z0 position, Tool #0, Z0 must be set before setting any tool length offsets. Refer to **Section 3 - Manual Operation and Machine Setup** for instructions on setting the location of Tool# 0, Z0.

NOTE: All of the CNC's Jog features can be run from the Tool Page.

Method 1, From Tool Page, Using Calib Z

1. Activate the Tool Page and position the highlight to select the tool number.
2. Load tool and manually position its tip at the part Z0 position.
3. Press **Calib Z** (F8) (Z position is inserted into length offset column).

Method 2, From Tool Page, Manually Keying In Z Position

1. Retract the Z axis to the **Tool#** 0, Z0 position.
2. Load tool and manually position its tip at the part Z0 position.
3. Manually key in the Z position, with sign, as it appears in the position display and press (ENTER) (Z position is entered in offset column).

NOTE: The actual value of a tool length offset is usually a negative number.

Setting Tool Length Offset for Ball End Mills

When using a ball-end mill to cut contoured surfaces, tool diameter compensation must be used with a tool length offset. Unlike an end cutter, tool length offset for a ball-end mill is not set to position the tip of the tool. The tool length offset for a ball-end mill should be set to position the point ½ of the tool's diameter from the tip.

To set the offset for a ball-end mill:

1. Set the tool length offset using the procedure outlined for a standard end mill.
2. Add ½ of the tool's diameter to the value listed in the **Length** column.
3. Using the (ARROW)  keys, move the insertion point (the underline) to the **Length** column key in the new value.

Section 9 - Communications and DNC

Communications

The 1100M is equipped to exchange data with other RS-232 compatible devices. The baud rate, parity, data bits, stop bits and software parameters of the CNC and the device it is communicating with must agree for RS-232 communications to work.

Default communication parameters are entered in the machine's setup utility when the CNC is first setup. These same parameters can also be changed from the Communications screen. Parameter changes made in the Communications screen are not permanent. Every time the control is powered up the CNC's default settings are reloaded.

CAUTION: Only qualified personnel should attempt to alter settings in the CNC's setup utility. Incorrect settings may disable the machine.

NOTE: The setup file is reloaded whenever setup changes are made and every time the CNC is powered on.

Installing the RS-232 Cable

RS-232 communication requires a cable connection between the sending and receiving machines. On the 1100M, connect the cable to the DE-9 connector located on the relay cabinet. The other end of the cable must be connected to the other machine or computer.

NOTE: The exact location of the DE-9 connector is determined by the machine builder.

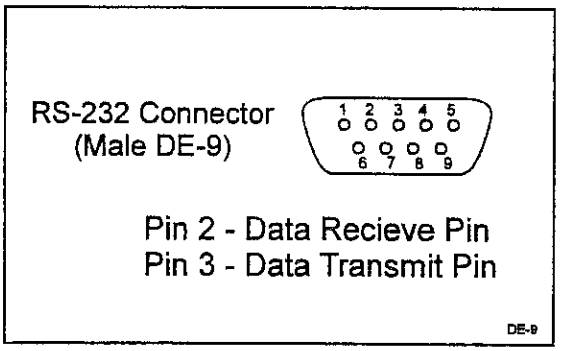


Figure 9 - 1, RS-232 Communications Connector

Refer to **Figure 9 - 1, RS-232 Communications Connector**. It is important to use a cable designed for RS-232 communication. Cables designed for RS-232 communications have the wires between pins 2 & 3 internally crossed.

Data sent from pin 3 (transmit) of one machine must go to pin 2 (receive) of the other. This is the only way RS-232 can work.

NOTE: The same requirements apply for pins 2 & 3 when one of the connectors is a 25 pin serial connector.

Computer cables designed as extension cords for computer peripherals cannot be used for RS-232 communications (pins 2 & 3 are not crossed).

Accessing the Communication Package

To access the Communication screen:

1. With the CNC in the Manual Mode press **PROGRAM** (F2) (Program Directory activates).
2. Press **Utility** (F9) (utility pop-up displays).
3. Refer to **Figure 9 - 2, Communications Screen**. Move highlight to select **Communications** and press (ENTER) [ENTER] (communication screen is displayed).

NOTE: The program marked by the highlight when the Communications screen is activated is the default selection for sending.

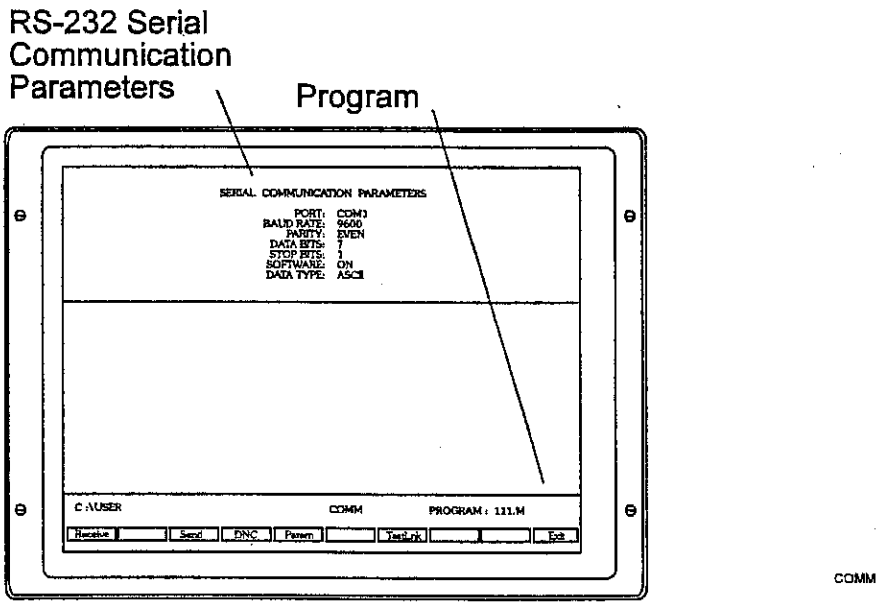


Figure 9 - 2, Communications Screen

PROGRAM	The program selected for transmission.
SERIAL COMMUNICATIONS PARAMETERS	Settings required for the two machines to send and receive programs.

Setting Communication Parameters

This manual does not attempt to discuss the merits of the different parameter choices in depth. Refer to an appropriate computer communications reference for this type of information.

NOTE: Both sending and receiving devices must have the same baud rate, parity, data bits, stop bits and software parameter settings.

Selecting the Communications Port

Most CNC installations have at least one RS-232 connector. The connector is usually mounted somewhere on the cabinet labeled "**RS-232**". The CNC is setup to send signals from it's **COM1** port to the RS-232 connector. This setting cannot be changed on an 1100M machine installation. Personal computers running the off-line software will be able to use **COM2** if the hardware supports it.

To select the communications port:

1. With the Communications screen active, press **Param** (F5) (parameter softkeys activate).
2. Press **Port** (F1) (**PORT:** setting toggles between **COM1** and **COM2**).

Setting the Baud Rate

The 1100M supports the following baud rates:

- 110 baud
- 150 baud
- 300 baud
- 600 baud
- 1200 baud
- 2400 baud
- 4800 baud
- 9600 baud
- 19200 baud

Pressing **Baud** (F2) cycles the setting through the available choices. The current setting is displayed on the Communications screen.

To set the baud rate:

1. With the Communications screen active press **Param** (F5) (parameter softkeys activate).
2. Press **Baud** (F2) (**BAUD RATE:** setting changes).

Setting Parity

The 1100M supports the following parity settings:

- Odd
- Even
- None

Pressing **Parity** (F3) cycles the setting through the available choices. The current setting is displayed on the Communications screen.

To set the parity:

1. With the Communications screen active press **Param** (F5) (parameter softkeys activate).
2. Press **Parity** (F3) (**PARITY:** setting changes).

Setting Data Bits

The 1100M supports the following data bit settings:

- 5 bit
- 6 bit
- 7 bit
- 8 bit

Pressing **Data_b** (F4) cycles the setting through the available choices. The current choice is displayed on the Communications screen.

To set the number of data bits:

1. With the Communications screen active press **Param** (F5) (parameter softkeys activate).
2. Press **Data_b** (F4) (**DATA BITS:** setting changes).

Setting Stop Bits

The 1100M supports the following stop bit settings:

- 0 bit
- 1 bit
- 2 bit

Pressing **Stop_b** (F5) cycles the setting through all available choices. The current choice is displayed on the Communications screen.

To set the number of stop bits:

1. With the Communications screen active press **Param** (F5) (parameter softkeys activate).
2. Press **Stop_b** (F5) (**STOP BITS:** setting changes).

Software Setting

The 1100M supports the following protocol settings:

- ON
- OFF

Software protocol is frequently referred to as "Xon" or "Xoff" or "handshaking" in commercial communications packages.

Pressing **Softw** (F7) cycles the setting through the available choices. The current choice is displayed on the Communications screen.

To set the protocol:

1. With the Communications screen active press **Param** (F5) (parameter softkeys activate).
2. Press **Softw** (F7) (**SOFTWARE:** setting changes).

Setting Data Type

The 1100M supports the following data display types:

- ASCII
- binary

This setting does not affect the data exchanged. Only how it is displayed on the screen during transmission.

Pressing **D_Type** (F8) cycles the setting through the available choices. The current choice is displayed on the Communications screen.

To set data type:

1. With the Communications screen active press **Param** (F5) (parameter softkeys activate).
2. Press **D_Type** (F8) (**DATA TYPE:** setting changes).

Testing the Data Link

The CNC has a utility for testing the data link. The CNC must be connected to another machine for testing. 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 9 - 3, Test Link Screen**. With the Communications screen active, press **TestLnk** (F7) (Link Test screen displays).

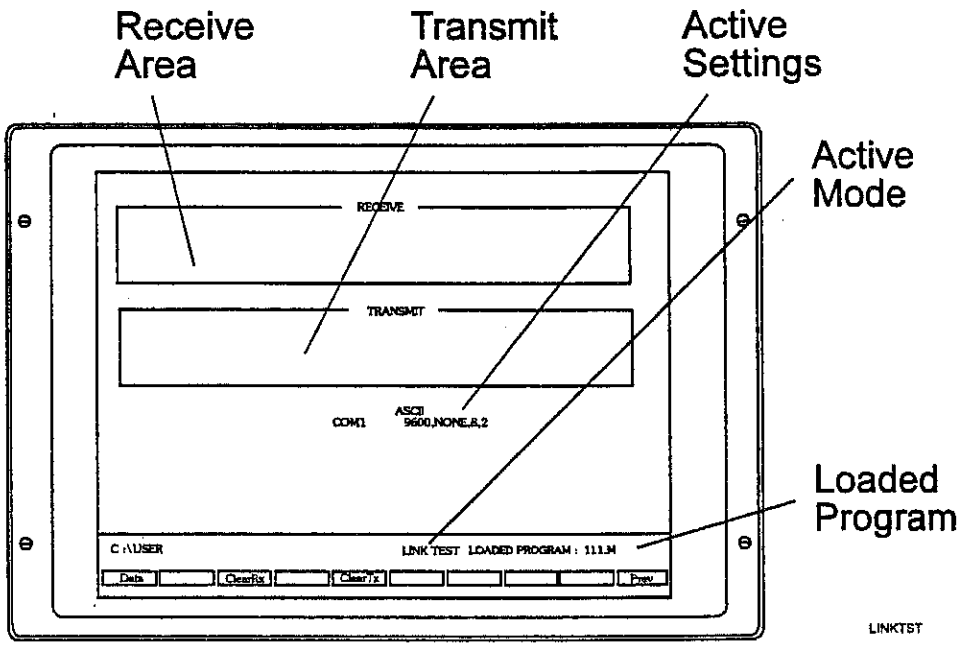


Figure 9 - 3, Test Link Screen

Setting Test Link Display Modes

The operator tests the link by visually verifying that the test data sent matches the test data received. The data type setting determines how characters appear 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 equivalent is displayed. Set both machines to use the same data display type.

NOTE: Hexadecimal characters appear as pairs of numbers or numbers and letters.

To change the Link Test screen data display:

1. From the Link Test screen, press **Data** (F1) to toggle between ASCII and binary modes. The current mode is displayed in the settings area of the Link Test screen.

Testing 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 type in any series of numbers (these characters will appear in the transmit area).

4. Verify the characters are correctly received at the other machine.
5. Set the other machine (or computer) to transmit.
6. Manually transmit a series of characters from the other machine (or computer).
7. Verify the characters sent to the CNC correctly appear in the receive area (test is complete).

Clearing the Receive Area

Clear the receive area by pressing the **ClearRx** (F3) key.

Clearing the Transmit Area

Clear the transmit area by pressing the **ClearTx** (F5) key.

Sending a Program

The name of the program marked by the highlight when the Communications screen was activated, is listed next to the **PROGRAM** label on the screen. The CNC will send this program unless another program is selected. The CNC will prompt the operator to select another program during the sending process.

1. With the Communications screen active, the parameters set and the link tested, press **Send** (F3) (CNC prompts the operator to send the selected program or to change the program being sent).
2. Press **Select** (F1) (CNC sends program).

   - or -

   Press **Change** (F3) (ASCII softkeys appear to key in the name of the desired program).

NOTE: Refer to **Section 2 - CNC Console And Software Basics** for ASCII chart instructions.

Receiving a Program

The operator must key in a name for received programs before receiving it from the other machine.

To receive a program:

1. With the Communications screen active, the parameters set and the link tested, press **Receive** (F1) (CNC prompts operator to key in a name for the incoming program, ASCII softkeys appear).
2. Key in the desired name and press (ENTER) [ENTER] (CNC displays a "**READY TO RECEIVE . . .**" message).
3. Start transmission from sending machine.

NOTE: Refer to **Section 2 - CNC Console And Software Basics** for ASCII chart instructions.

Setting the Transmission and Receiving Display

If the CNC is transmitting or receiving with the Text Mode active, the exchanged program will be displayed on the screen. If the Text Mode is off, the display area will remain blank.

The **Text** (F4) softkey label is highlighted when the Text Mode is active.

The "Transfer In Progress" symbol (just above the softkey line) cycles when data is exchanged.

Toggle the CNC in and out of the Text Mode by pressing **Text** (F4) while transmitting or receiving.

Holding Transmission/Receiving Operations

When sending or receiving programs the operation can be paused by pressing the **Hold** (F1) key. Press **Resume** (F2) to continue the exchange.

NOTE: If the CNC is receiving a program and either machine's software parameter (Xon/Xoff) is set to **OFF**, there is a possibility a hold will overload the CNC's buffer causing portions of the program to be lost. Operating with the software parameter (Xon/Xoff) set to **ON** is recommended.

Running in DNC

Under Direct Numeric Control (DNC) the incoming program is executed as it is received over the RS-232 link. The CNC can run incoming programs in the Single Step or Automatic Modes. Real Time Draw Mode is available when using DNC.

The 1100M CNC only executes programs written in the conversational format. Do not execute a program in the DNC Mode if the program is not in the correct format.

In the DNC Mode, the CNC cannot look ahead to verify that a subprogram is complete. Do not execute a program in the DNC Mode if it contains a call to a subprogram (contains **Call**, **Loop** or **RMS** blocks).

Refer to **Figure 9 - 4, DNC Screen**. The **DNC** screen is similar to other operating screens, but with communications information added.

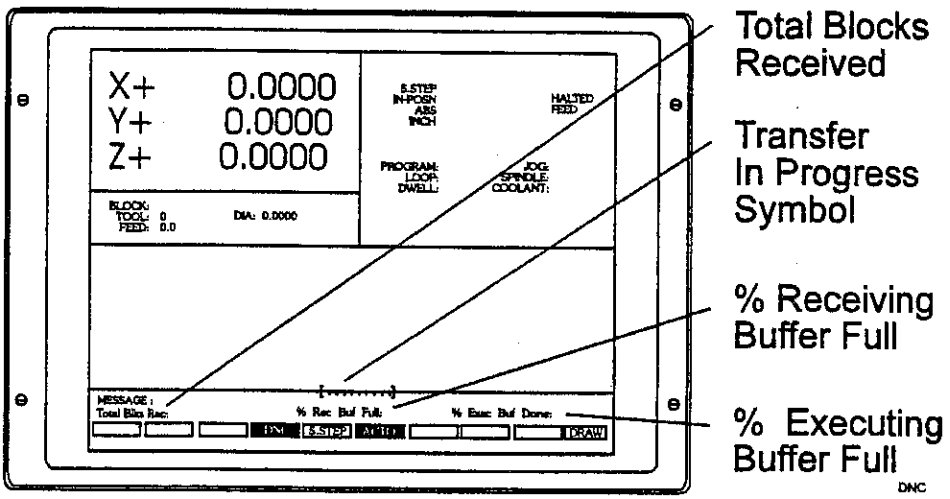


Figure 9 - 4, DNC Screen

Total Blks Rec: The total number of program blocks the CNC has received at that point.

% Receiving Buffer Full Percentage of receiving buffer used.

% Executing Buffer Full Percentage of executing buffer used.

Over an RS-232 link, programs are transmitted faster than they can be executed. This makes it necessary to manage the memory and timing between the two machines.

The CNC uses two buffers. One buffer receives the incoming program while the contents of the second buffer is executed. When the executing buffer is empty, they swap. The contents of the full buffer is executed while the empty buffer receives.

When the software protocol (Xon/Xoff) is used, the CNC can signal the sending machine to pause until it has a buffer with room to receive.

NOTE: Most machines default to the buffer mode for DNC operations. Some machines may be set to default to the Drip Feed Mode for DNC. In Drip Feed Mode program blocks are transmitted and executed one at a time, without buffering.

To put the CNC in the Direct Numeric Control Mode:

1. With the Communications screen active, parameters set and the link tested, press **DNC** (F4) (**DNC** softkey label highlights).
2. Press **Receive** (F1) (**DNC** screen activates, message line prompts operator to start transmission from other machine).
3. Press (START)  (CNC runs incoming program).

Section 10 - The Calculator

The 1100M CNC features a powerful calculator package that contains three separate calculators:

- The Mathematics Calculator
- The Right Triangle Calculator
- The Geometry Calculator

Calculator solutions can be recalled directly into the labeled fields of a Graphic Menu when programming. All three calculators have separate memory for storing and recalling solutions.

The Math Calculator

The math calculator performs all of the operations commonly found in a scientific calculator. These include basic math, trigonometry, unit conversion, logs, exponentials, angle/radian conversions and finding the inverse.

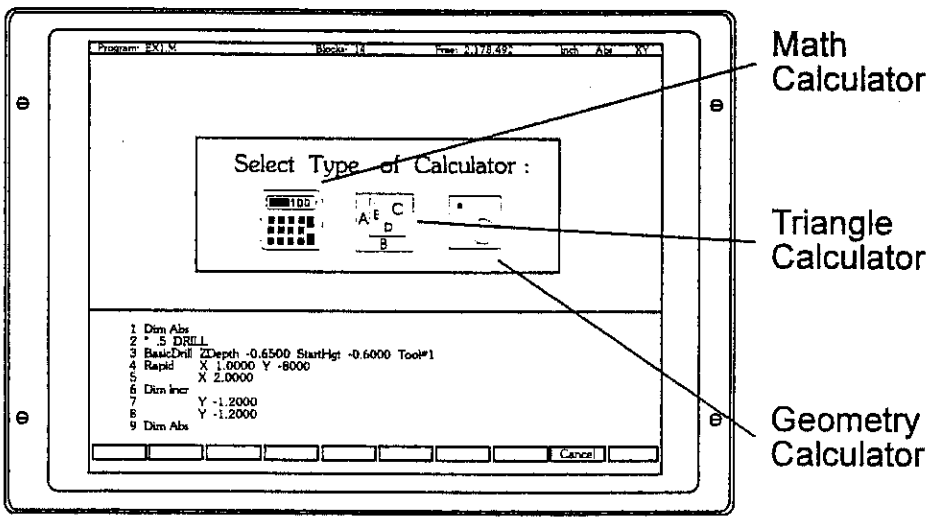


Figure 10 - 1, Calculator Selection Menu

Activating the Math Calculator

To activate the Math Calculator:

1. Refer to **Figure 10 - 1, Calculator Selection Menu**. With the CNC in the Edit or MDI Mode press **Calc** (F7) (calculator selection menu appears on the screen).

2. Using the (ARROW) keys move the highlight to select the (MATH CALCULATOR) template and press (ENTER) (math calculator activates).

Math Calculator Basics

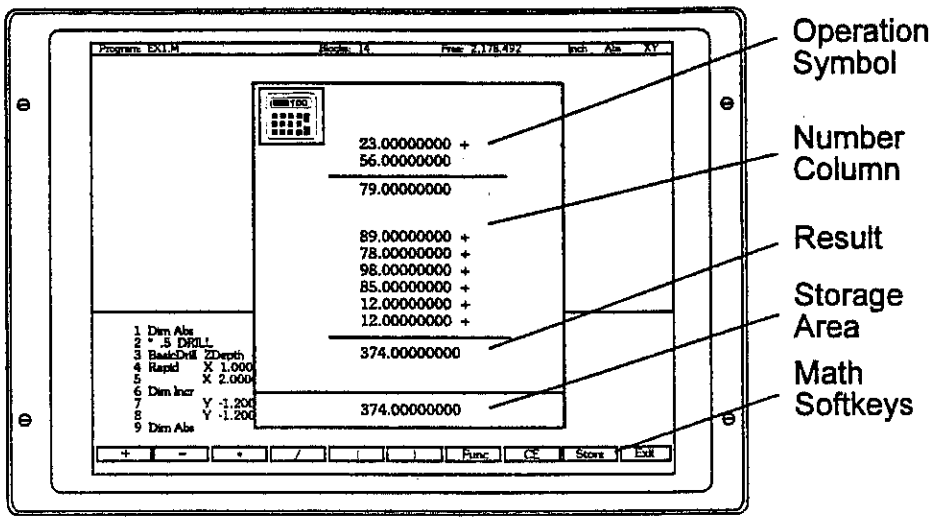


Figure 10 - 2, Math Calculator & Softkeys

Refer to **Figure 10 - 2, Math Calculator & Softkeys**. The math calculator appears in a pop-up box in the center of the screen.

Numbers appear in the storage area as they are keyed in. Math operations are selected from the softkeys.

Refer to **Table 10 - 1, Math Operation Softkeys**. After keying in the first number of a calculation, press an operation softkey to add the number to the column and select the operation performed with the next number.

After keying in the last number of a calculation, use the (ENTER) key to add the number to the column and find the result. Calculation results are displayed in the storage area.

Press **Store** (F9) to put the value displayed in the storage area into the calculator's memory.

Clear any previous calculations by pressing **CE** (F8) (clear entry).

If the calculator is asked to perform an illogical calculation, it will display either an error message or a row of asterisks.

Table 10 - 1, Math Operation Softkeys

Operation	Softkey Label	Softkey Number
Addition	+	(F1)
Subtraction	-	(F2)
Multiplication	*	(F3)
Division	/	(F4)
Left Hand Parenthesis	(	(F5)
Right Hand Parenthesis	)	(F6)
Misc. Function Pop-up Menu	**Func**	(F7)
Clear	**CE**	(F8)
Store Number For Recall	**Store**	(F9)

NOTE: Off-line keyboard users can use the keypad operators.

Adding Two Numbers

To add two numbers:

1. With the math calculator active, key in the first number and press + (F1) (number and operation symbol are added to the column).
2. Key in the second number and press (ENTER) (second number is added to the column, sum is displayed in storage area).

Subtracting a Number

To subtract one number from another:

1. With the math calculator active, key in the first number and press - (F2) (number and operation symbol are added to the column).
2. Key in the number being subtracted and press (ENTER) (second number is added to the column, resulting difference appears in storage area).

Multiplying Two Numbers

To multiply two numbers:

1. With the math calculator active, key in the first factor and press * (F3) (number and operation symbol are added to column).
2. Key in the second factor and press (ENTER) (second number is added to column, product appears in the storage area).

Dividing One Number Into Another

To divide one number into another:

1. With the math calculator active, key in the first number and press / (F4) (number and operation symbol are added to the column).
2. Key in the divisor and press (ENTER) ENTER (second number and operation symbol are added to the column, quotient (result) appears in the storage area).

Math with a Column of Numbers

Mathematics performed on a column of numbers is solved like a continuous equation. A column of numbers with mixed operations is calculated using the mathematic standard order of operations. The result is generated after the (ENTER) ENTER key is pressed.

When parentheses are used, the operation within parentheses is solved when the parentheses are closed. The solved value is substituted it into the column (equation) at that point.

NOTE: The mathematic standard order of operations in an equation (without parentheses) is: multiplication first, division second, addition third, and subtraction fourth.

Using Parentheses

When it is necessary to perform mathematic operations in a non-standard order, parentheses are used. For instance, finding a sum before using it as a multiplication factor. Operations within parentheses are performed before the standard order of operations begins.

Parentheses must always be used in pairs. No result will be calculated if the parentheses are not closed. Parentheses can be nested inside of each other provided they are used in pairs.

Operations within parentheses are performed first to last, (top to bottom) inner most pairs first.

Using Additional Functions

Refer to **Table 10 - 2, Function Selection Pop-up Listing**. The **Func** (F7) key activates a pop-up menu that provides access to the additional math functions listed on the table. These functions all perform their listed operation on a single number.

To use an additional function:

1. With the math calculator active, key in number and press **Func** (F7) (function pop-up menu appears to the right of the calculator).

2. Using the (ARROW)  keys, move the highlight to select a function and press (ENTER)  (result appears in the storage area).

Table 10 - 2, Function Selection Pop-up Listing

Pop-up Box Label	Function
Sine	Sine Function
Cosine	Cosine Function
Asine	Arcsine Function
Acosine	Arcosine 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

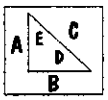
Storing Numbers from the Math Calculator

Press the **Store** (F9) softkey to put the number from the storage area into the calculator's memory. The math calculator's memory holds 64 numbers. Numbers stored in memory can be recalled directly into a program.

The Right Triangle Calculator

The Right Triangle Calculator solves the angles and sides of a right triangle once given: any two sides, any two angles, or an angle and a side. Any or all of the values can be put in the calculator's memory for recall into a program.

Activating the Triangle Calculator

1. Refer to **Figure 10 - 1, Calculator Selection Menu**. With the CNC in the Edit or MDI Mode, press **Calc** (F7) (calculator selection menu appears on the screen).
2. Using the (ARROW)  keys, move the highlight to select the (TRIANGLE CALCULATOR)  template and press (ENTER)  (Triangle Calculator activates).

Using the Triangle Calculator

Refer to **Figure 10 - 3, Right Triangle Calculator Screen**. The Right Triangle Calculator only solves right triangles. The Right Triangle Calculator's pop-up screen contains three main areas: the entry/solution area, the diagram area and the message area.

The entry fields in the entry/solution area are labeled to correspond to the sides and angles shown in the labeled diagram.

To use the Right Triangle Calculator, the operator keys values into the field for any two angles, any two sides or one angle and one side.

Clear a single incorrect value by using the highlight to select the value and press (CLEAR) CLEAR.

Clear all of the values by pressing **ClrAll** (F8).

Solved angles and sides are displayed in the unused fields followed by an asterisk. A scaled drawing of the solved triangle is also displayed in the diagram area.

If illogical values are keyed in the calculator will display an error message.

Right Triangle Calculator solutions can be used (recalled) directly into a program when writing positive (+) incremental moves. These same solutions can be adjusted to produce absolute coordinates by recalling the values into the Math Calculator and adding an offset value.

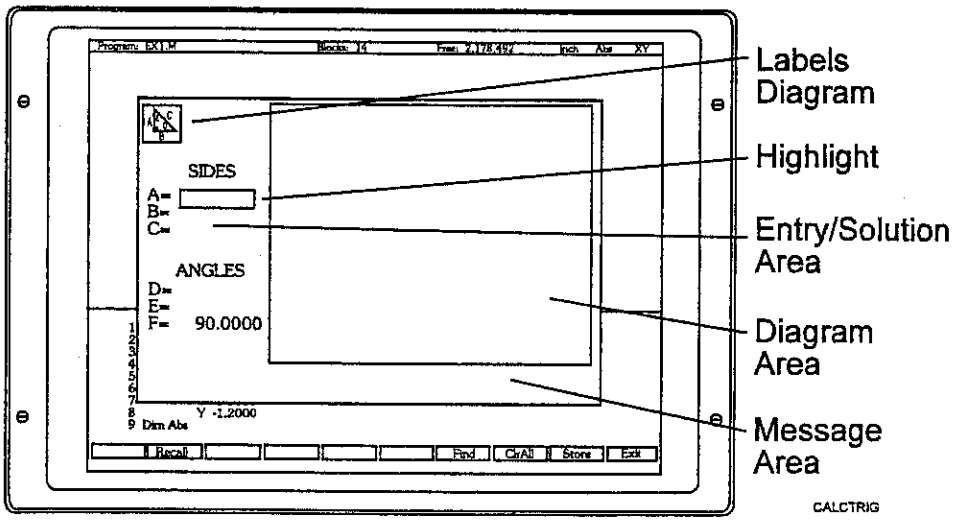


Figure 10 - 3, Right Triangle Calculator Screen

To solve for the sides and angles of a right triangle:

1. With the Right Triangle Calculator active, use the (ARROW) keys to move the highlight to select the field for the first side or angle and key in the value.
2. Use the (ARROW) keys to move the highlight to select a second side or angle and key in the value.
3. Press **Find** (F7) (scaled drawing of triangle appears in the solution box, calculated sides and angles are displayed in unused fields).

Storing Right Triangle Calculator Results

The length of any side or the measure of any angle can be saved to memory. The Right Triangle Calculator can store up to 64 values.

To store a value:

1. Solve the required triangle.
2. Use the (ARROW) keys to position the highlight to select the value being stored.
3. Press (ENTER) (selected value is stored in memory).

The Geometry Calculator

Most positions in a CNC program are defined using Cartesian coordinates (X, Y, Z axis values). Sometimes the coordinates of a position on the print must be determined from the location of the other objects on the print. Unknown positions can usually be defined using a geometric construction of lines, circles and angles that use known positions as a reference.

The Geometry Calculator gives the operator an assortment of line, circle, angle and point templates. These templates are used to sketch a geometry construction that identifies the unknown position. A point is inserted at the required position on the construction. The calculator automatically solves the coordinates of all points. Coordinate positions can be stored in memory for recall into a program.

Activating the Geometry Calculator

1. Refer to **Figure 10 - 1, Calculator Selection Menu**. With the CNC in the Edit or MDI Mode, press **Calc** (F7) (calculator selection menu appears on the screen).

2. Use the (ARROW) keys to move the highlight to select the (GEOMETRY CALCULATOR) template and press (ENTER) (Geometry Calculator activates).

Geometry Calculator Screen

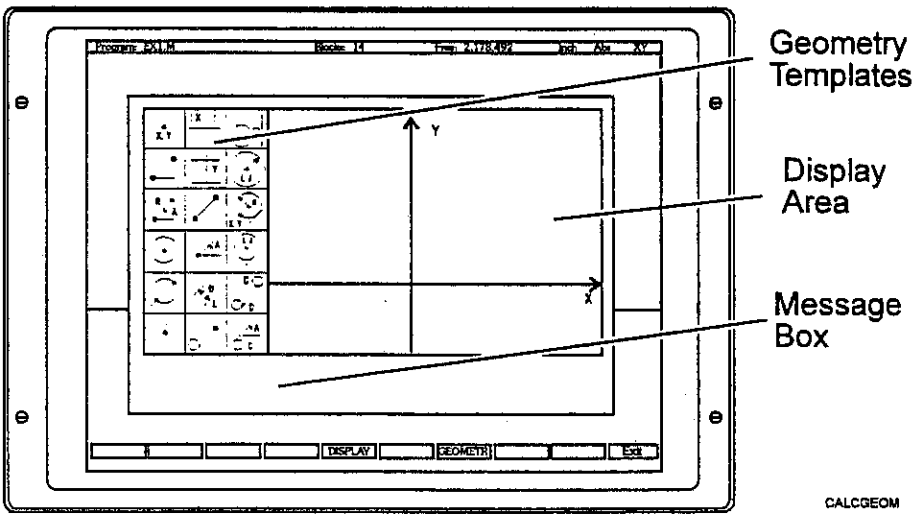


Figure 10 - 4, Geometry Calculator

Refer to **Figure 10 - 4, Geometry Calculator**. The Geometry Calculator consists of a pop-up screen with a display area, a message area and eighteen geometry templates. The geometry sketch is constructed in the display area.

Use the **DISPLAY** (F5) softkey 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 displays messages to prompt the operator through the procedure required by each tool. Pay close attention to the prompts.

Using the Geometry Calculator

Points, lines and circles are the basic elements of all sketches. These elements are inserted or defined using the eighteen geometry templates.

Use the (ARROW) keys to move the highlight for tool selection.

Press the (ENTER) key to activate the selected tool.

Points, lines and circles can all be defined more than one way. Each geometry tool defines an element a different way.

Sometimes more than one tool is required to define an element. Templates that require a pre-existing point will prompt the operator to activate one of the point identification templates.

Display area grid lines are solid. Lines and circles are drawn with dotted lines. Points are marked with a "**x**" (small case x).

All elements in a sketch are assigned a number. Sketches can contain a maximum of 50 elements.

When an element is added to a sketch, it is also added to 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 it crosses X and/or Y axis.

Points - Absolute positions.

The important details of every element are automatically numbered and saved in the calculators memory. The memory contains the following details about each element:

Points - Absolute position.

Circles - Absolute position of center.

NOTE: The coordinates of an intersection between two elements are not automatically saved in memory. Insert a point at the intersection to save its coordinates.

Deleting an element from the sketch removes the element from the geometry list and removes its position from the calculator's memory.

There are three categories of Geometry templates.

- Point templates. Refer to **Table 10 - 3, Point Templates**.
- Line templates. Refer to **Table 10 - 4, Line Templates**.
- Circle templates. Refer to **Table 10 - 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. Operators are advised to experiment to get a feel for how each template operates.

NOTE: After a series of deletions and additions, the display may appear incomplete. Press **Display** (F5) and select **Redraw** to refresh the screen.

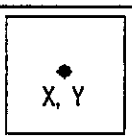
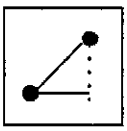
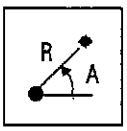
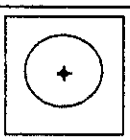
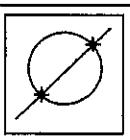
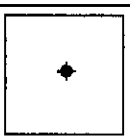
Point Templates

Some point templates insert points at a position defined by the operator. Other point templates use the other elements as a reference.

Many line and circle templates display a **"Select point definition . . ."** message when activated. This tells the operator that the selected tool requires the selection (or creation) of a reference point.

Templates that insert points at circle centers and element intersections will prompt the operator for the element number(s) involved. Once entered, all possible intersections are displayed and the user prompted to select one.

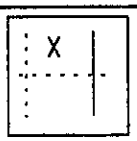
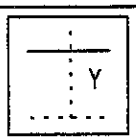
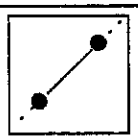
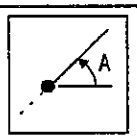
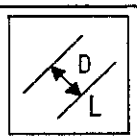
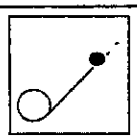
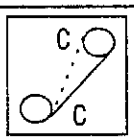
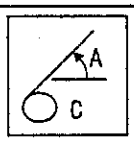
Table 10 - 3, Point Templates

Template	Purpose	Requirements
	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.	– Existing point must already be an element of the sketch. – Must know incremental X and Y distances from existing point.
	Defines a point at radius and angle from existing point.	– Existing point must already be an element of the sketch. – Must know distance from existing point. – Must know number of degrees and direction (+/-) from 3 o'clock position.
	Defines a point at 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 which intersection to use when more than one intersection exists.	– Sketch must contain two intersecting (or tangent) 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

The line templates all use other elements or axes positions as a reference. Templates that draw lines tangent to circles will display all possible tangent lines and prompt the operator to select one.

Table 10 - 4, Line Templates

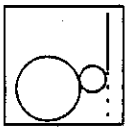
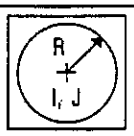
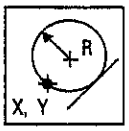
Template	Purpose	Requirements
	Constructs a line parallel to Y axis.	– Must know absolute X axis position of the line.
	Constructs a line parallel to X axis.	– Must know absolute Y axis position the line.
	Constructs a line between any two points. Prompts user to select a point tool to define each endpoint.	– Use any point tool to locate the two endpoints.
	Constructs a line through a point, rotated specified number of degrees from 3 o'clock position. Prompts user to select any point tool to define point of rotation.	– Must know number of degrees and direction (+/-) from 3 o'clock position. – Use any point tool 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 that is tangent to an existing circle . Prompts user to select any point tool 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 tool to locate the point.
	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 3 o'clock position and tangent to existing circle.	– Circle must already be an element of the sketch. – Must know number of degrees of rotation from 3 o'clock position.

Circle Templates

The circle templates all use other elements as a positioning reference.

Templates that draw circles tangent to other circles, lines or points will display all of the possible tangent circles and prompt the operator to select one.

Table 10 - 5, Circle Templates

Template	Purpose	Requirements
	Constructs a circle of specified radius tangent to any two existing elements. All possible tangent circles are displayed. The user selects 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 tool to define the center point.	– Use any point tool 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 tool to define point.	– Line must already be an element of the sketch. – Use any point tool to locate the point.
	Constructs a circle centered on a point and tangent to an existing line. Prompts user to select any point tool to define center point.	– Line must already be an element of the sketch. – Use any point tool to locate the center.

Deleting Selected Elements

To delete an element from the sketch:

1. With the Geometry Calculator active, press **GEOMETR** (F7) (pop-up menu displays).
2. Using (ARROW)  keys, position highlight to select **Delete Item** and press (ENTER)  (screen prompts for element number being deleted).
3. Key in element number and press (ENTER)  (element is removed from screen).

Deleting all Elements

To clear all elements from the display area:

1. With the Geometry Calculator active, press **GEOMETR** (F7) (pop-up menu displays).
2. Using (ARROW)  keys, position highlight to select **Delete All** and press (ENTER)  (screen prompts user to confirm deletion).
3. Press **Yes** (F1) (display area is cleared).

   - or -

   Press **No** (F2) (no change is made).

Last Position Recall

The Geometry Calculator has a feature that permits the operator to recall the last X position, Y position or circle used by the calculator. This feature is available whenever the calculator prompts for an X position, a Y position or a circle.

When used, the calculator will recall a value of a type asked for by the prompt.

To recall a position:

1. When the calculator prompts for an X position, a Y position or a circle number, press the (UP or DOWN ARROW)  key to recall the last X position, Y position or circle number used.

Recalling Values into a Program

Refer to **Figure 10 - 5, Calculator Recall Menu**. The program editor always displays a **Recall** (F2) softkey when a Graphic Menu is activated. Calculator solutions that have been stored in memory can be recalled directly into the entry fields of a Graphic Menu. Saved values must be recalled from the same calculator they were saved in. Calculator recall selection is made from the recall selection pop-up menu.

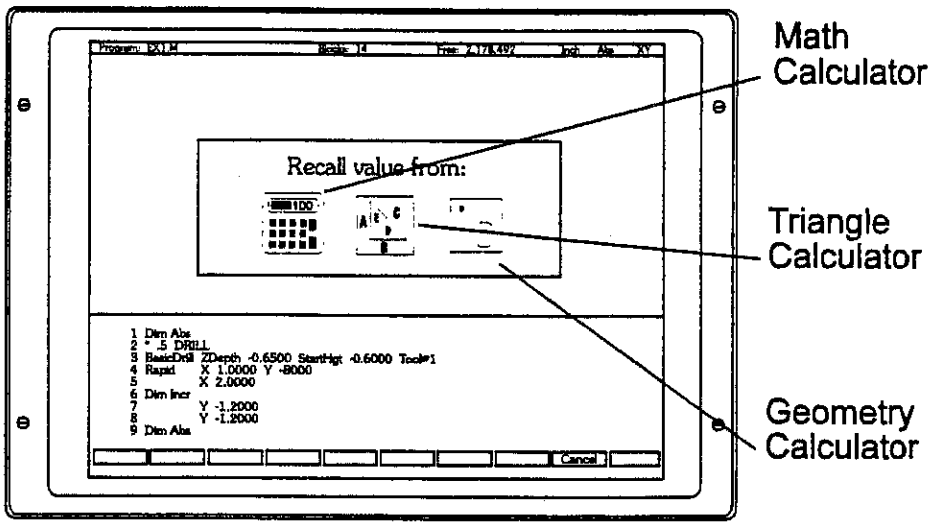


Figure 10 - 5, Calculator Recall Menu

Recalling Values from the Math Calculator

To recall a value from the Math Calculator

1. Refer to **Figure 10 - 5, Calculator Recall Menu**.
2. From the Graphic Menu for the block being edited, use the (ARROW) keys to move the highlight to select the field and press **Recall** (F2) (calculator recall selection menu appears).
3. Refer to **Figure 10 - 6, Math Calculator Select Value Pop-up**.
4. Using the (ARROW) keys, position the highlight to select the (MATH CALCULATOR) template and press (ENTER) (**Select value:** pop-up appears, most recently saved value is shown at the top).
5. Using the (ARROW) keys, position the highlight to select the required value and press (ENTER) (stored value is copied into the Graphic Menu).

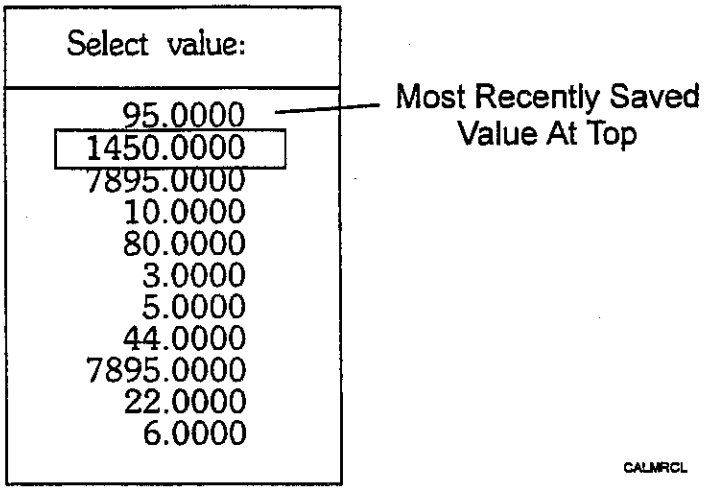


Figure 10 - 6, Math Calculator Select Value Pop-up

Recalling Values from the Right Triangle Calculator

Refer to **Figure 10 - 5, Calculator Recall Menu**.

To recall values from the Right Triangle Calculator:

1. From the Graphic Menu for the block being edited, use the (ARROW) keys to move the highlight and select the field receiving the recalled value.
2. Press **Recall** (F2) (**Select value:** menu appears).
3. Refer to **Figure 10 - 7, Triangle Calculator Recall Pop-up**.
4. Using the (ARROW) keys, position the highlight to select the (TRIANGLE CALCULATOR) template and press (ENTER) (Triangle Calculator memory selection pop-up appears).
5. Using the (ARROW) keys, position the highlight to select the required value and press (ENTER) (stored value is copied into the Graphic Menu).

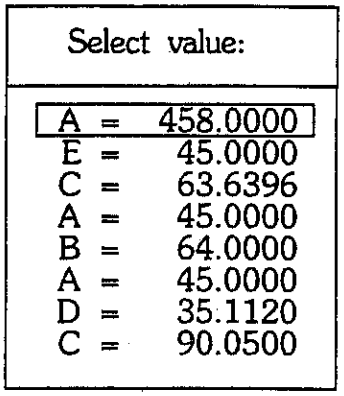


Figure 10 - 7, Triangle Calculator Recall Pop-up

Recalling Values from the Geometry Calculator

Geometry Calculator values are recalled 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 is numbered to correspond with the elements on the sketch.

When the position on the recall list is followed by a **[C]**, it is the center of one of the circles in the sketch.

When the position on the recall list is followed by a **[P]**, it is the location of a point in the sketch.

Either one or both of the position coordinates can be recalled.

Refer to **Figure 10 - 5, Calculator Recall Menu**.

To recall a value from the Geometry Calculator:

1. From the Graphic Menu for the block being edited use the (ARROW) keys to move the highlight and select the field receiving the recalled value.
2. Press **Recall** (F2) (calculator recall selection menu appears).
3. Refer to **Figure 10 - 8, Geometry Calculator Select point Pop-up Menu**.
4. Using the (ARROW) keys, position the highlight to select the (GEOMETRY CALCULATOR) template and press (ENTER) (The **Select point:** pop-up menu appears).
5. Using the (ARROW) keys, position the highlight to select the required values and press (ENTER) (**Select term:** pop-up menu prompts for **Both X and Y** values, **X only** or **Y only**).

6. Use the (ARROW) keys to select the required terms and press (ENTER) (selected values are copied to the Graphic Menu).

NOTE: Position coordinates can only be recalled into Graphic Menu entry fields that require X and/or Y coordinate values.

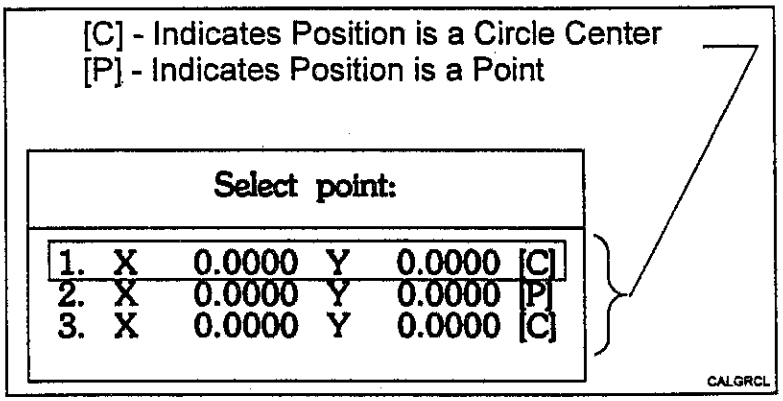


Figure 10 - 8, Geometry Calculator Select point Pop-up Menu

Recalling Values from One Calculator into Another

Both the Math Calculator and the Triangle Calculator can recall values from other calculators.

In the Triangle Calculator, recall is activated by pressing **Recall** (F2). In the Math Calculator, recall is activated by pressing any of the (ARROW) keys.

Section 11 - Software, Keyboard, and Printer Installation

Off-line Software Installation

The off-line version of the 1100M CNC software requires a computer equipped with a 386 SX processor with matching coprocessor (or better). Intel math coprocessors are preferred. A minimum of 2M of RAM is required. The Hard disk drive must have a minimum of 2M of space available.

The software is compatible with Microsoft Himem.sys and requires Microsoft DOS (version 3.3 or greater). No other extended memory managers or operating systems are recommended.

The off-line software is not compatible with multitasking environments such as Microsoft Windows 3.1, Windows NT, Windows 95 and OS/2. The off-line software may also conflict with certain terminate and stay resident programs.

To install the software:

1. Insert the 3.5 inch floppy disk into an appropriate drive.
2. From the DOS prompt, type in [drive]:\install and press (ENTER) (installation process begins).
3. Follow the installation program's instructions until installation is complete. The computer will automatically reboot after setup.

NOTE: During the software installation process, the control will prompt the operator to install sample programs. The sample programs installed include the programs listed in **Section 12 - Sample Programs and Practice Examples** that are not demonstrated in the exercises. The names for 2 axis sample programs begin with "2AX-". The names for three axis sample programs begin with "3AX-".

Running Off-line Software

To run the off-line program:

1. Go to the **S1100** directory.
2. Type in **PS11** and press (ENTER) (program starts)

Machine Software Installation

CNC software is initially loaded when the machine is built. Installation should only be necessary for software updates.

CAUTION: During the installation procedure, the control may prompt to overwrite the existing configuration file. Choose NO. Do not overwrite the configuration file under any circumstances. Overwriting the configuration file may disable the machine.

1. With the CNC in Manual Mode, press **EXIT** (F10) (CNC's startup screen is displayed).
2. Position the highlight to select **Setup Utility** and press (ENTER) (**Setup Options** menu appears).
3. Position the highlight bar to select **Machine/Installation** and press (ENTER) (**Machine/Installation** menu appears).
4. Position the highlight to select **Software Upgrade** and press (ENTER) (installation sequence begins, follow the instructions on the screen).

NOTE: During the software installation process, the control will prompt the operator to install sample programs. The sample programs installed include the programs listed in **Section 12 - Sample Programs and Practice Examples** that are not demonstrated in the exercises. The names for 2 axis sample programs begin with "2AX-". The names for three axis sample programs begin with "3AX-".

Software Option Kit Installation

Software Options Kits are kits that 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 the operation of the purchased options. The key must be installed for the options to work.

Procedure:

CAUTION: During the installation procedure, the control may prompt to overwrite the existing configuration file. Choose NO. Do not overwrite the configuration file under any circumstances. Overwriting the configuration file may disable the machine.

1. Use the shipping document to verify that the kit contains the ordered options.
2. Gain access to the female DB-25 printer port connector located inside the computer cabinet. Install the hardware key.

NOTE: If an Anilam Electronics hardware key is already installed, remove it before installing the new key.

3. With the CNC in Manual Mode, press **EXIT** (F10) (CNC's startup screen is displayed).

4. Position the highlight to select **Setup Utility** and press (ENTER) (**Setup Options** menu appears).
5. Position the highlight bar to select **Machine/Installation** and press (ENTER) (**Machine/Installation** menu appears).
6. Position the highlight to select **Software Upgrade** and press (ENTER) (installation sequence begins, follow the instructions on the screen).

NOTE: After installing the software, additional setup may be required to enable the purchased option.

NOTE: With the installation of an option kit, the original program disk and hardware key are no longer useful and should be discarded.

Printer Installation

To install a printer, plug the printer cable into the DB-25 connector located on the computer chassis inside the CNC Cabinet. If a software key is already installed in the connector, leave the key in place and plug the printer into the back of the key.

Keyboard Installation

To install a standard PC keyboard, plug the keyboard connector into the DIN 41524, 5 pin socket located on the back of the console. Installing a keyboard deactivates the keypad on the console. To reactivate the keypad, unplug the keyboard.

The keyboard cannot be used to perform an emergency shutdown or reactivate the machine servos. These functions can only be performed from the console.

NOTE: The DIN 41524, 5-pin connector is a standard PC keyboard connector. A size adapter may be required for some installations.

WARNING: Emergency shutdown cannot be made using a keyboard. Use the (E - STOP) button to perform emergency machine shutdowns.

NOTE: Industrial grade keyboards are recommended for use in shop environments.

Using Softkeys from a keyboard

The (F1) through (F10) keyboard keys are equivalent to the CNC's (F1) through (F10) softkeys. The shift key activates secondary softkey functions when they are available. Use secondary functions by pressing the (SHIFT) key and the appropriate function key at the same time.

Keypad Equivalent Keyboard Keys

Refer to **Table 11 - 1, Keyboard Keystroke Equivalents**, **Table 11 - 2, Off-line Keyboard, Tool Page Utilities** and **Table 11 - 3, Off-line Keyboard, Program File Directory Utilities** for the keyboard equivalents to the keypad keys. Ordinary letter, number and decimal entries are simply typed in. All other keyboard equivalents require two keys be pressed at once (ALT + another key).

Making Jog Moves from a Keyboard

To make jog moves on a keyboard use the ALT+ F1 through ALT + F5 keys to select the increment. Pressing the axis/direction key starts the jog.

To make continuous jog moves use the ALT+ F1 through ALT + F2 keys to select the move rate (Rapid or Feed). Press the axis/direction key to start the jog. Pressing any keyboard key stops a continuous jog.

Table 11 - 1, Keyboard Keystroke Equivalents

Key Name	Key Face	Keystroke Equivalent
(ABS/INCR)		(Alt + E)
(+/-)		(+) or (-)
(CLEAR)		(Alt + C) or (Delete)
(ARROWS)		(ARROWS)
(ENTER)		(Enter)
RAPID		(Alt + F1)
FEED		(Alt + F2)
JOG: 100		(Alt + F3)
JOG: 10		(Alt + F4)
JOG: 1		(Alt + F5)
(STOP CONTINUOUS JOG)	none	(Alt + T)
(X+)		(Alt + X)
(X-)		(Alt + K)
(Y+)		(Alt + Y)
(Y-)		(Alt + L)
(Z+)		(Alt + Z)
(Z-)		(Alt + N)
(START)		(Alt + S)
(HOLD)		(Alt + H)

Table 11 - 1 (Continued), Keyboard Keystroke Equivalents

Key Name	Key Face	Keystroke Equivalent
(FEEDRATE OVERIDE) 0%		(Alt + 0)
(FEEDRATE OVERIDE) 10%		(Alt + 1)
(FEEDRATE OVERIDE) 20%		(Alt + 2)
(FEEDRATE OVERIDE) 30%		(Alt + 3)
(FEEDRATE OVERIDE) 40%		(Alt + 4)
(FEEDRATE OVERIDE) 50%		(Alt + 5)
(FEEDRATE OVERIDE) 60%		(Alt + 6)
(FEEDRATE OVERIDE) 70%		(Alt + 7)
(FEEDRATE OVERIDE) 80%		(Alt + 8)
(FEEDRATE OVERIDE) 90%		(Alt + 9)
(FEEDRATE OVERIDE) 100%		(Alt + A)
(FEEDRATE OVERIDE) 110%		(Alt + B)
(FEEDRATE OVERIDE) 120%		(Alt + D)
(SPINDLE FORWARD)	SPDL FWD	(Alt + F)
(SPINDLE OFF)	SPDL OFF	(Alt + O)
(SPINDLE REV)	SPDL REV	(Alt + G)

Table 11 - 2, Off-line Keyboard, Tool Page Utilities

Function	Keystroke	Operation
Clear all entries on tool page.	(Shift + F3)	Clears all entries from tool page, operator is prompted for confirmation before page is cleared.
Exit tool page without saving changes.	(Shift + F10)	Exits tool page, any edits made since last save are lost.

Table 11 - 3, Off-line Keyboard, Program File Directory Utilities

Function	Keystroke	Operation
Allows creation of a subdirectory.	(Shift + F2)	Prompt for **NEW DIR:__** appears above softkey labels.
Allows deletion of any program in specified path.	(Shift + F3)	Prompt for PROGRAM TO DELETE: _ appears above softkey labels.

Section 12 - Sample Programs and Practice Exercises

This section of the manual contains sample programs and practice exercises.

First time programmers are encouraged to learn about the CNC by doing the exercises. Experienced operators may sometimes find it useful to look at one of the sample programs.

Practice Exercises #1 - #5 list all the keystrokes required to create, program and check Sample Program #1 - Drill Program.

Practice Exercises #6 - #8 list all the keystrokes required to create (using the Geometry Calculator), program and check Sample Program #4 - Irregular Pocket Program.

NOTE: During the software installation process, the control will prompt the operator to install sample programs. The sample programs installed include the programs from this section that are not demonstrated in the Practice Exercises. The names for 2 axis sample programs begin with "2AX-". The names for three axis sample programs begin with "3AX-".

Sample Program #1 - Drill Program

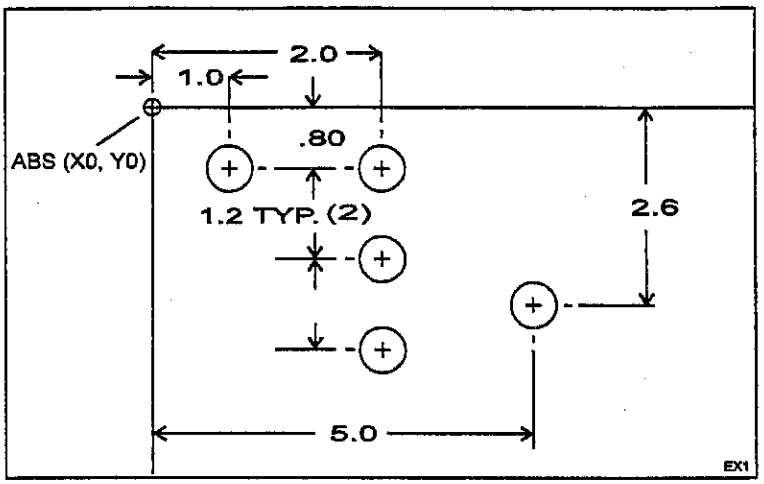


Figure 12 - 1, Layout for Sample Program #1

Refer to **Figure 12 - 1, Layout for Sample Program #1**. This program drills 5 holes in a surface using a 3/8 inch diameter twist drill (Tool #1). The holes are drilled with a Pecking Drill cycle. Go to the Tool Page and enter a 0.375 inch diameter for Tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

Detailed instructions for programming and viewing this program are outlined in Practice Exercises #1 - #5.

Sample Program #1

```
1  Dim Abs
2  * .375 DRILL
3  PeckDrill  ZDepth -0.6500 StartHgt 0.1000 Peck 0.1500 Feed 12.8
       Tool# 1
4  Rapid      X 1.0000 Y -.8000
5             X 2.0000
6  Dim Incr
7             Y -1.2000
8             Y -1.2000
9  Dim Abs
10            X 5.0000 Y -2.6000
11 DrillOff
12 Rapid      Z 0.0000 Tool# 0
13            X 0.0000 Y 0.0000
14 EndMain
15 <End Of Program>
```

Sample Program #2 - Rectangular Pocket Program

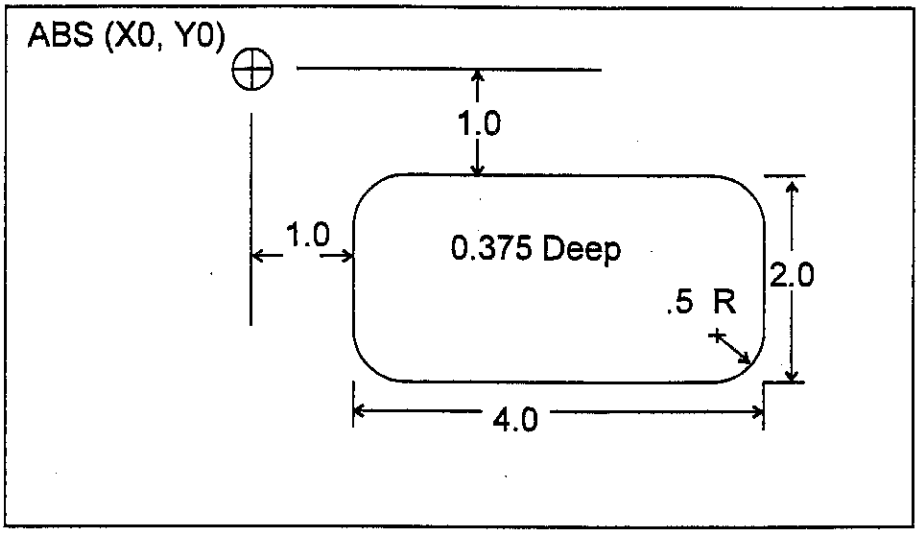


Figure 12 - 2, Layout for Sample Program #2

Refer to **Figure 12 - 2, Layout for Sample Program #2**. This program uses a 1/2 inch diameter endmill (Tool #1) to mill out a rectangular pocket.

Go to the Tool Page and enter a 0.50 inch diameter for tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

Sample Program #2

```
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      Rapid Z 0.0000 Tool# 0
4      X -3.0000 Y 3.0000
5  EndMain
6  <End Of Program>
```

Sample Program #3 - Irregular Pocket Program

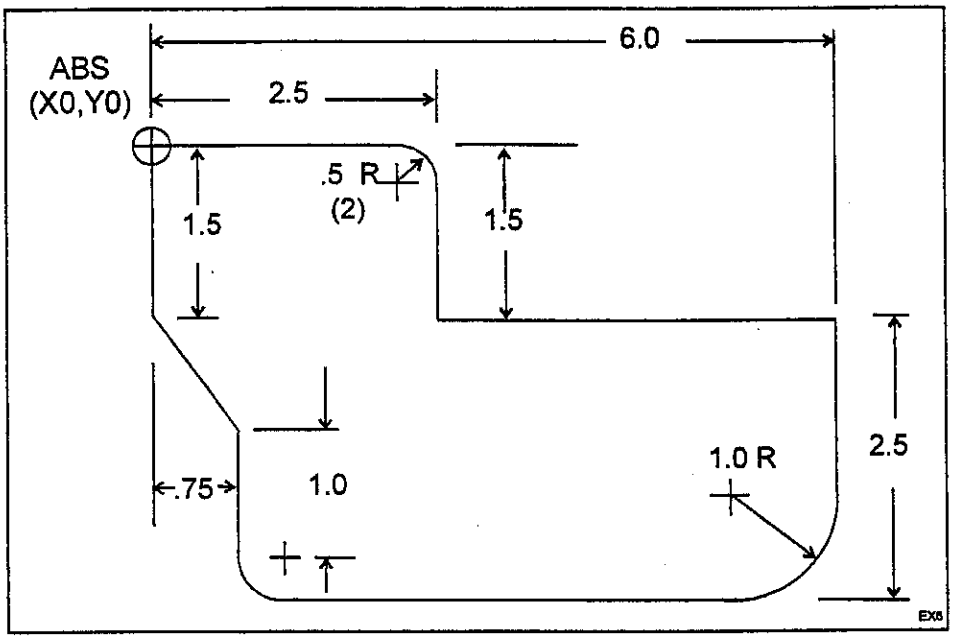


Figure 12 - 3, Layout for Sample Program #3

Refer to **Figure 12 - 3, Layout for Sample Program #3**. This program uses a 1/2 inch diameter endmill (Tool #1) to mill out the irregular pocket shown. Go to the Tool Page and enter a 0.50 inch diameter for Tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

The pocket outline is defined by subprogram #1. The first cut starts at the same position as the first Line (feed) move in the subprogram, and moves in a direction parallel to the first straight Line (feed) move in the subprogram.

Sample Program #3

```
1  Dim Abs
2  Pocket     Sub# 1 StartHgt 0.1000 ZDepth -0.1250 XStart 1.0000
           YStart -1.0000 Stepover 0.1500 FinStock 0.0100 RoughFeed 12.5
           FinFeed 9.5 Tool# 1
3  Rapid      Z 0.0000 Tool# 0
4          X -2.0000 Y 2.0000
5  EndMain
6  Sub 1
7  Rapid      X 0.0000 Y 0.0000
8  Line       X 2.5000 CornerRad 0.5000
9             Y -1.5000
10            X 6.0000
11 Dim Incr
12 Line       Y -2.5000 CornerRad 1.0000
13 Dim Abs
14 Line       X 0.7500 CornerRad 0.5000
15            Y -2.5000
16            X 0.0000 Y -1.5000
17            Y 0.0000
18 EndSub
19 <End Of Program>
```

Sample Program #4 - Irregular Pocket Program

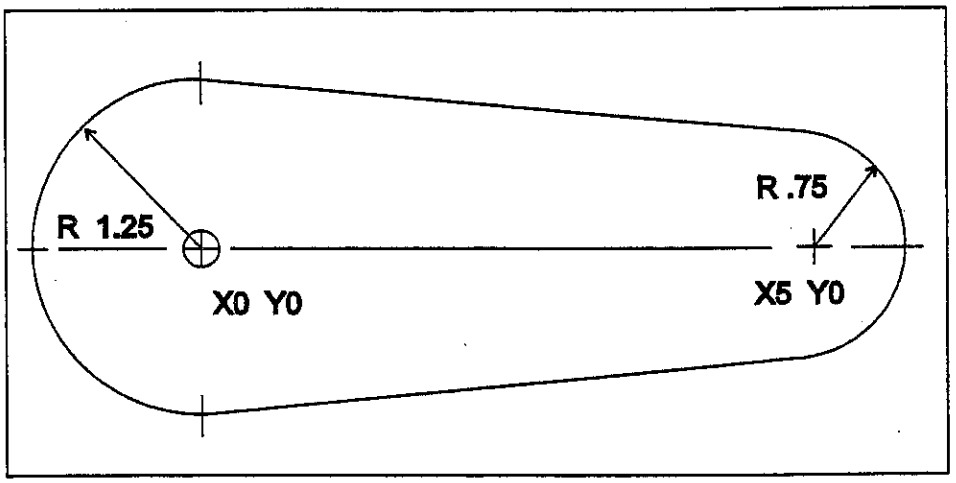


Figure 12 - 4, Layout for Sample Program #4

Refer to **Figure 12 - 4, Layout for Sample Program #4**. This program uses a 3/8 inch diameter endmill (Tool #1) to mill out an irregular pocket. Go to the Tool Page and enter a 0.375 inch diameter for Tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

The shape of the pocket is defined by the moves in subprogram #99. The starting position of the first cut is the (compensated) intersection of the first and last feed moves (Arcs) in the subprogram. The direction of the first cut is forced to 89.9 degrees (0.01 degrees inside of the Arc) by the Angle value.

The keystrokes for programming and viewing Sample Program #4 are presented in Practice Exercises #6 - #8.

Sample Program #4

```
1  Dim Abs
2  Pocket     Sub 99 StartHgt 0.1000 ZDepth -0.2500 Angle 89.9000
              XStart 0.0000 YStart 0.0000 Stepover 0.1600 DepthCut 0.1250
              FinStock 0.0100 RoughFeed 12.5 FinFeed 9.5 Tool# 1
3  Rapid      Z 0.0000 Tool# 0
4             X -2.0000 Y 2.0000
5  EndMain
6  Sub 99
7  Rapid      X -1.2500 Y 0.0000
8  Arc Cw     X 0.1250 Y 1.2437 Radius 1.2500
9  Line       X 5.0750 Y 0.7462
10 Arc Cw     X 5.0750 Y -0.7462 Radius 0.7500
11 Line       X 0.1250 Y -1.2437
12 Arc Cw     X -1.2500 Y 0.0000 Radius 1.2500
13 EndSub
14 <End Of Program>
```

Sample Program #5 - Program with Looping Subprogram

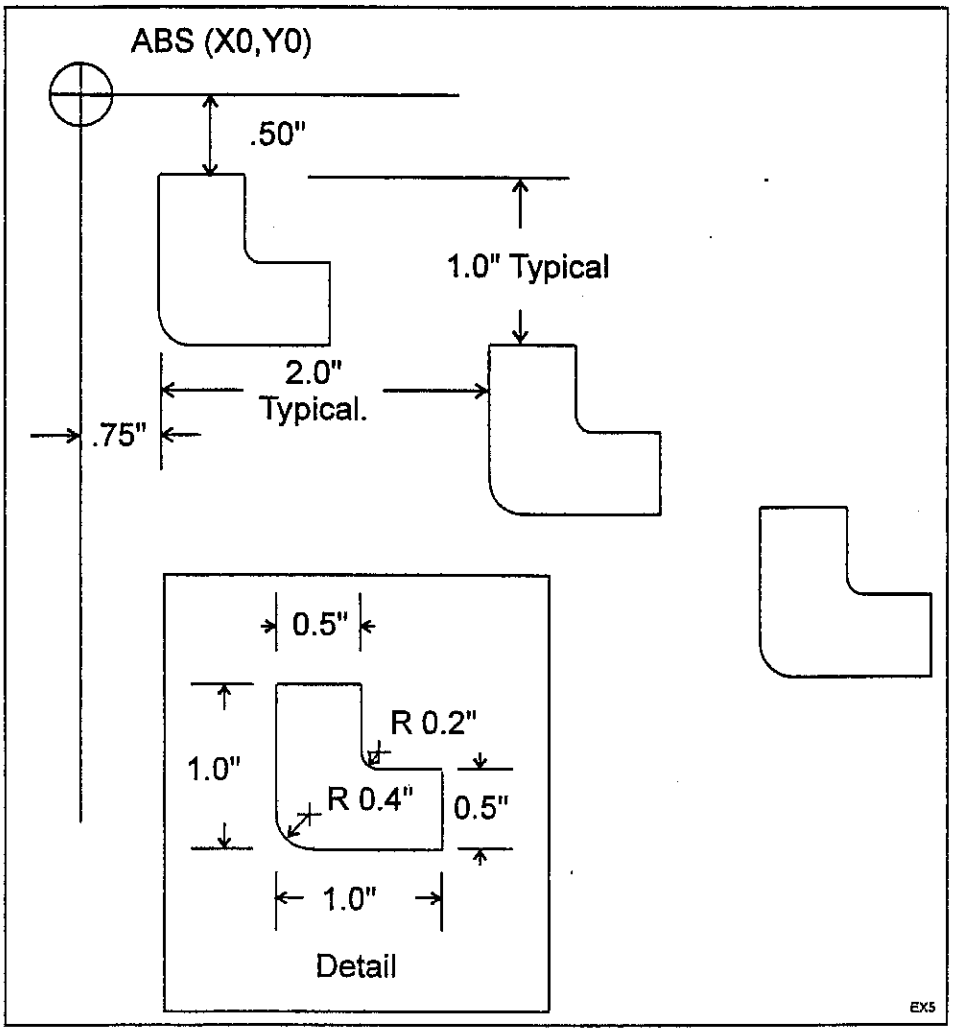


Figure 12 - 5, Layout for Sample Program #5

Refer to **Figure 12 - 5, Layout for Sample Program #5**. This program loops subprogram #1 to cut three L shaped pockets. The pockets are cut using a 1/8 inch diameter end mill (Tool#1). Each loop starts at a position within the pocket.

All of the XY axis moves in the subprogram are written for incremental positioning. This lets the CNC cut the pocket anywhere on the workpiece. Go to the Tool Page and enter a 0.125 inch diameter for tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

Sample Program #5

```
1  Dim Abs
2  Rapid     X 0.7500 Y -0.7500 Tool# 1
3  Loop      Sub 1 Loops 3 XIncr 2.0000 YIncr -1.0000
4  Rapid     Z 0.0000 Tool# 0
5            X -2.0000 Y 2.0000
6  EndMain
7  Sub 1
8  Dim Abs
9  Rapid     Z 0.1000
10 Line      Z -0.0620 Feed 1.5
11 Dim Incr
12 Line      Y 0.2500 ToolComp Left Feed 2.1
13 Line      X -0.2500
14 Line      Y -1.0000 CornerRad 0.4000
15 Line      X 1.0000
16 Line      Y 0.5000
17 Line      X -0.5000 CornerRad 0.2000
18 Line      Y 0.5000
19 Line      X -0.2500
20 Line      Y -0.2500 ToolComp Off
21 Dim Abs
22 Rapid     Z 0.1000
23 EndSub
24 <End Of Program>
```

Sample Program #6 - Program with Rotating Subprogram

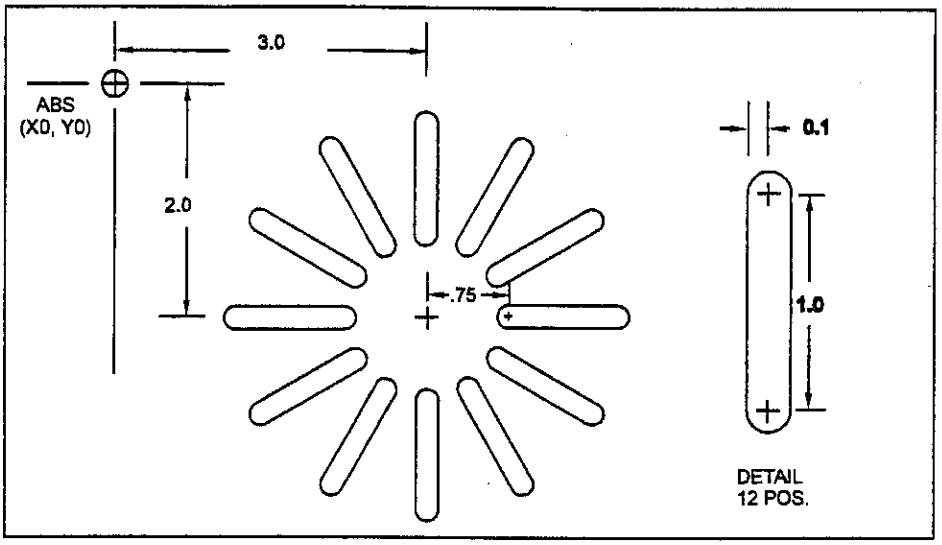


Figure 12 - 6, Layout for Sample Program #6

Refer to **Figure 12 - 6, Layout for Sample Program #6**. This program rotates subprogram #1 to cut 12 slots. The slots are cut using a 1/8 inch endmill (Tool #1).

Go to the Tool Page and enter a 0.125 inch diameter for tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

Sample Program #6

```
1  Dim Abs
2  RMS       Sub# 1 #Loops 12 StartAngle 0.0000 Angle 30.0000
          XCenter 3.0000 YCenter -2.0000 Tool# 1
3  Rapid     Z 0.0000 Tool# 0
4            X -2.0000 Y 2.0000
5  EndMain
6  Sub 1
7  Dim Abs
8  Rapid     X 3.7500 Y -2.0000
9            Z 0.1000
10 Line      Z -0.1250 Feed 1.2
11 Dim Incr
12 Line      Y 0.1000 ToolComp Left Feed 2.2
13 Arc Ccw   X 0.0000 Y -0.2000 Radius 0.1000
14 Line      X 1.0000
15 Arc Ccw   X 0.0000 Y 0.2000 Radius 0.1000
16 Line      X -1.0000
17 Line      Y -0.1000 ToolComp Off
18 Dim Abs
19 Rapid     Z 0.1000
20 EndSub
21 <End Of Program>
```

Sample Program #7 - X Axis Mold Rotation Program

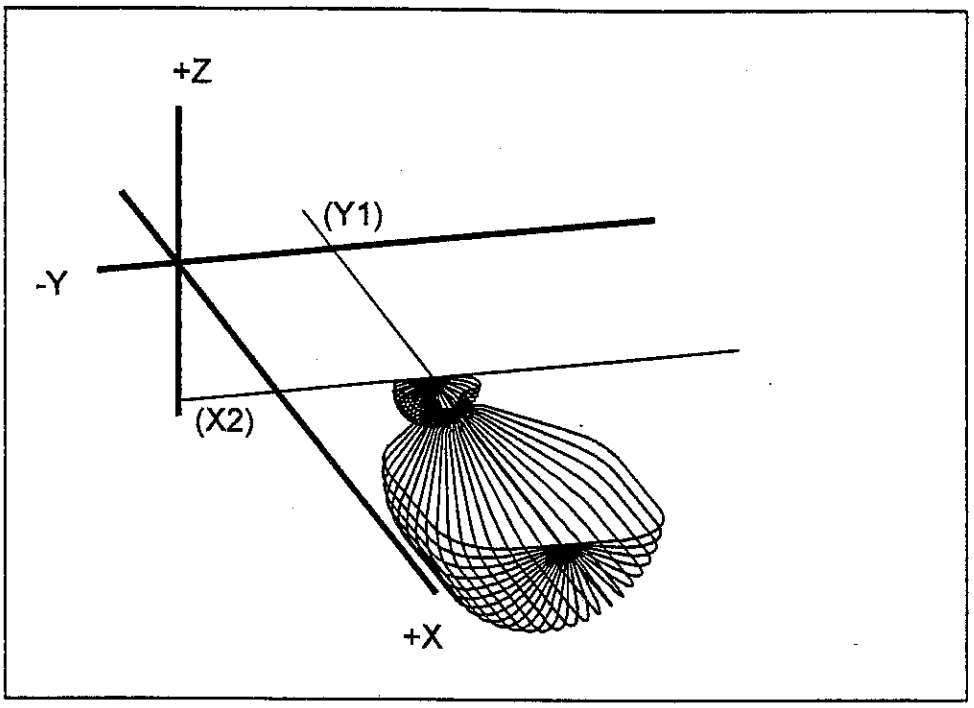


Figure 12 - 7, Layout for Sample Program #7 - Mold Rotation

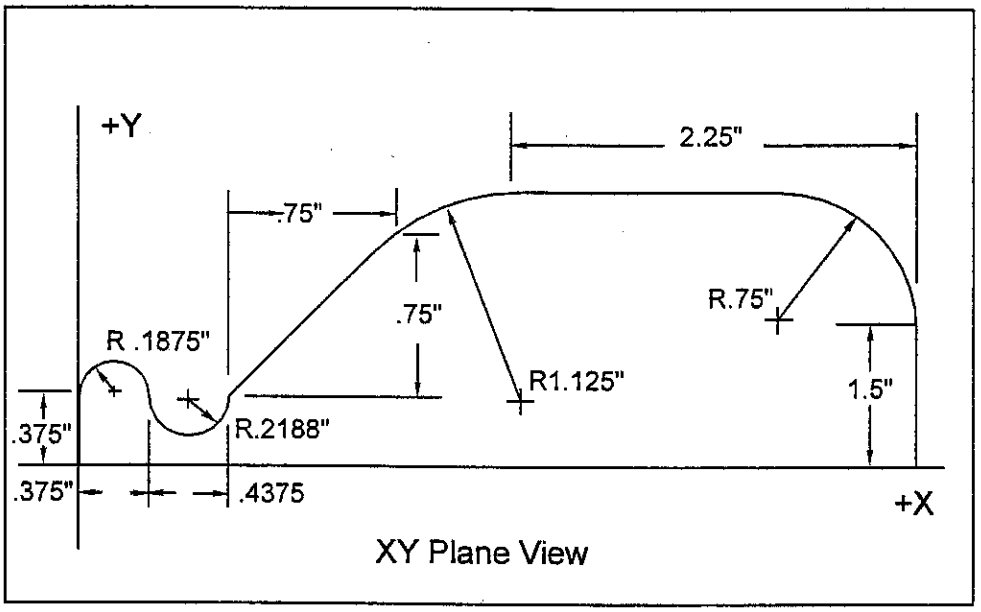


Figure 12 - 8, Subprogram Layout for Sample Program #7

Refer to **Figure 12 - 7, Layout for Sample Program #7 - Mold Rotation** and **Figure 12 - 8, Subprogram Layout for Sample Program #7**. This program uses a ball endmill to cut a bottle mold. Subprograms #1 and #2 define the forward and reverse paths of the tool in the XY plane. The mold rotation is cut by alternately executing subprograms #1 and #2 while rotating around the X axis.

This example does not use diameter compensation. The size of Tool #1 will determine the finished size mold in this example. Go to the Tool Page and enter an appropriate diameter for Tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page.

Sample Program #7

```
1  Dim Abs
2  Rapid      X 2.0000 Y 1.0000 Tool# 1
3             Z 0.5000
4  Line       Z 0.0000 Feed 10.0
5  MoldRot    StartAngle 0.0000 EndAngle -180.0000 Cycles 12 FwdSub 1
        RevSub 2 AxisRot X BAxisCL 1.0000
6  Rapid      Z 0.0000 Tool# 0
7  EndMain
8  Sub 1
9  Dim Incr
10 Line       Y 0.3750
11 Arc Cw     X 0.3750 Y 0.0000 Radius 0.1875
12 Arc Ccw    X 0.4375 Y 0.0000 Radius 0.2188
13 Line       X 0.7500 Y 0.7500
14 Arc Cw     X 0.7500 Y 0.3750 Radius 1.1250
15 Line       X 2.2500 CornerRad 0.7500
16 Line       Y -1.5000
17 EndSub
18 Sub 2
19 Line       Y 1.5000 CornerRad 0.7500
20 Line       X -2.2500
21 Arc Ccw    X -0.7500 Y -0.3750 Radius 1.1250
22 Line       X -0.7500 Y -0.7500
23 Arc Cw     X -0.4375 Y 0.0000 Radius 0.2188
24 Arc Ccw    X -0.3750 Y 0.0000 Radius 0.1875
25 Line       Y -0.3750
26 EndSub
27 <End Of Program>
```

Sample Program #8 - Y Axis Mold Rotation Program

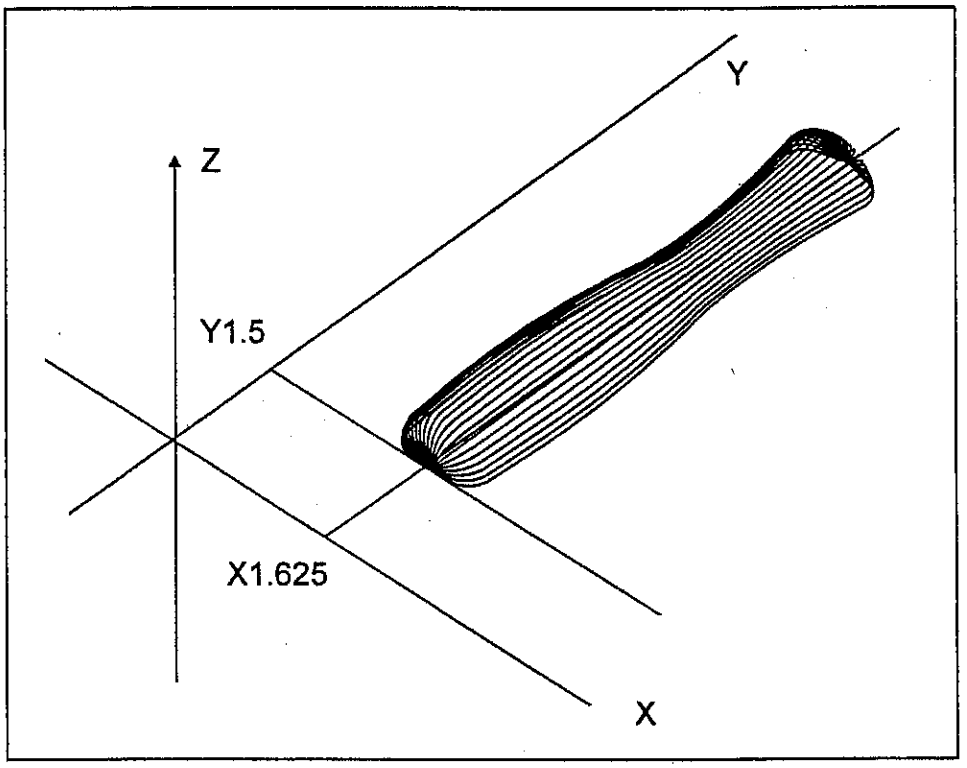


Figure 12 - 9, Layout for Sample Program #8 - Mold Rotation

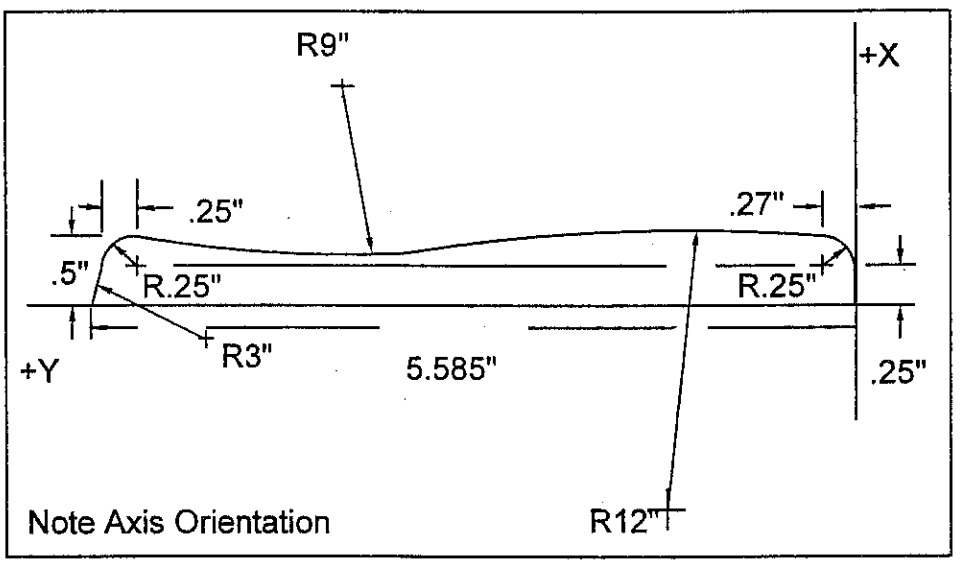


Figure 12 - 10, Subprogram Layout for Sample Program #8

Refer to **Figure 12 - 9, Layout for Sample Program #8 - Mold Rotation** and **Figure 12 - 10, Subprogram Layout for Sample Program #8**. This program uses a ball endmill to cut a bottle mold. Subprogram #15 defines the forward path of the tool. Subprogram #16 defines the reverse. The mold rotation is cut by alternately executing subprograms #15 and #16 while rotating around the Y axis.

A mold rotation would normally be cut with a ball end mill. This example does not use diameter compensation. The size of tool #1 will determine the finished size of the mold in this example. Go to the Tool Page and enter an appropriate diameter for tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page and setting tool length offsets for ball endmills.

Sample Program #8

```
1  Dim Abs
2  Rapid     X 1.6250 Y 1.5000 Tool# 1
3            Z 0.5000
4  Line      Z 0.0000 Feed 10.0
5  MoldRot   StartAngle 0.0000 EndAngle 180.0000 Cycles 10
        FwdSub 15 RevSub 16 AxisRot Y BAxisCL 1.6250
6  Rapid     Z 0.0000 Tool# 0
7  EndMain
8  Sub 15
9  Dim Incr
10 Line      X -0.2500
11 Arc Cw    X -0.2500 Y 0.2700 Radius 0.2500
12 Arc Cw    X 0.1250 Y 3.0000 Radius 12.0000
13 Arc Ccw   X -0.1250 Y 2.0000 Radius 9.0000
14 Arc Cw    X 0.2500 Y 0.2500 Radius 0.2500
15 Arc Cw    X 0.2500 Y 0.0625 Radius 3.0000
16 EndSub
17 Sub 16
18 Arc Ccw   X -0.2500 Y -0.0625 Radius 3.0000
19 Arc Ccw   X -0.2500 Y -0.2500 Radius 0.2500
20 Arc Cw    X 0.1250 Y -2.0000 Radius 9.0000
21 Arc Ccw   X -0.1250 Y -3.0000 Radius 12.0000
22 Arc Ccw   X 0.2500 Y -0.2700 Radius 0.2500
23 Line      X 0.2500
24 EndSub
25 <End Of Program>
```

Sample Program #9 - Z Axis Mold Rotation Program

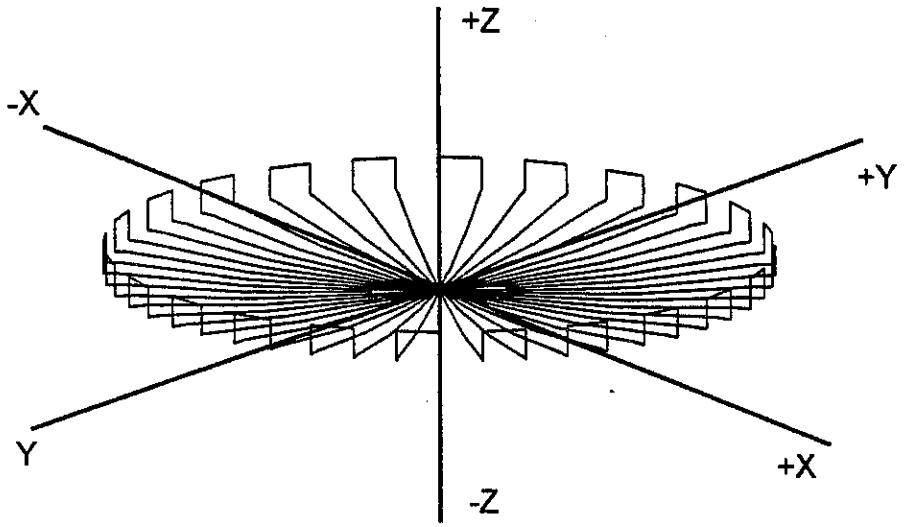


Figure 12 - 11, Layout for Sample Program #9, Mold Rotation

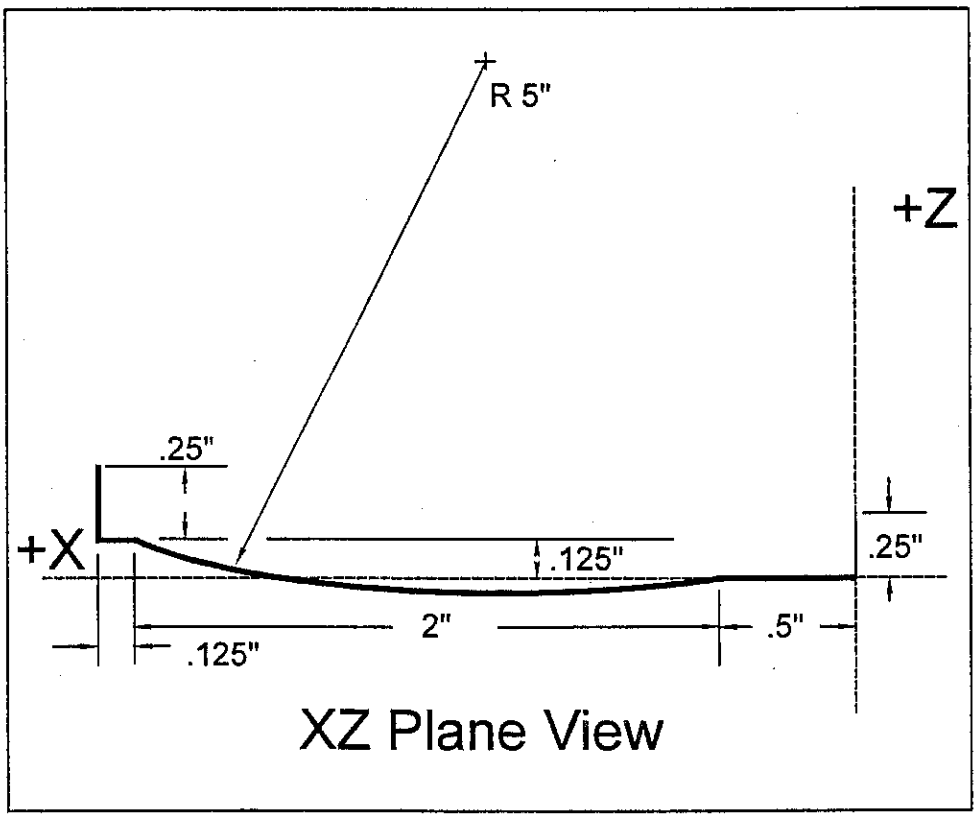


Figure 12 - 12, Subprogram Layout for Sample Program #9

Refer to **Figure 12 - 11, Layout for Sample Program #9, Mold Rotation** and **Figure 12 - 12, Subprogram Layout for Sample Program #9**. This program uses a ball endmill to cut a dish. Subprogram #1 defines the forward path of the tool. Subprogram #2 defines the reverse. The mold rotation is cut by alternately executing subprograms #1 and #2 while rotating around the Z axis. In order to rotate around the Z axis, the subprograms are programmed in the XZ plane.

This example uses tool diameter compensation. Each subprogram includes an additional ramp move to activate the diameter compensation. Note that moves in subprogram #1 are compensated right while moves in subprogram #2 are compensated left. This is because the moves in the subprograms are executed in opposite directions.

Go to the Tool Page and enter an appropriate diameter for tool #1 before running the program in Draw. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page and setting tool length offsets for ball endmills.

Sample Program #9

```
1  Dim Abs
2  Rapid      X 0.0000 Y 0.0000 Tool# 1
3             Z 0.5000
4  Line       Z 0.0000 Feed 5.0
5  MoldRot    StartAngle 0.0000 EndAngle 350.0000 Cycles 13 FwdSub 1
        RevSub 2 AxisRot Z Feed 5.7
6  Rapid      Z 0.0000 Tool# 0
7  Plane XY
8  EndMain
9  Sub 1
10 Plane XZ
11 Dim Incr
12 Line       Z -0.2500 ToolComp Right
13 Line       X 0.5000
14 Arc Cw     X 2.0000 Z 0.1250 Radius 5.0000
15 Line       X 0.1250
16 Line       Z 0.2500 ToolComp Off
17 EndSub
18 Sub 2
19 Line       Z -0.2500 ToolComp Left
20 Line       X -0.1250
21 Arc Ccw    X -2.0000 Z -0.1250 Radius 5.0000
22 Line       X -0.5000
23 Line       Z 0.2500 ToolComp Off
24 EndSub
25 <End Of Program>
```

Sample Program # 10 - Elbow Milling

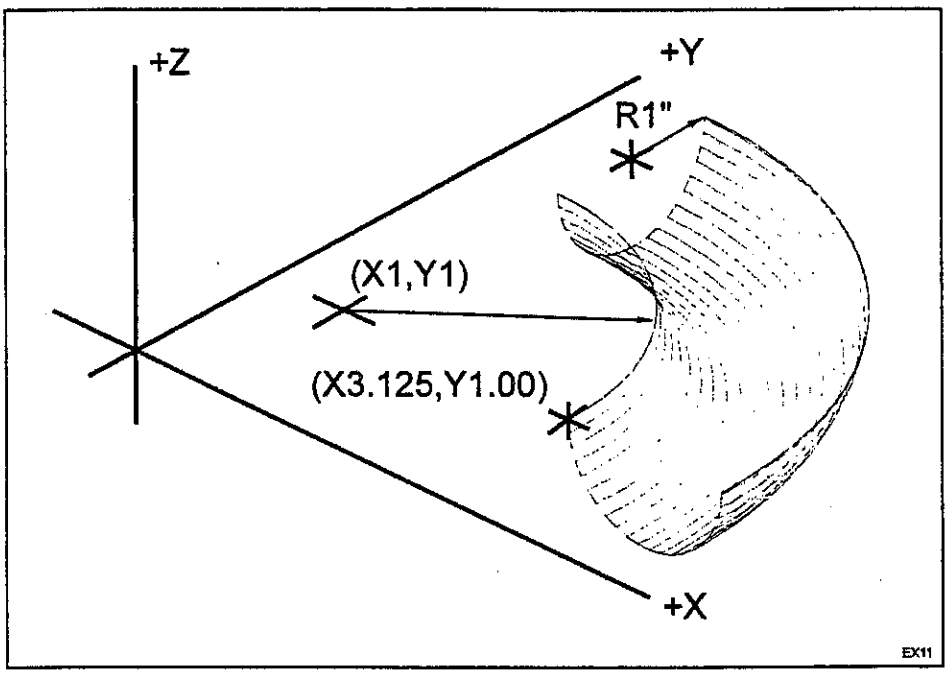


Figure 12 - 13, Layout for Sample Program #10, Elbow Milling

This program uses a ball endmill to cut an elbow cavity. Tool compensation cannot be used with elbow milling, the starting position must be calculated. Refer to **Section 8 - Tool Management** for instructions on working with the Tool Page and setting tool length offsets for ball endmills.

Sample Program #10

```
1  Dim Abs
2  Rapid      X 3.1250 Y 1.0000 Tool# 1
3             Z 0.5000
4  Line       Z 0.0000 Feed 6.5
5  Elbow      Ccw StartRad 1.0000 EndRad 1.0000 InclAngle 90.0000
   Cycles 10 XCenter 1.0000 YCenter 1.0000 Feed 25.5
6  Rapid      Z 0.0000 Tool# 0
7  EndMain
8  <End Of Program>
```

Using the Practice Exercises

The exercises consist of a series of tables. Each table contains keystrokes and instructions required to accomplish a single operation. The keystrokes in each table are numbered. Perform the keystrokes in the order shown on the table.

Menu selections are made by using the (ARROW) keys to highlight to the required choice. Press (ENTER) to activate a highlighted choice. Toggled selections are changed by pressing the (+/-) key.

A brief overall description of the procedure is provided at the start of each exercise. Be sure to start each exercise from the CNC screen listed in the description.

When an exercise contains more than one table, each table will leave the CNC in the correct screen for starting the next table.

Starting Practice Exercises #1 - #5

Practice Exercises #1 - #5 are a series of related exercises designed to coach a first time programmer through the development of Sample Program #1 - Drill Program. These exercises should be done in order.

Each practice exercise coaches the operator through a different step of the process. Practice Exercise #1 creates the new program. Practice Exercise #2 opens the program for editing. The program blocks are written in Practice Exercise #3. The tooling is entered in the Tool Page in Practice Exercise #4 and the program is checked in Practice Exercise #5.

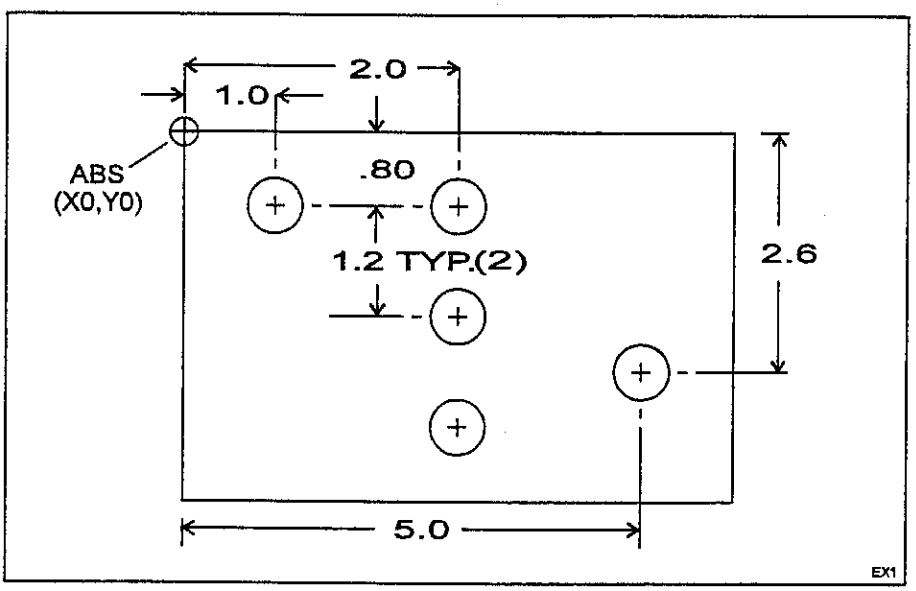


Figure 12 - 14, Layout for Sample Program #1

Sample Program #1 - Drill Program

```
1  Dim Abs
2  * .375 DRILL
3  PeckDrill  ZDepth -0.6500 StartHgt 0.1000 Peck 0.1500 Feed 12.8
       Tool# 1
4  Rapid      X 1.0000 Y -.8000
5             X 2.0000
6  Dim Incr
7             Y -1.2000
8             Y -1.2000
9  Dim Abs
10            X 5.0000 Y -2.6000
11 DrillOff
12 Rapid      Z 0.0000 Tool# 0
13            X 0.0000 Y 0.0000
14 EndMain
15 <End Of Program>
```

Practice Exercise #1 - Creating a Program Named SP1.M

Refer to **Figure 12 - 14, Layout for Sample Program #1**. This exercise outlines the steps required to create a new program named "SP1.M" (Sample Program 1). Before starting, make sure no program named "SP1.M" already exists. Start this exercise from the **MANUAL** screen.

Create a new program named "SP1.M".					
1	Activate Program Directory.	**PROGRAM** (F2)	6	Highlight required character.	Select **P**
2	Create new program.	**Create** (F2)	7	Add character to name.	ENTER
3	Activate ASCII Chart.	**ASCII** (F2)	8	Key in 1.	1 RAPID
4	Highlight required character.	Select **S**	9	Close ASCII Chart.	**ASCII** (F2)
5	Add character to name.	ENTER	10	Add new program to Program Directory USER list.	ENTER

Practice Exercise #2 - Activating Edit to Program SP1.M

This exercise describes the steps required to activate the editor to write program blocks in program "SP1.M." Start this exercise from the **PROGRAM** screen.

Activate the Program Editor to write blocks in program **SP1.M**.					
1	Highlight "**SP1**.M".	Select **SP1.M**	2	Activate the program editor.	**Edit** (F4)

Practice Exercise #3 - Writing the Blocks in SP1.M

This exercise illustrates the steps used to write the blocks contained in Sample Program #1 - Drill Program. Start this exercise from the **Edit** screen.

Activate the Absolute Mode.					
Block - 1 Dim Abs					
1	Activate ABS/INC Graphic Menu.	ABS/INC	2	Add block to program listing.	ENTER

Write a comment to describe Tool #1.					
Block - 2 * .375 DRILL					
1	Activate Misc softkeys.	(F9) **Misc**	12	Highlight required character.	Select **R**
2	Activate comment tool.	(F2) **Comment**	13	Add character to comment.	ENTER
3	Key in decimal point.	.	14	Highlight required character.	Select **I**
4	Key in a 3.	3 ARC	15	Add character to comment.	ENTER
5	Key in a 7.	7 UNIT	16	Highlight required character.	Select **L**
6	Key in a 5.	5 TOOL	17	Add character to comment.	ENTER
7	Key in a space.		18	Add character to comment.	ENTER
8	Key in a space.		19	Close ASCII Chart.	(F2) **ASCII**
9	Activate ASCII Chart.	(F2) **ASCII**	20	Add block to program.	ENTER
10	Highlight required character.	Select **D.**	21	Restore default softkeys.	(F9) **Prev**
11	Add character to comment.	ENTER			

Write block for Peckdrill cycle to drill to a ZDepth of .65 inch using .015 in pecks with Tool #1. Same block activates the CNC in a Drill Mode.					
Block - 3 PeckDrill ZDepth -0.6500 StartHgt 0.1000 Peck 0.1500 Feed 12.8 Tool# 1					
1	Activate Drill pop-up menu.	**Drill** (F3)	12	Key in decimal.	.
2	Highlight required selection.	Select **Pecking**	13	Key in 1.	1 RAPID
3	Activate Pecking cycle Graphic Menu.	ENTER	14	Key in 5.	5 TOOL
4	Change sign.	+/- Toggle Negative	15	Advance to **Feed**.	ENTER
5	Key in decimal.	.	16	Key in 1.	1 RAPID
6	Key in 6.	6 MODE	17	Key in 2.	2 LINE
7	Key in 5	5 TOOL	18	Key in decimal.	.
8	Advance to **StartHgt**.	ENTER	19	Key in 8.	8 DWELL
9	Key in decimal.	.	20	Advance to **Tool#**.	ENTER
10	Key in 1.	1 RAPID	21	Key in 1.	1 RAPID
11	Advance to required field.	Select **Peck**	22	Add block to program listing.	**Save** (F10)

Write rapid move to position of first hole.					
Block - 4 Rapid X 1.0000 Y -.8000					
1	Activate Rapid Graphic Menu.	1 RAPID	5	Key in decimal.	.
2	Key in 1.	1 RAPID	6	Key in 8.	8 DWELL
3	Advance to **Y**.	ENTER	7	Add block to program listing.	**Save** (F10)
4	Change sign.	+/- Toggle Negative			

Write modal move.					
Block - 5 X 2.0000					
1	Activate Modal Move Graphic Menu.	X	3	Add block to program listing.	**Save** (F10)
2	Key in 2.	2 LINE			

Activate the Incremental Mode.					
Block - 6 Dim Incr					
1	Activate ABS/INC Graphic Menu.	ABS INC	3	Add block to program listing.	**Save** (F10)
2	Toggle setting	+/- Toggle for **Incr**			

Write modal move.					
Block - 7 Y-1.2000					
1	Activate Modal Move Graphic Menu.	Y	4	Key in decimal.	.
2	Change sign.	+/- Toggle Negative	5	Key in 2.	2 LINE
3	Key in 1.	1 RAPID	6	Add block to program listing.	**Save** (F10)

Write modal move.					
Block - 8 Y-1.2000					
1	Activate Modal Move Graphic Menu.	Y	4	Key in decimal.	.
2	Change sign.	+/- Toggle Negative	5	Key in 2.	2 LINE
3	Key in 1.	1 RAPID	6	Add block to program listing.	**Save** (F10)

Activate the Absolute Mode.					
Block - 9 Dim Abs					
1	Activate ABS/INC Graphic Menu.	ABS INC	2	Add block to program listing.	**Save** (F10)

Write modal move.					
Block - 10 X 5.0000 Y -2.6000					
1	Activate Modal Move Graphic Menu.	X	5	Key in 2.	2 LINE
2	Key in 5.	5 TOOL	6	Key in decimal.	.
3	Advance to **Y**.	ENTER	7	Key in 6.	6 MODE
4	Change sign.	+/- Toggle Negative	8	Add block to program listing.	**Save** (F10)

Deactivate the Drill Mode.					
Block - 11 DrillOff					
1	Activate Drill pop-up menu.	**Drill** (F3)	3	Add block to program listing.	ENTER
2	Highlight required selection.	Select **Drilling Off**			

Fully retract the quill.					
Block - 12 Rapid Z 0.0000 Tool# 0					
1	Activate Rapid Graphic Menu.	1 RAPID	4	Advance to required field.	Select **Tool#.**
2	Advance to required field.	Select **Z**	5	Key in 0.	0
3	Key in 0.	0	6	Add block to program listing.	**Save** (F10)

Make Modal move to part change position.					
Block - 13 X 0.0000 Y 0.0000					
1	Activate Modal Move Graphic Menu.	X	4	Key in 0.	0
2	Key in 0.	0	5	Add block to program listing.	**Save** (F10)
3	Advance to **Y**.	ENTER			

Add EndMain block.					
Block - 14 EndMain					
1	Activate the subprogram softkeys.	**Sub** (F8)	3	Restore the default softkey line.	**Prev** (F9)
2	Add EndMain block to the program.	**EndMain** (F4)	4	Exit the program editor saving the program.	**Exit** (F10)

Practice Exercise #4 - Entering SP1.M Tooling on the Tool Page

Tool length offsets are not needed to run programs in **Draw**. If this program were used to cut a part, the operator would need to set the machine zero (absolute zero) and a tool length offset for Tool# 1.

Sample Program #1 is written for a 3/8 inch diameter twist drill. Start this exercise from the **PROGRAM** screen.

Enter 0.375 inch diameter on Tool Page for tool #1.					
1	Highlight "SP1.M".	Select **SP1.M**	6	Key in a 7.	7 UNIT
2	Activate the program editor.	**Edit** (F4)	7	Key in a 5.	5 TOOL
3	Activate the Tool Page	**Tool** (F6)	8	Exit the Tool Page.	**Exit** (F10)
4	Key in a decimal point.	.	9	Exit the editor.	**Exit** (F10)
5	Key in a 3.	3 ARC			

Practice Exercise #5 - Running SP1.M in Draw

This exercise uses Draw to view the moves contained in SP1.M program.

Refer to **Section 5 - Viewing Programs with Draw** for more information on working with the Draw Mode.

This exercise will adjust four of the display parameters before running the program. Adjusting these parameters ensures the CNC will operate the way the exercises are written. The adjustments also provide a good view of the programmed moves. Some of the settings may already be set and will not need adjustment.

A program to cut a large part must be scaled to fit in the Draw Window. Tool display and Rapid move display can be on or off. This exercise turns them on. This moves in this exercise will display both an XY plane view and an isometric view of the moves.

Start this exercise from the **PROGRAM** screen.

Activate Draw. Turn Tool on, Rapid on, set the Mode to Auto and turn Run off. Set the view to isometric, fit the program to the Window and run it. Change the view to the XY plane and run it again.

1	Highlight "SP1.M".	Select **SP1.M**
2	Activate the program editor.	**Edit** (F4)
3	Activate Draw.	**Draw** (F2)
4	Activate Parms pop-up menu.	**Parms** (F9)
5	Highlight required selection.	Select **Tool**
6	Toggle parameter (if required).	Toggle **On**
7	Highlight required selection.	Select **Rapid**
8	Toggle parameter (if required).	Toggle **On**
9	Highlight required selection.	Select **Mode**
10	Activate **Mode** pop-up menu.	
11	Highlight required selection.	Select **Auto**
12	Activate **Auto** (if required).	
13	Highlight required selection.	Select **Run**
14	Toggle parameter (if required).	Toggle **Off**
15	Close Parms pop-up menu.	**Parms** (F9)
16	Activate VIEW pop-up menu.	**VIEW** (F4)
17	Highlight required selection.	Select **Iso**
18	Activate **Iso** view.	
19	Activate DISPLAY.	**DISPLAY** (F5)
20	Activate selection and pause for the **"Program ended normally"** message.	
21	Run the program and verify the programmed moves.	**Run** (F3)
22	Activate VIEW pop-up menu.	**VIEW** (F4)
23	Highlight required selection.	Select **XY**
24	Activate **XY** view.	
25	Run the program and verify it's moves.	**Run** (F3)
26	Close Draw.	**Exit** (F10)
27	Close the editor.	**Exit** (F10)
28	Close the Program Directory.	**Exit** (F10)

Starting Practice Exercises #6 - #8

Practice Exercises #6 - #8 are a series of related exercises designed to coach a first time programmer through the development of Sample Program #4 - Irregular Pocket Program. These exercises should be done in order.

Each exercise coaches the operator through a different step of the process. Practice Exercise #6 creates the new program. In Practice Exercise #7, the Geometry Calculator is used to find the coordinates of the start and end points required in the subprogram. In Practice Exercise #8 the positions saved in the Geometry Calculator are recalled directly into the program as it is written.

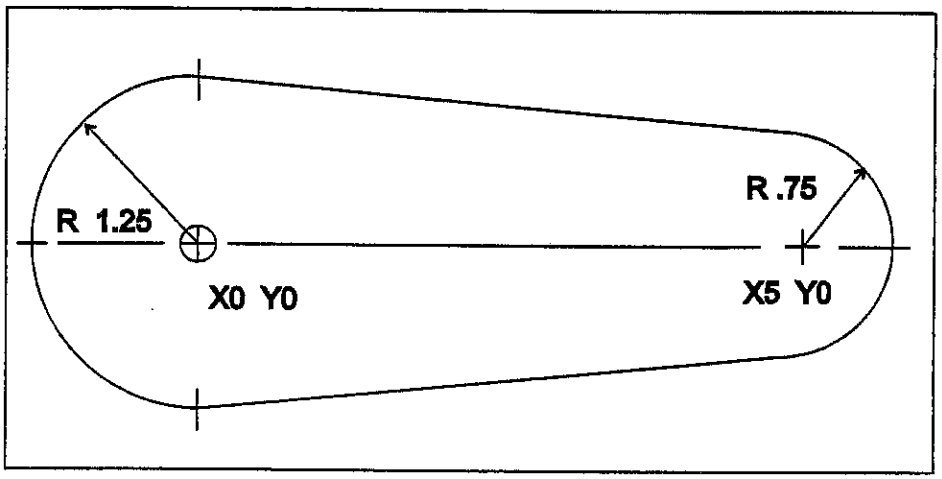


Figure 12 - 15, Layout for Sample Program #4

Sample Program #4 - Irregular Pocket Program

```
1  Dim Abs
2  Pocket     Sub 99 StartHgt 0.1000 ZDepth -0.2500 Angle 89.9000
              XStart 0.0000 YStart 0.0000 Stepover 0.1600 DepthCut 0.1250
              FinStock 0.0100 RoughFeed 12.5 FinFeed 9.5 Tool# 1
3  Rapid      Z 0.0000 Tool# 0
4             X -2.0000 Y 2.0000
5  EndMain
6  Sub 99
7  Rapid      X -1.2500 Y 0.0000
8  Arc Cw     X 0.1250 Y 1.2437 Radius 1.2500
9  Line       X 5.0750 Y 0.7462
10 Arc Cw     X 5.0750 Y -0.7462 Radius 0.7500
11 Line       X 0.1250 Y -1.2437
12 Arc Cw     X -1.2500 Y 0.0000 Radius 1.2500
13 EndSub
14 <End Of Program>
```

Practice Exercise #6 - Creating a New Program SP4.M

Ensure no program named "SP4.M" (Sample Program 4) already exists. Start this exercise from the **MANUAL** screen.

Create a new program named "SP4.M".					
1	Activate Program Directory.	**PROGRAM** (F2)	7	Add character to name.	ENTER
2	Create new program.	**Create** (F2)	8	Key in 4.	4 FEED
3	Activate ASCII Chart.	**ASCII** (F2)	9	Close ASCII Chart.	**ASCII** (F2)
4	Highlight required character.	Select **S**	10	Add new program to list USER program list.	ENTER
5	Add character to name.	ENTER	11	Close the Program Directory	**Exit** (F10)
6	Highlight required character.	Select **P**			

Practice Exercise #7 - Finding SP4.M Positions with Geometry Calculator

Refer to **Figure 12 - 16, Geometry Calculator Layout for Exercise #7**. This exercise clears the geometry calculator of any existing values and calculates the five positions required to program Sample Program #4. Once calculated, the position coordinates are stored in the CNC for recall into the program.

This exercise should be done before Exercise #8. Programs visible in the editor are not affected by running the calculator. Start this exercise from the **MANUAL** screen.

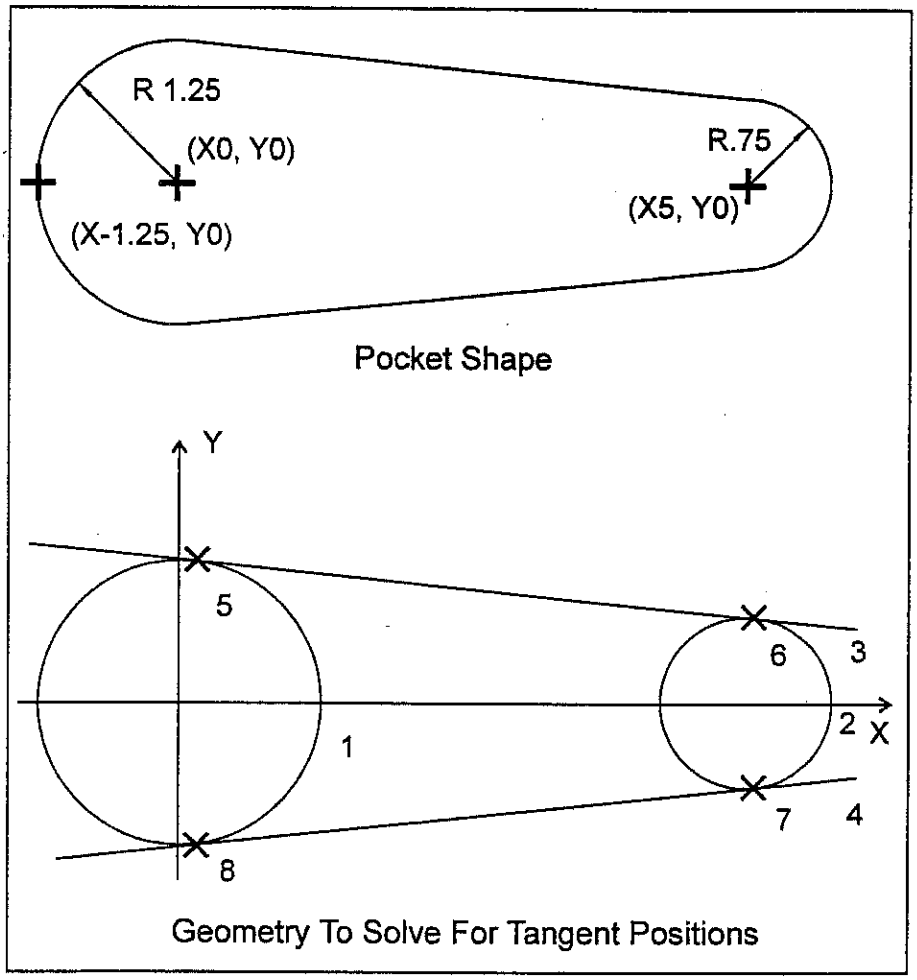


Figure 12 - 16, Geometry Calculator Layout for Exercise #7

Activate the Geometry Calculator.					
1	Activate Program Directory	**PROGRAM** (F2)	4	Activate the Calculator.	**Calc** (F7)
2	Highlight required program.	Select **SP4.M**	5	Select required calculator.	Select
3	Activate program	**Edit**	6	Activate Geometry	

	editor.	(F4)

	Calculator.	

Clear any existing objects from the Geometry Calculator.					
1	Activate Geometry pop-up.	**GEOMETR** (F7)	3	Activate deletion.	ENTER
2	Highlight required selection.	Select **Delete All**	4	Respond to safety prompt.	**Yes** (F1)

Create 1.25 inch circle centered on X 0, Y 0.					
1	Highlight tool to create a circle on a center.	Select	8	Activate highlighted tool. CNC prompts for X coordinate of center.	ENTER
2	Activate selected tool. CNC prompts for radius.	ENTER	9	Key in 0.	0
3	Key in 1.	1 RAPID	10	Enter value. CNC prompts for Y coordinate of center.	ENTER
4	Key in a decimal point.	.	11	Key in 0.	0
5	Key in a 2.	2 LINE	12	Enter value. CNC draws circle.	ENTER
6	Key in a 5.	5 TOOL	13	Activate DISPLAY pop-up menu.	**DISPLAY** (F5)
7	Enter value. CNC prompts for center definition tool.	ENTER	14	Activate **Fit**.	ENTER

Create 0.75 inch circle centered on X 5, Y 0.					
1	Highlight tool to create a circle on a center.	Select	8	Key in a 5.	5 TOOL
2	Activate selected tool. CNC prompts for radius.	ENTER	9	Enter value. CNC prompts for Y coordinate of center.	ENTER
3	Key in decimal point.	.	10	Key in 0.	0
4	Key in 7.	7 UNIT	11	Enter value. CNC draws circle.	ENTER
5	Key in 5.	5 TOOL	12	Activate DISPLAY pop-up menu.	**DISPLAY** (F5)
6	Enter value. CNC prompts for center definition.	ENTER	13	Activate selection to fit drawing.	ENTER

7	Activate highlighted tool. CNC prompts for X coordinate of center.	ENTER

Create line tangent to top of both circles.					
1	Highlight tool to create a line tangent to two circles.	Select	7	Key in 1.	1 RAPID
2	Activate selected tool. CNC prompts for first circle number.	ENTER	8	Enter value. CNC clears all but the selected line.	ENTER
3	Key in 1.	1 RAPID	9	Activate DISPLAY pop-up menu.	**DISPLAY** (F5)
4	Enter value. CNC prompts for second circle number.	ENTER	10	Highlight required selection.	Select **Redraw"**
5	Key in 2.	2 LINE	11	Activate selection to refresh drawing.	ENTER
6	Enter value. CNC draws all possible tangent lines and prompts for a selection.	ENTER			

Create line tangent to bottom of both circles.					
1	Highlight tool to create a line tangent to two circles.	Select	7	Key in 4.	4 FEED
2	Activate selected tool. CNC prompts for first circle number.	ENTER	8	Enter value. CNC clears all but the selected line.	ENTER
3	Key in 1.	1 RAPID	9	Activate DISPLAY pop-up menu.	**DISPLAY** (F5)
4	Enter value. CNC prompts for second circle number.	ENTER	10	Highlight required selection.	Select **Redraw**
5	Key in 2.	2 LINE	11	Activate **Redraw** to refresh drawing.	ENTER
6	Enter value. CNC draws all possible tangent lines and prompts for a selection.	ENTER			

Create point at intersection of circle #1 and tangent line #3.					
1	Highlight tool to create a point where two elements intersect.	Select	4	Enter value. CNC prompts for second element number.	ALT-USE
2	Activate selected tool. CNC prompts for first element number.	ALT-USE	5	Key in 3.	3 ARC
3	Key in 1.	1 RAPID	6	Enter value. CNC adds a numbered point at the intersection.	ALT-USE

Create point at intersection of circle #2 and tangent line #3.					
1	Highlight tool to create a point where two elements intersect.	Select	4	Enter value. CNC prompts for second element number.	ALT-USE
2	Activate selected tool. CNC prompts for first element number.	ALT-USE	5	Key in 3.	3 ARC
3	Key in 2.	2 LINE	6	Enter value. CNC adds a numbered point at the intersection.	ALT-USE

Create point at intersection of circle #2 and tangent line #4.					
1	Highlight tool to create a point where two elements intersect.	Select	4	Enter value. CNC prompts for second element number.	ALT-USE
2	Activate selected tool. CNC prompts for first element number.	ALT-USE	5	Key in 4.	4 FEED
3	Key in 2.	2 LINE	6	Enter value. CNC adds a numbered point at the intersection.	ALT-USE

Create point at intersection of circle #1 and tangent line #4.					
1	Highlight tool to create a point where two elements intersect.	Select	4	Enter value. CNC prompts for second element number.	ENTER
2	Activate selected tool. CNC prompts for first element number.	ENTER	5	Key in 4.	4 FEED
3	Key in 1.	1 RAPID	6	Enter value. CNC adds a numbered point at the intersection.	ENTER

Verify the coordinates of points #5, #6, #7, #8, and close calculator.					
1	Activate Geometry pop-up menu.	**GEOMETR** (F7)	3	Close the Geometry List.	**Cancel** (F9)
2	Activate the Geometry List and verify the listed position coordinates make sense. Positions on Geometry List are automatically stored in Geometry Calculator's memory.	ENTER	4	Close Geometry Calculator.	**Exit** (F10)

Practice Exercise #8 - Programming SP4.M Using Calculator Positions

This exercise programs the irregular pocket shown in Sample Program #4. Exercise # 5 and #6 should already be done. The tangent positions found using the Geometry Calculator in Exercise #6 will be recalled into the program in this exercise.

The program in this exercise uses an Irregular Pocket cycle. Careful planning of the moves in the subprogram makes programming an irregular pocket easier. Refer to **Section 4 - Writing Programs** for a detailed explanation on planning an Irregular Pocket cycle.

The main program must always come before any subprograms. In this exercise the subprogram is written and tested first. After testing, the main program is added in front of it.

This program is written in three steps.

- Step 1 Writing the subprogram.
- Step 2 Test the subprogram with Draw.
- Step 3 Write the main part of the program in front of the subprogram.

Step 1 - Define Irregular Outline By Writing Subprogram

Refer to **Figure 12 - 16, Geometry Calculator Layout for Exercise #7**. The easy way to program the pocket outline is to recall the positions from points #5, #6, #7 and #8 directly into the program. Start this step from the **PROGRAM** screen.

Activate the program editor and program a subprogram call.					
Sub 99					
1	Highlight required program.	Select **SP4.M**	5	Key in 9.	9 PLANE
2	Activate program editor.	**Edit** (F4)	6	key in 9.	9 PLANE
3	Activate the subprogram softkeys.	**Sub** (F8)	7	Add block to program.	**Save** (F10)
4	Activate Subprogram Graphic Menu.	**Sub** (F1)	8	Restore default softkeys.	**Prev** (F9)

Program a Rapid move to starting point.					
X -1.2500 Y 0.0000					
1	Activate Rapid Graphic Menu	1 RAPID	6	Key in a 5.	5 TOOL
2	Change sign.	+/- Toggle Negative	7	Advance to **Y**.	ENTER
3	Key in 1.	1 RAPID	8	Key in 0.	0
4	Key in a decimal.	.	9	Add block to program.	**Save** (F10)
5	Key in a 2.	2 LINE			

Program an Arc to the first tangent point (#5).					
Arc Cw X 0.1250 Y 1.2437 Radius 1.2500					
1	Activate Mill softkeys.	**Mill** (F5)	9	Activate for **Both X and Y.**	ENTER
2	Activate Arc Graphic Menu.	**Arc** (F4)	10	Highlight required field.	Select **Radius**
3	Advance to **X.**	ENTER	11	Key in 1.	1 RAPID
4	Activate Recall pop-up menu.	**Recall** (F2)	12	Key in a Decimal.	.
5	Select Geometry Calculator.	Select	13	Key in a 2.	2 LINE
6	Activate Geometry Calculator recall.	ENTER	14	Key in a 5.	5 TOOL
7	Highlight required point.	Select **5. X 0.125 Y 1.2437**	15	Add block to program.	**Save** (F10)
8	Activate pop-up menu to select term.	ENTER			

Program Line move to second tangent point (#6).					
Line X 5.0750 Y 0.7462					
1	Activate Line Graphic Menu.	2 LINE	5	Highlight required point.	Select **6. X 5.0750 Y 0.7462**
2	Activate Recall pop-up menu.	**Recall** (F2)	6	Activate pop-up menu to select term.	ENTER
3	Select Geometry Calculator.	Select	7	Activate for **Both X and Y.**	ENTER
4	Activate Geometry Calculator recall.	ENTER	8	Add block to program.	**Save** (F10)

Program an Arc to third tangent point (#7).					
Arc Cw X 5.0750 Y -0.7462 Radius 0.7500					
1	Activate Arc Graphic Menu.	**Arc** (F4)	8	Activate for **Both X and Y.**	ENTER
2	Advance to **X.**	ENTER	9	Highlight required field.	Select **Radius**
3	Activate Recall pop-up menu.	**Recall** (F2)	10	Key in a decimal point.	.
4	Select Geometry Calculator.	Select	11	Key in 7.	7 UNIT
5	Activate Geometry Calculator recall.	ENTER	12	Key in 5.	5 TOOL
6	Highlight required point.	Select **7. X 5.0750 Y - 0.7462**	13	Add block to program.	**Save** (F10)
7	Activate pop-up menu to select term.	ENTER			

Program Line move to fourth tangent point (#8).					
Line X 0.1250 Y -1.2437					
1	Activate Line Graphic Menu.	2 LINE	5	Highlight required point.	Select **8. X 0.1250 Y -1.2437**
2	Activate Recall pop-up menu.	**Recall** (F2)	6	Activate **Both X and Y.**	ENTER
3	Select Geometry Calculator.	Select	7	Add block to program.	**Save** (F10)
4	Activate Geometry Calculator recall.	ENTER			

Program an Arc to the Starting point (closing the shape of the pocket).					
Arc Cw X -1.2500 Y 0.0000 Radius 1.2500					
1	Activate the Arc Graphic Menu.	(F4) **Arc**	8	Highlight required field.	Select **Radius**
2	Advance to **X**.	ENTER	9	Key in 1.	1 RAPID
3	Change sign.	+/-	10	Key in a decimal.	.
4	Key in 1.	1 RAPID	11	Key in a 2.	2 LINE
5	Key in a decimal.	.	12	Key in 5.	5 TOOL
6	Key in a 2.	2 LINE	13	Add block to program.	**Save** (F10)
7	Key in 5.	5 TOOL			

Program a block to tell the CNC where the subprogram ends.					
EndSub					
1	Restore the default softkeys.	**Prev** (F9)	4	Restore the default softkeys.	**Prev** (F9)
2	Activate the subprogram softkeys.	**Sub** (F8)	5	Close the editor to save the program.	**Exit** (F10)
3	Add the **Endsub** block to the program.	**EndSub** (F2)			

Step 2 - Check Subprogram Using Draw

When the subprogram is written and saved, check it. A subprogram cannot run alone. To check it, a short main program (that only calls the subprogram) is temporarily keyed in. Once the subprogram is checked, the temporary blocks are deleted and the rest of the main program is written.

The last block of a main program is an EndMain block. For test purposes, the main program is a subprogram Call and an EndMain block. Start this step from the **PROGRAM** screen.

NOTE: The Draw parameter settings made in Practice Exercise #5 are required to ensure the CNC screens follow the procedure in this step. Accomplish Practice Exercise #5 up to keystroke 15 (turn Tool on, Rapid on, set the Mode to Auto and turn Run off), if these settings have been changed.

Activate the editor and write a subprogram call block.					
Call 99					
1	Highlight required program.	Select **SP4.M**	5	Key in 9.	9 PLANE
2	Activate program editor.	**Edit** (F4)	6	Key in 9.	9 PLANE
3	Activate the subprogram softkeys.	**Sub** (F8)	7	Add the block to the program.	**Save** (F10)
4	Activate the Call Graphic Menu.	**Call** (F3)			

Add EndMain block to signal the end of the main program. This block must be the last block in the main program (just in front of the first block in the subprogram).					
EndMain					
1	Add EndMain block to the program.	**EndMain** (F4)	2	Restore the default softkey line.	**Prev** (F9)

Activate Draw set the view and run the program.					
1	Activate Draw	**Draw** (F2)	6	Activate Selection and pause for the "Program ended normally" message.	ENTER
2	Activate VIEW pop-up.	**VIEW** (F4)	7	Run the program to verify the pocket shape.	**Run** (F3)
3	Activate highlighted selection.	ENTER	8	Exit to the program editor.	**Exit** (F10)
4	Activate DISPLAY pop-up.	**DISPLAY** (F5)	9	Close the editor to save the program.	**Exit** (F10)

Step 3 - Writing the Main Program

In an Irregular Pocket cycle, if no starting angle is specified, the first cut will be in the same direction as the first straight Line (feed) move in the subprogram. If no starting position is specified, the first cut will start at the starting point of the first feed move in the subprogram.

In this exercise the starting position is fine but the direction is not. An Angle value must be used to choose a direction that will let the CNC mill out the entire pocket. A starting angle of 90 degrees (from the three o'clock position) might seem convenient but a 90 degree line from the starting point (on the Arc) is tangent to the Arc. A Tangent line does not point to a position inside the pocket. An 89.9 degree Angle is used because it points to a position 0.1 degrees inside the Arc. Refer to **Section 4 - Writing Programs** for more information on planning Irregular Pocket cycles.

The EndMain block added in the previous step will be re-used. The Graphic Menu for the irregular pocket has its own Sub# field so the Call 99 block can be deleted. Start this step from the **PROGRAM** screen.

Delete the Call 99 block.					
1	Select the required program.	Select **SP4.M**	3	Clear the selected block.	CLEAR
2	Activate the program Editor.	**Edit** (F4)			

Use the first block in the main program to activate the Absolute Mode.					
Dim Abs					
1	Activate ABS/INC Graphic menu.	ABS INC	2	Add block to the program.	ENTER

Program irregular pocket block.					
Pocket Sub 99 StartHgt 0.1000 ZDepth -0.2500 Angle 89.9000 XStart 0.0000 YStart 0.0000 Stepover 0.1600 DepthCut 0.1250 FinStock 0.0100 RoughFeed 12.5 FinFeed 9.5 Tool# 1					
1	Activate Pocket Pop-up menu.	**Pocket** (F4)	19	Highlight required field.	Select **Stepover**
2	Highlight required selection.	Select **Irregular**	20	Key in a decimal point.	.
3	Activate Irregular Graphic Menu.	ENTER	21	Key in 1.	1 RAPID
4	Key in 9.	9 PLANE	22	Key in 6.	6 MODE
5	Key in 9.	9 PLANE	23	Advance to **DepthCut**.	ENTER

6	Highlight required field.	Select **StartHgt**
7	Key in a decimal point.	.
8	Key in 1.	1 RAPID
9	Advance to **ZDepth**.	ENTER
10	Change sign.	+/- Toggle Negative
11	Key in a decimal point.	.
12	Key in a 2.	2 LINE
13	Key in a 5.	5 TOOL
14	Advance to **Angle**.	ENTER
15	Key in 8.	8 DWELL
16	Key in 9.	9 PLANE
17	Key in a decimal point.	.
18	Key in 9.	9 PLANE
24	Key in a decimal point.	.
25	Key in 1.	1 RAPID
26	Key in 2.	2 LINE
27	Key in a 5.	5 TOOL
28	Advance to **FinStock**.	ENTER
29	Key in decimal point.	.
30	Key in 0.	0
31	Key in 1.	1 RAPID
32	Advance to required field.	Select **RoughFeed**
33	Key in 1.	1 RAPID
34	Key in 2.	2 LINE
35	Key in a decimal point.	.
36	Key in 5.	5 TOOL

- Table Continued -

Program irregular pocket block (continued).		
37	Advance to **FinFeed**.	ENTER
38	Key in a 9.	9 PLANE
39	Key in a decimal point.	.
40	Key in a 5.	5 TOOL
41	Advance to **Tool#**.	ENTER
42	Key in 1.	1 RAPID
43	Add block to program listing.	**Save** (F10)

Fully retract the quill.		
Rapid Z 0.0000 Tool# 0		
1	Activate Rapid Graphic Menu.	1 RAPID
	Advance to **Z**.	Select
	Key in 0.	0
	Add block to program listing.	**Save** (F10)

		Z
	Key in 0.	0
	Advance to **Tool#.**	Select **Tool#**

	Exit the editor saving the program.	**Exit** (F10)

Step 4 - Viewing the Finished Program with Draw

This program is written for a 3/8 inch endmill. In this step the tool diameter is entered on the Tool Page and the program is checked by running it in Draw. Start this step from the **PROGRAM** screen.

View the finished program in Draw.		
1	Select the required program.	Select **SP4.M**
2	Activate the editor.	**Edit** (F4)
3	Activate the Tool Page	**Tool** (F6)
4	Key in a decimal point.	.
5	Key in a 3.	3 ARC
6	Key in a 7.	7 UNIT
7	Key in a 5.	5 TOOL
8	Exit the Tool Page.	**Exit** (F10)
9	Activate Draw.	**Draw** (F2)
10	Run the program to verify the milling of the pocket.	**Run** (F3)
11	Close Draw.	**Exit** (F10)
12	Close the editor.	**Exit** (F10)
13	Exit to the MANUAL screen.	**Exit** (F10)

ANILAM

ANILAM Inc.
3300 Corporate Way
Miramar, FL
USA 33025

☎ (954) 430-6600
FAX (954) 450-6700

ANILAM Ltd.
16 Plover Close, Interchange Park
Newport Pagnell
Bucks, MK16 9PS.
England

☎(0) 1908 514500
FAX (0) 1908 610111

ANILAM Inc.
(Regional office) Petalling Jaya Ind. Park
A1 - A14, 1st/FL., Jalan Klemajuan
46200 Petalling Jaya, Malaysia

☎ (603) 757 7619
FAX (603) 757 4659

ANILAM GMBH
Kehisteinstrasse 2
D-84529 Tittmoning

☎ (08683) 8985-21
FAX (08683) 7347

ANILAM s.r.l.
10043 Orbassano
Strada Borgaretto 38
Torino, Italy

☎ (011) 900 2606
FAX (011) 900 2466

P/N 70000289
23-Aug-96