ANILAM

3000M
Student Workbook for
Three-Axis Systems

www.anilam.com

Warranty

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

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

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

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

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

Section 1 - Fundamentals of Programming

Section 2 - CNC Console

Section 3 - Programming Sample

Section 1 - Fundamentals of Programming

Welcome to ANILAM's Training Program!

This workbook and the training tape will take you on a step-by-step through the fundamentals of CNC programming and machine setup. This work book applies to Anilam 3000M CNC.

The 3000M is a closed-loop system. It receives positioning information from highly accurate measurement transducers and compares the actual position against the programmed positions. Simultaneously, the control regulates the speed and position of the controlled axis until each command is completed.

If you have already created CNC programs, you will probably not need to spend much time in this section and might want to move on to "Section 3 - Programming Sample." If this is new to you, make sure that you understand everything before you move on to the next section. For best results:

- Pay close attention to the explanation of positive and negative signs.
- Do all of the exercises in the workbook.
- View the accompanying video to understand the concepts discussed.

The 3000M cuts arcs and angles, helical shapes and 3-axis shapes of practically any configuration. In the hands of a good machinist, its speed and accuracy offer unlimited capabilities.

Your input as a programmer is critical to the CNC process.

NOTE: Use this workbook with the training video.

Tool Motion Orientation

Refer to Figure 1-1, Mill Axes of Motion (Tool Motion Orientation). The machine moves along its axes of motion. All movement along an axis is in either a positive or negative direction. Not all machines use the same system to identify axes. The descriptions here are most commonly used for 3-axis mills.

NOTE: When programming machine movements, always consider tool motion rather than table motion for the sake of clarity.

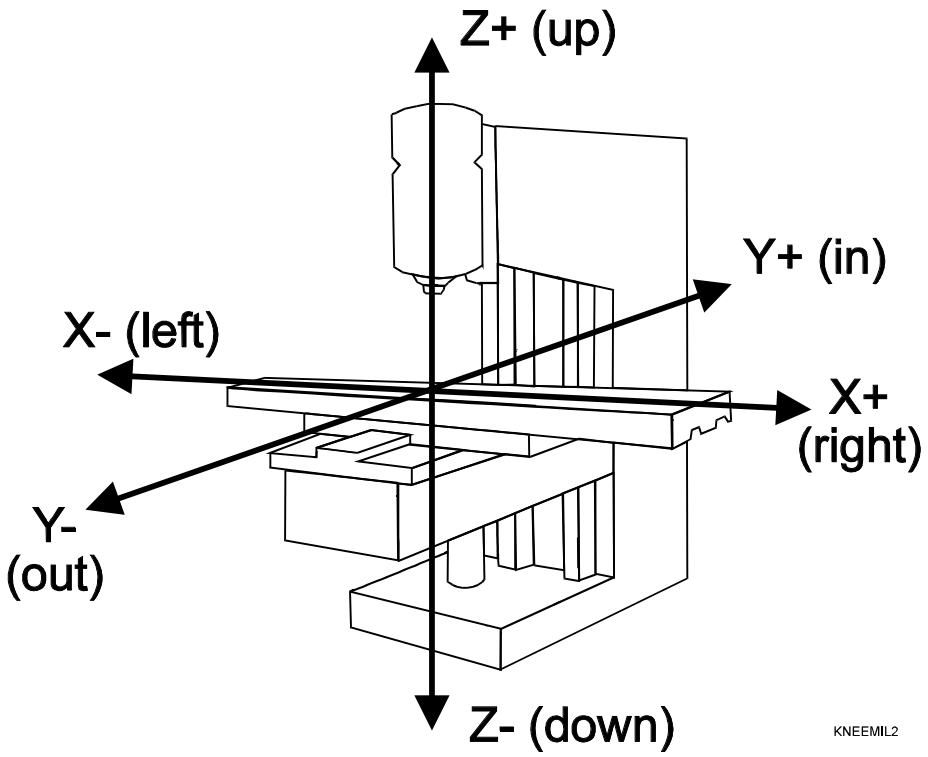


Figure 1-1, Mill Axes of Motion (Tool Motion Orientation)

X Axis

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

Y Axis

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

Z Axis

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

Absolute Positioning

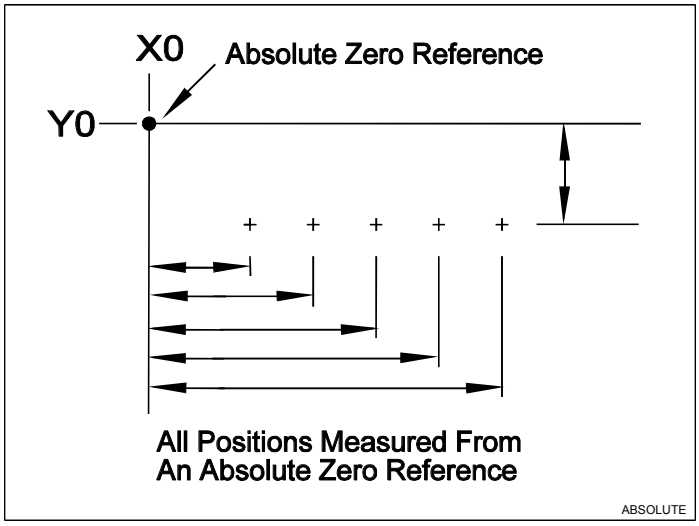


Figure 1-2, Absolute Positioning

Refer to **Figure 1-2**. In the Absolute Mode, all positions are measured from Absolute Zero. Absolute Zero is not a fixed position on the machine, but a point the operator selects.

You can set Absolute Zero (X0, Y0) anywhere. Set Absolute Zero at a position that makes it easy to use the dimensions on a blueprint. This is called “setting Part Zero”.

Incremental Positioning

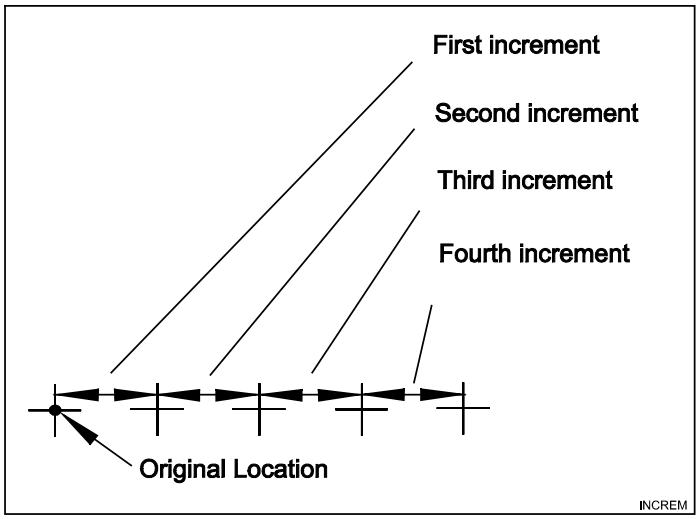


Figure 1-3, Incremental Positioning

Refer to **Figure 1-3**. Measure Incremental moves from the machine’s present position. This is convenient if you must perform an operation at regular intervals.

Defining Positions

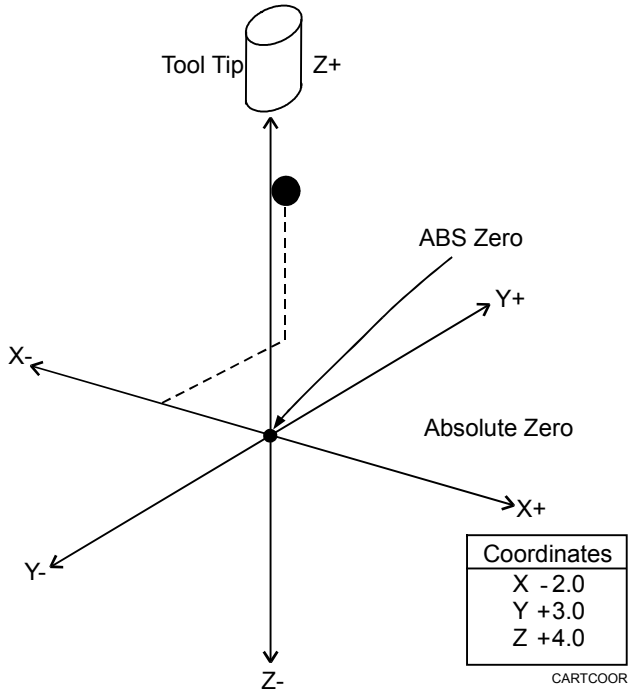


Figure 1-4, Locating Positions

Refer to **Figure 1-4**. The intersection of the X, Y and Z axes is the reference point that defines most positions. This point is the X0, Y0, Z0 position. It is usually Absolute Zero. Most positions are identified by their X, Y, and Z coordinates.

Example 1, Absolute Dimensions

A position two inches left, three inches back, and four inches up has the following coordinates:

X-2.0

Y3.0

Z4.0

Use this system of measurement, known as the Cartesian Coordinate System, to describe the location of any point within the range of motion. Refer to Figure 1-5, Cartesian Coordinates for examples.

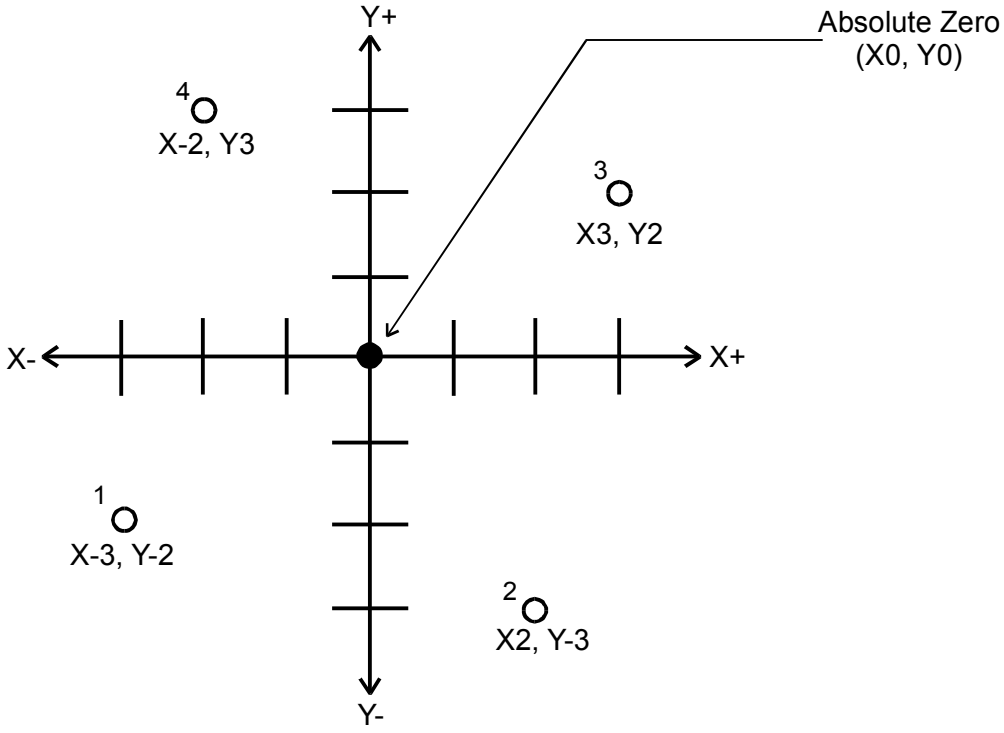


Figure 1-5, Cartesian Coordinates

The figure is a center-referenced blueprint with four hole locations. Each hole location describes a specific X, Y position, referenced from Absolute Zero (X0, Y0).

Point 1 is located at X negative 3, Y negative 2 (X-3, Y-2). Point 2 is located at X positive 2, Y negative 3 (X2, Y-3). Point 3 is located at X positive 3, Y positive 2 (X 3, Y2). Point 4 is located at X negative 2, Y positive 3 (X-2, Y3). These are **Absolute dimensions**. Absolute dimensions are referenced to Absolute Zero (X0, Y0).

Example 2, Incremental Dimensions

You can also describe these locations in **Incremental dimensions**. An incremental dimension is the distance from one point to the next.

A tool starts at Point 1 and travels to Point 4 (refer to **Figure 1-5**). Calculate the X, Y Incremental distance from Point 1 to Point 4. In X, the tool moves one inch in the positive direction. In Y, the tool moves five inches in the positive direction.

Therefore, the Incremental dimensions of the move from Point 1 to Point 4 are X1.0, Y5.0.

Quiz 1

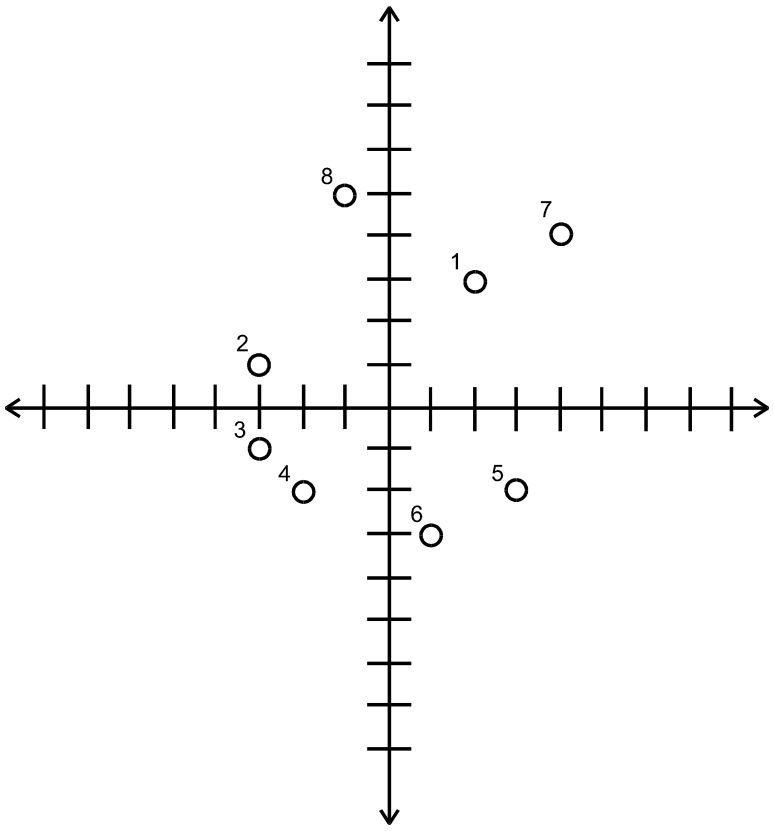


Figure 1-6, Quiz 1 Coordinates

Directions: Refer to **Figure 1-6**. Circle the correct answers.

I.

The X-axis runs:	The Y-axis runs:	The Z-axis runs:
a. in and out	a. in and out	a. in and out
b. back and forth	b. back and forth	b. back and forth
c. up and down	c. up and down	c. up and down

II. Circle the choice (a, b, c or d) that gives the correct X, Y coordinates of each point (1 to 8). Refer to **Figure 1-6**.

1. a. X3, Y2 b. X2, Y3 c. X-3, Y2 d. X-2, Y3	2. a. X3, Y-1 b. X1, Y-3 c. X-1, Y-3 d. X-3, Y1	3. a. X3, Y-1 b. X1, Y-3 c. X-1, Y-3 d. X-3, Y-1	4. a. X2, Y-2 b. X2, Y-2 c. X-2, Y-2 d. X-2, Y2
5. a. X2, Y-3 b. X-3, Y2 c. X3, Y-2 d. X-2, Y3	6. a. X1, Y-3 b. X-3, Y1 c. X-1, Y-3 d. X3, Y-1	7. a. X4, Y4 b. X-4, Y4 c. X4, Y-4 d. X-4, Y-4	8. a. X-1, Y5 b. X5, Y-1 c. X-5, Y-1 d. X1, Y-5

III. What is the incremental distance from Point 5 to Point 6?

Quiz 1 Answer Key

I.

X-axis: b. Y-axis: a. Z-axis: c

II.

1. b.	2. d.	3. d.	4. c.
5. c.	6. a.	7. a	8. a.

III. X-2, Y-1

Section 2 - CNC Console

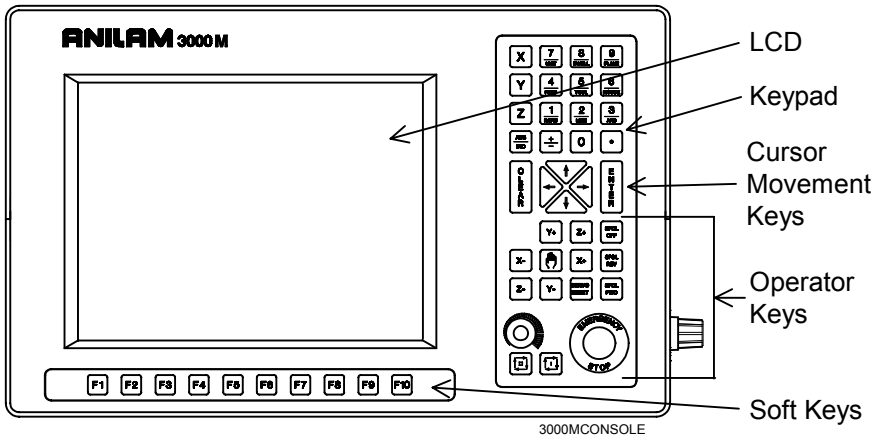


Figure 2-1, 3000M Console

Console Overview

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

- Alphanumeric Keys
- Cursor Movement Keys
- Operator Keys
- Soft Keys

The console has a 12.1" LCD screen that displays the programming functions and canned cycles.

Keyboard Layout

The Alphanumeric Keys at the top of the keyboard include the X, Y, and Z dimension keys and the number keys. The number keys are hotkeys that have dual purposes that will be discussed later. Use these to program moves. Press **ABS/INC** to switch between the Absolute and Incremental Modes. Cursor Movement Keys include: **CLEAR**, **ENTER**, and the **ARROWS**.

Operator Keys control machine and spindle movements manually. You can use the Jog keys to manually move the machine. The **JOG SELECTOR** key (shaped like a hand) selects the Rapid, Feed, or Jog (1, 10, or 100) speed at which the machine will travel during a Jog move. The X+, Y+, Z+, X-, Y-, and Z- keys indicate the axis and direction of the Jog move. The **SERVO RESET** powers up the servo motors. Spindle keys control spindle movement (spindle off, spindle forward and spindle reverse) on machines equipped with those functions.

The **START** and **STOP** keys initiate and halt machine operation. The **E-STOP** performs an emergency shutdown of all functions. **FEEDRATE OVERRIDE** overrides the active feedrate to increase or decrease machine speed. Soft keys **F1** to **F10** beneath the console correspond to the on-screen labels.

Manual Mode Screen

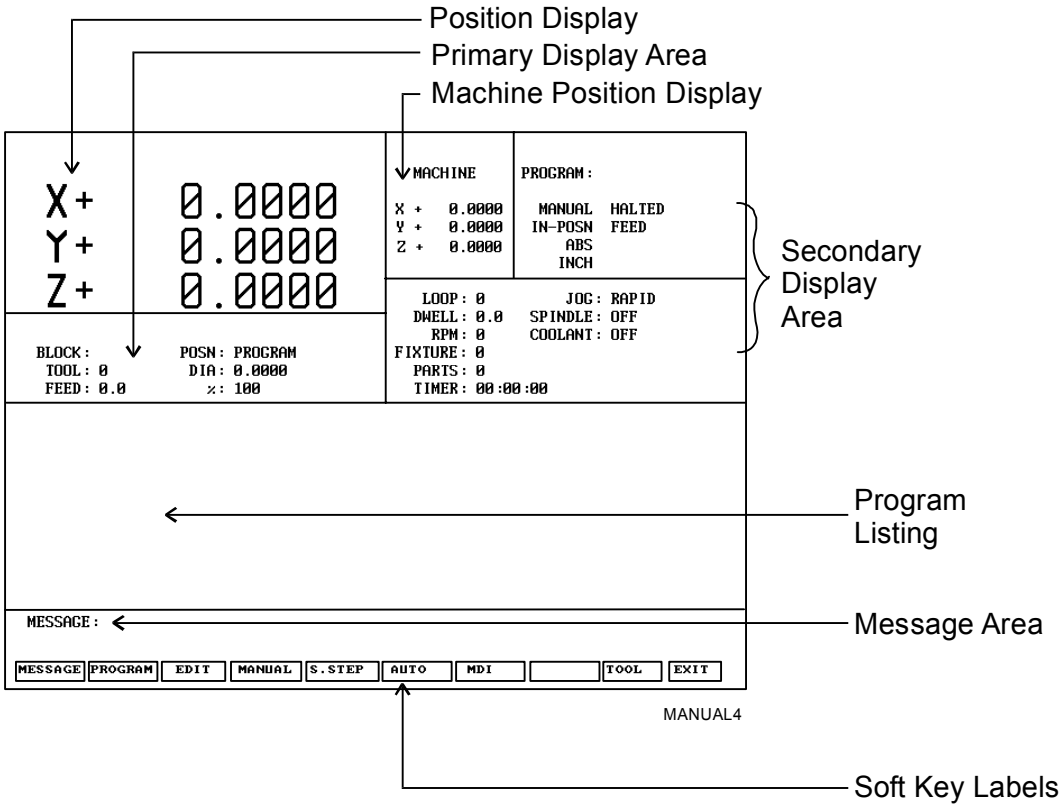


Figure 2-2, Manual Mode Screen

Refer to **Figure 2-2**. The Manual screen is the main CNC screen. All other operating screens activate from the Manual screen. In Manual Mode, the **MANUAL** (**F4**) soft key label highlights.

The Manual screen features:

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

Machine Position Display
This area of the screen shows the axis positions in reference to Machine Home.

Primary Display Area
Displays essential operating information.

Secondary Display Area
Displays additional operating information.

Message Area Displays messages, prompts, and reminders.

Soft Key Labels	Identify the function of the soft key directly underneath. Labels change from screen to screen; a highlighted label indicates an active mode.
Program Listing	Displays program blocks as they run.

Primary Display Area Labels

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

Secondary Display Area Labels

PROGRAM:	Name of selected program.
MANUAL/AUTO/S.STEP:	Current operating mode
IN-POSN:	Tells operator whether machine has reached target (IN-POSN) or not.
ABS / INC:	Current positioning mode.
INCH / MM:	Current units mode.
HALTED/*HALTED/RUNNING:	Without asterisk: machine is in a programmed hold, or has completed its program. With asterisk: hold was activated by an event, or **HOLD** was pressed. Running: indicates normal program run.
FEED/RAPID/ARC:	Current move mode.
LOOP:	Number of loops remaining (when running a subprogram that has loops).
DWELL:	Seconds remaining in a dwell.
RPM:	Spindle RPM (optional). May display programmed RPM or actual RPM. Refer to builder's documentation for details.
FIXTURE:	Indicates the active fixture offset (1 to 9). "0" indicates no fixture offset is active.
JOG:	Current jog mode.

SPINDLE:FWD/REV/OFF:

Spindle status. Optional.

COOLANT: Coolant status. Optional.

PARTS: Counts the number of successfully completed parts. (Increments by one every time the CNC encounters **EndMain** in a program run.) The counter resets to zero when you start a new program.

TIMER: Total program run time from START to **EndMain** execution. If the CNC holds, the counter pauses until the program restarts. The counter resets to zero when you start a new program.

Position Display

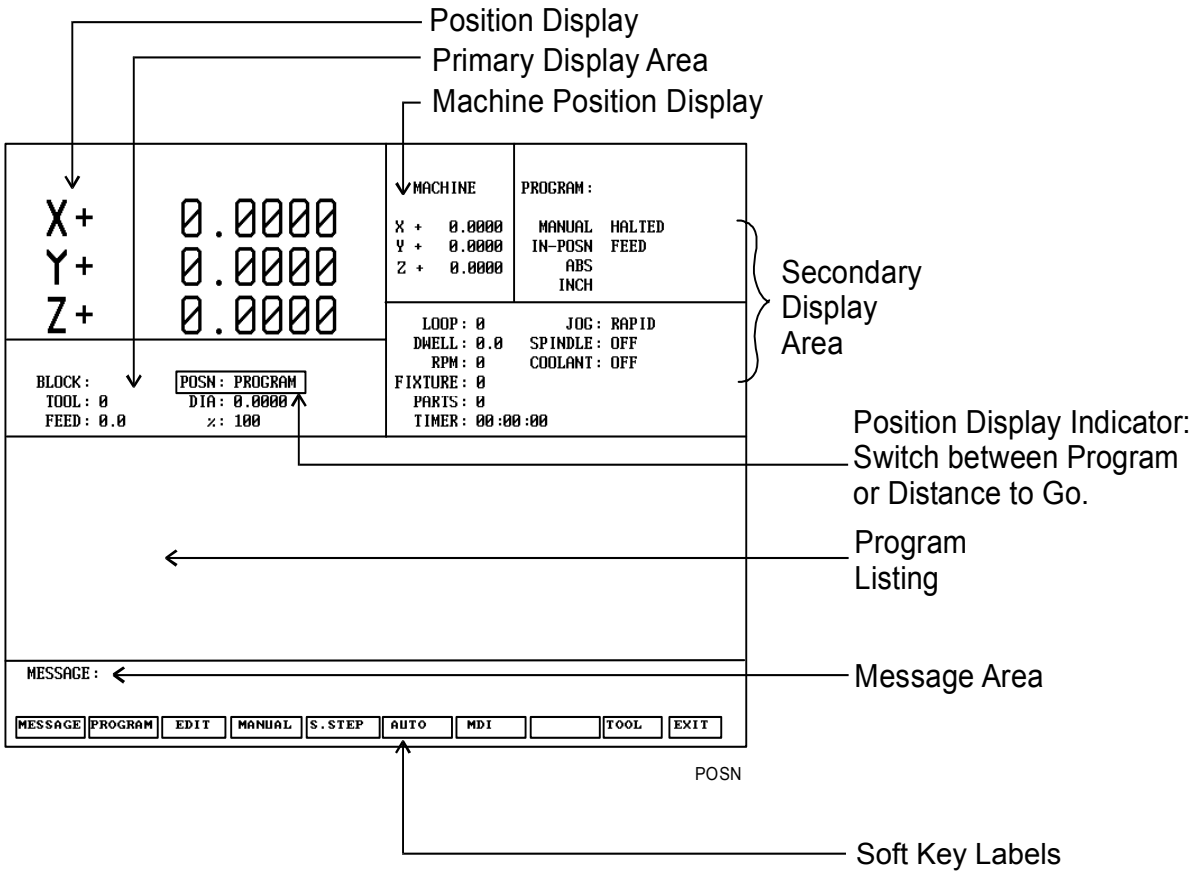


Figure 2-3, Position Display Options

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

Program Position Display shows the programmed position.

Distance To Go	Position Display shows the remaining distance to the commanded position.

To switch the **POSN** setting:

1. In Manual, S. Step, or Auto Mode, press **0** to switch the setting.

Jog Moves

Enable Jog moves when:

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

NOTE: Ensure that the CNC **POSN:** setting is in Program Mode.

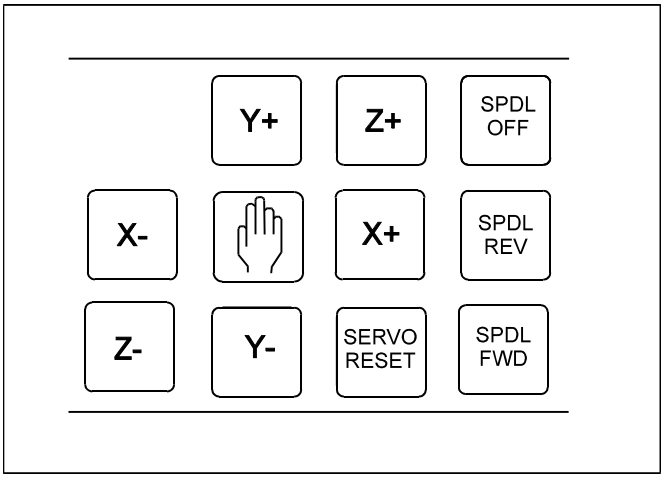


Figure 2-4, Manual Operation Keys

Refer to **Figure 2-4**. Use the Manual Operation Keys to make jog moves. There are two keys for each axis of motion: one for the positive direction and one for the negative direction.

Refer to **Table 2-1**. There are five move modes available. The machine builder determines the rate for each mode (Jog Rapid and Jog Feed) at machine setup. Press **JOG**  to cycle through the available Jog settings.

Table 2-1, Move Mode Selections

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

Table 2-1, Move Mode Selections (Continued)

Mode	Description
Jog: 10	Conventional Jog Mode, increment set to 10 times machine resolution.
Jog: 1	Conventional Jog Mode, increment set to actual machine resolution.

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

Soft Key Overview

Manual Soft Keys

In Manual Mode, note the active soft keys (**F1** to **F10**) at the bottom of the screen. Refer to **Table 2-2**.

Table 2-2, Manual Mode Soft Keys

Key	Function
Message (**F1**)	Displays the last eight messages from the CNC in the Program Listing area of the screen.
Program (**F2**)	Activates the Program Directory.
Edit (**F3**)	Activates Edit Mode for the selected program. **NOTE:** To edit a program, highlight the program name in the Program Listing and press **Edit** (**F3**).
Manual (**F4**)	Activates Manual Mode. Deactivates the active S.Step, Auto, or MDI Mode.
S.Step (**F5**)	Activates S.Step Mode.
Auto (**F6**)	Activates Auto Mode.
MDI (**F7**)	Activates MDI Mode.
Handwheel (**F8**)	Activates handwheel selection window. **NOTE:** This soft key will only be active if the handwheel setting has been enabled in the Setup Utility. This is a purchased option.
Tool (**F9**)	Activates the Tool Page.
Exit (**F10**)	Exits the Manual screen.

Program Soft Keys

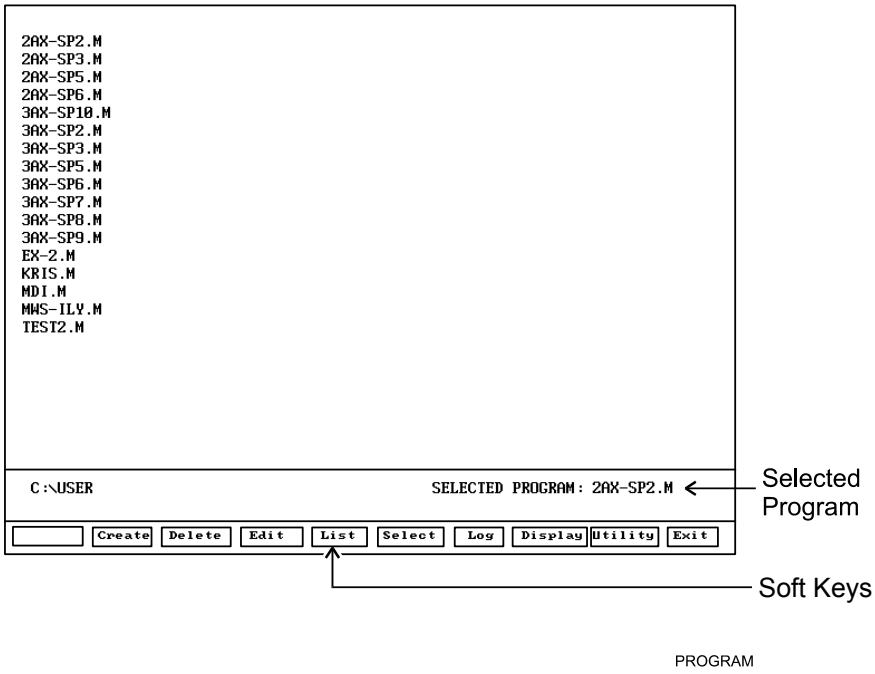


Figure 2-5, Program Directory

NOTE: Refer to *3000M CNC Programming and Operations Manual for Three- and Four-Axis Systems*, P/N 70000504, "Section 9 - Program Management" for details.

Table 2-3, Program Soft Keys and Shift Soft Keys

Key	Function
Create (**F2**)	Create a new program.
Delete (**F3**)	Delete a program.
Edit (**F4**)	Open a program to edit.
List (**F5**)	Open a program to view. Program cannot be edited.
Select (**F6**)	Select a program. A program must be selected before you can run it.
Log (**F7**)	Log onto another drive, such as the floppy drive (A:).
Display (**F8**)	Press the **F8** switch the Display Mode. The Program Listing will alternately display more or less information about the program (last edited date, file size, etc.). It will also switch the type(s) of programs displayed (*.M, *.S, and/or other extensions).
Utility (**F9**)	Choose from a pop-up listing program and file management utilities.
Exit (**F10**)	Exit to the Manual screen.
Sub Dir (**F2**)	Create a sub directory.
Del ? (**F3**)	Delete a program or directory.

Edit Soft Keys

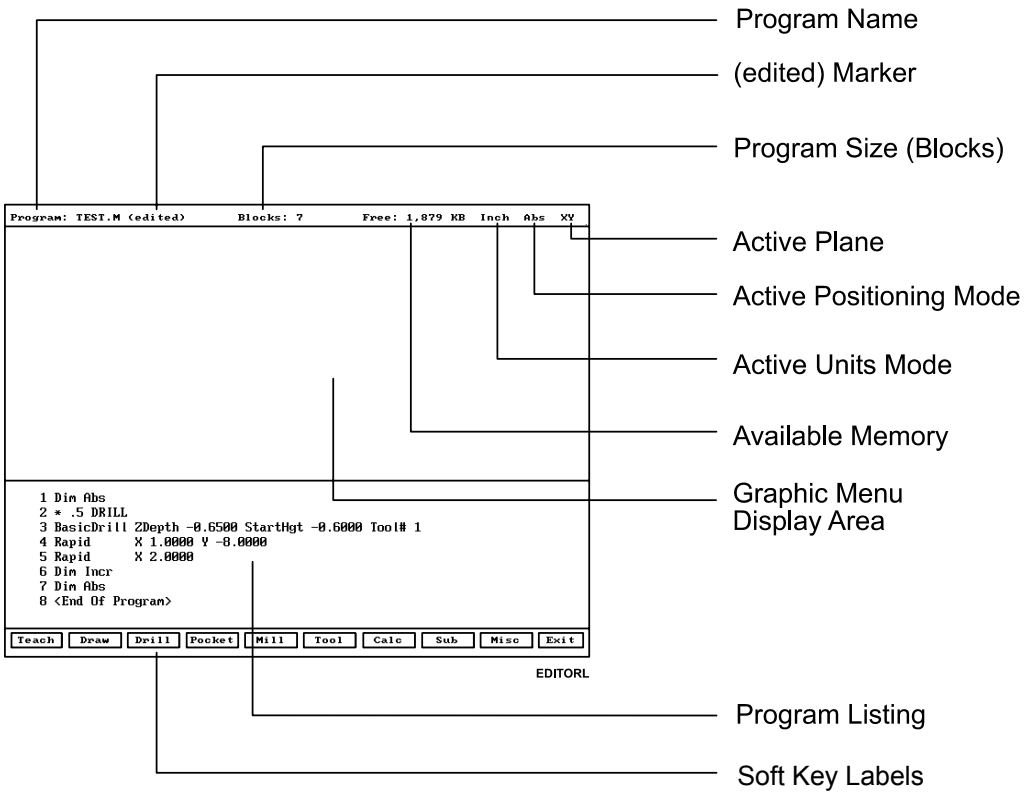


Figure 2-6, Program Editor

In Edit Mode, note the active soft keys (**F1** to **F10**) at the bottom of the screen. Refer to **Table 2-4**.

NOTE: Refer to *3000M CNC Programming and Operations Manual for Three- and Four-Axis Systems*, P/N 70000504, "Section 6 - Editing Programs" for details.

Table 2-4, Edit Mode Soft Keys

Key	Function
Teach (**F1**)	Activates/deactivates Teach Mode.
Draw (**F2**)	Activates/deactivates Draw Mode.
Drill (**F3**)	Enables you to program a drilling cycle.
Pocket (**F4**)	Enables you to program a pocket cycle.
Mill (**F5**)	Activates the Mill soft keys. Press **Rapid** (**F2**) to program a rapid move. Press **Line** (**F3**) to program a line move. Press **Arc** (**F4**) to program an arc move. Press **More** (**F7**) to activate a pop-up with the following options: Feed, Plane, Unit, Offset, SetZero, Home, Ellipse, and Spiral. Press **Prev.** (**F9**) to return to the Edit screen and soft keys.

(Continued…)

Table 2-4, Edit Mode Soft Keys (Continued)

Key	Function
Tool (**F6**)	Activates the Tool Page.
Calc. (**F7**)	Activates the calculator menu for the Math, Triangle or Geometry Calculator.
Sub (**F8**)	Activates the Subprogram soft keys. Press **Sub** (**F1**) to define a subprogram number. Press **EndSub** (**F2**) to insert an EndSub block. Press **Call** (**F3**) to program a subprogram call. Press **EndMain** (**F4**) to insert an EndMain block. Press **Loop** (**F5**) to loop a subprogram. Press **RMS** (**F6**) to rotate, mirror or scale a subprogram. Press **Dwell** (**F7**) to program a Dwell. Press **MCode** (**F8**) to program a Machine Code, if installed on the machine. Press **Prev** (**F9**) to return to the Edit screen.
Misc (**F9**)	Activates Miscellaneous soft keys. Press **Comment** (**F2**) to create a comment block. Press **Search** (**F3**) to search for a block number or specified text within the program. Use **PgUp** (**F4**) and **PgDown** (**F5**) to scroll up and down the Program Listing one page at a time (about 9 blocks). Press **Begin** (**F6**) or **End** (**F7**) to return to the first block or advance to the last block in the program. Press **Quit** (**F8**) to exit the program without saving changes. Press **Prev.** (**F9**) to return to the Edit Mode.
Exit (**F10**)	Exits Edit Mode and returns to the Manual screen. (Auto, S.Step, or Manual).

Quiz 2

Exercise 1:

Review the section on the console keyboard. Then, locate the alphanumeric section, the **CLEAR** key, the cursor control keys and the **ENTER** key. As you identify the keys, name their functions. Locate the machine movement keys. Make sure you can identify every key and its function.

Exercise 2:

Study the display. Review the section on the displayed information, including the location of programming, positioning and tooling information. Go into Manual Mode and switch through the five Jog Modes (Rapid, Feed, Jog 1, Jog 10, and Jog 100).

Exercise 3:

In Edit Mode, review the functions of soft keys **F1** to **F10**. Go through the various layers of menus to find all available cycles. Activate two or three of the canned cycle graphic menus to become familiar with these screens.

To activate a graphic menu, highlight the cycle name and press **ENTER**.

For example, to activate the Basic Drill Cycle's Graphic Menu:

1. In the Edit Mode, press **Drill** (**F3**). The Drill Pop-up Menu activates.
2. Highlight **Basic** and press **ENTER**. The Basic Drilling Cycle's Graphic Menu activates. Note the entry fields and accompanying graphic.

True or False:

1. Limit switches, when installed, limit the maximum feed rate of each axis.
2. The **X+**, **X-**, **Y+**, **Y-**, **Z+**, **Z-** keys on the console keypad are used only to program moves.
3. The Bolt Hole Pattern canned cycle is located in the Drill Pop-up Menu.
4. If you press **E-STOP**, the CNC halts movement on all axes and removes power from the servo motors.
5. In Edit Mode, press **Tool** (**F6**) to access the Tool Page.

Answer Key (True/False Only)

1. F
2. F
3. T
4. T
5. T

Section 3 - Programming Sample

The Part

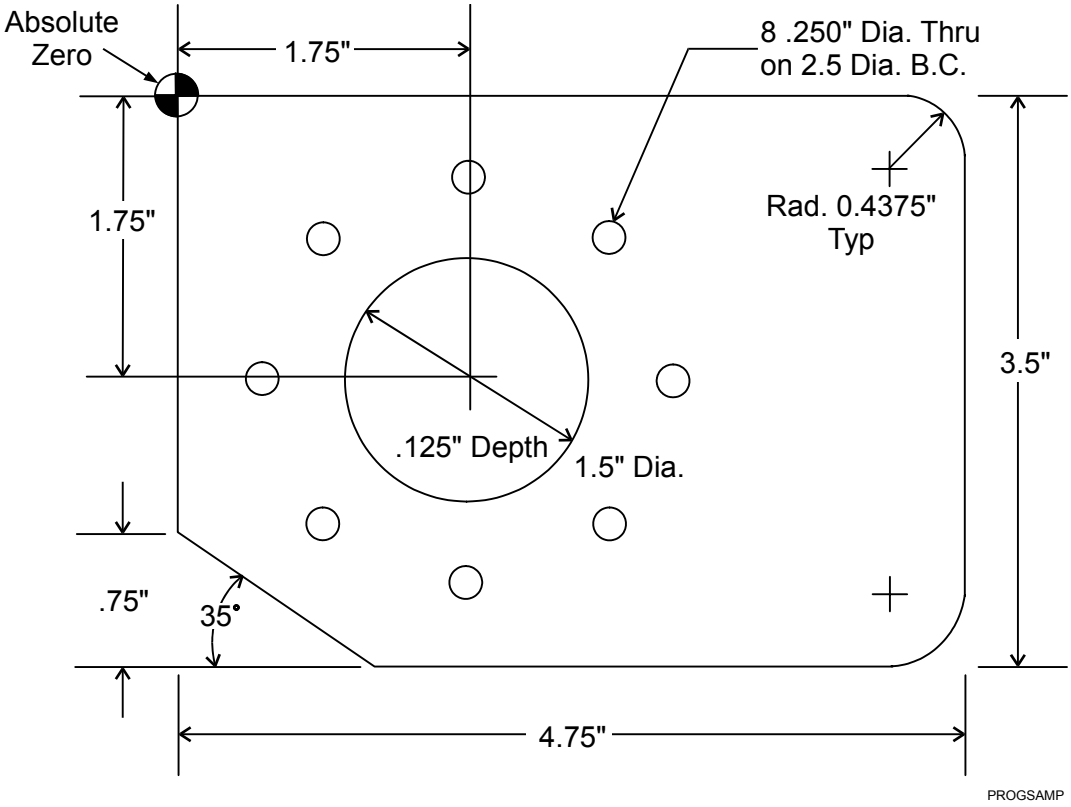


Figure 3-1, Programming Sample Part

Refer to **Figure 3-1** for all examples discussed in this section. The part contains a bolt hole circle around the circular pocket, a circular pocket near the part center and a contour around the outside.

Developing Part Programs

Summary

First, decide how to clamp the part and where to set Part Zero (X0, Y0). Absolute positions are measured from Part Zero. Therefore, locate Part Zero at a point that corresponds to dimensions on the part print.

Required Tools

For the Sample Part:

Required Tools	Tool #1 - 1/4" Drill
	Tool #2 - 7/16" End Mill

Creating a Program Name

To create a new program:

1. In Manual Mode, press **PROGRAM** (**F2**). Program Directory activates (C:\USER).
2. Press **Create** (**F2**). The Message line displays a prompt: "**NEW PROGRAM: _**"
3. If the program name has letters, press **ASCII** (**F2**). The ASCII Chart activates. Use the console keypad for numeric entries.
4. Use the ASCII Chart and the number keypad to type the new program's name. (Use "EX-1" for the Sample Part Program.)
5. Press **ASCII** (**F2**). The ASCII pop-up closes.
6. Press **ENTER** to place the new program in the Program Directory in alphabetical order.
7. Press **Select** (**F6**) to select the program.

Editing Blocks

To edit an existing program block:

1. In Edit Mode, highlight a block.
2. Press **ENTER** if the existing block is a move or cycle. The appropriate graphic menu opens.
3. Highlight the entry fields that require changes.
4. Make the appropriate changes. Press **Save** (**F10**) to close the block.

NOTE: When the program block's graphic menu offers a default entry (for example, Cw/Ccw), highlight the block and press the **+/-** key to change the selection. Use the **+/-** key to switch non-numeric settings, such as **ToolComp** (Right/Left/Off).

Write Your Own Program, Block-by-Block

Refer to Table 3-1, Sample Part Program EX-1. In the following procedure, you will go through the steps necessary to write a program that will cut a part.

NOTE: In a graphic menu, press **CLEAR** to delete an incorrect entry from a highlighted entry field.

NOTE: In Edit Mode, press **Save** (**F10**) to save a program block. In some screens, the soft keys change. In these screens, press **Prev.** (**F9**) to activate the main soft keys. (This also changes the screen). Then, press **Save** (**F10**) to save the block.

Follow the step-by-step instructions to program each block.

Table 3-1, Sample Part Program EX-1

Block #	Block	Description
1	Dim Abs	Activate Absolute Mode.
2	Rapid Z 0.0000 Tool# 0	Rapid to Z0 (tool change position). Cancel Tool Length Compensation.
3	Tool#1	Activate Tool #1 (1/4” drill).
4	PeckDrill ZDepth -0.3000 StartHgt 0.1000 Peck 0.0700 Feed 10.0	Activate Peck Drill.
5	DrillBHole XCenter 1.7500 YCenter -1.7500 #Holes 8 Diameter 2.5000 StartAngle 0.0000	Activate Bolt Hole Pattern Cycle.
6	DrillOff	Turn off Drill Cycle.
7	Rapid Z 0.0000 Tool# 0	Rapid to Z0 (tool change position). Cancel Tool 1.
8	X -1.0000 Y 0.0000	Rapid move.
9	Tool #2	Activate Tool #2 (7/16” flute end mill).
10	CircPock XCenter 1.7500 YCenter -1.7500 StartHgt 0.1000 Diameter 1.5000 ZDepth -0.1250 Ccw Stepover 0.3000 FinStock 0.0150 RoughFeed 15.0 FinFeed 12.0	Activate Circular Pocket Cycle.
11	Rapid X -0.5000 Y 0.5000	Rapid off the edge of the workpiece.
12	Line Z -0.1300 Feed 20.0	Feed to Z starting depth.
13	Line Y 0.000 ToolComp Left	Line Y Move and Tool Compensation.
14	Line X 4.7500 CornerRad 0.4375 Feed 14.0	Feed in X. Activate Corner Rounding.
15	Line Y -3.5000 CornerRad 0.4375	Feed in Y. Activate Corner Rounding.
16	Line X 1.0711	Feed move. (Recall Right Triangle Calculator value.)
17	Line X 0.0000 Y -2.7500	Feed move.
18	Line Y 0.5000	Feed move.
19	Line X -0.5000 ToolComp Off	Feed in X. Cancel Tool Compensation.
20	Rapid Z 0.0000 Tool# 0	Rapid to Z0 (tool change position). Cancel Tool #2.
21	X -1.0000 Y 1.0000	Move away from the work.
22	EndMain	End of main program.

Block 1: Set Absolute Mode

Format: Dim Abs

To set the CNC to Absolute Mode:

1. Press **ABS/INCR**.
2. Press **Save** (**F10**) to save the block.

Block 2: Cancel Tool Compensation

Format: Rapid Z 0.0000 Tool# 0

Cancel any tool length offset and raise Z to the home position. Do this in a Rapid move as follows:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Type the following values:

Z	0.0000
Tool #	0

The CNC cancels the active tool and Rapids to the Z0 home position.

Block 3: Activate Tool #1

Format: Tool#1

To activate Tool #1:

1. Press **5/TOOL**. The Tool Mount Graphic Menu activates.
2. Type the following values:

Tool#	1

The CNC activates Tool #1.

Block 4: Drilling Canned Cycles

Format: PeckDrill ZDepth -0.3000 StartHgt 0.1000 Peck 0.0700 Feed 10.0

The Peck Drilling Cycle determines how the CNC will drill each of the eight holes in the Bolt Hole Cycle that follows it.

To program the appropriate Peck Drilling Cycle:

1. In Edit Mode, press **Drill** (**F3**). The Drill Pop-Up Menu is displayed.
2. Highlight **Pecking**. Press **ENTER**. The Peck Drilling Graphic Menu is displayed.
3. Type values in all entry fields that contain “0.0000” or the cycle will not work properly. Blank entry fields are optional. Fill in the following values:

ZDepth	-0.3 (drill through plate)
StartHgt	0.100 (above the work)
Peck	0.070
Feed	10 inches per minute

NOTE: Tool #1 is a 1/4” drill.

4. Press **Save** (**F10**) to save the block.

Block 5: Bolt Hole Canned Cycle

Format: DrillBHole XCenter 1.7500 YCenter -1.7500 #Holes 8 Diameter 2.5000 StartAngle 0.0000

Use a Bolt Hole Canned Cycle to inform the CNC where to drill eight equally spaced holes around a 2.5" diameter on the Sample Part.

To program the appropriate Bolt Hole Cycle:

1. In Edit Mode, press **Drill** (**F3**). The Drill Pop-Up Menu is displayed.
2. Highlight **Bolt Hole**. Press **ENTER**. The Bolt Hole Graphic Menu is displayed.
3. Type values in all entry fields that contain "0.0000". Otherwise, the cycle will not work properly. Blank entry fields are optional. Fill in the following values:

XCenter	1.75
YCenter	-1.75
#Holes	8
Diameter	2.5
Start Angle	0 (3-o'clock position)
Tool#	Unnecessary, previously activated Tool #1.

4. Press **Save** (**F10**) to save the block.

Block 6: Drill Off Cycle

Format: DrillOff

Since the program does not require any other holes, program a Drill Off block next, as follows:

1. Press **Drill** (**F3**). The Drill Pop-Up Menu is displayed.
2. Highlight **Drilling Off** and press **ENTER**. The **Drill Off** block is displayed in the Program Listing.
3. Press **Save** (**F10**) to save the block.

Block 7: Clear the Part

Format: Rapid Z 0.0000 Tool# 0

Cancel the Tool Offsets for Tool #1 and raise Z to the home position. Do this in a Rapid move, as follows:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Fill in the following values:

Z	0
Tool	0

The CNC cancels the active tool and rapids to the Z0 home position (fully-retracted quill position).

3. Press **Save** (**F10**) to save the block.

Block 8: Move to the Tool Change Position

Format: X -1.0000 Y 0.0000

Now, move off to the side of the work to change the tool.

To program this move via the console keypad:

1. Press **X**. The Modal Move Graphic Menu activates. The **X** entry field is already highlighted. Press **+/-** to switch the sign to negative. Press **1**.
2. Highlight **Y** on the screen or press **Y** on the keypad. Press **0**.
3. Press **Save** (**F10**) to save the block.

Block 9: Activate Tool #2

Format: Tool #2

To activate Tool #2:

1. Press **5/TOOL**. The Tool Mount Graphic Menu activates.
2. Type the following values:

Tool#	2

The CNC activates Tool #2.

Block 10: Circular Pocket Milling

Format: CircPock XCenter 1.7500 YCenter -1.7500 StartHgt 0.1000 Diameter 1.5000 ZDepth -0.1250 Ccw Stepover 0.3000 FinStock 0.0150 RoughFeed 15.0 FinFeed 12.0

To program the Circular Pocket:

1. In Edit Mode, press **Pocket** (**F4**). The Pocket Pop-Up Menu activates.
2. Highlight **Circular** (Pocket). Press **ENTER**. The Circular Pocket's Graphic Menu activates.
3. Fill in the following entry field values:

XCenter	1.75
YCenter	-1.75
StartHgt	0.100 (above the part)
Diameter	1.5
ZDepth	-0.125
Direction	Ccw (counterclockwise, climb mill)
Stepover	0.300
Depthcut	leave blank (complete in one pass)
FinStock	0.015
RoughFeed	15 inches per minute
FinFeed	12 inches per minute

NOTE: Make sure you have entered the appropriate offsets in the Tool Page for Tool #2, a 7/16" end mill.

4. Press **Save** (**F10**) to save the block.

NOTE: The program is now ready for the outside contour moves.

Block 11: Rapid Move

Format: Rapid X -0.5000 Y 0.5000

Program the rapid move off to the upper left side of the part.

To program the rapid move:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Fill in the following values:

X	-0.5 (This leaves room to plunge the tool down in mid-air without plunging into any material.)
Y	0.5

3. Press **Save** (**F10**) to save the block.

Block 12: Line Z Move to Cutting Depth

Format: Line Z -0.130 Feed 20.0

Feed Z to the cutting depth with the following block:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Fill in the following values:

Z	-0.130
Feed	20 (inches per minute)

3. Press **Save** (**F10**) to save the block.

Block 13: Line Y Move and Tool Compensation

Format: Line Y 0.000 ToolComp Left

Program the feed move to the edge of the part. Enable Tool Compensation in the same block. To climb mill (clockwise tool path) around the outside of the part, enable Left-of-Path Compensation.

To program the Y feed move to the edge of the part and enable Left-of-Path Compensation:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Fill in the following values:

Y	0
ToolComp	Left
Tool#	Unnecessary. (#2 Active from Circular Pocket Cycle)

NOTE: Press the **+/-** key to switch the **Tool Comp** setting.

3. Press **Save** (**F10**) to save the block.

Block 14: Line X Move and Corner Rounding

Format: Line X 4.7500 CornerRad 0.4375 Feed 14.0

Activate Corner Rounding in the same block that programs the line along X at the top edge of the part. Corner Rounding automatically blends the intersections of two moves by the given radius.

To program the block:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Fill in the following values:

X	4.75 (full X dimension on the print)
CornerRad	0.4375
Feed	14 (inches per minute)

3. Press **Save** (**F10**) to save the block.

Block 15: Line Y Move and Corner Rounding

Format: Line Y -3.5000 CornerRad 0.4375

Activate Corner Rounding in the same block that programs the line along Y at the right-hand edge of the part. Corner Rounding automatically blends the intersections of two moves by the given radius.

To program the block:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Fill in the following values:

Y	-3.5 (full X dimension on the print)
CornerRad	0.4375

3. Press **Save** (**F10**) to save the block.

Recalling Values from the Right Triangle Calculator

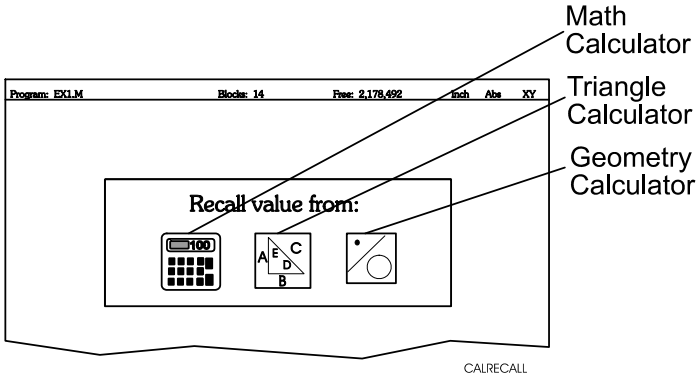


Figure 3-2, Recall Selection Pop-up

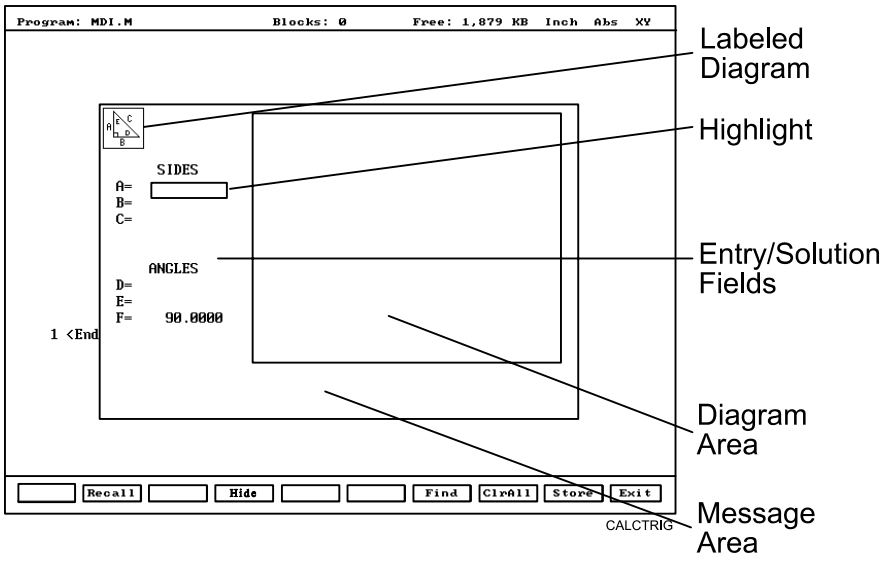


Figure 3-3, Right Triangle Calculator Screen

To recall values from the Right Triangle Calculator:

1. Open the graphic menu for the block to be edited. Highlight the entry field to which you want to recall the Triangle Calculator value.
2. Press **Recall** (**F2**). The **Select value:** menu is displayed. Refer to Figure 3-2, Recall Selection Pop-up.
3. Highlight the **Triangle Calculator** template and press **ENTER**. The Triangle Calculator memory selection pop-up is displayed. Refer to **Figure 3-4**.
4. Highlight the required value and press **ENTER** to copy the stored value to the Graphic Menu.

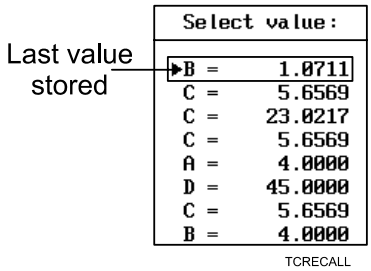


Figure 3-4, Triangle Calculator Recall Pop-up

Block 16: Line Move from Right Triangle Calculation

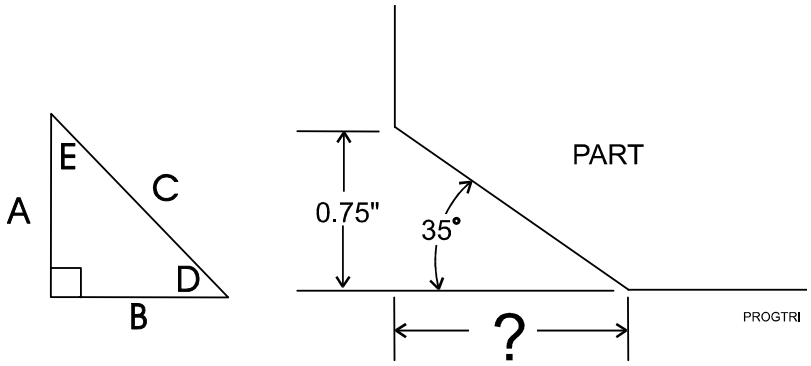


Figure 3-5, Triangle Calculation

Format: Line X 1.0711

Refer to **Figure 3-5**. The next move is also a line move. The Y dimension does not change (-3.5). However, you must calculate the X endpoint from the information given on the print before you can program the move.

I. To use the Right Triangle Calculator to solve for the X dimension:

1. Activate the Right Triangle Calculator.
2. On the print, Side A (0.75) and Angle D (35°) are given. Type the given values:

   A 0.75

   D 35
3. Press **Find** (**F7**). The CNC calculates and displays all other values (B=1.0711, C=1.3076, E=55, F=90).
4. **B** is the missing X dimension. To copy the value to memory, highlight **B** and press **Store** (**F9**).
5. Press **Exit** (**F10**) to return to the Edit screen.

II. Now, program the line move:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Highlight **X**. Recall the B Triangle Calculator value to the X entry field. Refer to "Recalling Values from the Right Triangle Calculator."
3. Press **Save** (**F10**) to save the block.

Block 17: Linear Interpolation-Angle Move

Format: Line X 0.0000 Y -2.7500

X moves to 0, feeding along an angle. X is 0. Y is -2.75 (3.5 minus 0.75).

To program the line move:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Type the X (0) and Y (-2.75) values.
3. Press **Save** (**F10**) to save the block.

Block 18: Make the Last Cut and Move Away from the Part

Format: Line Y 0.5000

Make another line move in Y to make the last cut and end up clear of the part. Move the tool away from the work:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Highlight **Y**. Type 0.5.
3. Press **Save** (**F10**) to save the block.

Block 19: Cancel Tool Compensation

Format: Line X -0.5000 ToolComp Off

NOTE: Always cancel compensation before programming Tool #0.

Cancel Tool Compensation after the CNC has machined the last contour. In the same move, move X away from the workpiece. (A move is required to cancel compensation.)

To cancel Tool Compensation:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Type the following values:

X	-.5
ToolComp	Off

3. Press **Save** (**F10**) to save the block.

Block 20: Cancel Tool #2 and Return to Z0

Format: Rapid Z 0.0000 Tool# 0

To cancel Tool #2 and move the tool to Z Home (Z0):

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Type the following values:

Z	0
Tool#	0

3. Press **Save** (**F10**) to save the block.

Block 21: Move Away from the Workpiece

Format: X -1.0000 Y 1.0000

To move away from the work:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Type the following values:

X	-1
Y	1

3. Press **Save** (**F10**) to save the block.

Block 22: Program EndMain

Format: EndMain

To add an EndMain block:

1. Press **Sub** (**F8**). The soft keys change.
2. Press **EndMain** (**F4**). The CNC adds an EndMain block to the program.

The program is finished.

Checking the Program

Refer to Table 3-1, Sample Part Program EX-1. Review the program block by block.

Draw Graphics

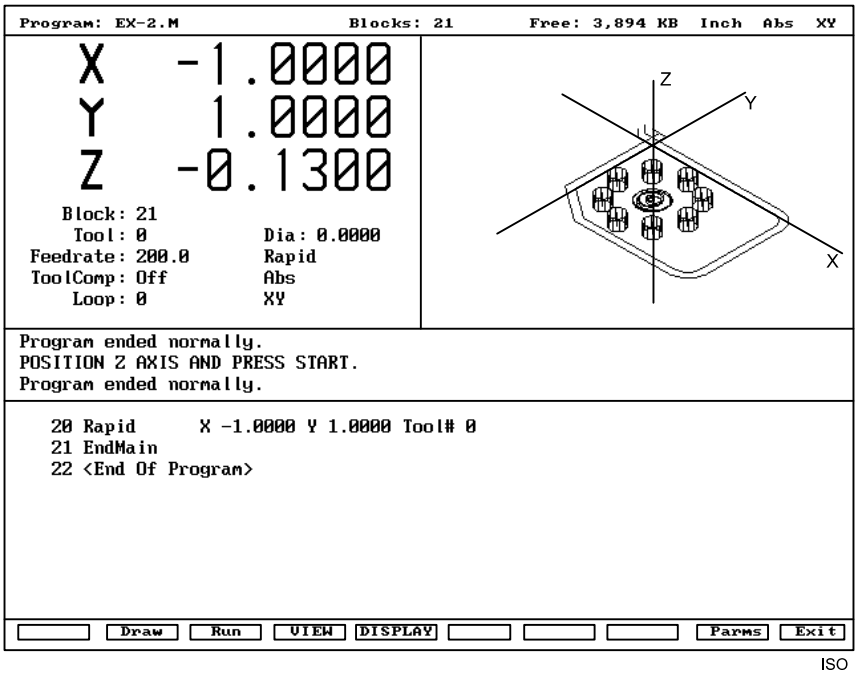


Figure 3-6, Draw Mode (Iso View) Showing Sample Part

Starting Draw

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

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

NOTE: In the Tool Page, set the tool diameter for Tool #1 to .25 in. Set the tool diameter for Tool #2 to .4375 in. Add a temporary tool length offset of .1 in. for both tools. This enables the Draw Graphics Mode to display tool movement away from the part in Z.

To activate Draw Simulation Mode:

1. In Edit Mode, select the program.
2. Press **Draw** (**F2**). The viewing displayed in upper-right corner of the screen. Draw soft keys activate.
3. Press **DISPLAY** (**F5**). A pop-up is displayed, with **Fit** highlighted.
4. Press **ENTER**. **Fit** scales the image to fit in the viewing area.
5. Press **Run** (**F3**) to run the program. The CNC traces the tool path in the viewing area, but the machine remains idle.

In Run Mode, the soft keys change to allow you to change the way the draw simulation runs. Press **Auto** (**F1**), **S. Step** (**F2**) or **Motion** (**F3**) to switch the operating mode. Run in Motion (motion-to-motion) or S. Step (block-by-block) Mode to check axis position at the end of every motion or block.

NOTE: To clear the Draw display and return to the Edit screen, press **Draw** (**F2**) or **Exit** (**F10**).

Refer to Figure 3-6, Draw Mode (Iso View) Showing Sample Part. Run the completed program in Draw Graphics (also called Simulation Mode) to verify the moves. Refer to the *3000M CNC Programming and Operations Manual for Three- and Four-Axis Systems*, P/N 70000504, "Section 7 - Viewing Programs with Draw" for more information on Draw Graphics.

The CNC simulates drilled holes as cylinders fixed on the hole position.

NOTE: The CNC will simulate drilled holes only if you have entered a Tool Diameter for the active tool (in the Tool Page).

Draw Graphics usually runs the program twice; first without Tool Compensation, then with Tool Compensation. The first drawing shows the actual programmed tool path. The second drawing is the compensated tool path that the machine will actually follow. Compare these views with the blueprint to make sure each move begins and ends where it should, and that Tool Compensation activates and deactivates as required.

View the program in Isometric view (3D), then in the XY (top) view for the best results (refer to **Figure 3-7**). Look at it from the front (XZ) or end (YZ) views to examine the depths of the cuts. If you detect any errors, return to the Edit Mode and correct the program as necessary. Re-run the program in Draw to verify any corrections before you run the program.

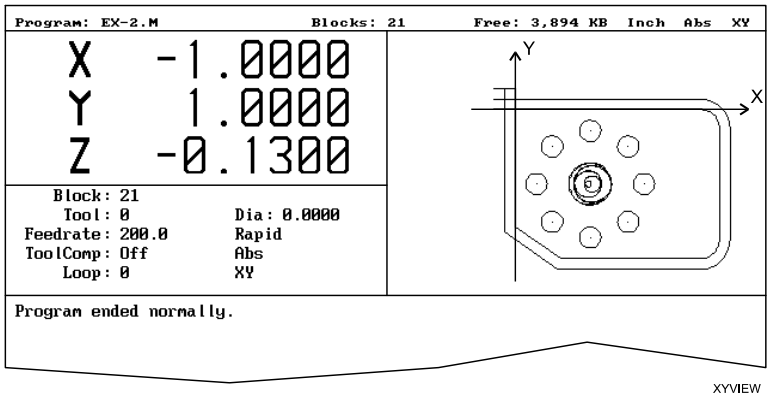


Figure 3-7, Draw Mode XY View (Top)

NOTE: If the operator sets Draw to display both compensated and uncompensated moves, Draw runs the program twice: once with then once without compensated moves. For comparison, the tool paths of both versions appear on the screen.

Putting Draw in Hold

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

Selecting the View Mode

View Draw from one of the following viewpoints:

- **XY** plane (top view)
- **XZ** plane (front view)
- **YZ** plane (side or end view)
- **Iso** (3D, with gridlines)

To set View Mode:

1. In Draw Mode, press **VIEW** (**F4**). The View Pop-Up Menu is displayed.
2. Highlight **XY**, **XZ**, **YZ**, or **Iso**. Press **ENTER**. Draw orients the display to the selected View Mode.

Tool On or Off

Turn **Tool On** to display a drawing of the tool as it moves through the part. Draw displays only the active tool. The tool must have a diameter on the Tool Page or it will not appear in Draw. Draw scales the displayed tool (cylinder) to the corresponding diameter. With **Tool Off**, Draw runs the program faster.

Default: **On**.

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

1. In Draw Mode, press **Parms** (**F9**). The Parameter Pop-Up Menu is displayed.
2. Highlight **Tool** and press **ENTER**. **Tool** switches between **On** and **Off**.
3. Press **Parms** (**F9**). The Parameter Pop-Up closes.

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

Select a Program to Run

You must select a program before you can run it. To Select a program:

1. In the Program Directory, highlight a program name. Press **Select** (**F6**). The CNC selects the program and the "**SELECTED PROGRAM**" label is displayed at the bottom of the screen with the selected program name.

Running Programs

There are three ways to run a program:

Single-Step Mode	One block at a time (stops on every block).
Motion Mode	One motion at a time (does not stop on non-motion blocks such as **Dim Abs/Inc** blocks).
Automatic Mode	Automatically runs the whole program, pausing only for tool changes.

The Automatic and Single-Step screens are based on the Manual Mode screen. Use the soft key labels to distinguish between modes. The CNC highlights the soft key for the active mode.

NOTE: The CNC will only run the currently selected program. Use **Select** (**F6**) in the Program Directory to select a program.

Running a Program One Step at a Time

The Single-Step screen provides access to the Single-Step Mode (S.Step) and the Motion Mode (Motion) screens. Either mode allows the operator to step through the program and verify the moves before production.

NOTE: ANILAM recommends Motion Mode.

The S.Step screen differs from the Manual screen as follows:

- There are fewer active soft keys.
- The **S.STEP** (**F5**) soft key highlights.
- The **S.STEP** indicator is displayed in the status box (upper right corner of screen).

To run a program in Single-Step Mode:

1. Select the required program and return to the Manual screen.
2. Press **S.STEP** (**F5**). Single-Step Mode activates.
3. Press **START**. The CNC executes a single block or motion.

NOTE: In Auto Mode, press **S.STEP** (**F5**) to activate Single-Step Mode.

Switching Between Motion and Single-Step Mode

To switch the CNC between Single-Step (S.Step) and Motion Modes, press **MOTION** (**F7**). Active soft keys highlight.

- In Single-Step Mode, the CNC holds after each block, even if a block does not include a move command. Press **START** to execute the following block.
- In Motion Mode, the CNC holds after each machine move. Press **START** to execute each machine move.

Holding or Canceling a Single-Step Run

Press **HOLD** to pause program execution. To restart the program, press **START**. To cancel a program that is on hold, press **MANUAL** (**F4**). This cancels active canned cycles and Tool Compensation. All other modal settings remain active.

Single-Step Execution of Selected Program Blocks

To select a starting block with **ARROWS**:

1. Press **S.STEP** (**F5**) to activate Single-Step Mode. **MOTION** (**F7**) is the default.
2. Highlight the desired starting block.
3. Press **START**. The CNC executes the next block or motion.

Switching from Single-Step to Auto Mode

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

1. In Single-Step Mode, press **AUTO** (**F6**). The CNC completes the current move then holds.
2. Press **START**. The CNC restarts and runs the rest of the program in Automatic Mode.

Automatic Program Execution

Auto Mode is the CNC's production mode. Execute all or part of a program in Auto Mode. Activate Auto Mode from the Manual or Single-Step screens.

The Auto screen differs from the Manual screen as follows:

- There are fewer active soft keys.
- The **AUTO** (**F6**) highlights.
- The **AUTO** indicator is displayed in the status box (upper right corner of the screen).

To run a program in Auto Mode:

1. Select the required program and return to the Manual screen.
2. Press **AUTO** (**F6**). Auto Mode activates.
3. Press **START**. The CNC runs the entire program for production. It stops only for tool changes.

Holding or Canceling an Auto Run

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

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

Starting at a Specific Block

CAUTION:	**Choose the specified starting block carefully. Modes and compensations enabled in the program before the specified block may no longer be effective. We recommend starting at a tool mount.**

To select a starting block before you run the program, use **ARROWS** or the Search function. **Search** directs the CNC to search the program for a block number, a block containing a number or a block containing specific text. The CNC highlights and displays the first block found that contains the search criteria information. **Search** only seeks forward in the program.

To select a starting block:

1. Press **AUTO** (**F6**).
2. Highlight the required starting block.

   – or –

   Press **Search** (**F3**). Type the search text. Search all occurrences of the text until you find the required starting block.
3. Press **START**. The CNC automatically runs a program from the selected block.

Clearing a Halted Program

When the CNC encounters a program block that generates an error, it displays a Warning message and stops the program. Press **MANUAL** (**F4**) to reactivate the keypad. Press **CLEAR** to clear the messages. Press **START** to re-start the program.

Program Run Status

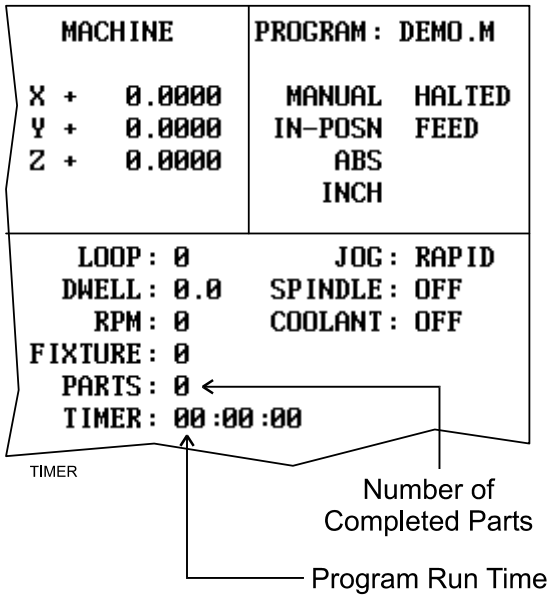


Figure 3-8, Program Timer and Parts Counter

Refer to Figure 3-8, Program Timer and Parts Counter. The CNC keeps track of program run time (**TIMER**) and the number of successfully completed parts (**PARTS**). Run time is displayed in hours, minutes and seconds. These two features are available in Manual, Auto and S. Step Modes.

The timer begins timing the program run when you press **START**. It stops when it encounters an **EndMain** block. Therefore, ensure that an **EndMain** block has been included at the end of the program.

The timer pauses if the CNC holds and during a tool change. The timer stops if the operator switches to Manual Mode. The timer value remains the same until the operator switches to Auto or S.Step Mode again. Then, the timer resets to zero.

The Parts Counter starts at zero and increments by one every time the CNC runs an **EndMain** block. Therefore, ensure that an **EndMain** block has been included at the end of the program. The CNC continues to count parts until you switch to Manual Mode. The counter resets to zero when you switch to Auto or Single-Step Mode.

Quiz 3

True or False

1. After the program is written, you must decide how the work will be held, what tools you will use, and in what order you will use them.
2. You must tell the control how to drill before you tell it where to drill.
3. In the entry menus, you press **F10** to save programming blocks.
4. In the entry menus, you must fill in all “blank” entry fields.
5. To determine whether Tool Compensation is Left or Right, stand behind the tool and look in the direction the tool is traveling.
6. In order to turn off Tool Compensation after the last compensated move has been made, you must program a move in the same block as a Tool #0 (cancel Tool Compensation).
7. You can only run programs one block at a time in Draw Graphics.

Exercise 1

Use the blueprint for the Sample Part (refer to Figure 3-1, Programming Sample Part) to program a Bolt Hole Pattern, Circular Pocket and contours on your own. Use Draw Graphics to check your work. Try to create the program without referring to the text.

Answer Key (True/False Only)

1. F
2. T
3. T
4. F
5. T
6. F
7. F

Section 4 - Machine Setup

In this section, you will:

- Set Absolute Zero.
- Set the Z Home Position.
- Set Tool Length Offsets for Drill and End Mill Tools.
- Set Tool Diameter Offsets.

Set Absolute Zero

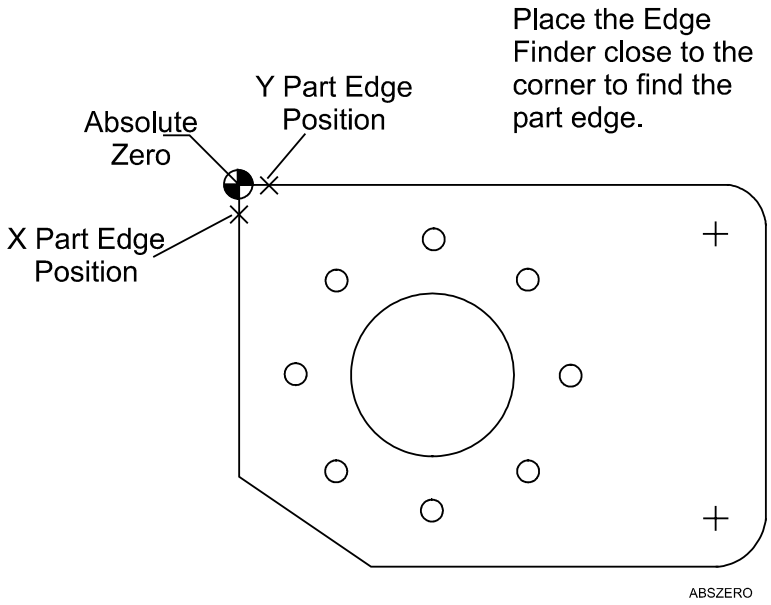


Figure 4-1, Set Absolute Zero

Refer to **Figure 4-1**. All Absolute dimensions are referenced to Absolute Zero (X0, Y0). For the Sample Part, place Absolute Zero in the upper-left corner of the workpiece held in a vise.

1. Place a 0.200” diameter edge finder in the spindle.
2. Set it to the proper height. The tip of the edge finder should clear the top of the workpiece.
3. Hand-tighten the edge finder in the spindle.
4. To ensure that the servos are on, press **SERVO RESET**. The CNC turns on power to the servos.

NOTE:	Press the **FEEDRATE OVERRIDE** switch to change the Jog Feedrate (Rapid, Feed, Jog 100, Jog 10, Jog 1). The Jog Mode is displayed on screen.

5. Find the X Part Edge:
   A. Press the **X-** and **Y- JOG** keys to move the table into position. Move to the X Part Edge Position. The edge finder must clear the left side of the part. Use Rapid Jog Mode.

B. Switch to Feed Jog Mode. Press **Z-** until the edge finder is slightly below the work surface.

C. Jog in X+. As the tool moves closer to the work surface, decrease the Jog resolution to Jog 100 (0.010" per move).

D. Press **SPDL FWD**. The spindle turns on.

E. Switch to Jog 10 (0.001" per move) and move the tool in until the edge finder kicks out of concentricity. Turn off the spindle.

F. The edge of the spindle is now 0.100" to the left of the workpiece. Set X to -0.100". To do this, press **X**, press the **+/-** key so that the sign is negative, and type 0.100. Press **ENTER**. The CNC displays -0.100 in the X Axis Display.

NOTE: Do not press **START**, which commands a move to the entered position.

6. Find the Y Part Edge:

A. Press the **X-** and **Y+ JOG** keys to move the table into position. Move to the Y Part Edge Position. (You may need to raise the edge finder in Z to clear the work holding device.) Use Rapid Jog Mode.

B. Switch to Feed Jog Mode. Press **Z-** until the edge finder is slightly below the work surface.

C. Jog in Y-. As the tool moves closer to the work surface, decrease the Jog resolution to Jog 100 (0.010" per move).

D. Press **SPDL FWD**. The spindle turns on.

E. Switch to Jog 10 (0.001" per move) and move the tool in until the edge finder kicks out of concentricity. Turn off the spindle.

F. The edge of the spindle is 0.100" away from the workpiece. Set Y to 0.100". To do this, press **Y** and type 0.100. Press **ENTER**. The CNC displays 0.100 in the Y Axis Display.

NOTE: Do not press **START**, which commands a move to the entered position.

7. Visually inspect the spindle center position.

A. Switch to Jog Rapid Mode.

B. Jog Z+ to clear the work surface.

C. Return to X0, Y0. (Press **X**. Press **0**. Press **Y**. Press **0**. Press **START**.)

D. Make sure the spindle is centered over the upper-left corner of the part. If not, repeat the procedure to correct any errors until the spindle is centered over the upper-left corner of the part at X0, Y0.

8. Return to Z 0.100. (Press **Z**. Press **.100**. Press **START**.)

Using the DRO Mode

You can use DRO (Digital Readout) Mode to set X0, Y0 and Tool Length Offsets. Use the hand cranks to position the axes manually, instead of turning on the servos. Set the switch on the cabinet to **MANUAL**. The CNC displays the positions of the axes but will not execute programmed commands.

Canceling Tool Length Offsets

Before you type the first Tool Length Offset, make sure there are no Tool Length Offsets active from a previous program. The active tool (**TOOL**) is displayed under the Axis Display on the screen. If the CNC displays **TOOL: 0**, then no Tool Length Offsets are active. If the CNC displays any other Tool #, you will need to cancel the active tool before setting the Tool Length Offsets.

To cancel the active Tool #:

1. Press **5/TOOL**. The Tool Mount Graphic Menu is displayed.
2. Press **0**.
3. Press **Save** (**F10**) to save the block.
4. Press **START** to activate Tool #0.

Setting a New Z Home

Set Z Home (Tool #0, Z0) at the fully retracted quill position, almost on the limit switch.

To set Z0:

1. Make sure Tool #0 is active.
2. Jog the quill up, until it almost reaches the limit switch.
3. Press **Z**. Press **0**. Press **ENTER**. The CNC sets Z0 at the current location.

NOTE: Do not press **START**, which commands a move to Z0.

Setting Tool Length Offsets for Drilling Tools

To find the Tool Length Offset for Tool #1, a 1/4" drill:

1. Make sure Tool #0 is active.
2. Put Tool 1 in the spindle.
3. Press the **JOG** key to switch to Jog Feed Mode.
4. Jog to an XY position over the top of the workpiece.
5. Press **Z-** to jog to a position close to the work surface.
6. Jog to a position very close to the work surface.
7. Place a scrap of paper between the tip of the tool and the work surface.

8. Press the **JOG** key to switch to Jog 100 (0.010" per move). Press **Z-** to move the tool closer to the part.
9. Move the paper back and forth. If it moves freely, press the **JOG** key to switch to Jog 10 (0.001" per move). Press **Z-** until the tip of the tool pinches the paper and it no longer moves.

   The value displayed in the Z Axis Display is the Tool #1 Length Offset. Enter the offset in the Tool Page so that every time Tool #1 is activated, the programmed Z position will be referenced to that point.
10. Enter the Tool Page. Highlight Tool #1 (row 1). Press **Calib** (**F8**). The CNC enters the Z Axis Display value as the Tool #1 Length Offset.

Setting the Tool Length Offset for Milling Tools

> **NOTE:** Normally, in a CNC retrofit, there is a quick change spindle so that the tool goes to the same position for every tool change. In those cases, the correct tool length offset is automatically maintained after you enter the offsets.

With end mills, the Tool Length Offset needs to be more accurate than for a drill. Therefore, the procedures are different.

To set the Tool Length Offset for Tool #2, a 7/16" end mill:

1. Make sure Tool #0 is active.
2. Jog away from the workpiece.
3. Remove Tool #1 from the spindle.
4. Place Tool #2 in the spindle.
5. Touch the tip of the tool to the work surface.
6. Press the **JOG** key to switch to Jog Feed Mode.
7. Jog to an XY position over the top of the workpiece.
8. Press **Z-** to jog to a position close to the work surface.
9. Jog to a position very close to the work surface.
10. Press the **JOG** key to switch to Jog 100 (0.010" per move). Press **Z-** to move the tool closer to the part.
11. Press the **JOG** key to switch to Jog 10 (0.001" per move); or switch to Jog 1 (0.0001" per move). Press **Z**- until the tip of the tool touches the work surface.
12. Turn off the spindle.

    The value displayed in the Z Axis Display is the Tool #2 Length Offset. Enter the offset in the Tool Page so that every time Tool #2 is activated, the programmed Z position will be referenced to the point where the tip of the tool meets the work surface.
13. Enter the Tool Page. Highlight Tool #2 (row 2). Press **Calib** (**F8**). The CNC enters the Z Axis Display value as the Tool #2 Length Offset.

Setting Tool Diameters

Tool #1 has a diameter of 1/4" (0.25"). Tool #2 has a diameter of 7/16" (0.4375").

To enter the tool diameter offset in the Tool Page:

1. In the Tool Page, use the **ARROWS** to move the cursor to the appropriate row (Row 1 for Tool #1 and Row 2 for Tool #2).
2. Move the cursor to the Diameter column.
3. Type the appropriate diameter (0.25 for Tool #1 and 0.4375 for Tool #2).
4. Press **Exit** (**F10**) to exit the Tool Page and save the diameters.
5. Press **Z**. Press **0**. Press **START**. The CNC returns to Z0.

Machining the Part

To machine the part:

1. Select the required program.
2. In the Manual screen, press **AUTO** (**F6**) to activate the Auto Mode.
3. Place Tool #1 in the spindle.
4. Press **START**. The CNC runs the program.

Quiz 4

True or False:

1. Before you machine the workpiece, you must select (load) the required program in the Edit Mode.
2. Absolute Zero is always used as the tool change position.
3. In DRO Mode, you must use the hand cranks to position the table.
4. Tool #0, Z0 is also known as the Z Home position.
5. Before you enter the first Tool Length Offset, Tool #0 must be the active tool.
6. The **CLEAR** key cancels all active Tool Length Offsets.
7. The CNC supports 16 tool numbers.

Exercise 1:

Use an edge finder to set Absolute Zero for X and Y.

Exercise 2:

Set a new Z Home position. Set the Tool Length Offsets for Tools #1 and #2.

Exercise 3:

Machine the workpiece programmed in "Section 3."

Answer Key (True/False Only)

1. F
2. F
3. T
4. T
5. T
6. F
7. F

Section 5 - Calculators

The CNC features a powerful calculator package that contains three separate calculators:

- Math Calculator
- Right Triangle Calculator
- Geometry Calculator

The programmer can recall calculator solutions directly into the labeled fields of a graphic menu. Each of the three calculators has separate memory space to store and recall solutions.

Math Calculator

The Math Calculator has the same features commonly found in a scientific calculator, including: basic math, trigonometry, unit conversion, logs, exponential operations, angle/radian conversions, and inverse calculation.

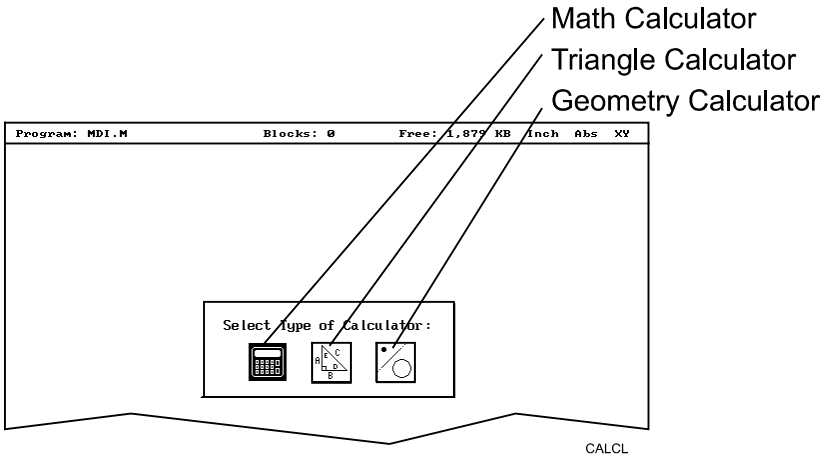


Figure 5-1, Calculator Selection Menu

Activating the Math Calculator

To activate the Math Calculator:

1. Refer to **Figure 5-1**. In Edit or MDI Mode, press **Calc** (**F7**). The Calculator Selection Menu is displayed on the screen.
2. Highlight the **Math Calculator** template and press **ENTER**. The Math Calculator activates.

Math Calculator Basics

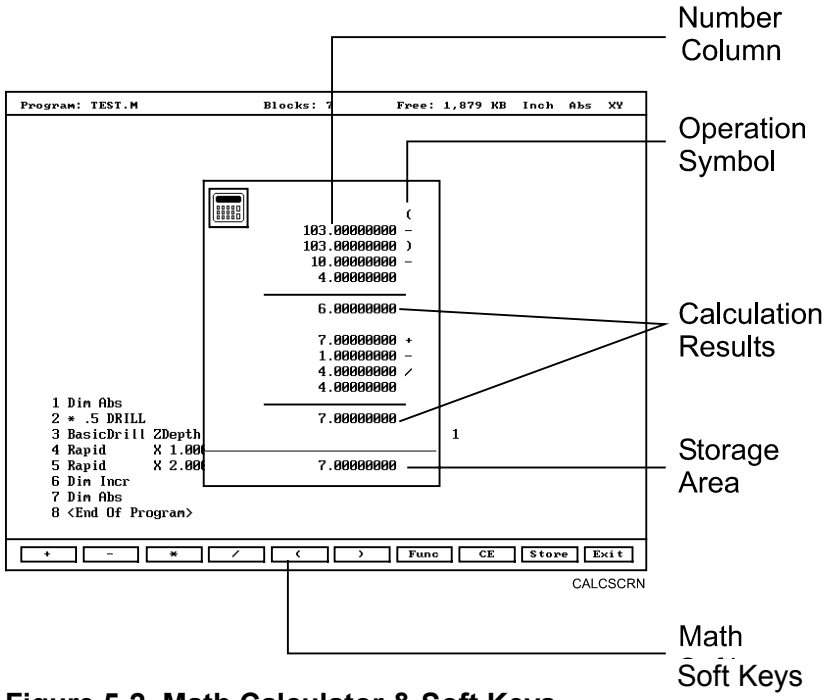


Figure 5-2, Math Calculator & Soft Keys

Refer to **Figure 5-2**. The Math Calculator is displayed as a pop-up in the center of the screen.

Numbers appear in the storage area, as entered. Select math operations from the soft keys.

Refer to Table 5-1, Math Operation Soft Keys. Type the first number of the calculation, then press an operation soft key. The CNC places the number in the column. Now, select the operation to be performed with the next number. Type the second number, and so forth.

After you type the last number of a calculation, press **ENTER**. The CNC places the final number in the column and calculates the answer. The result is displayed in the storage area. Press **Store** (**F9**) to copy the result to the calculator's memory.

Press **CE** (**F8**) to clear previous calculations.

When the operator types illogical combinations of operations and numbers, the CNC displays an error message or a row of asterisks (*******).

Table 5-1, Math Operation Soft Keys

Operation	Soft Key Label	Soft Key Number
Add	**+**	(**F1**)
Subtract	**-**	(**F2**)
Multiply	*****	(**F3**)
Divide	**/**	(**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 appropriate keyboard keys.

Operations Involving Two Numbers

To add, subtract, multiply or divide two numbers:

1. With the Math Calculator active, type the first number, followed by the appropriate function hot key: **+** (**F1**), **-** (**F2**), ***** (**F3**), or **/** (**F4**). The CNC adds the number and operation symbol to the column.
2. Type the second number and press **ENTER**. The CNC adds the second number to the column and displays the solution in the storage area.

Using Parentheses

Use parentheses to indicate arithmetical operations in a non-standard order. (Example: finding a sum before using it as a multiplication factor.) The CNC performs operations within parentheses first.

When the CNC encounters parentheses, it solves the operation within the parentheses immediately, and substitutes that value for the parenthetical expression in the column.

Use parentheses in pairs, if at all. Make sure the parenthetical expression contains a left parenthesis "(" at the beginning and a right parenthesis ")" at the end. Otherwise, the CNC cannot calculate the result.

The CNC performs operations within parentheses top to bottom, as they appear in the column, with innermost expressions solved first.

For example, the following expression:

(7 + 4 + ((6 * 9) -1)) / 8

Generates a result of 8.

Order of calculation:

6 * 9 = 54

54 -1 = 53

53 + 7 = 60

60 + 4 = 64

64 / 8 = 8

Using Additional Functions

Refer to **Table 5-2**. The **Func** (**F7**) key activates a pop-up menu that provides access to additional math functions. These functions perform their listed operations on a single number.

To use an additional function:

1. With the Math Calculator active, type a number and press **Func** (**F7**). The Function Pop-Up Menu is displayed to the right of the calculator.
2. Highlight a function and press **ENTER**. The result is displayed in the storage area.

Table 5-2, 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 **Store** (**F9**) to copy the number from the storage area to the calculator's memory. The Math Calculator 's memory holds 64 numbers. Numbers stored in memory can be recalled directly to a program.

NOTE: If accessed from a graphic menu, the Math Calculator will auto paste values into an entry field.

Right Triangle Calculator

The Right Triangle Calculator solves the unknown angles and sides of a right triangle, given any two sides, any two angles or an angle and a side. Store any or all of the values in the calculator's memory.

Activating the Triangle Calculator

1. Refer to Figure 5-1, Calculator Selection Menu. In Edit or MDI Mode, press **Calc** (**F7**). The Calculator selection menu is displayed.
2. Highlight the Triangle Calculator template and press **ENTER**. The Triangle Calculator activates.

Using the Triangle Calculator

Refer to Figure 5-3, Right Triangle Calculator Screen. The Right Triangle Calculator's pop-up screen contains three main areas:

- Entry/solution area
- Diagram area
- Message area

The labeled entry fields in the entry/solution area correspond to the sides and angles shown in the diagram.

To use the Right Triangle Calculator, type the known values: any two angles, any two sides or one angle and one side.

When you type two known elements and press **Find** (**F7**), the CNC calculates and displays values for all remaining sides and angles. The CNC places an asterisk after solved element values. A scaled drawing of the solved triangle is also displayed in the diagram area.

To clear a single incorrect value, highlight that value and press **CLEAR**. To clear all displayed values, press **ClrAll** (**F8**).

An illogical entry generates an error message.

Recall Right Triangle Calculator solutions directly to a program (for positive incremental moves only). To adjust these solutions to produce Absolute coordinates, recall the values to the Math Calculator and add an offset value.

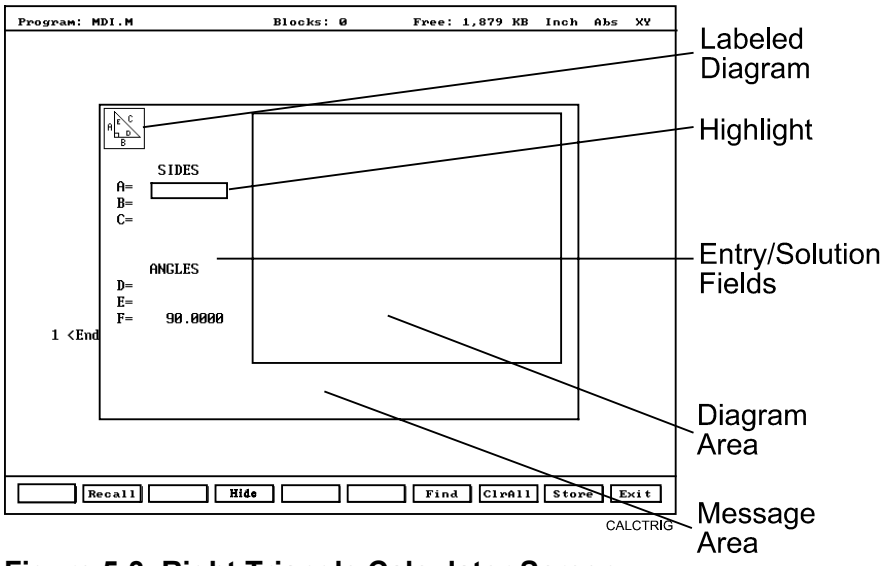


Figure 5-3, Right Triangle Calculator Screen

To solve for the sides and angles of a right triangle:

1. With the Right Triangle Calculator active, highlight the field for one side or angle and type the value.
2. Highlight another side or angle and type the value.
3. Press **Find** (**F7**). A scaled drawing of the triangle is displayed in the solution box. Calculated sides and angles are displayed in all fields.

Storing Right Triangle Calculator Results

The CNC will save any side or angle value.

To store a value:

1. Solve the required triangle.
2. Highlight the value to be stored.
3. Press **Store** (**F9**). The selected value is stored in memory.

Hiding the Right Triangle Calculator Screen

You can hide the Right Triangle Calculator to view the program, without exiting the calculator. To hide the calculator, press **Hide** (**F4**). To show the calculator again, press **Find** (**F7**) or any other key on the console or keypad.

Geometry Calculator

The CNC uses Cartesian coordinates (X, Y, Z-axis values) to define most positions. However, the operator must sometimes determine position coordinates based on the known construction of other elements on the print, including lines, circles and angles.

The Geometry Calculator provides an assortment of line, circle, angle and point templates. Use these templates to sketch a geometry construction that identifies the unknown position. The Geometry Calculator inserts a point at the required position. The calculator automatically solves the coordinates of all points. Recall stored coordinates as necessary in a program.

Activating the Geometry Calculator

1. Refer to Figure 5-1, Calculator Selection Menu. In Edit or MDI Mode, press **Calc** (**F7**). Calculator Selection Menu is displayed on the screen.
2. Highlight the **Geometry Calculator** template and press **ENTER**. The Geometry Calculator activates.

Geometry Calculator Screen

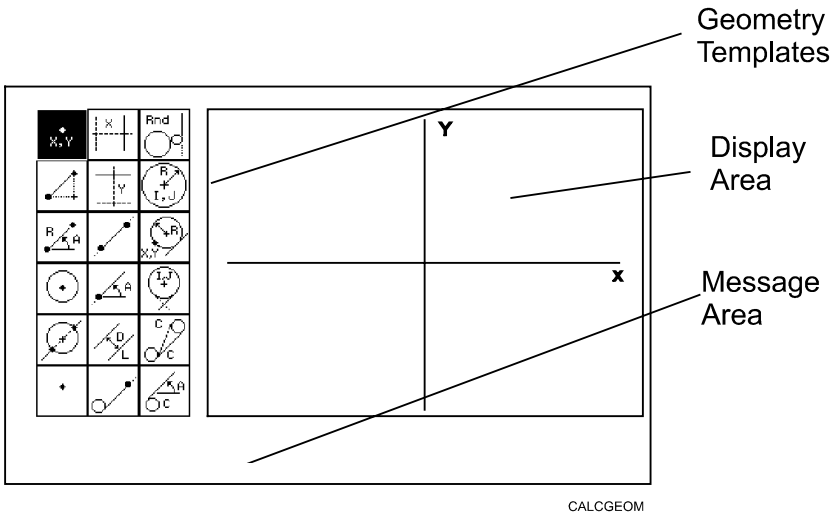


Figure 5-4, Geometry Calculator

Refer to **Figure 5-4**. The Geometry Calculator is a pop-up screen with a display area, a message area and eighteen geometry templates. The geometry construction is displayed in the display area.

Use the **DISPLAY** (**F5**) soft key selections to alter the view of the display area. The **DISPLAY** (**F5**) options work the same as they do in the Draw Mode display.

The calculator prompts for required values and selections. Pay close attention to the prompts.

Using the Geometry Calculator

Use the **ARROWS** to select a template. Press **ENTER** to activate the selected template. Points, lines and circles are the basic elements of all sketches. Use the eighteen geometry templates to define these elements. Each geometry tool defines an element differently. You will probably need more than one tool to define the required geometry construction. Templates that require pre-existing points will prompt you to activate one of the point identification templates.

The X- and Y-axis lines appear as solid lines; constructed lines and circles appear as dotted lines; points are marked with an "**x**" (lowercase x).

NOTE: Absolute Zero is at the intersection of the X and Y-axis lines. Therefore, construct geometry elements based on where you will set X0, Y0 on the part.

The CNC assigns a number to each element in a sketch.

Every element in the sketch also is displayed on the Geometry List. The element numbers on the Geometry List correspond to the element numbers in the sketch.

The Geometry List includes the following details about each element:

Circles	Absolute position of center and radius
Lines	Orientation (angle) and Absolute position at which it crosses X and/or Y axis
Points	Absolute positions

The calculator automatically numbers each element and stores the following details in memory:

Points	Absolute position
Circles	Absolute position of center

NOTE: The Geometry Calculator does not automatically save the coordinates of an intersection between two elements. Insert a point at the intersection to save its coordinates.

When you delete an element from the sketch, the CNC deletes all stored information regarding that element.

There are three columns of geometry templates shown on the screen. Point templates are in the left-hand column. Line templates are in the center column. Circle templates are in the right-hand column.

- For point templates (left column), refer to Table 5-3, Point Templates.
- For line templates (center column), refer to Table 5-4, Line Templates.
- For circle templates (right column), refer to Table 5-5, Circle Templates.

Review the tables to see the requirements for each template. Some templates require some feature of an existing element as a reference. Experiment to understand how each template operates.

NOTE: After a series of deletions and additions, the display could appear incomplete. Press **Display** (**F5**) and select **Redraw** to refresh the screen.

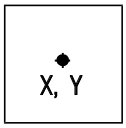
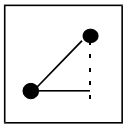
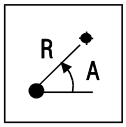
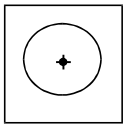
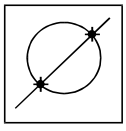
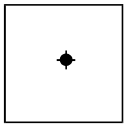
Point Templates

Some point templates insert points at positions defined by the operator. Some use other elements as references.

Many line and circle templates display a "**Select point definition . . .**," message when activated. This indicates that the selected template requires the operator to select (or create) a reference point.

Templates that insert points at circle centers and element intersections will prompt the operator for the required element number(s). The CNC displays all possible intersections. The user selects one.

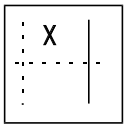
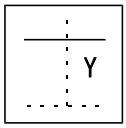
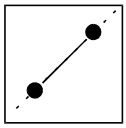
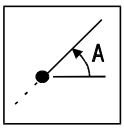
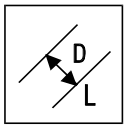
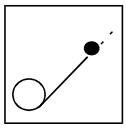
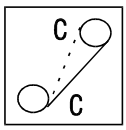
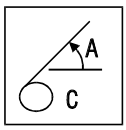
Table 5-3, Point Templates

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.	– Must know Incremental X and Y distances from existing point.
	Defines a point at radius and angle from existing point.	– Must know number of degrees and direction (±) from the 3 o'clock position.
	Defines a point at the center of an existing circle.	– Circle must already be an element of the sketch.
	Defines the point of intersection between two existing elements. Prompts user to select an intersection when more than one intersection exists.	– Sketch must contain two intersecting (or tangential) elements.
	Identifies an existing point. Usually used when the construction of a new element requires a reference point.	– Point must already be an element of the sketch.

Line Templates

Line templates use other elements or axis positions as references. Templates that draw lines tangent to circles display all possible tangent lines and prompt the operator to select one.

Table 5-4, Line Templates

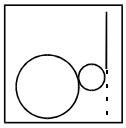
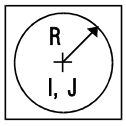
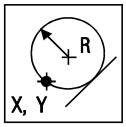
Template	Purpose	Requirements
	Constructs a line parallel to Y-axis, at an X location.	– Must know Absolute X-axis position of the line.
	Constructs a line parallel to X-axis, at a Y location.	– Must know Absolute Y-axis position of the line.
	Constructs a line between any two points. Prompts user to select a point tool to define each endpoint.	– Use any point tool to locate the two endpoints.
	Constructs a line through a point, rotated specified number of degrees from the 3 o'clock position. Prompts user to select any point tool to define point of rotation.	– Must know number of degrees and direction (±) from the 3 o'clock position. – Use any point tool to locate point of rotation.
	Constructs a line parallel to an existing line at specified (positive or negative) distance.	– Existing line must already be an element of the sketch.
	Constructs a line through a selected point and tangent to an existing circle. Prompts user to select any point 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 the 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 the 3 o'clock position.

Circle Templates

Circle templates use other elements as positioning references.

Templates that draw circles tangent to other circles, lines or points display all possible tangent circles and prompt the operator to select one.

Table 5-5, Circle Templates

Template	Purpose	Requirements
	Constructs a circle of a specified radius tangent to any two existing lines and/or circles. All possible tangential circles are displayed. The user selects the required tangent.	– Sketch must contain at least two lines and/or circles.
	Constructs a circle of a 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 a specified radius tangent to an existing line and through an existing point. Prompts user to select a point tool to define point. Finds center.	– 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. Finds radius.	– 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**). A pop-up menu is displayed.
2. Highlight **Delete Item** and press **ENTER**. The screen prompts for the element number being deleted.
3. Type the element number and press **ENTER** to delete the item.

Deleting All Elements

To clear all elements from the display area:

1. With the Geometry Calculator active, press **GEOMETR** (**F7**). A pop-up menu is displayed.
2. Highlight **Delete All** and press **ENTER**. The screen prompts the user to confirm the deletion.
3. Press **Yes** (**F1**) to clear the screen. Press **No** (**F2**) to cancel the command.

Listing All Geometry Elements

The CNC stores information on all points, circles and lines created in the Geometry Calculator in the **Geometry List**.

- For a point, the CNC lists the X,Y coordinates.
- For a line, the CNC lists where the line crosses the X and/or Y-axes; and, sometimes, the angle in reference to the 3 o'clock, 0-degree position.
- For a circle, the CNC lists the circle center and radius.

To display the Geometry List:

1. With the Geometry Calculator open, press **GEOMETR** (**F7**). A pop-up menu is displayed.
2. Highlight **Geometry List** and press **ENTER**. The Geometry List is displayed.

Calculating the Distance between Two Elements

Use **Calc. Distance** to calculate the shortest distance between any two elements drawn with the Geometry Calculator.

NOTE: For circles, the CNC calculates the shortest distance from the circle center to the other element.

To calculate the distance between two elements with the Geometry Calculator:

1. With the Geometry Calculator open, press **GEOMETR** (**F7**). A pop-up menu is displayed.
2. Highlight **Calc. Distance** and press **ENTER**. The CNC prompts, "**Enter number of first element:**"
3. Type the first element number. The CNC prompts, "**Enter number of second element:**"
4. Type the second element number. The CNC calculates and displays the shortest distance between the two elements.

Last Position Recall

Whenever the calculator prompts for an X position, Y position or circle, the operator can recall the last X position, Y position or circle used by the calculator.

To recall a position:

1. When the calculator prompts for an X position, Y position or circle number, press the up and down **ARROWS** to recall the last position or circle number used.

Recalling Values to a Program

Refer to **Figure 5-5**. The Program Editor displays **Recall** (**F2**) when a graphic menu activates. You can now recall calculator solutions stored in memory directly to the entry fields of a graphic menu.

NOTE: The operator can recall saved values only from the same calculator in which they were saved. Use the menu to select the type of recall.

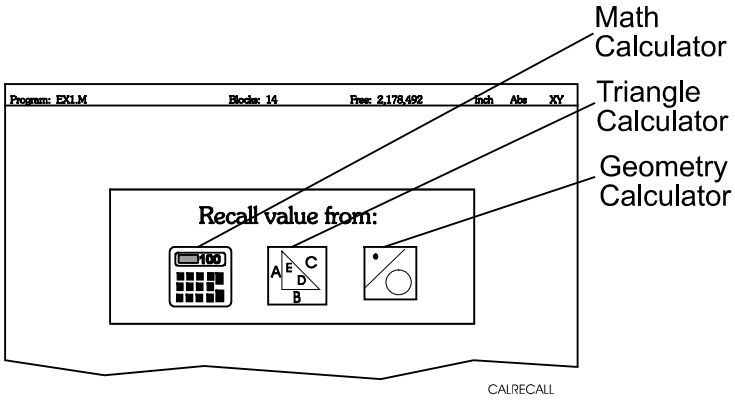


Figure 5-5, Recall Selection Pop-Up

Recalling Values from the Math Calculator

To recall a value from the Math Calculator:

1. From the graphic menu for the block being edited, highlight the field and press **Recall** (**F2**). The Calculator Recall Selection Menu is displayed. Refer to **Figure 5-5**.
2. Highlight the **Math Calculator** template and press **ENTER**. **Select value:** pop-up is displayed, with the most recently saved value displayed at the top of the pop-up menu. Refer to **Figure 5-6**.
3. Highlight the required value and press **ENTER** to copy the stored value to the graphic menu.

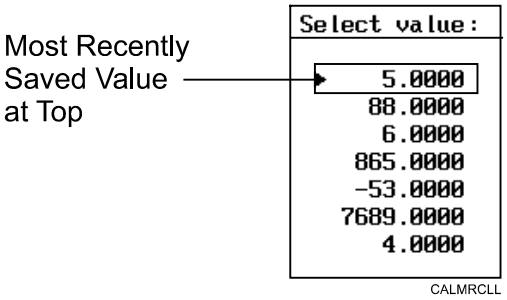


Figure 5-6, Math Calculator Select Value Pop-Up

Recalling Values from the Right Triangle Calculator

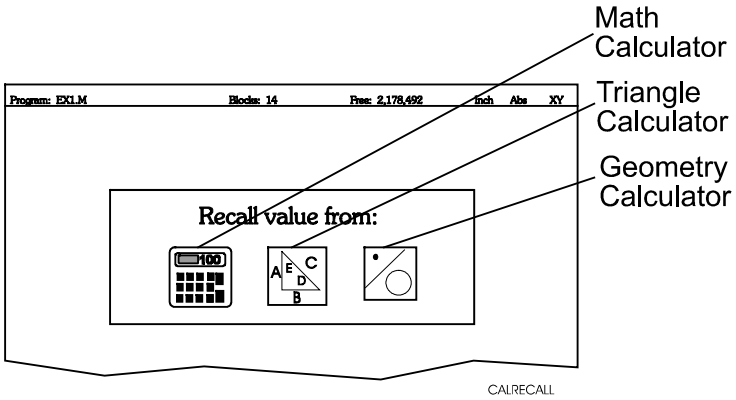


Figure 5-7, Recall Selection Pop-Up

To recall values from the Right Triangle Calculator:

1. From the graphic menu for the block being edited, highlight the field to receive the recalled value.
2. Press **Recall** (**F2**). The **Select value:** menu is displayed. Refer to Figure 5-5, Recall Selection Pop-Up.
3. Highlight the **Triangle Calculator** template and press **ENTER**. The Triangle Calculator Memory Selection Pop-Up is displayed. Refer to **Figure 5-8**.
4. Highlight the required value and press **ENTER** to copy the stored value to the graphic menu.

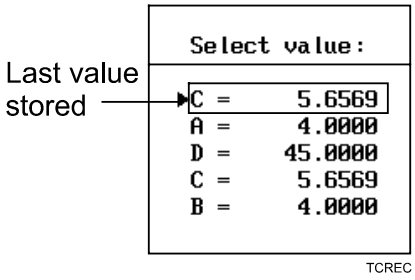


Figure 5-8, Triangle Calculator Recall Pop-Up

Recalling Values from the Geometry Calculator

You can recall Geometry Calculator values from the calculator's **Select point:** pop-up menu. This menu is displayed next to a copy of the sketch that generated the points. The recall listing corresponds to the numbered elements on the sketch.

Positions on the recall list followed by **[C]** are circle centers.

Positions on the recall list followed by **[P]** are points.

Either one or both of the position coordinates can be recalled.

To recall a value from the Geometry Calculator:

1. From the graphic menu for the block being edited, highlight the field to receive the recalled value.
2. Press **Recall** (**F2**). The Calculator Recall Selection Menu is displayed. Refer to Figure 5-5, Recall Selection Pop-Up.
3. Highlight the **Geometry Calculator** template and press **ENTER**. The CNC opens the **Select point:** pop-up menu and displays the saved sketch. Refer to **Figure 5-9**.
4. Highlight the required values and press **ENTER**. The **Select term:** pop-up menu prompts for **Both X and Y** values, **X only**, or **Y only**.
5. Select the required terms and press **ENTER**. CNC copies the selected values to the graphic menu.

[C] Indicates Circle Center Coordinates
[P] Indicates Point Coordinates

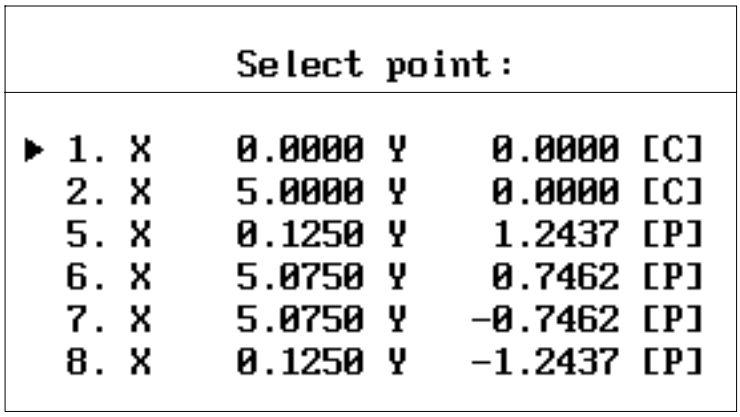


Figure 5-9, Geometry Calculator Select Point Pop-Up Menu

Recalling Values from One Calculator into Another

The Triangle Calculator can recall values from other calculators. Press **Recall** (**F2**) to recall values in the Geometry Calculator or Math Calculator.

Geometry Calculator Example

The example uses these Geometry Templates:

- Center and Radius Template
- XY Point Definition Template
- Line Tangent to Two Circles Template
- Point Definition: Intersection of Two Elements Template

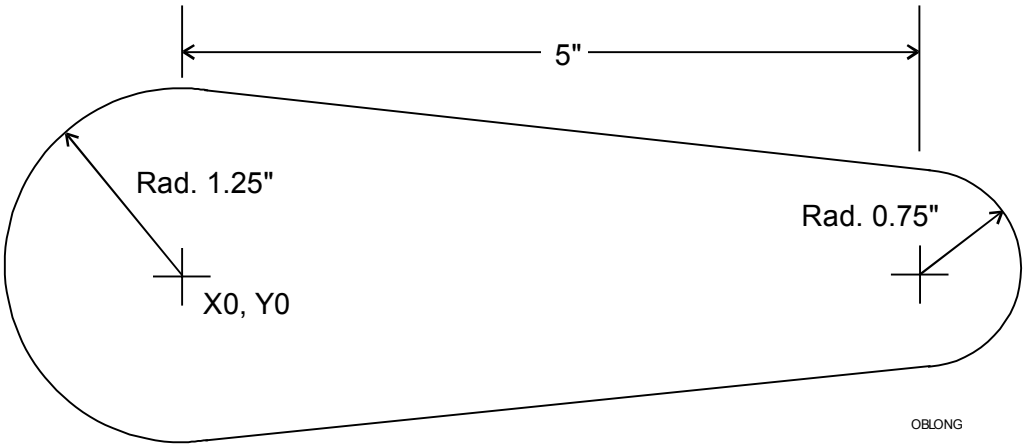


Figure 5-10, Oblong Geometry Calculator Example

Refer to **Figure 5-10**. Use the Geometry Calculator to locate programming points for the oblong shape in the figure. Follow the directions below.

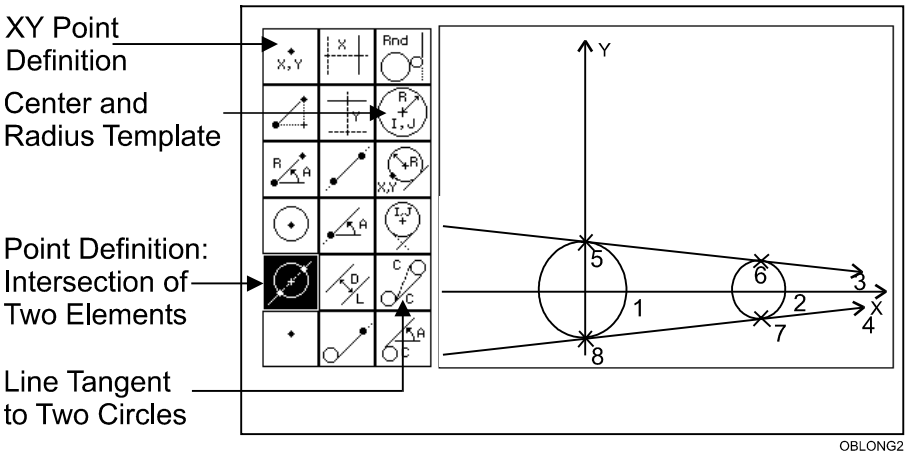
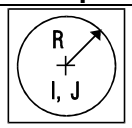
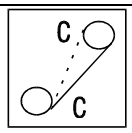
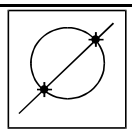


Figure 5-11, Completed Geometry Calculator Display

Refer to **Table 5-6** for a list of the elements you will create.

Table 5-6, Geometry Calculator Example-Elements

Element	Template	Template Name
Circle 1		Center and Radius Template
Circle 2		Center and Radius Template
Line 3		Line Tangent to Two Circles
Line 4		Line Tangent to Two Circles
Point 5		Point Definition: Intersection of Two Elements
Point 6		Point Definition: Intersection of Two Elements
Point 7		Point Definition: Intersection of Two Elements
Point 8		Point Definition: Intersection of Two Elements

I. Circle 1

To create Circle 1:

1. Refer to Figure 5-11, Completed Geometry Calculator Display. Activate the Geometry Calculator.
2. Highlight the **Center and Radius Template**. Press **ENTER**. The CNC prompts for the **R value** (radius).
3. Type 1.25. Press **ENTER**. The CNC prompts for a **center definition**.
4. The **XY Point Template** is highlighted. Press **ENTER**. The CNC prompts for the X value.
5. Type 0. Press **ENTER**. The CNC prompts for a **Y value**.
6. Type 0. Press **ENTER**. The CNC displays Circle 1.
7. Press **Display** (**F5**). **Fit** is already highlighted. Press **ENTER**. The CNC configures the display to show all constructed elements in the viewing area.

II. Circle 2

To create Circle 2:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the **Center and Radius Template**. Press **ENTER**. The CNC prompts for the **R value** (radius).
3. Type 0.75. Press **ENTER**. The CNC prompts for a **center definition**.
4. Select the **XY Point Template**. Press **ENTER**. The CNC prompts for the X value.
5. Type **5**. Press **ENTER**. The CNC prompts for a **Y value**.
6. Type **0**. Press **ENTER**. The CNC displays Circle 2.
7. Press **Display** (**F5**). **Fit** is already highlighted. Press **ENTER**. The CNC configures the display to show all constructed elements in the viewing area.

III. Line 3

To create Line 3:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the **Line Tangent to Two Circles Template**. Press **ENTER**. The CNC prompts for a **circle number**.
3. Type 1. Press **ENTER**. The CNC prompts for a **circle number**.
4. Type 2. Press **ENTER**. The CNC displays all possible lines tangent to both circles (labeled 1 to 4) and prompts you to **Select 1-4**.
5. Type **1**. Press **ENTER**. The CNC displays the top tangent line.

IV. Line 4

To create Line 4:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the **Line Tangent to Two Circles Template**. Press **ENTER**. The CNC prompts for a **circle number**.
3. Type **1**. Press **ENTER**. The CNC prompts for a **circle number**.
4. Type **2**. Press **ENTER**. The CNC displays all possible lines tangent to both circles (labeled 1 to 4) and prompts you to **Select 1-4**.
5. Type **4**. Press **ENTER**. The CNC displays the bottom tangent line.
6. Press **Display** (**F5**). Highlight **Redraw**. Press **ENTER**. The CNC redraws the display to show all constructed elements.

V. Point 5

To create Point 5:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the **Point Definition: Intersection of Two Elements Template**. Press **ENTER**. The CNC prompts for the **first element**.
3. Type **1**. Press **ENTER**. The CNC prompts for the **second element**.
4. Type **3**. Press **ENTER**. The CNC displays Point 5 tangent to the entered elements.

VI. Point 6

To create Point 6:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the **Point Definition: Intersection of Two Elements Template**. Press **ENTER**. The CNC prompts for the **first element**.
3. Type **3**. Press **ENTER**. The CNC prompts for the **second element**.
4. Type **2**. Press **ENTER**. The CNC displays Point 6 tangent to the entered elements.

VII. Point 7

To create Point 7:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the Point Definition: Intersection of Two Elements Template. Press **ENTER**. The CNC prompts for the **first element**.
3. Type **2**. Press **ENTER**. The CNC prompts for the **second element**.
4. Type **4**. Press **ENTER**. The CNC displays Point 7 tangent to the entered elements.

VIII. Point 8

To create Point 8:

1. Refer to Figure 5-11, Completed Geometry Calculator Display.
2. Highlight the **Point Definition: Intersection of Two Elements Template**. Press **ENTER**. The CNC prompts for the **first element**.
3. Type **1**. Press **ENTER**. The CNC prompts for the **second element**.
4. Type **4**. Press **ENTER**. The CNC displays Point 8 tangent to the entered elements.

NOTE: The Geometry Calculator stores all values. These values can be recalled to a program. Keep in mind that these Absolute values might need to be adjusted according to how you use them in the program.

Sample Program with Recalled Geometry Points

Refer to **Table 5-7**. In the following procedure, you will recall values from the Geometry Calculator based on the preceding Geometry Calculator Example. Follow the step-by-step instructions to program each block. Refer Figure 5-11, Completed Geometry Calculator Display to for Points referenced in the procedure.

Table 5-7, Program Using Recalled Geometry Calculator Values

Blk. #	Format
1	Dim Abs
2	Rapid Z 0.0000 Tool# 0
3	Tool# 1
4	Rapid X 0.1250 Y 1.2437
5	Rapid Z 0.1000
6	Line Z -0.1000
7	Line X 5.0750 Y 0.7462
8	Arc Cw X 5.0750 Y -0.7462 Radius 0.7500
9	Line X 0.1250 Y -1.2437
10	Arc Cw X 0.1250 Y 1.2437 XCenter 0.0000 YCenter 0.0000
11	Rapid Z 0.0000 Tool# 0
12	EndMain

Block 1: Set Absolute Mode

Format: Dim Abs

To set the CNC to Absolute Mode:

1. Press **ABS/INCR**.
2. Press **Save** (**F10**) to save the block.

Block 2: Cancel Tool Length Compensation

Format: Rapid Z 0.0000 Tool# 0

To cancel tool length compensation:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Type the following values:

Z	0.0000
Tool #	0

The CNC cancels length compensation and moves the tool to Z0.

Block 3: Activate Tool #1

Format: Tool#1

To activate Tool #1:

1. Press **5/TOOL**. The Tool Mount Graphic Menu activates.

Type the following values:

Tool#	1

The CNC activates Tool #1.

Block 4: Move to XY Starting Position with Recalled Values

Format: Rapid X 0.1250 Y 1.2437

Move to the starting position in X and Y. These values will be recalled from the Geometry Calculator (Point 5).

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Press **Recall** (**F2**). Press **ENTER**. The Recall Selection Menu activates.
3. Refer to Figure 5-7, Recall Selection Pop-Up. Highlight the **GEOMETRY CALCULATOR TEMPLATE**. Press **ENTER**. The Geometry Calculator Recall Menu activates.
4. Highlight the stored values for Point 5 (element 5). Press **ENTER**. The Select Term Pop-up activates.
5. Select **Both X and Y**. (Both values are required.) Press **ENTER**. The CNC transfers the X (0.1250) and Y (1.2437) coordinate values to the Rapid Graphic Menu.
6. Press **Save** (**F10**) to save the block.

Block 5: Rapid to Z Start Height

Format: Rapid Z 0.1000

Rapid to Z Start Height, as follows:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Fill in the following values:
   Z 0.1000
3. Press **Save** (**F10**) to save the block.

Block 6: Feed to Z Depth of Cut

Format: Line Z- 0.1000

Feed to Z depth of cut, as follows:

1. Press **2/LINE**. The Line Graphic Menu activates.
2. Fill in the following values:
   Z -0.1000
3. Press **Save** (**F10**) to save the block.

Block 7: Linear Feed with Recalled Values

Format: Line X5.0750 Y0.7462

Move to the required XY endpoint (Point 5 to Point 6), as follows:

1. Press **1/LINE**. The Rapid Graphic Menu activates.
2. Press **Recall** (**F2**). Press **ENTER**. The Recall Selection Menu activates.

3. Refer to Figure 5-7, Recall Selection Pop-Up. Highlight the **GEOMETRY CALCULATOR TEMPLATE**. Press **ENTER**. The Geometry Calculator Recall Menu activates.
4. Highlight the stored values for Point 6 (element 6). Press **ENTER**. The Select Term Pop-up activates.
5. Select **Both X and Y**. (Both values are required.) Press **ENTER**. The CNC transfers the X (5.0750) and Y (0.7462) coordinate values to the Rapid Graphic Menu.
6. Press **Save** (**F10**) to save the block.

Block 8: Cw Arc Move with Recalled Values

Format: Arc Cw X5.0750 Y-0.7462 Radius 0.7500

Program a Cw arc move with recalled values for X and Y. You will type the radius (.75). Move to the required XY endpoint (Point 6 to Point 7), as follows:

1. Press **3/ARC**. The Arc Graphic Menu activates. The Cw direction is already selected.
2. Press **Recall** (**F2**). Press **ENTER**. The Recall Selection Menu activates.
3. Refer to Figure 5-7, Recall Selection Pop-Up. Highlight the **GEOMETRY CALCULATOR TEMPLATE**. Press **ENTER**. The Geometry Calculator Recall Menu activates.
4. Highlight the stored values for Point 7 (element 7). Press **ENTER**. The Select Term Pop-up activates.
5. Select **Both X and Y**. (Both values are required.) Press **ENTER**. The CNC transfers the X (5.0750) and Y (-0.7462) coordinate values to the Rapid Graphic Menu.
6. Refer to Figure 5-11, Completed Geometry Calculator Display. The required radius is the radius of Circle 1 in the figure, or 0.75. In the Arc Graphic Menu, highlight **Radius**. Type 0.75. Press **ENTER**.
7. Press **Save** (**F10**) to save the block.

Block 9: Linear Feed with Recalled Values

Format: Line X 0.1250 Y -1.2437

Move to the required XY endpoint (Point 7 to Point 8), as follows:

1. Press **1/LINE**. The Rapid Graphic Menu activates.
2. Press **Recall** (**F2**). Press **ENTER**. The Recall Selection Menu activates.
3. Refer to Figure 5-7, Recall Selection Pop-Up. Highlight the **GEOMETRY CALCULATOR TEMPLATE**. Press **ENTER**. The Geometry Calculator Recall Menu activates.
4. Highlight the stored values for Point 8 (element 8). Press **ENTER**. The Select Term Pop-up activates.

5. Select **Both X and Y**. (Both values are required.) Press **ENTER**. The CNC transfers the X (0.1250) and Y (-1.2437) coordinate values to the Rapid Graphic Menu.
6. Press **Save** (**F10**) to save the block.

Block 10: Cw Arc Move with Recalled Values

Format: Arc Cw X 0.1250 Y 1.2437 XCenter 0.0000 YCenter 0.0000

Move from Point 8 to Point 5 in a Cw arc move. This arc is defined by the XY arc center and the XY endpoint. Recall values for X and Y (Point 5). The arc center is X0, Y0. Therefore, use the default for **XCenter** (0.0000) and **YCenter** (0.0000).

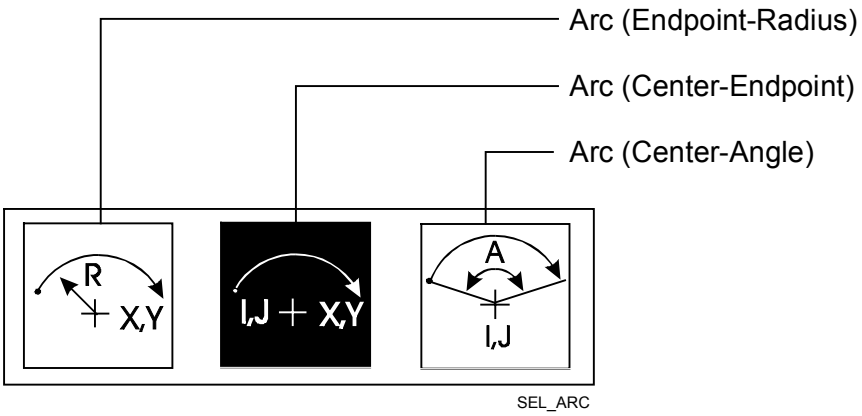


Figure 5-12, Arc Definition Selection Menu

1. Press **3/ARC**. The soft keys change.
2. Press **More…**(**F4**). The Arc Definition Selection Menu activates.
3. Refer to **Figure 5-12**. Select the **ARC (CENTER-ENDPOINT)** template. Press **ENTER**. The Arc (Center-Endpoint) Graphic Menu activates.

   The required **Cw** direction is already selected. The following steps will program the remaining arc parameters.
4. Highlight **X**. Press **Recall** (**F2**). The Recall Selection Menu activates.
5. Refer to Figure 5-7, Recall Selection Pop-Up. Highlight the **GEOMETRY CALCULATOR TEMPLATE**. Press **ENTER**. The Geometry Calculator Recall Menu activates.
6. Highlight the stored values for Point 5 (element 5). Press **ENTER**. The Select Term Pop-Up activates.
7. Select **Both X and Y**. Press **ENTER**. The CNC transfers the X (0.1250) and Y (1.2437) coordinate values to the Rapid Graphic Menu.

   The **XCenter** and **YCenter** default to 0.0000. This places the arc center at X0, Y0.
8. Press **Save** (**F10**) to save the block.

Block 11: Cancel Tool Length Compensation

Format: Rapid Z 0.0000 Tool# 0

To cancel tool length compensation:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.
2. Type the following values:
   Z 0.0000
   Tool # 0
3. The CNC cancels length compensation and moves the tool to Z0.

Block 12: Program EndMain

Format: EndMain

To add an EndMain block:

1. Press **Sub** (**F8**). The soft keys change.
2. Press **EndMain** (**F4**). The CNC adds an EndMain block to the program.

Refer to Table 5-7, Program Using Recalled Geometry Calculator Values for the complete program listing.

Quiz 5

Exercise 1:

Use the Math and Trigonometry Calculators to find the cosine of 35 degrees.

Exercise 2:

Refer to **Figure 5-13**. Use the Right Triangle Calculator to find all sides and angles of the following triangle. Where A = 0.2500 and D = 25.0000.

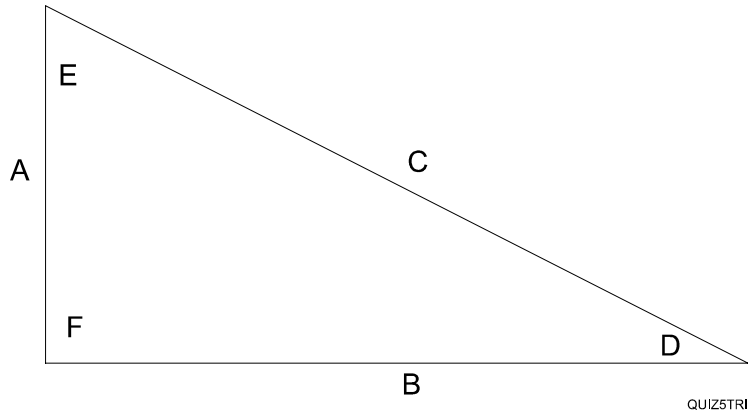


Figure 5-13, Exercise 2 Triangle

Exercise 3:

In this exercise, you will repeat the example provided in the text. Do not refer to the step-by-step instructions in the workbook. Use the Geometry Calculator to create the oblong slot. Store the four tangential points (where the lines and circles meet). In Edit Mode, recall these values to an open program.

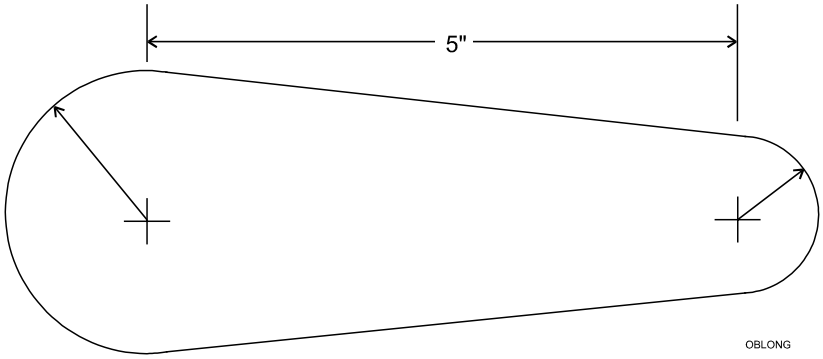


Figure 5-14, Exercise 3 Oblong Slot

Exercise 4:

Use Draw Graphics to verify the program that contains the recalled points from "Exercise 3."

Section 6 - Practice Exercises and Sample Programs

Using the Practice Exercises

The practice exercises consist of a series of tables. Each table contains the keystrokes and instructions required for a single operation. The keystrokes in each table are numbered. Perform the keystrokes in the order shown on the table.

To make menu selections, use the **ARROWS** to highlight to the required choice. Press **ENTER** to activate a highlighted choice. To switch selections, press (**+/-**).

A brief 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 at the correct screen to start the next table.

Starting Practice Exercises #1 - #5

Practice Exercises #1 to #5 are a series of related exercises that coach a first-time programmer through the development of Sample Program #1 - Drill Program. Do these exercises 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.

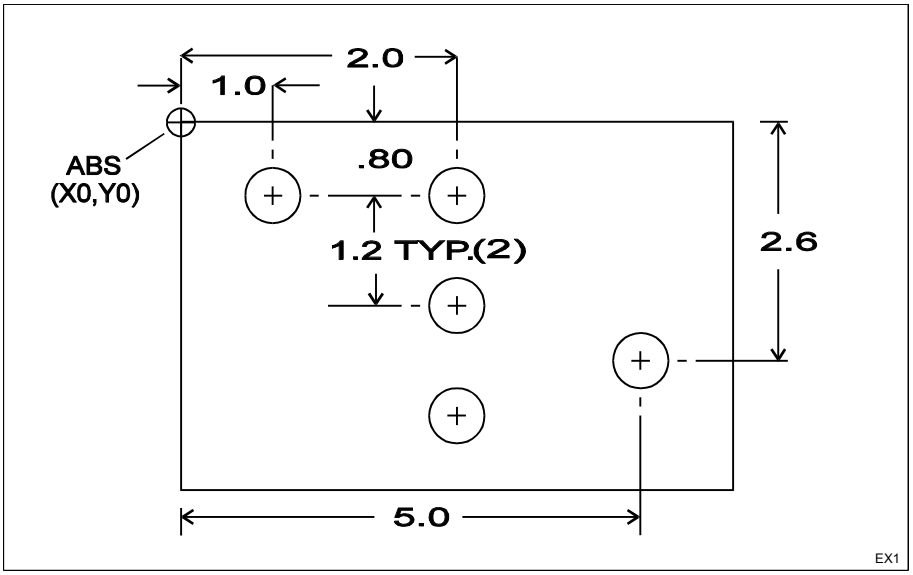


Figure 6-1, Layout for Sample Program #1

Sample Program #1 - Drill Program

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   *  .375  DRILL
4   PeckDrillZDepth  -0.6500  StartHgt  0.1000  Peck  0.1500  Feed  12.8  Tool#  1
5   Rapid X  1.0000  Y  -.8000
6         X  2.0000
7   Dim Incr
8         Y  -1.2000
9         Y  -1.2000
10  Dim Abs
11        X  5.0000  Y  -2.6000
12  DrillOff
13  Rapid Z  0.0000  Tool#  0
14        X  0.0000  Y  0.0000
15  EndMain
16  <End Of Program>
```

Practice Exercise #1 - Creating a Program Named SP1.M

Refer to Figure 6-1, 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	Enter 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 start 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 describes the steps required to write the blocks 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 soft keys.	(**F9**) **Misc**	12	Highlight required character.	Select **R**
2	Activate comment tool.	(**F2**) **Comment**	13	Add character to comment.	ENTER
3	Enter decimal point.	.	14	Highlight required character.	Select **I**
4	Enter 3.	3 ARC	15	Add character to comment.	ENTER
5	Enter 7.	7 UNIT	16	Highlight required character.	Select **L**
6	Enter 5.	5 TOOL	17	Add character to comment.	ENTER
7	Enter space.		18	Add character to comment.	ENTER
8	Enter 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 soft key.	(**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. The 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	Enter decimal point.	.
2	Highlight required selection.	 Select **Pecking**	13	Enter 1.	1 RAPID
3	Activate Pecking cycle Graphic Menu.	ENTER	14	Enter 5.	5 TOOL
4	Change sign.	+/- Switch Negative	15	Advance to **Feed**.	ENTER
5	Enter decimal point.	.	16	Enter 1.	1 RAPID
6	Enter 6.	6 MCODE	17	Enter 2.	2 LINE
7	Enter 5	5 TOOL	18	Enter decimal point.	.
8	Advance to **StartHgt**.	ENTER	19	Enter 8.	8 DWELL
9	Enter decimal point.	.	20	Advance to **Tool#**.	ENTER
10	Enter 1.	1 RAPID	21	Enter 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 -.800					
1	Activate Rapid Graphic Menu.	1 RAPID	5	Enter decimal point.	.
2	Enter 1.	1 RAPID	6	Enter 8.	8 DWELL
3	Advance to **Y**.	ENTER	7	Add block to Program Listing.	**Save** (**F10**)
4	Change sign.	+/- Switch 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	Enter 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	Switch setting.	+/- Switch for **Incr**			

Write modal move.					
Block - 7 Y-1.2000					
1	Activate Modal Move Graphic Menu.	Y	4	Enter decimal point.	.
2	Change sign.	+/- Switch Negative	5	Enter 2.	2 LINE
3	Enter 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	Enter decimal point.	.
2	Change sign.	+/- Switch Negative	5	Enter 2.	2 LINE
3	Enter 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	Enter 2.	2 LINE
2	Enter 5.	5 TOOL	6	Enter decimal point.	.
3	Advance to **Y**.	ENTER	7	Enter 6.	6 MCODE
4	Change sign.	+/- Switch 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	Enter 0.	0
3	Enter 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	Enter 0.	0
2	Enter 0.	0	5	Add block to Program Listing.	**Save** (**F10**)
3	Advance to **Y**.	ENTER			

Add EndMain block.					
Block - 14 EndMain					
1	Start the subprogram soft keys.	**Sub** (**F8**)	3	Restore the default soft key 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" 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	Enter 7.	7 UNIT
2	Start the Program Editor.	**Edit** (**F4**)	7	Enter 5.	5 TOOL
3	Start the Tool Page.	**Tool** (**F6**)	8	Exit the Tool Page.	**Exit** (**F10**)
4	Enter decimal point.	.	9	Exit the Editor.	**Exit** (**F10**)
5	Enter 3.	3 ARC			

Practice Exercise #5 - Running SP1.M in Draw

This exercise uses Draw to view the moves contained in SP1.M program.

This exercise will adjust four of the display parameters before the program runs. Adjust these parameters to ensure that the CNC will operate as specified in the exercises. The adjustments also provide a good view of the programmed moves. Some of the settings might 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.

Start Draw. Turn Tool on, turn Rapid on, set the Mode to Auto and turn off Run. 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**	15	Close Parms pop-up menu.	**Parms** (**F9**)
2	Activate the Program Editor.	**Edit** (**F4**)	16	Activate VIEW pop-up menu.	**VIEW** (**F4**)
3	Activate Draw.	**Draw** (**F2**)	17	Highlight required selection.	 Select **Iso**
4	Activate Parms pop-up menu.	**Parms** (**F9**)	18	Activate **Iso** view.	
5	Highlight required selection.	 Select **Tool**	19	Activate DISPLAY.	**DISPLAY** (**F5**)
6	Switch parameter (if required).	 **Switch On**	20	Activate selection and pause for the "**Program ended normally**" message.	
7	Highlight required selection.	 Select **Rapid**	21	Run the program and verify the programmed moves.	**Run** (**F3**)
8	Switch parameter (if required).	 **Switch On**	22	Activate VIEW pop-up menu.	**VIEW** (**F4**)
9	Highlight required selection.	 Select **Mode**	23	Highlight required selection.	 Select **XY**
10	Activate **Mode** pop-up menu.		24	Activate **XY** view.	
11	Highlight required selection.	 Select **Auto**	25	Run the program and verify its moves.	**Run** (**F3**)
12	Activate **Auto** (if required).		26	Close Draw.	**Exit** (**F10**)
13	Highlight required selection.	 Select **Run**	27	Close the Editor.	**Exit** (**F10**)
14	Switch parameter (if required).	 **Switch Off**	28	Close the Program Directory.	**Exit** (**F10**)

Starting Practice Exercises #6 to #8

Practice Exercises #6 to #8 are a series of related exercises that coach a first-time programmer through the development of Sample Program #4 - Irregular Pocket Program. Do these exercises 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 finds 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 being written.

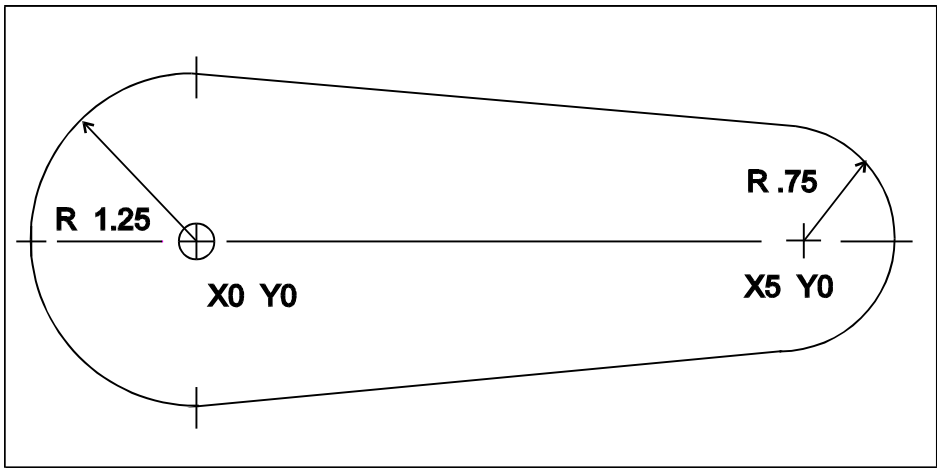


Figure 6-2, Layout for Sample Program #4

Sample Program #4 - Irregular Pocket Program

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   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
4   Rapid Z  0.0000  Tool#  0
5       X  -2.0000  Y  2.0000
6   EndMain
7   Sub 99
8   Rapid X  -1.2500  Y  0.0000
9   Arc Cw X  0.1250  Y  1.2437  Radius  1.2500
10  Line X  5.0750  Y  0.7462
11  Arc Cw X  5.0750  Y  -0.7462  Radius  0.7500
12  Line X  0.1250  Y  -1.2437
13  Arc Cw X  -1.2500  Y  0.0000  Radius  1.2500
14  EndSub
15  <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	Start Program Directory.	**PROGRAM** (**F2**)	7	Add character to name.	ENTER
2	Create new program.	**Create** (**F2**)	8	Enter 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 6-3, Geometry Calculator Layout for Exercise #7. This exercise clears the all values stored in the Geometry Calculator 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.

Do this exercise before Exercise #8. Programs visible in the Editor are not affected by running the calculator. Start this exercise from the Manual screen.

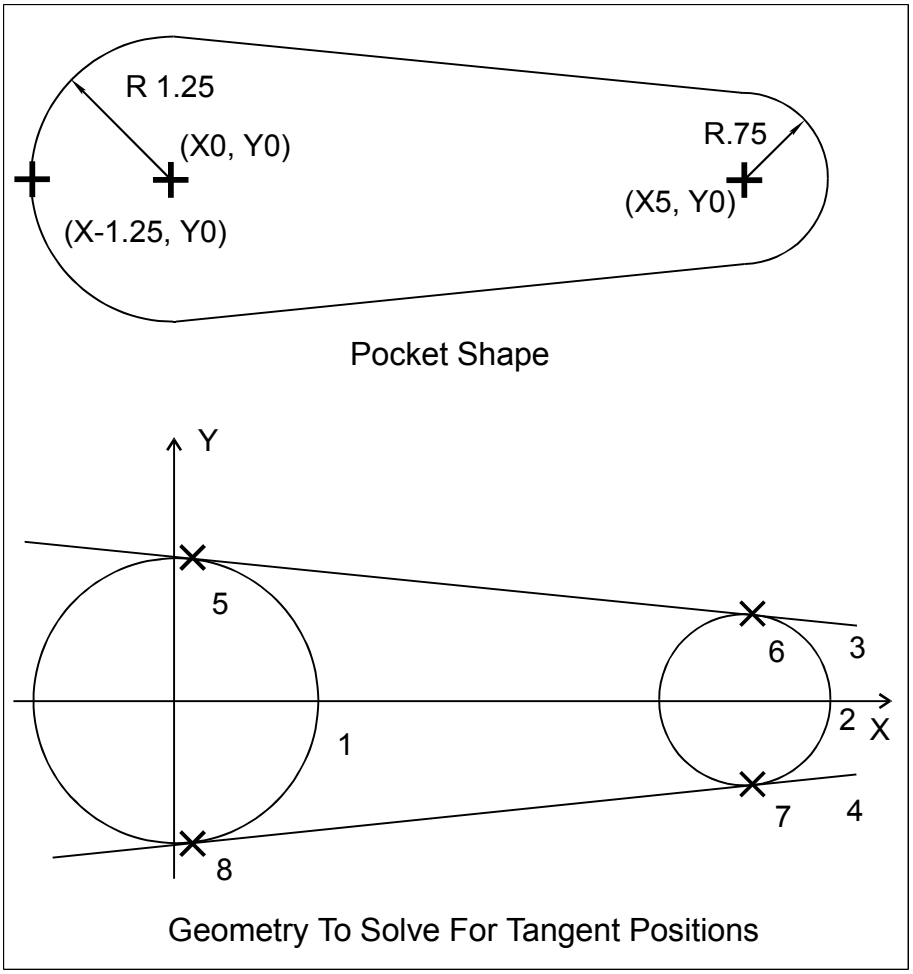
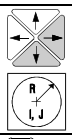
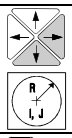


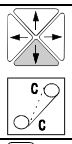
Figure 6-3, Geometry Calculator Layout for Exercise #7

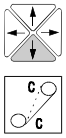
Activate the Geometry Calculator.					
1	Start Program Directory.	**PROGRAM** (**F2**)	4	Activate the Calculator.	**Calc** (**F7**)
2	Highlight required program.	Select **SP4.M**	5	Select required calculator.	Select
3	Start Program Editor.	**Edit** (**F4**)	6	Activate Geometry Calculator.	

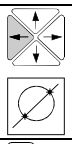
Clear any existing objects from the Geometry Calculator.					
1	Activate Geometry Pop-Up.	**GEOMETR** (**F7**)	3	Activate deletion.	
2	Highlight required selection.	Select **Delete All**	4	Respond to safety prompt.	**Yes** (**F1**)

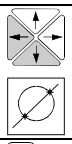
Create 1.25 inch circle centered on X 0, Y 0.					
1	Highlight tool to create a circle on a center.	 Select	8	Activate highlighted tool. CNC prompts for X coordinate of center.	ENTER
2	Activate selected tool. CNC prompts for radius.	ENTER	9	Enter 0.	0
3	Enter 1.	1 RAPID	10	Enter value. CNC prompts for Y coordinate of center.	ENTER
4	Enter decimal point.	.	11	Enter 0.	0
5	Enter 2.	2 LINE	12	Enter value. CNC draws circle.	ENTER
6	Enter 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

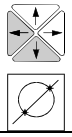
Create 0.75 inch circle centered on X 5, Y 0.					
1	Highlight tool to create a circle on a center.	 Select	8	Enter 5.	5 TOOL
2	Activate selected tool. CNC prompts for radius.	ENTER	9	Enter value. CNC prompts for Y coordinate of center.	ENTER
3	Enter decimal point.	.	10	Enter 0.	0
4	Enter 7.	7 UNIT	11	Enter value. CNC draws circle.	ENTER
5	Enter 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			

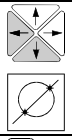
Create line tangent to top of both circles.					
1	Highlight tool to create a line tangent to two circles.	 Select	7	Enter 1.	1 RAPID
2	Activate selected tool. CNC prompts for first circle number.	ENTER	8	Enter value. CNC clears all but the selected line.	ENTER
3	Enter 1.	1 RAPID	9	Activate DISPLAY pop-up menu.	**DISPLAY** (**F5**)
4	Enter value. CNC prompts for second circle number.	ENTER	10	Highlight required selection.	 Select **Redraw**”
5	Enter 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	Enter 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	Enter 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	Enter 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.	ENTER
2	Activate selected tool. CNC prompts for first element number.	ENTER	5	Enter 3.	3 ARC
3	Enter 1.	1 RAPID	6	Enter value. CNC adds a numbered point at the intersection.	ENTER

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.	ENTER
2	Activate selected tool. CNC prompts for first element number.	ENTER	5	Enter 3.	3 ARC
3	Enter 2.	2 LINE	6	Enter value. CNC adds a numbered point at the intersection.	ENTER

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.	ENTER
2	Activate selected tool. CNC prompts for first element number.	ENTER	5	Enter 4.	4 FEED
3	Enter 2.	2 LINE	6	Enter value. CNC adds a numbered point at the intersection.	ENTER

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	Enter 4.	4 FEED
3	Enter 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. Do this exercise after you have completed Exercises # 5 and #6. The tangential positions found with the Geometry Calculator in Exercise #6 will be recalled to the program in this exercise.

The program in this exercise uses an Irregular Pocket cycle. Plan the moves in the subprogram to make programming an irregular pocket easier.

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 Write 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 a Subprogram

Refer to Figure 6-3, 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 to 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	Enter 9.	9 PLANE
2	Activate Program Editor.	**Edit** (**F4**)	6	Enter 9.	9 PLANE
3	Activate the subprogram soft keys.	**Sub** (**F8**)	7	Add block to program.	**Save** (**F10**)
4	Activate Subprogram Graphic Menu.	**Sub** (**F1**)	8	Restore default soft keys.	**Prev** (**F9**)

Program a Rapid move to starting point.					
X -1.2500 Y 0.0000					
1	Activate Rapid Graphic Menu.	1 RAPID	6	Enter 5.	5 TOOL
2	Change sign.	+/- Switch Negative	7	Advance to **Y**.	
3	Enter 1.	1 RAPID	8	Enter 0.	0
4	Enter decimal.	.	9	Add block to program.	**Save** (**F10**)
5	Enter 2.	2 LINE			

Program an Arc to the first tangential point (#5).					
Arc Cw X 0.1250 Y 1.2437 Radius 1.2500					
1	Activate Mill soft keys.	**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	Enter 1.	1 RAPID
4	Activate Recall pop-up menu.	**Recall** (**F2**)	12	Enter decimal.	.
5	Select Geometry Calculator.	Select	13	Enter 2.	2 LINE
6	Activate Geometry Calculator recall.	ENTER	14	Enter 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 tangential 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 tangential 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	Enter decimal point.	.
4	Select Geometry Calculator.	Select	11	Enter 7.	7 UNIT
5	Activate Geometry Calculator recall.	ENTER	12	Enter 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 tangential 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**.	W2-W3	9	Enter 1.	1 RAPID
3	Change sign.	+/-	10	Enter decimal point.	.
4	Enter 1.	1 RAPID	11	Enter 2.	2 LINE
5	Enter decimal point.	.	12	Enter 5.	5 TOOL
6	Enter 2.	2 LINE	13	Add block to program.	**Save** (**F10**)
7	Enter 5.	5 TOOL			

Program a block to tell the CNC where the subprogram ends.					
EndSub					
1	Restore the default soft keys.	**Prev** (**F9**)	4	Restore the default soft keys.	**Prev** (**F9**)
2	Activate the subprogram soft keys.	**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

Once the subprogram is written and saved, check it carefully. A program cannot run by itself. To check it, temporarily enter a short main program that only calls the subprogram. Once the subprogram is checked, delete the temporary blocks and write the rest of the main program.

The last block of a main program must be an EndMain block. For the test, the main program will consist of 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. Complete Practice Exercise #5 up to keystroke 15 (turn Tool on, turn Rapid on, set the mode to Auto and turn off Run), if these settings were changed.

Activate the Editor and write a subprogram call block.					
Call 99					
1	Highlight required program.	 Select **SP4.M**	5	Enter 9.	9 PLANE
2	Activate Program Editor.	**Edit** (**F4**)	6	Enter 9.	9 PLANE
3	Activate the subprogram soft keys.	**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 soft key line.	**Prev** (**F9**)

Start Draw, set the view and run the program.					
1	Start 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 acceptable but the direction is not. Use an angle value to choose a direction that will let the CNC mill out the entire pocket. A starting angle of 90° (from the 3 o'clock position) might seem convenient, but a 90° line from the starting point (on the arc) is tangent to the arc. A tangential line does not point to a position inside the pocket. An 89.9-degree angle points to a position 0.1 degrees inside the arc.

The EndMain block added in the previous step will be re-used. The graphic menu for the irregular pocket has its own Sub# field. Therefore, 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	Enter decimal point.	.
3	Activate Irregular Graphic Menu.	ENTER	21	Enter 1.	1 RAPID
4	Enter 9.	9 PLANE	22	Enter 6.	6 MCODE
5	Enter 9.	9 PLANE	23	Advance to **DepthCut**.	ENTER
6	Highlight required field.	Select **StartHgt**	24	Enter decimal point.	.
7	Enter decimal point.	.	25	Enter 1.	1 RAPID
8	Enter 1.	1 RAPID	26	Enter 2.	2 LINE
9	Advance to **ZDepth**.	ENTER	27	Enter 5.	5 TOOL
10	Change sign.	+/- Switch Negative	28	Advance to **FinStock**.	ENTER
11	Enter decimal point.	.	29	Enter decimal point.	.
12	Enter 2.	2 LINE	30	Enter 0.	0
13	Enter 5.	5 TOOL	31	Enter 1.	1 RAPID
14	Advance to **Angle**.	ENTER	32	Advance to required field.	Select **RoughFeed**
15	Enter 8.	8 DWELL	33	Enter 1.	1 RAPID
16	Enter 9.	9 PLANE	34	Enter 2.	2 LINE
17	Enter decimal point.	.	35	Enter decimal point.	.
18	Enter 9.	9 PLANE	36	Enter 5.	5 TOOL

Program irregular pocket block (continued).					
37	Advance to **FinFeed**.	ENTER	41	Advance to **Tool#**.	ENTER
38	Enter 9.	9 PLANE	42	Enter 1.	1 RAPID
39	Enter decimal point.	.	43	Add block to Program Listing.	**Save** (**F10**)
40	Enter 5.	5 TOOL			

Fully retract the quill.					
Rapid Z 0.0000 Tool# 0					
1	Activate Rapid Graphic Menu.	1 RAPID	5	Enter 0.	0
2	Advance to **Z**.	Select **Z**	6	Add block to Program Listing.	**Save** (**F10**)
3	Enter 0.	0	7	Exit the Editor saving the program.	**Exit** (**F10**)
4	Advance to **Tool#.**	Select **Tool#**			

Step 4 - Viewing the Finished Program with Draw

This program is written for a 3/8” endmill. In this step the tool diameter is entered on the Tool Page. To check the program, run it in Draw. Start this step from the Program screen.

View the finished program in Draw.					
1	Select the required program.	Select **SP4.M**	8	Exit the Tool Page.	**Exit** (**F10**)
2	Activate the Editor.	**Edit** (**F4**)	9	Activate Draw.	**Draw** (**F2**)
3	Activate the Tool Page.	**Tool** (**F6**)	10	Run the program to verify the milling of the pocket.	**Run** (**F3**)
4	Enter decimal point.	.	11	Close Draw.	**Exit** (**F10**)
5	Enter 3.	3 ARC	12	Close the Editor.	**Exit** (**F10**)
6	Enter 7.	7 UNIT	13	Exit to the Manual screen.	**Exit** (**F10**)
7	Enter 5.	5 TOOL			

Practice Exercises

This section contains sample programs and practice exercises.

If you are a first-time programmer, we encourage you to do the exercises to learn about the CNC. Even if you are an experienced operator, you may find it useful to look at one of the sample programs.

Practice Exercises #1 to #5 list all keystrokes required to create, program and check **Sample Program #1 - Drill Program**.

Practice Exercises #6 to #8 list all the keystrokes required to create (with the Geometry Calculator), program and check Sample Program #4 - Irregular Pocket Program.

NOTE: During software installation, the CNC prompts 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 3-axis sample programs begin with "3AX-".

Sample Program #1 - Drill Program

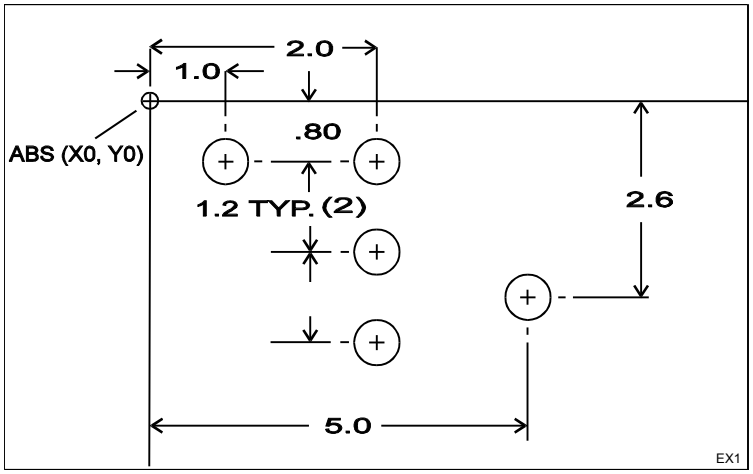


Figure 6-4, Layout for Sample Program #1

Refer to **Figure 6-4**. This program drills five holes in a surface using a 3/8" diameter twist drill (Tool #1). The holes are drilled with a Pecking Drill Cycle. Go to the Tool Page and enter 0.375" diameter for Tool #1 before you run the program in Draw.

Detailed instructions for how to program and view this program are outlined in Practice Exercises #1 to #5.

Sample Program #1

```
1   Dim Abs
2   Rapid  Z 0.0000  T 0.0000
3   *  .375  DRILL
4   PeckDrill ZDepth  -0.6500  StartHgt  0.1000  Peck  0.1500  Feed  12.8
        Tool#  1
5   Rapid X  1.0000  Y  -.8000
6         X  2.0000
7   Dim Incr
8         Y  -1.2000
9         Y  -1.2000
10  Dim Abs
11        X  5.0000  Y  -2.6000
12  DrillOff
13  Rapid Z  0.0000  Tool#  0
14        X  0.0000  Y  0.0000
15  EndMain
16  <End Of Program>
```

Sample Program #2 - Rectangular Pocket Program

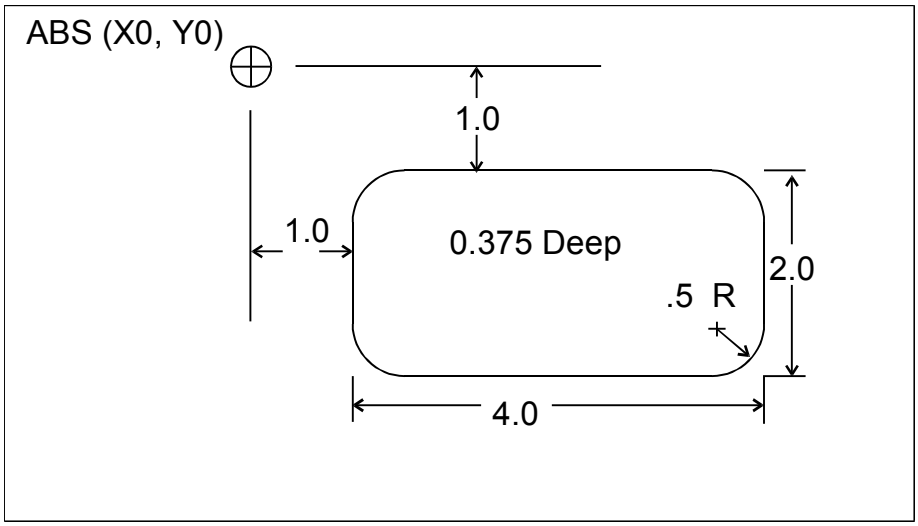


Figure 6-5, Layout for Sample Program #2

Refer to **Figure 6-5**. This program uses a 1/2” diameter endmill (Tool #1) to mill out a rectangular pocket.

Go to the Tool Page and enter 0.50” diameter for Tool #1 before you run the program in Draw.

Sample Program #2

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   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
4       Rapid  Z  0.0000  Tool#  0
5       X  -3.0000  Y  3.0000
6   EndMain
7   <End Of Program>
```

Sample Program #3 - Irregular Pocket Program

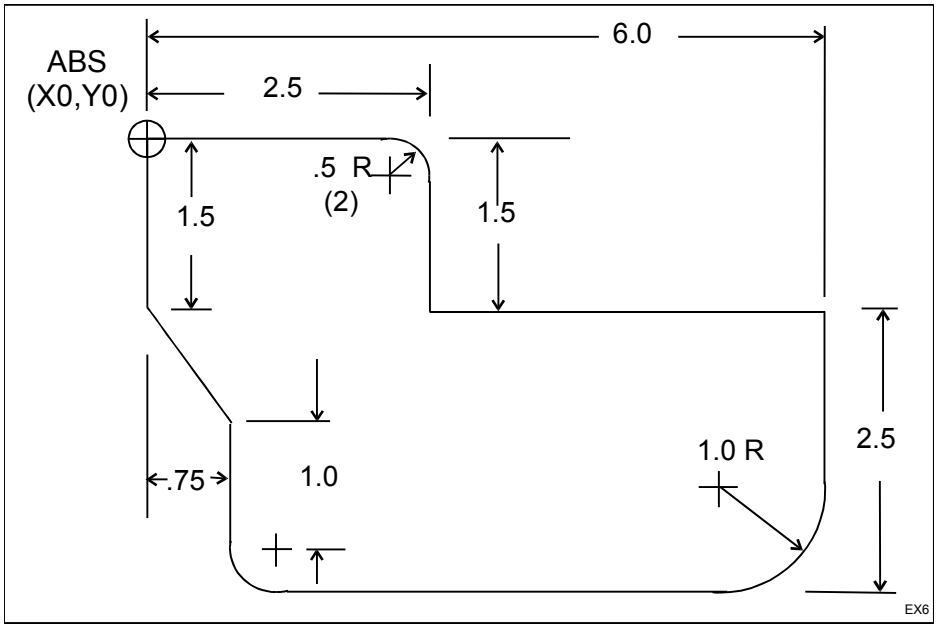


Figure 6-6, Layout for Sample Program #3

Refer to **Figure 6-6**. This program uses a 1/2“ diameter endmill (Tool #1) to mill out the irregular pocket shown. Go to the Tool Page and enter 0.50“ diameter for Tool #1 before you run the program in Draw.

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   Rapid Z 0.0000  T 0.0000
3   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
4   Rapid Z  0.0000  Tool#  0
5       X  -2.0000  Y  2.0000
6   EndMain
7   Sub 1
8   Rapid X  0.0000  Y  0.0000
9   Line X  2.5000  CornerRad  0.5000
10      Y  -1.5000
11      X  6.0000
12  Dim Incr
13  Line Y  -2.5000  CornerRad  1.0000
14  Dim Abs
15  Line X  0.7500  CornerRad  0.5000
16      Y  -2.5000
17      X  0.0000  Y  -1.5000
18      Y  0.0000
19  EndSub
20  <End Of Program>
```

Sample Program #4 - Irregular Pocket Program

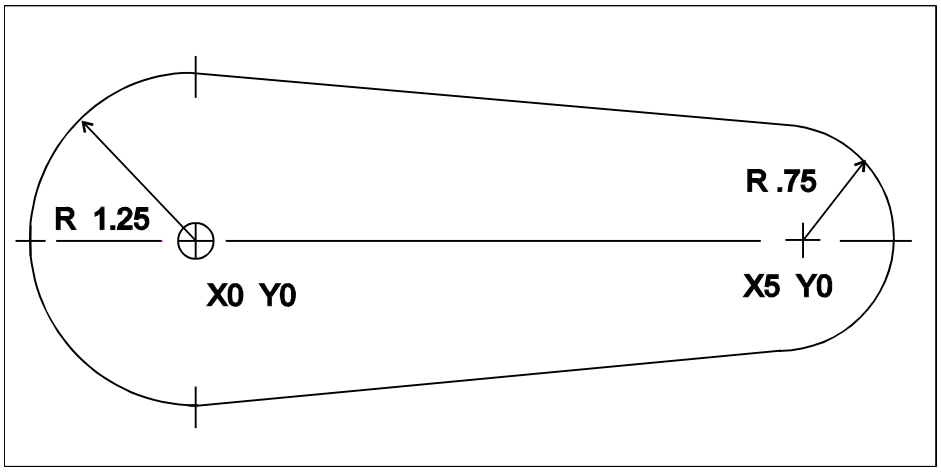


Figure 6-7, Layout for Sample Program #4

Refer to **Figure 6-7**. This program uses a 3/8“ diameter endmill (Tool #1) to mill out an irregular pocket. Go to the Tool Page and enter 0.375” diameter for Tool #1 before you run the program in Draw.

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° inside the Arc) by the Angle value.

The keystrokes used to program and view Sample Program #4 are shown in Practice Exercises #6 to #8.

Sample Program #4

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   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
4   Rapid Z  0.0000  Tool#  0
5   X  -2.0000  Y  2.0000
6   EndMain
7   Sub 99
8   Rapid X  -1.2500  Y  0.0000
9   Arc Cw X  0.1250  Y  1.2437  Radius  1.2500
10  Line X  5.0750  Y  0.7462
11  Arc Cw X  5.0750  Y  -0.7462  Radius  0.7500
12  Line X  0.1250  Y  -1.2437
13  Arc Cw X  -1.2500  Y  0.0000  Radius  1.2500
13  EndSub
14  <End Of Program>
```

Sample Program #5 - Program with Looping Subprogram

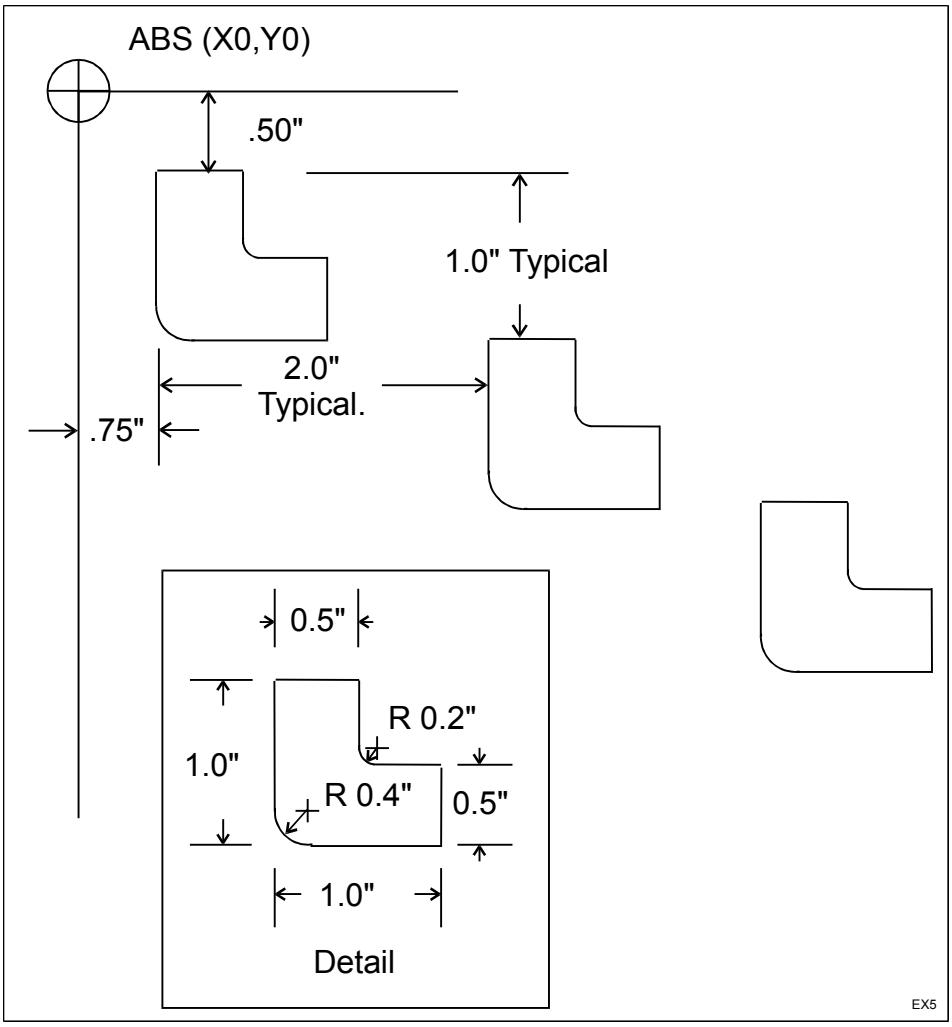


Figure 6-8, Layout for Sample Program #5

Refer to Figure 6-7, Layout for Sample Program #4. This program loops Subprogram #1 to cut three L-shaped pockets. The pockets are cut with a 1/8" diameter endmill (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 enables the CNC cut the pocket anywhere on the work. Go to the Tool Page and enter 0.125" diameter for Tool #1 before you run the program in Draw.

Sample Program #5

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   Rapid X 0.7500 Y -0.7500 Tool#  1
4   Loop Sub  1 Loops 3 XIncr 2.0000 YIncr -1.0000
5   Rapid Z 0.0000 Tool# 0
6         X -2.0000 Y 2.0000
7   EndMain
8   Sub 1
9   Dim Abs
10  Rapid Z 0.1000
11  Line Z -0.0620 Feed 1.5
12  Dim Incr
13  Line Y 0.2500 ToolComp Left Feed 2.1
14  Line X -0.2500
15  Line Y -1.0000 CornerRad 0.4000
16  Line X 1.0000
17  Line Y 0.5000
18  Line X -0.5000 CornerRad 0.2000
19  Line Y 0.5000
20  Line X -0.2500
21  Line Y -0.2500 ToolComp Off
22  Dim Abs
23  Rapid Z 0.1000
24  EndSub
25  <End Of Program>
```

Sample Program #6 - Program with Rotating Subprogram

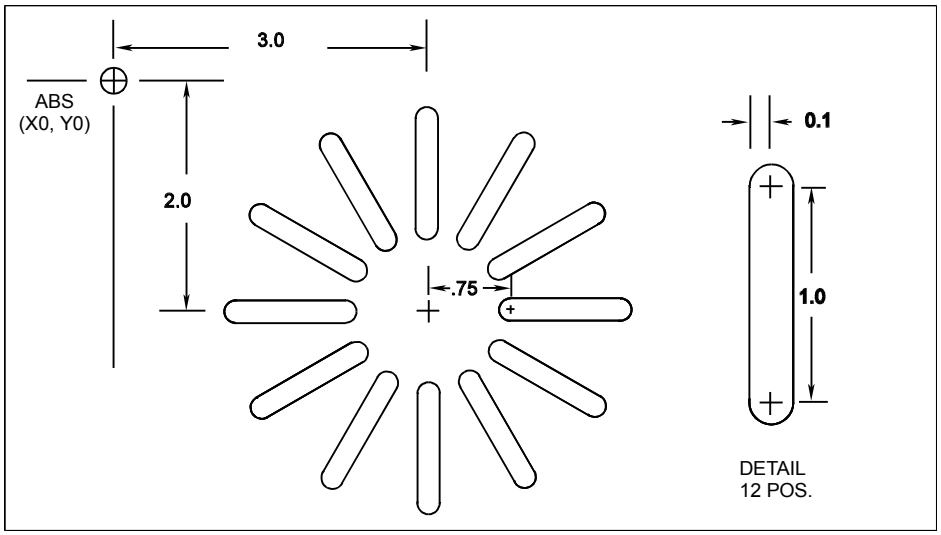


Figure 6-9, Layout for Sample Program #6

Refer to **Figure 6-9**. This program rotates subprogram #1 to cut twelve slots. The slots are cut using a 1/8“ endmill (Tool #1).

Go to the Tool Page and enter 0.125” diameter for Tool #1 before you run the program in Draw.

Sample Program #6

```
1   Dim Abs
2   Rapid  Z 0.0000  T 0.0000
3   RMS Sub# 1 #Loops 12 StartAngle 0.0000 Angle 30.0000 XCenter 3.0000
        YCenter -2.0000 Tool# 1
4   Rapid Z  0.0000  Tool# 0
5         X  -2.0000  Y  2.0000
6   EndMain
7   Sub 1
8   Dim Abs
9   Rapid X  3.7500  Y  -2.0000
10        Z  0.1000
11  Line Z  -0.1250  Feed  1.2
12  Dim Incr
13  Line Y  0.1000  ToolComp  Left  Feed  2.2
14  Arc Ccw X  0.0000  Y  -0.2000  Radius  0.1000
15  Line X  1.0000
16  Arc Ccw X  0.0000  Y  0.2000  Radius  0.1000
17  Line X  -1.0000
18  Line Y  -0.1000  ToolComp  Off
19  Dim Abs
20  Rapid Z  0.1000
21  EndSub
22  <End Of Program>
```

Sample Program #7 - X Axis Mold Rotation Program

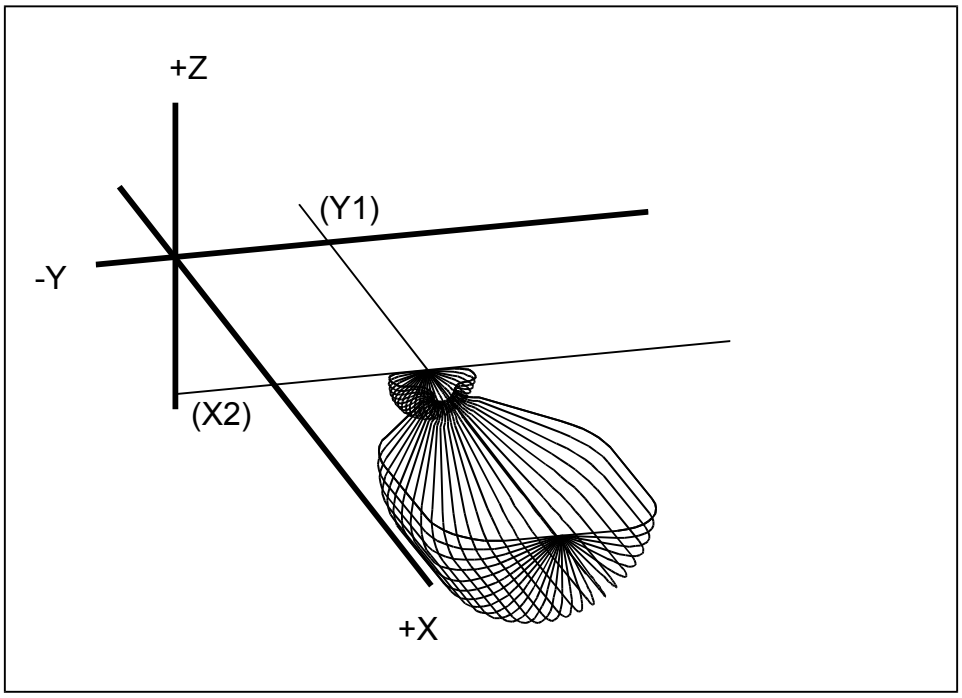


Figure 6-10, Layout for Sample Program #7 - Mold Rotation

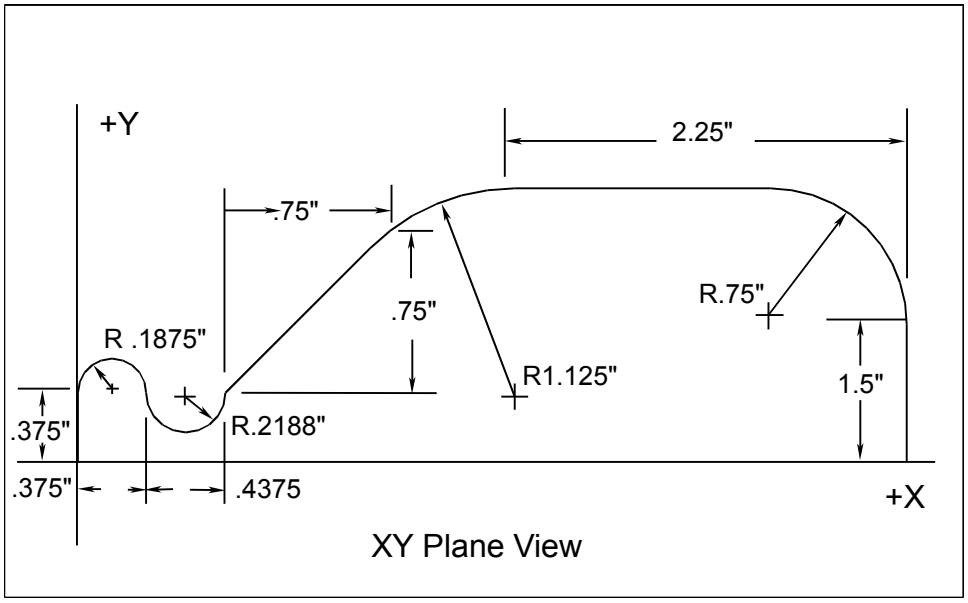


Figure 6-11, Subprogram Layout for Sample Program #7

Refer to Figure 6-10, Layout for Sample Program #7 - Mold Rotation and Figure 6-11, 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. In this example, the size of Tool #1 will determine size of the finished mold. Go to the Tool Page and enter an appropriate diameter for Tool #1 before you run the program in Draw.

Sample Program #7

```
1.  Dim Abs
1   Rapid  Z 0.0000  T 0.0000
1.  Rapid X  2.0000  Y  1.0000  Tool#  1
2.        Z  0.5000
3.  Line Z  0.0000  Feed  10.0
4.  MoldRotStartAngle  0.0000  EndAngle  -180.0000  Cycles 12 FwdSub  1
        RevSub  2  AxisRot  X  BAxisCL  1.0000
5.  Rapid Z  0.0000  Tool# 0
6.  EndMain
7.  Sub 1
8.  Dim Incr
9.  Line Y  0.3750
10. Arc  Cw X  0.3750  Y  0.0000  Radius  0.1875
11. Arc  Ccw X  0.4375  Y  0.0000  Radius  0.2188
12. Line X  0.7500  Y  0.7500
13. Arc Cw X  0.7500  Y 0.3750  Radius  1.1250
14. Line X  2.2500  CornerRad  0.7500
15. Line Y  -1.5000
16. EndSub
17. Sub 2
18. Line Y  1.5000  CornerRad  0.7500
19. Line X  -2.2500
20. Arc Ccw X  -0.7500  Y  -0.3750  Radius  1.1250
21. Line X  -0.7500  Y  -0.7500
22. Arc Cw X  -0.4375  Y  0.0000  Radius  0.2188
23. Arc Ccw X  -0.3750  Y  0.0000  Radius  0.1875
24. Line Y  -0.3750
25. EndSub
26. <End Of Program>
```

Sample Program #8 - Y Axis Mold Rotation Program

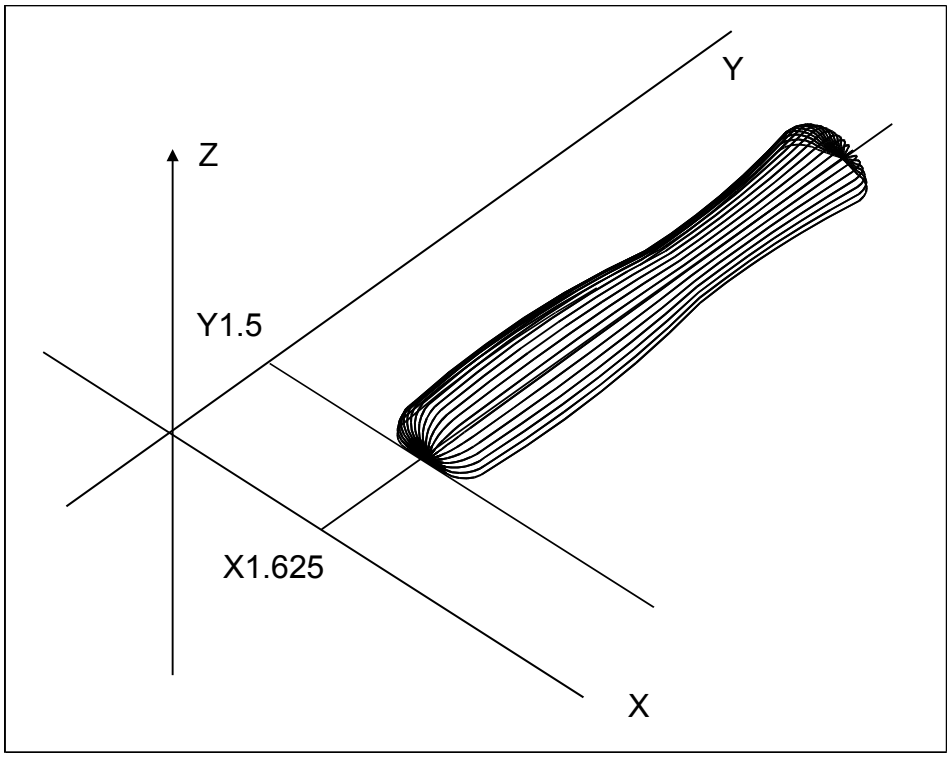


Figure 6-12, Layout for Sample Program #8 - Mold Rotation

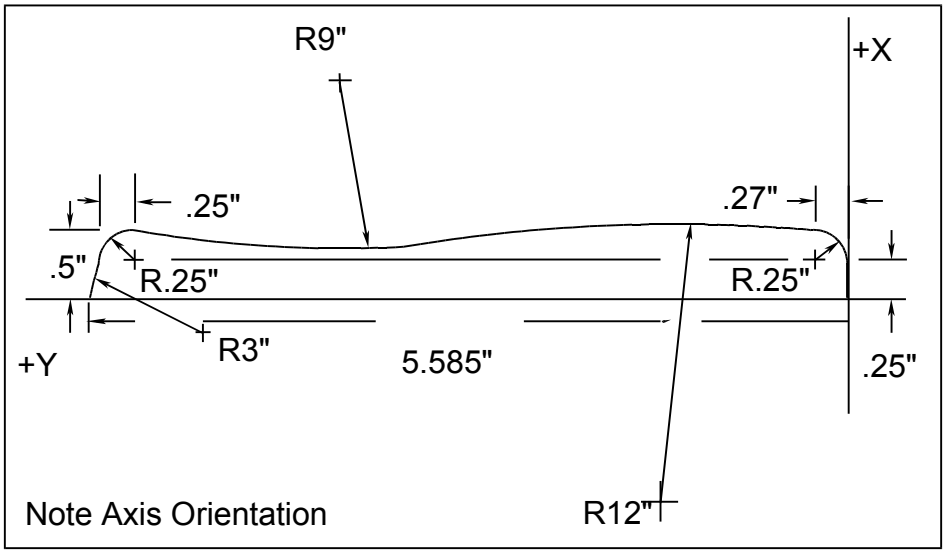


Figure 6-13, Subprogram Layout for Sample Program #8

Refer to Figure 6-12, Layout for Sample Program #8 - Mold Rotation and Figure 6-13, 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 path. 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 endmill. This example does not use diameter compensation. In this example, the size of Tool #1 will determine the size of the finished mold. Go to the Tool Page and enter an appropriate diameter for Tool #1 before you run the program in Draw.

Sample Program #8

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   Rapid X  1.6250  Y  1.5000  Tool#  1
4         Z  0.5000
5   Line Z  0.0000  Feed  10.0
6   MoldRot  StartAngle  0.0000  EndAngle  180.0000  Cycles 10
        FwdSub  15  RevSub  16  AxisRot  Y  BAxisCL  1.6250
7   Rapid Z  0.0000  Tool#  0
8   EndMain
9   Sub 15
10  Dim Incr
11  Line X  -0.2500
12  Arc Cw X  -0.2500  Y  0.2700  Radius  0.2500
13  Arc Cw X  0.1250  Y  3.0000  Radius  12.0000
14  Arc Ccw X  -0.1250  Y  2.0000  Radius  9.0000
15  Arc Cw X  0.2500  Y  0.2500  Radius  0.2500
16  Arc Cw X  0.2500  Y  0.0625  Radius  3.0000
17  EndSub
18  Sub 16
19  Arc Ccw X  -0.2500  Y  -0.0625  Radius  3.0000
20  Arc Ccw X  -0.2500  Y  -0.2500  Radius  0.2500
21  Arc Cw X  0.1250  Y  -2.0000  Radius  9.0000
22  Arc Ccw X  -0.1250  Y  -3.0000  Radius  12.0000
23  Arc Ccw X  0.2500  Y  -0.2700  Radius  0.2500
24  Line X  0.2500
25  EndSub
26  <End Of Program>
```

Sample Program #9 - Z Axis Mold Rotation Program

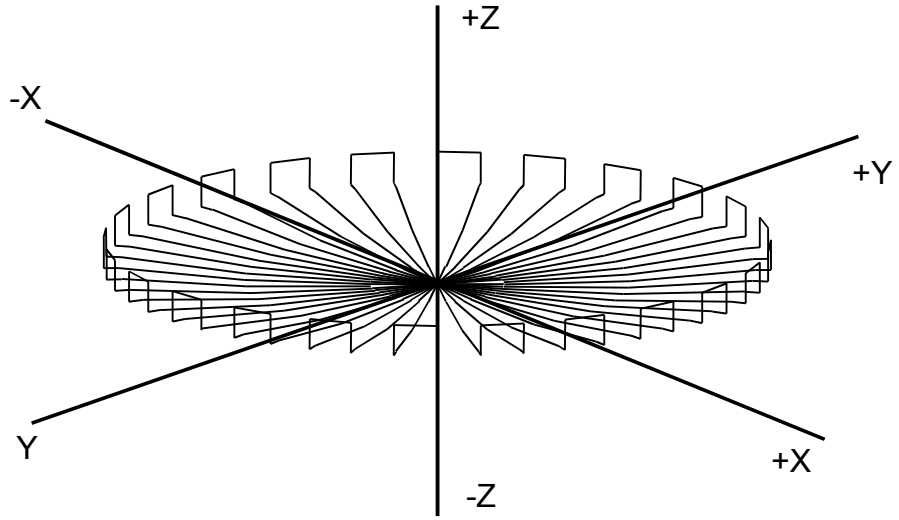


Figure 6-14, Layout for Sample Program #9, Mold Rotation

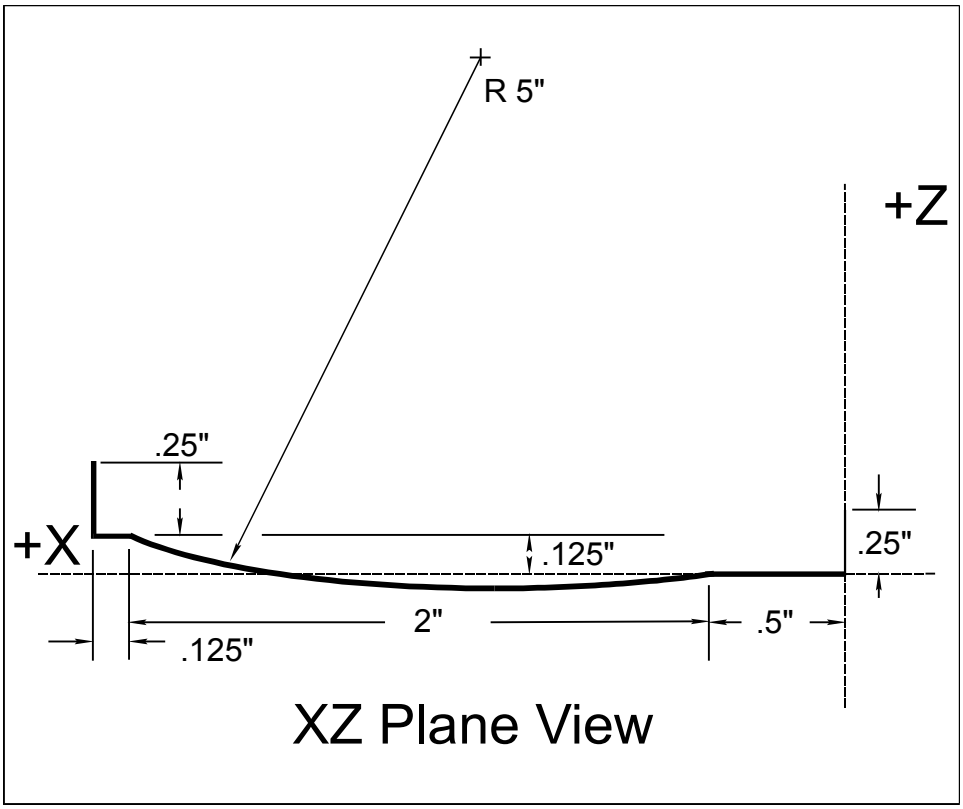


Figure 6-15, Subprogram Layout for Sample Program #9

Refer to Figure 6-14, Layout for Sample Program #9, Mold Rotation and Figure 6-15, 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 path. 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 and 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 you run the program in Draw.

Sample Program #9

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   Rapid X  0.0000  Y  0.0000  Tool#  1
4          Z 0.5000
5   Line  Z 0.0000 Feed 5.0
6   MoldRot StartAngle  0.0000  EndAngle  350.0000  Cycles  13
    FwdSub  1 RevSub  2  AxisRot  Z  Feed  5.7
7   Rapid Z 0.0000 Tool# 0
8   Plane XY
9   EndMain
10  Sub 1
11  Plane XZ
12  Dim Incr
13  Line  Z  -0.2500  ToolComp  Right
14  Line  X  0.5000
15  Arc Cw X  2.0000  Z  0.1250  Radius  5.0000
16  Line  X  0.1250
17  Line  Z  0.2500  ToolComp  Off
18  EndSub
19  Sub 2
20  Line  Z  -0.2500  ToolComp  Left
21  Line  X  -0.1250
22  Arc Ccw X  -2.0000  Z  -0.1250  Radius  5.0000
23  Line  X  -0.5000
24  Line  Z  0.2500  ToolComp  Off
25  EndSub
26  <End Of Program>
```

Sample Program # 10 - Elbow Milling

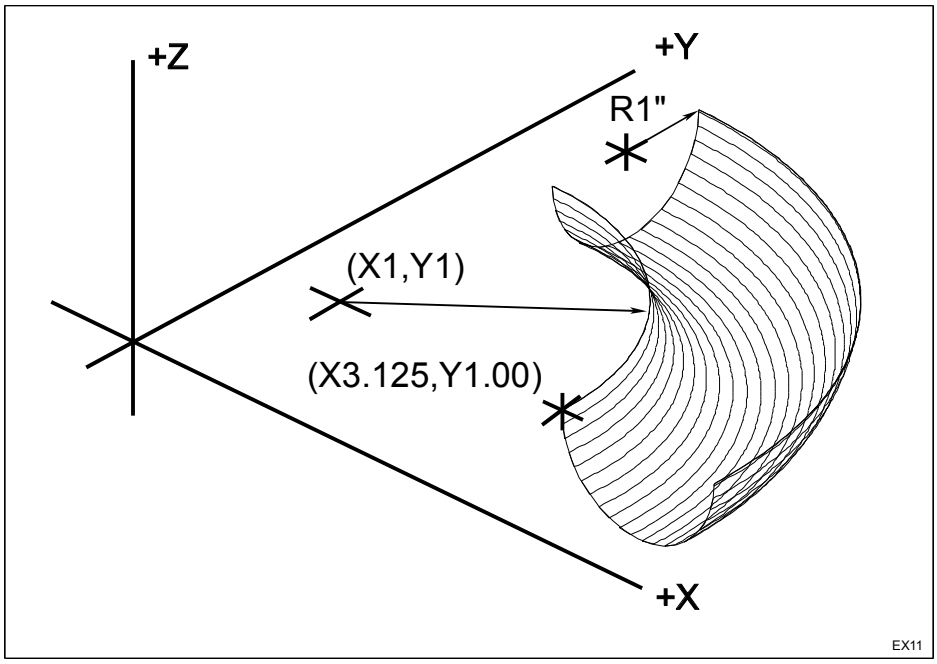


Figure 6-16, Layout for Sample Program #10, Elbow Milling

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.

Sample Program #10

```
1   Dim Abs
2   Rapid Z 0.0000  T 0.0000
3   Rapid X  3.1250  Y  1.0000  Tool#  1
4         Z 0.5000
5   Line Z  0.0000  Feed  6.5
6   Elbow..... Ccw  StartRad  1.0000  EndRad  1.0000  InclAngle 90.0000
        Cycles  10  XCenter  1.0000  YCenter  1.0000  Feed  25.5
7   Rapid Z 0.0000  Tool#   0
8   EndMain
9   <End Of Program>
```

ANILAM

U.S.A.
ANILAM
One Precision Way
Jamestown, NY 14701
☎ (716) 661-1899
FAX (716) 661-1884
✉ anilaminc@anilam.com

ANILAM, CA
16312 Garfield Ave., Unit B
Paramount, CA 90723
☎ (562) 408-3334
FAX (562) 634-5459
✉ anilamla@anilam.com

Dial "011" before each number when calling from the U.S.A.

France
ANILAM S.A.R.L.
2 Ave de la Cristallerie
B.P. 68-92316
Serves Cedex, France
☎ +33-1-46290061
FAX +33-1-45072402
✉ courrier@acu-rite.fr

Germany
ANILAM GmbH
Fraunhoferstrasse 1
D-83301 Traunreut
Germany
☎ +49 8669 856110
FAX +49 8669 850930
✉ info@anilam.de

Italy
ANILAM Elettronica s.r.l.
10043 Orbassano
Strada Borgaretto 38
Torino, Italy
☎ +39 011 900 2606
FAX +39 011 900 2466
✉ info@anilam.it

Taiwan
ANILAM, TW
No. 246 Chau-Fu Road
Taichung City 407
Taiwan, ROC
☎ +886-4 225 87222
FAX +886-4 225 87260
✉ anilamtw@anilam.com

United Kingdom
ACI (UK) Limited
16 Plover Close, Interchange Park
Newport Pagnell
Buckinghamshire, MK16 9PS
England
☎ +44 (0) 1908 514 500
FAX +44 (0) 1908 610 111
✉ sales@aciuk.co.uk

China
Acu-Rite Companies Inc.(Shanghai Representative Office)
Room 1986, Tower B
City Center of Shanghai
No. 100 Zunyi Lu Road
Chang Ning District
200051 Shanghai P.R.C.
☎ +86 21 62370398
FAX +86 21 62372320
✉ china@anilam.com