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**

P/N 70000497
31-August-01
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.

Copyright 2001 ACU-RITE Companies, Inc.

Section 1 - Fundamentals of Programming

Tool Motion Orientation 1-1
Absolute Positioning 1-3
Incremental Positioning 1-3
Defining Positions 1-4
Quiz 1 1-6

Section 2 - CNC Console

Console Overview 2-1
Keyboard Layout 2-1
Manual Mode Screen 2-2
  Primary Display Area Labels 2-3
  Secondary Display Area Labels 2-3
  Position Display 2-4
Jog Moves 2-5
Soft Key Overview 2-6
  Manual Soft Keys 2-6
  Program Soft Keys 2-7
  Edit Soft Keys 2-8
Quiz 2 2-10

Section 3 - Programming Sample

2-Axis CNC Operator's Role 3-1
The Part 3-2
Developing Part Programs 3-2
  Summary 3-2
  Required Tools 3-2
  Creating a Program Name 3-3
  Editing Blocks 3-3
Write Your Own Program, Block-by-Block 3-3
  Block 1: Set Absolute Mode 3-5
  Block 2: Drilling Canned Cycles 3-5
  Block 3: Bolt Hole Pattern Canned Cycle 3-5
  Block 4: Drill Off Cycle 3-6
  Block 5: Clear the Part (Manual Z Move) 3-6
  Block 6: Move to the Tool Change Position 3-6
  Block 7: Circular Pocket Milling 3-7
  Block 8: Rapid Move and Tool Compensation 3-8
  Block 9: Manual Z Move to Cutting Depth 3-8
  Block 10: Line X Move and Corner Rounding 3-9
  Block 11: Line Y Move and Corner Rounding 3-9
    Recalling Values from the Right Triangle Calculator 3-10
  Block 12: Line Move from Right Triangle Calculation 3-11
  Block 13: Linear Interpolation-Angle Move 3-12
  Block 14: Make the Last Cut and Move Away from the Part 3-12
  Block 15: Cancel Tool Compensation 3-12
  Block 16: Manual Return to Z0 3-13
  Block 17: Move Away from the Work and Cancel Tool #2 3-13
  Block 18: Program EndMain 3-13
  Checking the Program 3-13

Draw Graphics 3-14
  Starting Draw 3-14
  Putting Draw in Hold 3-16
  Selecting the View Mode 3-16
  Tool On or Off 3-16
Select a Program to Run 3-16
Running Programs 3-17
  Running a Program One Step at a Time 3-17
  Switching Between Motion and Single-Step Mode 3-17
  Holding or Canceling a Single-Step Run 3-18
  Single-Step Execution of Selected Program Blocks 3-18
  Switching from Single-Step to Auto Mode 3-18
  Automatic Program Execution 3-18
  Holding or Canceling an Auto Run 3-18
  Starting at a Specific Block 3-19
  Clearing a Halted Program 3-19
  Program Run Status 3-19
Quiz 3 3-21

Section 4 - Machine Setup

Set Absolute Zero 4-1
Using the DRO Mode 4-3
Canceling Tool Length Offsets 4-3
Setting a New Z Home 4-3
Setting Tool Length Offsets for Drilling Tools 4-3
Setting the Tool Length Offset for Milling Tools 4-4
Setting Tool Diameters 4-4
Machining the Part 4-5
Quiz 4 4-6

Section 5 - Calculators

Math Calculator 5-1
  Activating the Math Calculator 5-1
  Math Calculator Basics 5-2
  Operations Involving Two Numbers 5-3
  Using Parentheses 5-3
  Using Additional Functions 5-4
  Storing Numbers from the Math Calculator 5-5
Right Triangle Calculator 5-5
  Activating the Triangle Calculator 5-5
  Using the Triangle Calculator 5-5
  Storing Right Triangle Calculator Results 5-6
  Hiding the Right Triangle Calculator Screen 5-6
Geometry Calculator 5-7
  Activating the Geometry Calculator 5-7
  Geometry Calculator Screen 5-7
  Using the Geometry Calculator 5-8
  Point Templates 5-9
  Line Templates 5-10
  Circle Templates 5-11
  Deleting Selected Elements 5-11
  Deleting All Elements 5-11

Listing All Geometry Elements 5-12
Calculating the Distance between Two Elements 5-12
Last Position Recall 5-12
Recalling Values to a Program 5-13
Recalling Values from the Math Calculator 5-13
Recalling Values from the Right Triangle Calculator 5-14
Recalling Values from the Geometry Calculator 5-14
Recalling Values from One Calculator into Another 5-15
Geometry Calculator Example 5-15
Sample Program with Recalled Geometry Points 5-20
Block 1: Set Absolute Mode 5-20
Block 2: Manual Z Move 5-20
Block 3: Cancel Offsets 5-20
Block 4: Activate Tool #1 5-21
Block 5: Move to XY Starting Position with Recalled Values 5-21
Block 6: Manual Move to Z Start Height 5-21
Block 7: Manual Move to Z Depth of Cut 5-21
Block 8: Linear Feed with Recalled Values 5-22
Block 9: Cw Arc Move with Recalled Values 5-22
Block 10: Linear Feed with Recalled Values 5-22
Block 11: Cw Arc Move with Recalled Values 5-23
Block 12: Manual Move to Z0 5-24
Block 13: Cancel Tool Compensation 5-24
Block 14: Program EndMain 5-24
Quiz 5 5-25

Section 6 - Sample Programs and Practice Exercises

Sample Program #1 - Drill Program 6-2
Sample Program #2 - Rectangular Pocket Program 6-3
Sample Program #3 - Irregular Pocket Program 6-4
Sample Program #4 - Irregular Pocket Program 6-6
Sample Program #5 - Program with Looping Subprogram 6-7
Sample Program #6 - Program with Rotating Subprogram 6-9
Using the Practice Exercises 6-10
Starting Practice Exercises #1 - #5 6-10
Practice Exercise #1 - Creating a Program Named SP1.M 6-11
Practice Exercise #2 - Activating Edit to Program SP1.M 6-11
Practice Exercise #3 - Writing the Blocks in SP1.M 6-12
Practice Exercise #4 - Entering SP1.M Tooling on the Tool Page 6-16
Practice Exercise #5 - Running SP1.M in Draw 6-16
Starting Practice Exercises #6 - #8 6-18
Practice Exercise #6 - Creating a New Program SP4.M 6-19
Practice Exercise #7 - Finding SP4.M Positions with Geometry Calculator 6-19
Practice Exercise #8 - Programming SP4.M Using Calculator Positions 6-24
Step 1 - Define Irregular Outline by Writing Subprogram 6-25
Step 2 - Check Subprogram Using Draw 6-28
Step 3 - Writing the Main Program 6-29
Step 4 - Viewing the Finished Program with Draw 6-32

Index Index-1

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.

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 2-axis mills.

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

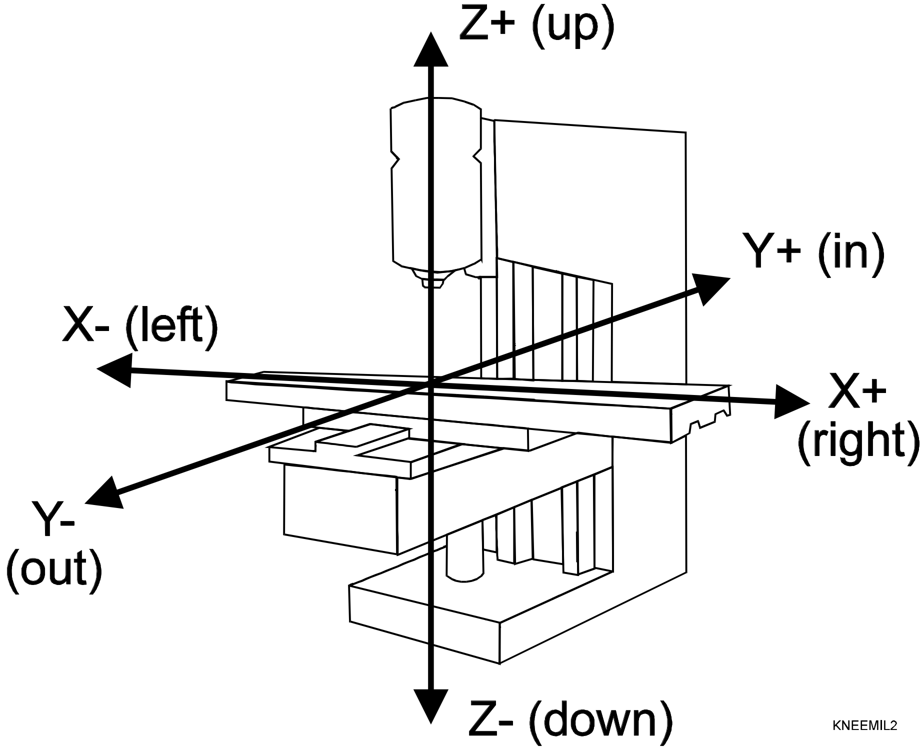


Figure 1-1, Mill Axes of Motion (Tool Motion Orientation)

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

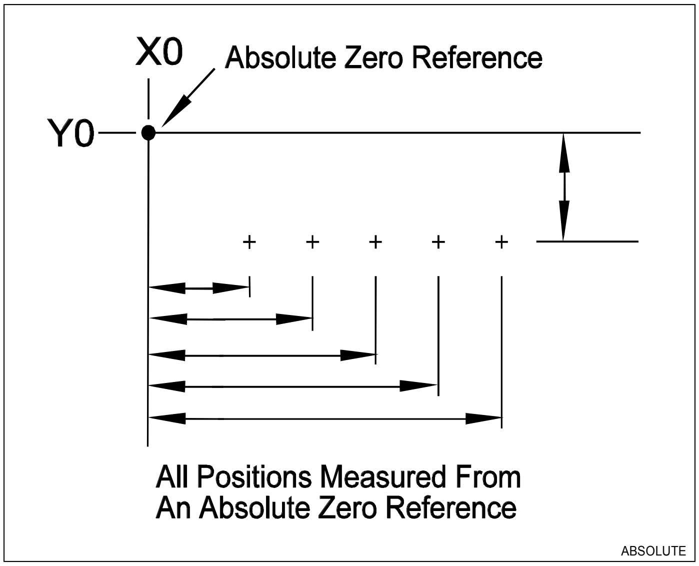

X0
Absolute Zero Reference
Y0
All Positions Measured From
An Absolute Zero Reference
ABSOLUTE

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

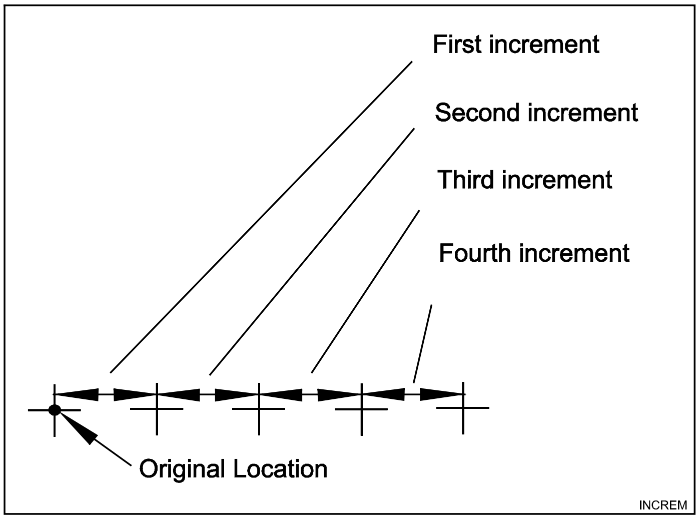

First increment
Second increment
Third increment
Fourth increment
Original Location
INCREM

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.

> **NOTE:** An incremental "0" move commands a move to the current position; thus there is no motion.

Defining Positions

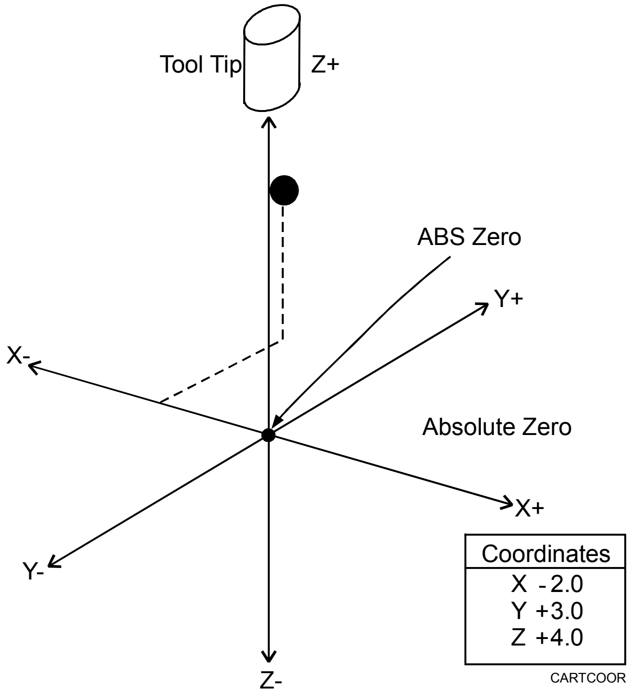


Figure 1-4, Locating Positions

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.

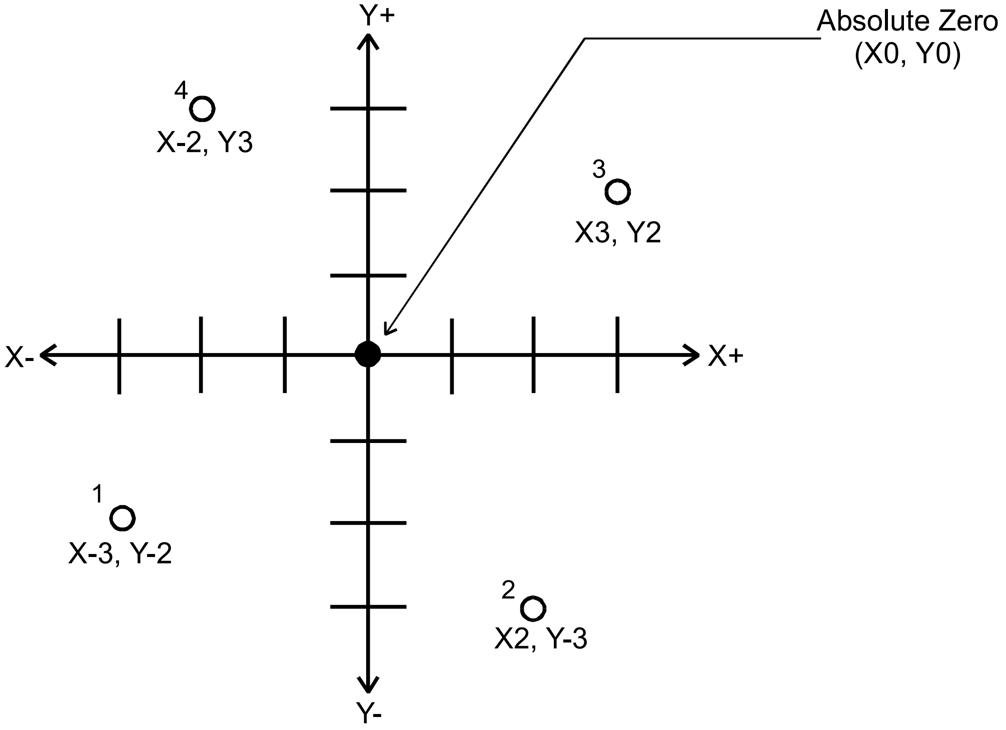


Figure 1-5, Cartesian Coordinates

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

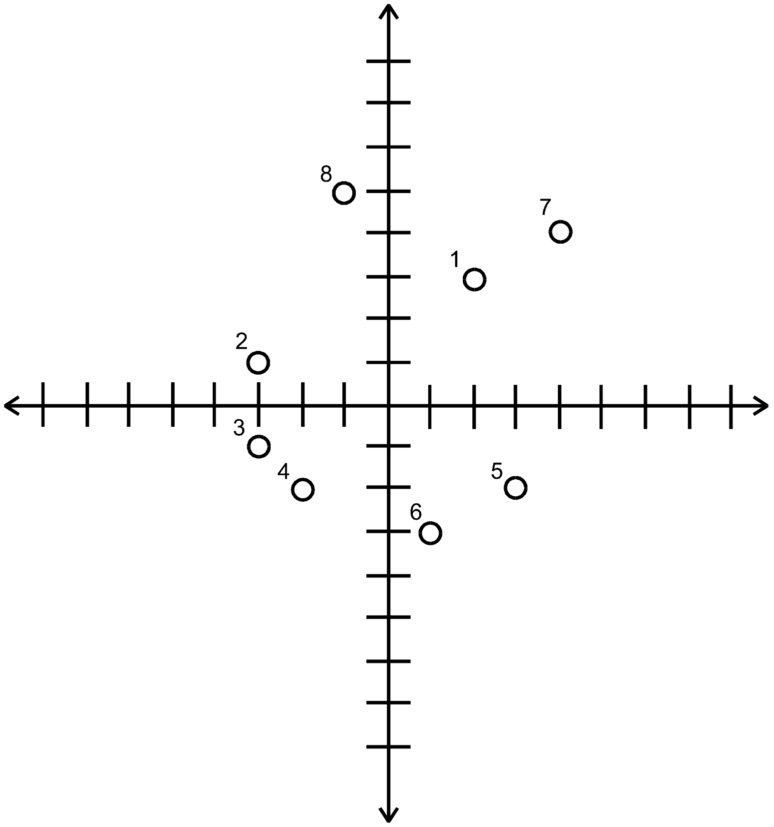


Figure 1-6, Quiz 1 Coordinates

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

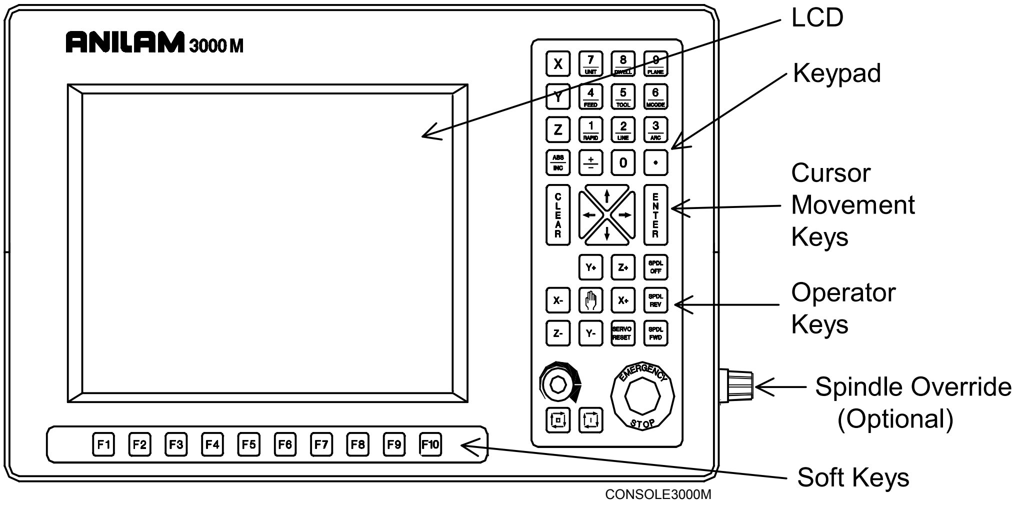

ANILAM 3000M
LCD
Keypad
Cursor Movement Keys
Operator Keys
Spindle Override (Optional)
Soft Keys
F1
F2
F3
F4
F5
F6
F7
F8
F9
F10
CLEAR
ENTER
EMERGENCY STOP
CONSOLE3000M

Figure 2-1, 3000M Console

Console Overview

Refer to **Figure 2-1**. The console consists of the 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" color, flat-panel liquid crystal display (LCD) screen that displays the programming functions and canned cycles.

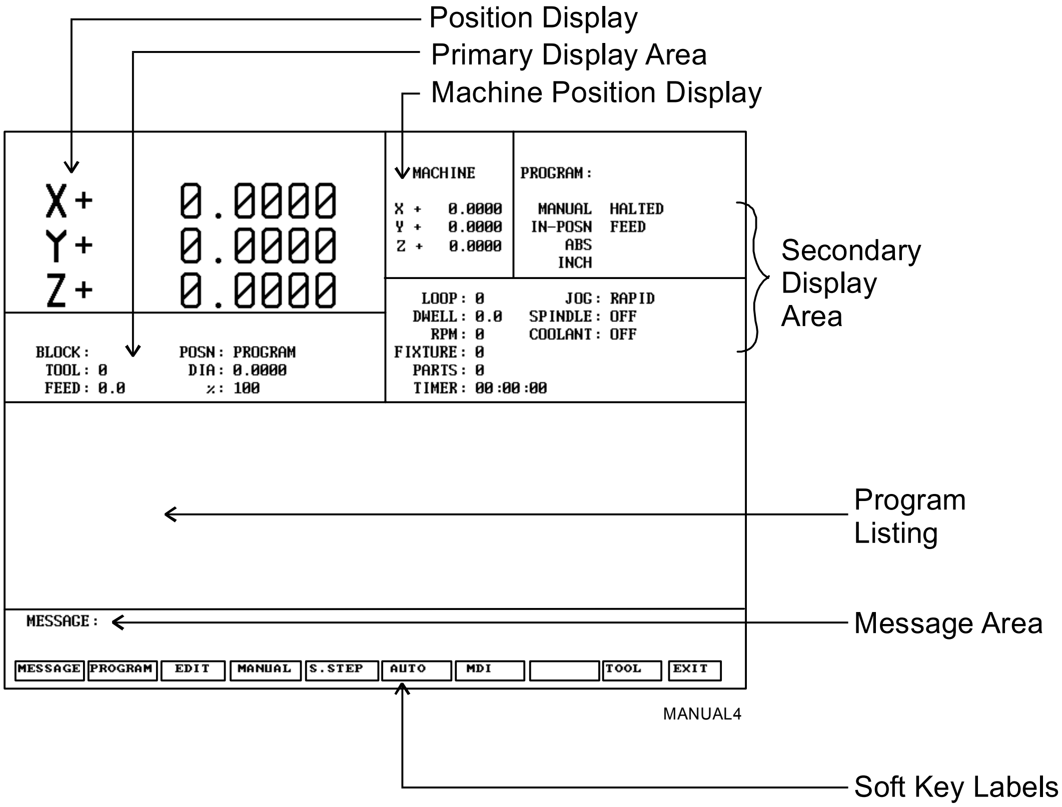
Keyboard Layout

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+, X-, and Y- 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



X+ 0.0000
Y+ 0.0000
Z+ 0.0000

MACHINE
X + 0.0000
Y + 0.0000
Z + 0.0000

PROGRAM:
MANUAL HALTED
IN-POSN FEED
ABS
INCH

LOOP: 0
JOG: RAPID
DWELL: 0.0
SPINDLE: OFF
RPM: 0
COOLANT: OFF
FIXTURE: 0
PARTS: 0
TIMER: 00:00:00

BLOCK:
TOOL: 0
FEED: 0.0
POSN: PROGRAM
DIA: 0.0000
%: 100

MESSAGE:

Position Display
Primary Display Area
Machine Position Display
Secondary Display Area
Program Listing
Message Area
MESSAGE
PROGRAM
EDIT
MANUAL
S.STEP
AUTO
MDI
TOOL
EXIT
MANUAL4
Soft Key Labels

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

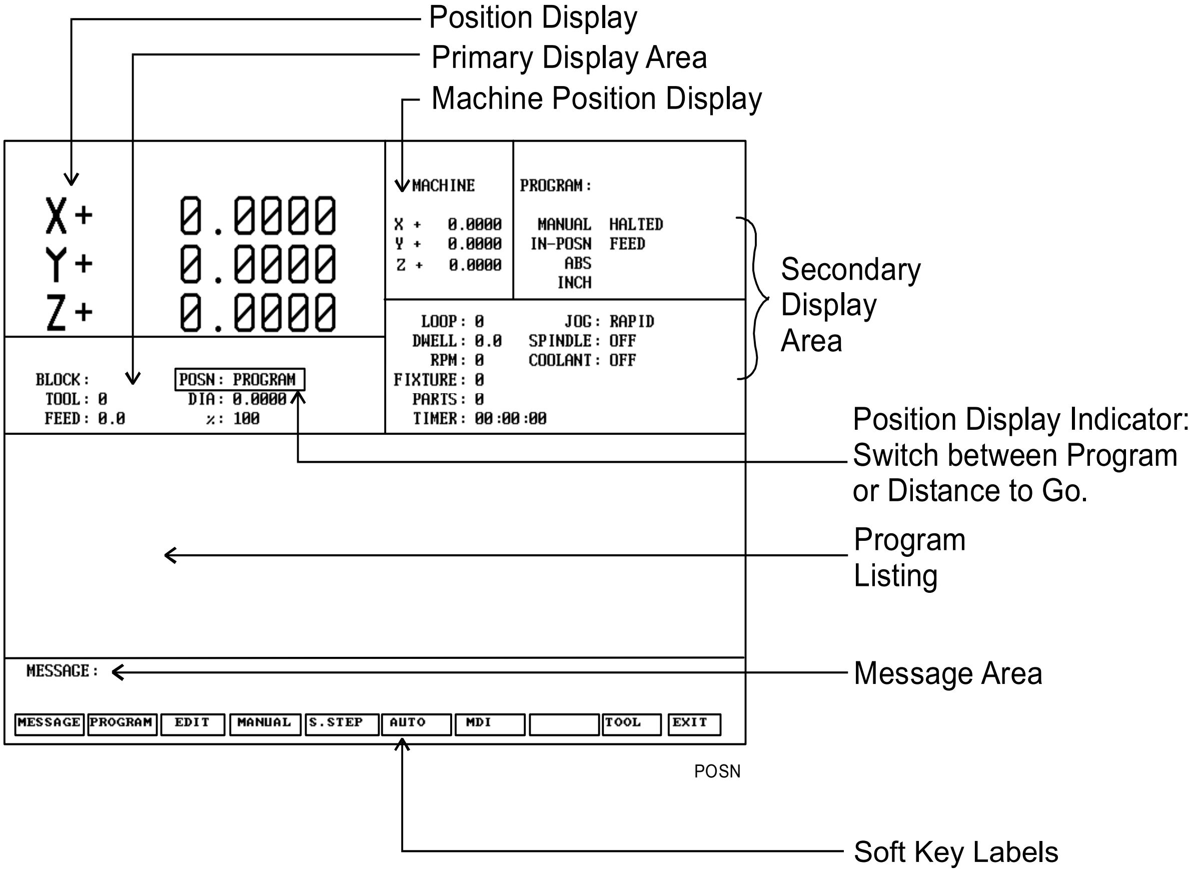
BLOCK:	Current program block number.
TOOL:	Active tool.
FEED:	Current feedrate.
POSN:	Position Display Mode (Program or Distance to Go).
DIA:	Active tool diameter.
%:	Feedrate override setting (0% to 120% for Feed moves; 0% to 100% for Rapid moves).

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



```
X+   0.0000    MACHINE         PROGRAM:
Y+   0.0000    X +  0.0000       MANUAL  HALTED
Z+   0.0000    Y +  0.0000     IN-POSN  FEED
               Z +  0.0000         ABS
                                  INCH
                  LOOP: 0          JOG: RAPID
                 DWELL: 0.0    SPINDLE: OFF
                   RPM: 0      COOLANT: OFF
BLOCK:       POSN: PROGRAM   FIXTURE: 0
 TOOL: 0      DIA: 0.0000      PARTS: 0
 FEED: 0.0      %: 100         TIMER: 00:00:00

MESSAGE:
```

Position Display
Primary Display Area
Machine Position Display
Secondary Display Area
Position Display Indicator: Switch between Program or Distance to Go.
Program Listing
Message Area
MESSAGE
PROGRAM
EDIT
MANUAL
S.STEP
AUTO
MDI
TOOL
EXIT
POSN
Soft Key Labels

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.

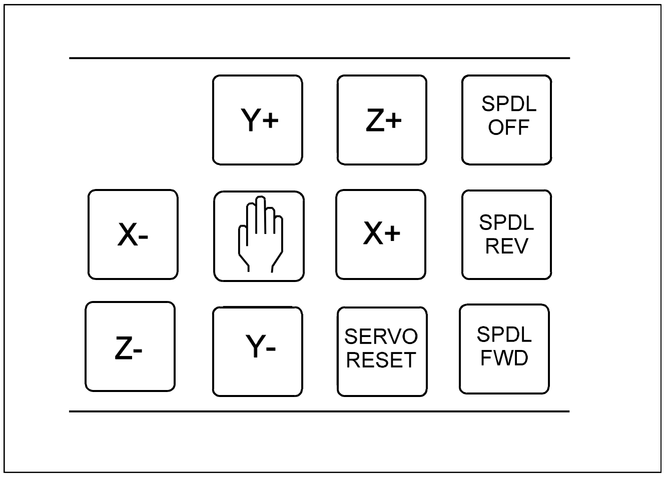


Figure 2-4, Manual Operation Keys

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, Move Mode Selections. 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.
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

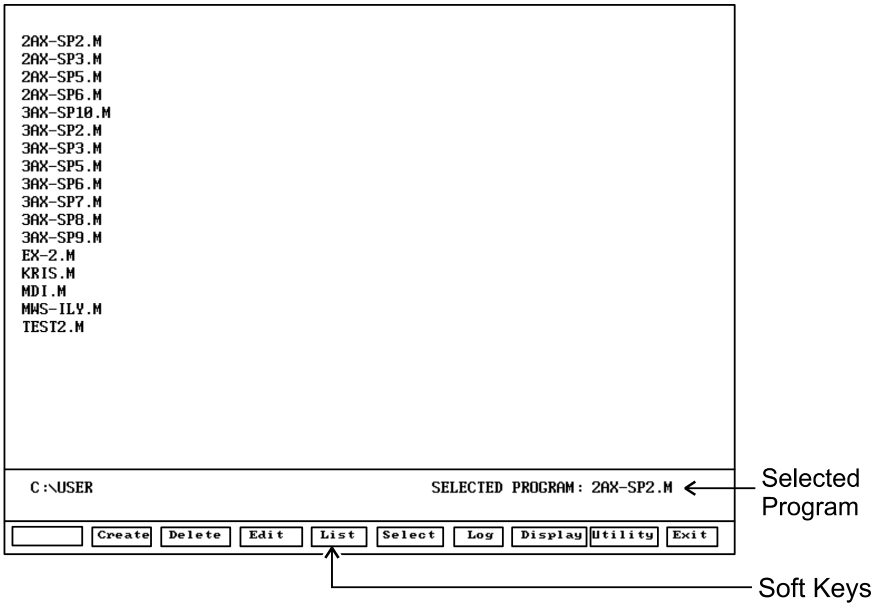
Manual Soft Keys

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



```
2AX-SP2.M
2AX-SP3.M
2AX-SP5.M
2AX-SP6.M
3AX-SP10.M
3AX-SP2.M
3AX-SP3.M
3AX-SP5.M
3AX-SP6.M
3AX-SP7.M
3AX-SP8.M
3AX-SP9.M
EX-2.M
KRIS.M
MDI.M
MWS-ILY.M
TEST2.M

C:\USER                                    SELECTED PROGRAM: 2AX-SP2.M
```

PROGRAM

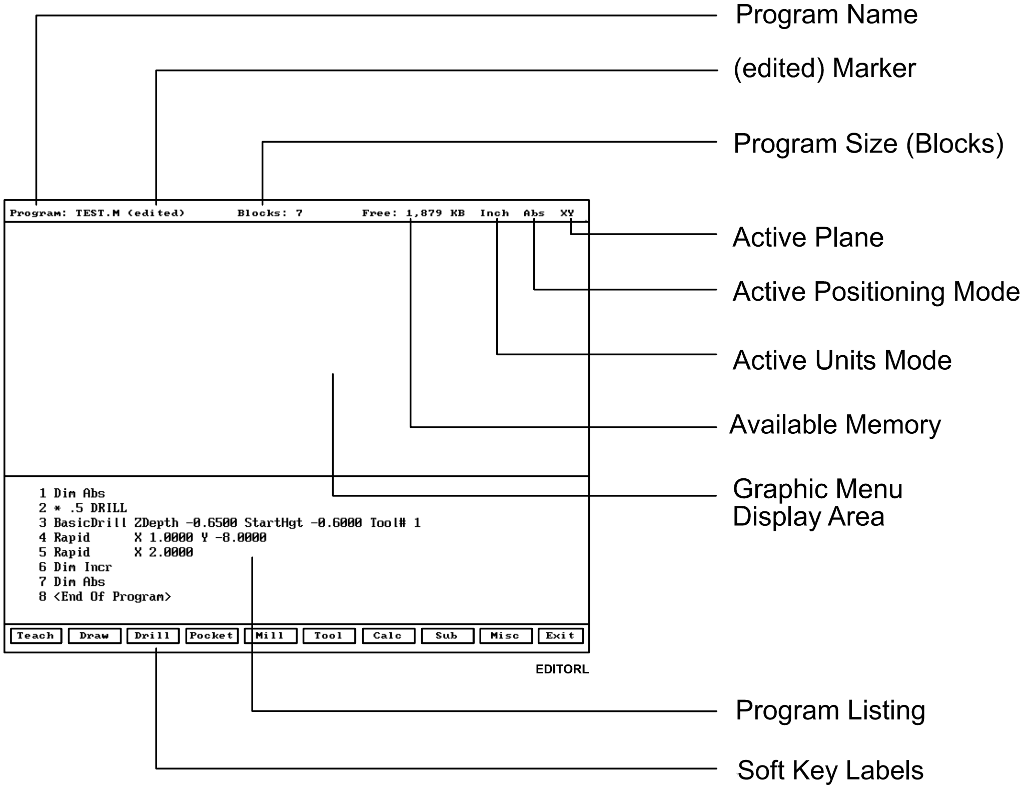
Figure 2-5, Program Directory

NOTE: Refer to *3000M CNC Programming and Operations Manual for Two-Axis Systems*, P/N 70000496, “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


Program Name
(edited) Marker
Program Size (Blocks)
Active Plane
Active Positioning Mode
Active Units Mode
Available Memory
Graphic Menu Display Area
Program Listing
Soft Key Labels
Teach
Draw
Drill
Pocket
Mill
Tool
Calc
Sub
Misc
Exit
EDITORL

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

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

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 Two-Axis Systems*, P/N 70000496, and "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.
Tool (**F6**)	Activates the Tool Page.

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

Key	Function
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 LCD. 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-** 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

2-Axis CNC Operator's Role

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

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

Operator and CNC roles and restrictions for running programs are as follows:

- ☐ The CNC will hold and prompt, **"POSITION Z AXIS AND PRESS START."** whenever it encounters a Z-axis command.
- ☐ In cycles that use a programmed starting height (StartHgt) and cutting depth (DepthCut), the control calculates the number of passes needed to peck from the StartHgt to the pocket depth (ZDepth) when incrementing the DepthCut on each pass. The CNC will hold and prompt, **"POSITION Z AXIS AND PRESS START."** on each pass.
- ☐ In cycles that use a programmed finish stock (FinStock), the control calculates the number of passes required to peck to the pocket depth (ZDepth) and leave enough stock for a finish pass. Leave finish stock on the pass before the finish pass and be sure to accurately position the Z axis on the finish pass.
- ☐ The CNC will hold and prompt, **"PREPARE FOR RAPID MOVE AND PRESS START."** every time the CNC changes from Rapid to Feed. This enables you to verify the Z-axis position before feeding into the part. This hold also occurs in canned cycles.
- ☐ The CNC will hold and prompt, **"PREPARE FOR RAPID MOVE AND PRESS START."** every time the CNC changes from Feed to Rapid. During this hold, verify the clearance before making a rapid move to another location. This hold also occurs in canned cycles.
- ☐ The CNC will hold and prompt, **"Program stopped on tool mount, press START."** at the very start of every block that activates a tool. This hold enables you to verify the Z position (and length of the new tool) before executing the block.

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

The Part

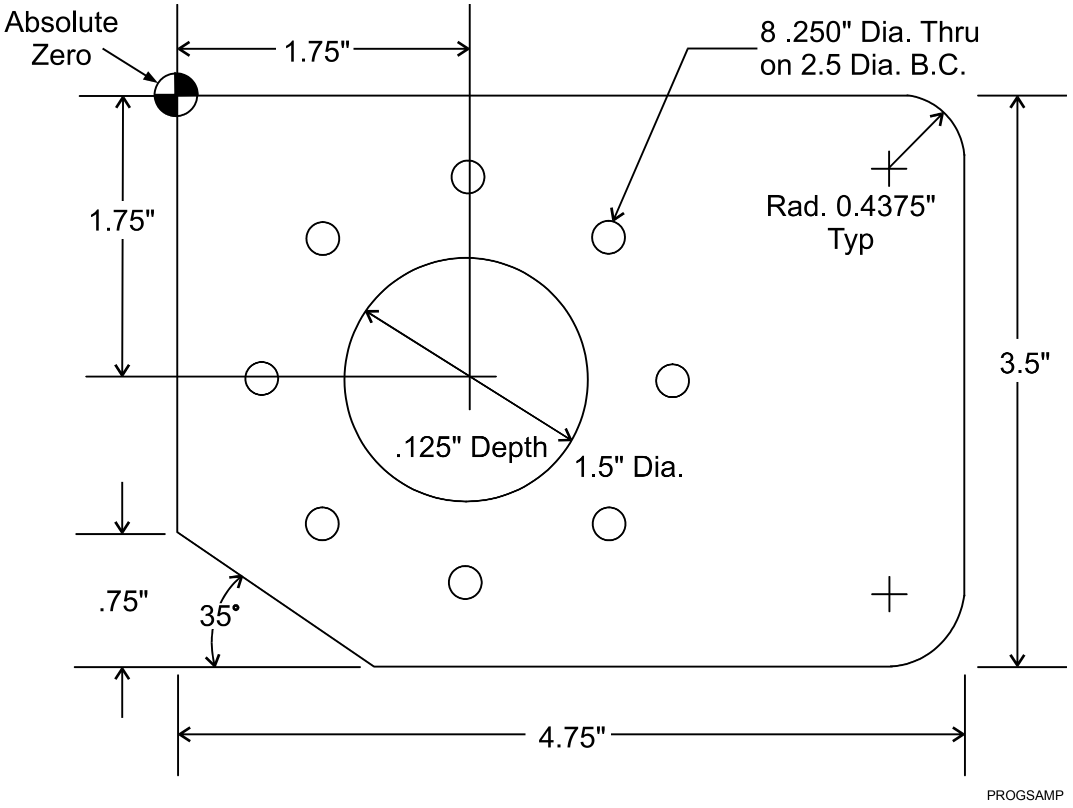

Absolute
Zero
1.75"
8 .250" Dia. Thru
on 2.5 Dia. B.C.
1.75"
Rad. 0.4375"
Typ
3.5"
.125" Depth
1.5" Dia.
.75"
35°
4.75"
PROGSAMP

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	BasicDrill ZDepth -0.300 Tool#1	Activate Basic Drill. Activate Tool #1 (1/4" drill).
3	DrillBHole XCenter 1.7500 YCenter -1.7500 #Holes 8 Diameter 2.5000 StartAngle 0.0000	Activate Bolt Hole Pattern Cycle.
4	DrillOff	Turn off Drill Cycle.
5	ZMove 0.0000	Manually retract Z.
6	Rapid X-1.0000 Y 0.0000 Tool#0	Rapid move. Cancel Tool#1.
7	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 Tool#2	Activate Circular Pocket Cycle. Activate Tool #2 (7/16" flute end mill). Position Z manually as required.
8	Rapid X -0.5000 Y 0.0000 ToolComp Left	Rapid off the edge of the workpiece and turn on Tool Compensation Left.
9	ZMove -0.1300	Manually position Z to starting depth.
10	Line X 4.7500 CornerRad 0.4375 Feed 14.0	Feed in X. Activate Corner Rounding.
11	Line Y -3.5000 CornerRad 0.4375	Feed in Y. Activate Corner Rounding.
12	Line X 1.0711	Feed move. (Recall Right Triangle Calculator value.)
13	Line X 0.0000 Y -2.7500	Feed move.
14	Line Y 0.5000	Feed move.
15	Line X -0.5000 ToolComp Off	Feed in X. Cancel Tool Compensation.
16	ZMove 0.0000	Manually retract Z to 0.0000 (tool change position).
17	Rapid X -1.0000 Y 0.0000 Tool# 0	Move away from the work. Cancel Tool #2.
18	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: Drilling Canned Cycles

NOTE: Do not use a Basic Drill command if the Z-axis on your machine is not a Digital Readout. Instead, perform a manual Z move to the required depth.

Format: BasicDrill ZDepth -0.3000 Tool# 1

A Basic Drill Cycle is similar to a Manual Z move. When executed, the CNC holds the program and prompts you to move the Z-axis. The ZDepth is only entered to remind you how deep to drill and does not actually initiate a Z move.

To program the appropriate Basic Drill Cycle:

1. In Edit Mode, press **Drill** (**F3**). The Drill Pop-Up Menu appears.
2. Highlight **Basic**. Press **ENTER**. The Basic Drilling Graphic Menu is displayed.
3. All entry fields are optional. Fill in the following values:

ZDepth	-0.3 (drill through plate)
Tool#	1

NOTE: Tool #1 is a 1/4” drill.

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

Block 3: Bolt Hole Pattern Canned Cycle

Format: DrillBHole XCenter 1.7500 YCenter -1.75 #Holes 8 Diameter 2.5000 StartAngle 0.0000

Use a Bolt Hole Canned Cycle to instruct 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 Basic Drilling Graphic Menu is displayed.

3. Enter 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)
End Angle	Unnecessary.
Index Angle	Unnecessary.
Tool#	Unnecessary, previously activated Tool #1.

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

Block 4: 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 appears.
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 5: Clear the Part (Manual Z Move)

Format: ZMove 0.0000

Program a Z Move, to be made manually by the operator.

1. Press **Z**. The Z Move Graphic Menu activates.
2. Fill in the following value:

ZMove	0.0000

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

Block 6: Move to the Tool Change Position

Format Rapid X -1.0000 Y 0.0000 Tool#0

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

To program this move via the console keypad:

1. Press **1/RAPID**. The Rapid Graphic Menu activates.

2. Fill in the following values:

X	-1
Y	0
Tool#	0

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

Block 7: 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
Tool#	2 (activate Tool #2)

NOTE: The program will halt and the CNC will prompt you to adjust the Z-axis several times during this cycle. First, move Z to -0.110 for the rough cut. Second, raise Z for the rapid move. Third, move Z to -0.125 for the finish cut.

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 8: Rapid Move and Tool Compensation

Format Rapid X -0.5000 Y 0.0000 ToolComp Left

Program the rapid move off to the upper left side 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 rapid move and enable Left-of-Path Compensation:

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
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 9: Manual Z Move to Cutting Depth

Format ZMove -0.1300

Program a Z Move block as a reminder of the depth of the cut. The CNC will halt the program when the block is executed and prompt you to move the Z-axis. Manually move Z to the cutting depth. Press **START** to clear the message and resume the program run.

1. Press **Z**. The Z Move Graphic Menu activates.
2. Fill in the following values:

Z	-0.13

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

Block 10: 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 11: 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

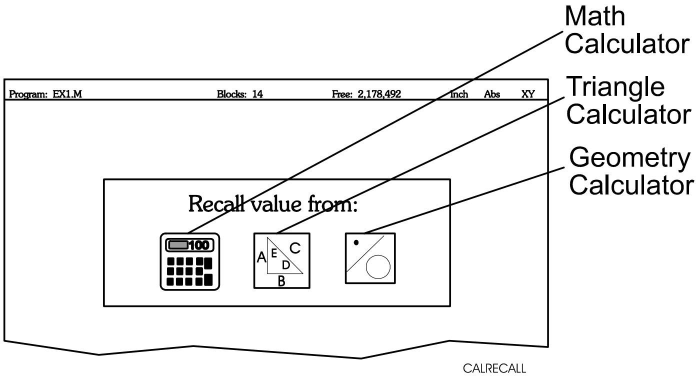

Program: EX1.M
Blocks: 14
Free: 2,178,492
Inch
Abs
XY
Recall value from:
Math Calculator
Triangle Calculator
Geometry Calculator
CALRECALL

Figure 3-2, Recall Selection Pop-up

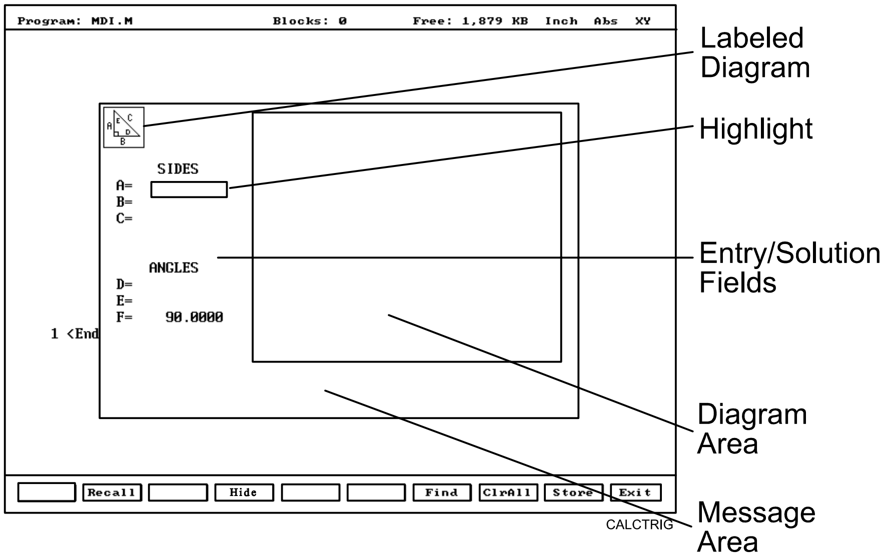
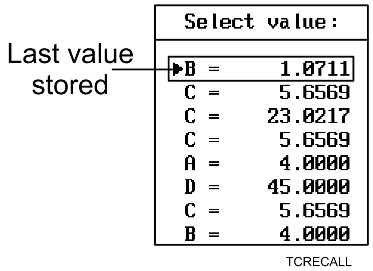

Program: MDI.M
Blocks: 0
Free: 1,879 KB
Inch
Abs
XY
SIDES
A=
B=
C=
ANGLES
D=
E=
F= 90.0000
1 <End
Recall
Hide
Find
ClrAll
Store
Exit
Labeled Diagram
Highlight
Entry/Solution Fields
Diagram Area
Message Area
CALCTRIG

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**.
3. Highlight the **Triangle Calculator** template and press **ENTER**. The Triangle Calculator memory selection pop-up is displayed. Refer to Figure 3-4, Triangle Calculator Recall Pop-up.
4. Highlight the required value and press **ENTER** to copy the stored value to the Graphic Menu.



```
Select value:

▶B =    1.0711
 C =    5.6569
 C =   23.0217
 C =    5.6569
 A =    4.0000
 D =   45.0000
 C =    5.6569
 B =    4.0000
```

Figure 3-4, Triangle Calculator Recall Pop-up

Block 12: Line Move from Right Triangle Calculation

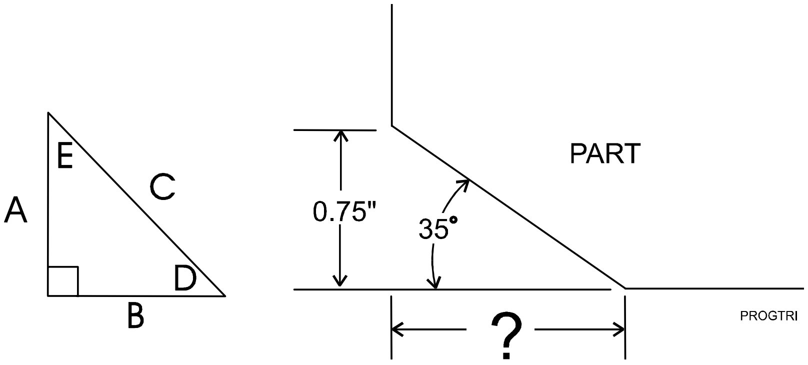


Figure 3-5, Triangle Calculation

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. Enter 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 13: 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 14: 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 15: 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. Enter the following values:

X	-.5
ToolComp	Off

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

Block 16: Manual Return to Z0

Format: ZMove 0.0000

Program a Z Move block as a reminder to manually return to Z0. The CNC will halt the program when the block is executed and prompt you to move the Z-axis. Manually move Z to 0.0000. Press **START** to clear the message and resume the program run.

1. Press **Z**. The Z Move Graphic Menu activates.
2. Enter the following values:

Z	0.0000

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

Block 17: Move Away from the Work and Cancel Tool #2

Format: Rapid X -1.0000 Y 0.0000 Tool# 0

Rapid to the XY position and cancel Tool #2.

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

X	-1
Y	1
Tool#	0

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

Block 18: Program EndMain

Format: EndMain

To add an EndMain block:

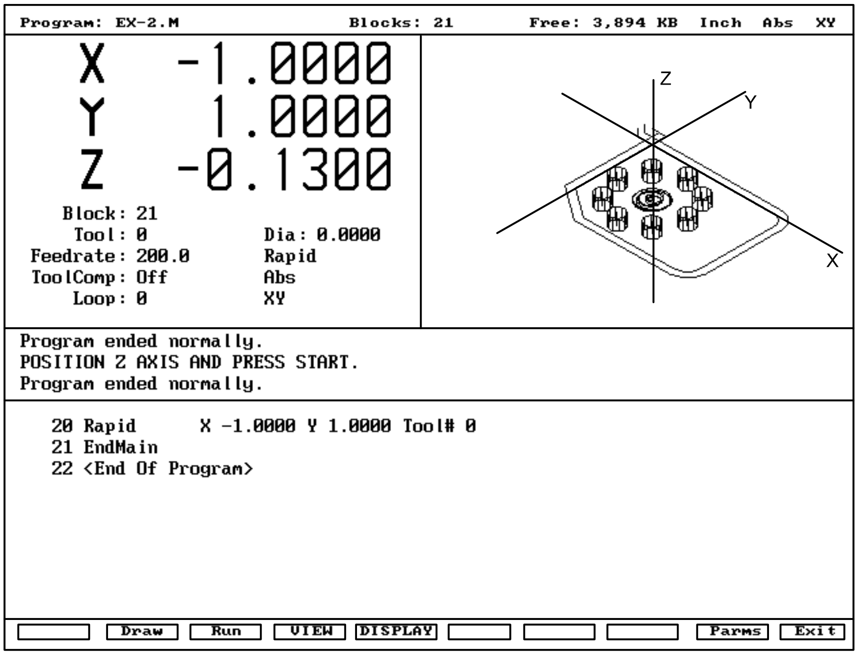
1. Press **Sub** (**F8**). The soft keys change.
2. Press **EndMain** (**F4**). The CNC adds an EndMain block to the program.

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



```
Program: EX-2.M              Blocks: 21       Free: 3,894 KB  Inch  Abs  XY

X  -1.0000
Y   1.0000
Z  -0.1300

     Block: 21
      Tool: 0            Dia: 0.0000
  Feedrate: 200.0        Rapid
  ToolComp: Off          Abs
      Loop: 0            XY

Program ended normally.
POSITION Z AXIS AND PRESS START.
Program ended normally.

   20 Rapid      X -1.0000 Y 1.0000 Tool# 0
   21 EndMain
   22 <End Of Program>
```

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**).

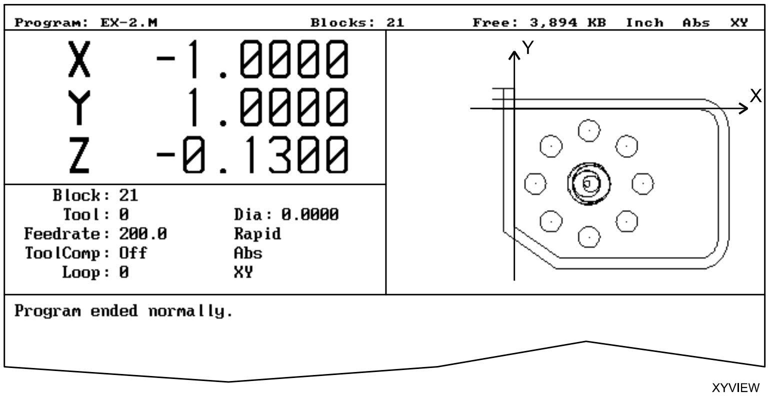
Refer to Figure 3-6, Draw Mode (Iso View) Showing Sample Part. Run the completed program in Draw Graphics (also called Simulation Mode) to verify the moves. Refer to the *3000M CNC Programming and Operations Manual for Two-Axis Systems*, P/N 70000496, "Section 7 - Viewing Programs with Draw" for more 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.


```
Program: EX-2.M                Blocks: 21        Free: 3,894 KB   Inch   Abs   XY

   X   -1.0000
   Y    1.0000
   Z   -0.1300

     Block: 21
      Tool: 0             Dia: 0.0000
  Feedrate: 200.0         Rapid
  ToolComp: Off           Abs
      Loop: 0             XY

Program ended normally.
```

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)
- ☐ **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** 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.

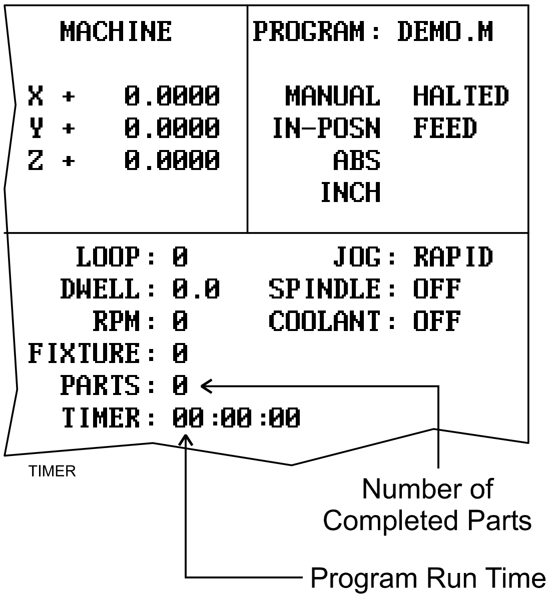
   – or –

   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



```
  MACHINE      PROGRAM: DEMO.M

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

    LOOP: 0           JOG: RAPID
   DWELL: 0.0     SPINDLE: OFF
     RPM: 0       COOLANT: OFF
 FIXTURE: 0
   PARTS: 0
   TIMER: 00:00:00
```

Number of Completed Parts

Program Run Time

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

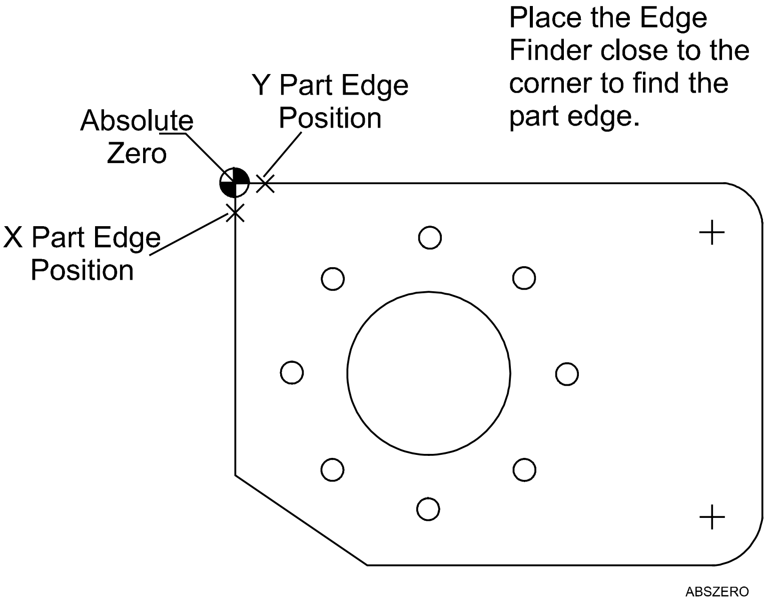

Place the Edge Finder close to the corner to find the part edge.
Y Part Edge Position
Absolute Zero
X Part Edge Position
ABSZERO

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. Manually lower 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.

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.

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. Move the quill down until the tool contacts the work surface lightly.

   The value displayed in the Z Axis Display is the Tool #1 Length Offset. It is the distance from the tip of Tool #1 at Z0 to the top of the part when Tool #1 is in the spindle. 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.
6. Enter the Tool Page. Tool #1 (row 1) is already highlighted. 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. Slowly move Z- closer and closer to the work surface until the tip of the tool touches the work surface.
9. 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 Z axis readout will be referenced to the point where the tip of the tool meets the work surface.
10. 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.

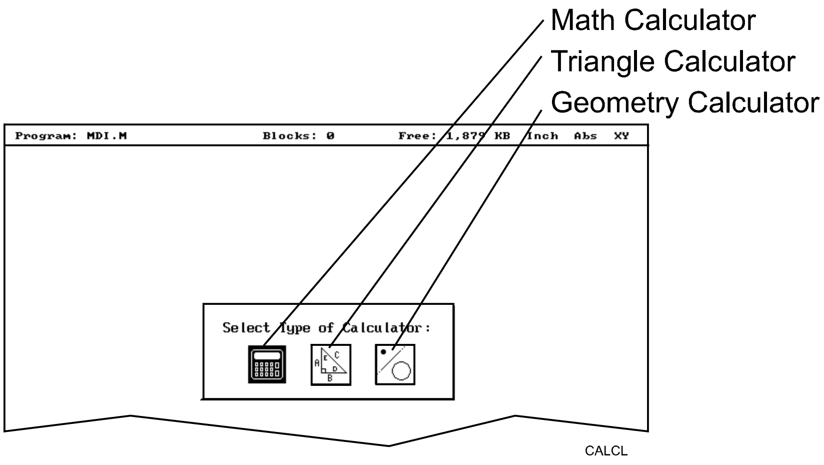

Math Calculator
Triangle Calculator
Geometry Calculator
Program: MDI.M
Blocks: 0
Free: 1,879 KB
Inch
Abs
XY
Select Type of Calculator:
CALCL

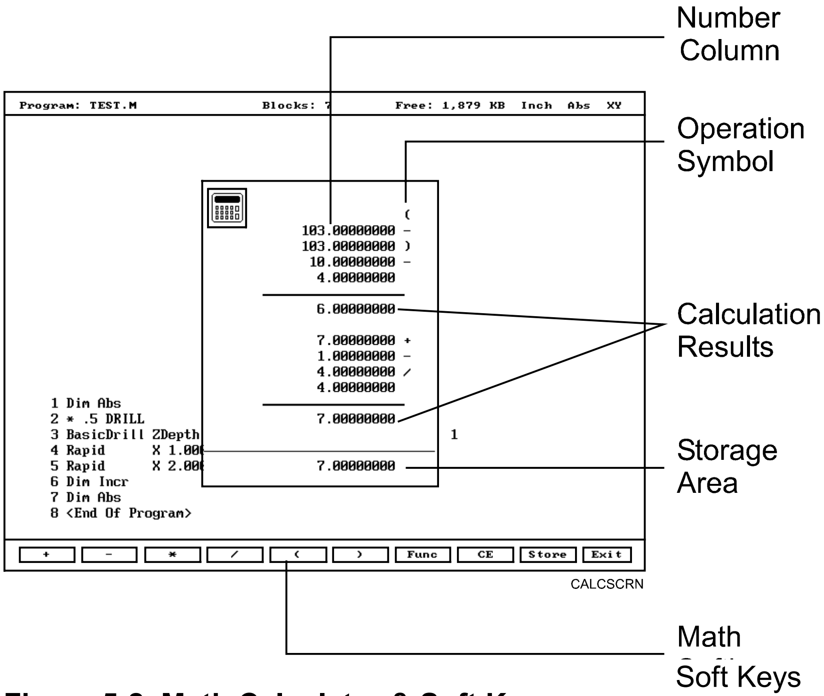
Figure 5-1, Calculator Selection Menu

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



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

                                                 (
                                   103.00000000 -
                                   103.00000000 )
                                    10.00000000 -
                                     4.00000000
                                   ______________
                                     6.00000000

                                     7.00000000 +
                                     1.00000000 -
                                     4.00000000 /
                                     4.00000000
                                   ______________
                                     7.00000000
 1 Dim Abs
 2 * .5 DRILL
 3 BasicDrill ZDepth                                  1
 4 Rapid      X 1.00
 5 Rapid      X 2.00             7.00000000
 6 Dim Incr
 7 Dim Abs
 8 <End Of Program>
```

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.

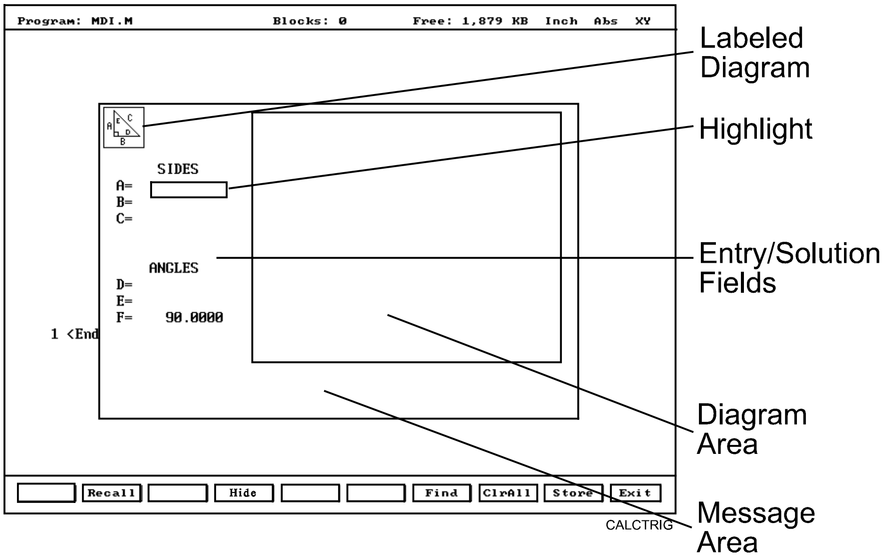
To use the Right Triangle Calculator, type the known values: any two angles, any two sides or one angle and one side.

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.



```
Program: MDI.M          Blocks: 0     Free: 1,879 KB  Inch  Abs  XY

SIDES
A=
B=
C=

ANGLES
D=
E=
F=    90.0000

1 <End
```

A
B
C
D
E
Recall
Hide
Find
ClrAll
Store
Exit
Labeled Diagram
Highlight
Entry/Solution Fields
Diagram Area
Message Area
CALCTRIG

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

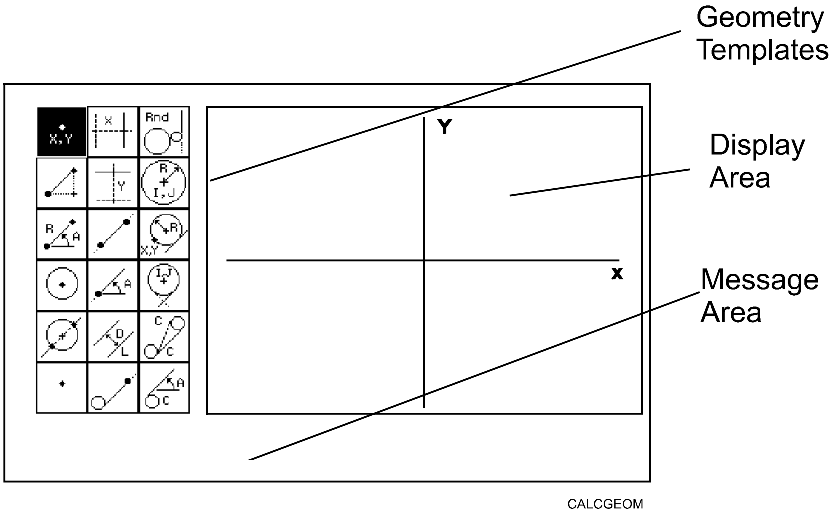


Figure 5-4, Geometry Calculator

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.

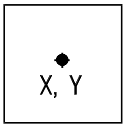
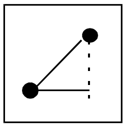
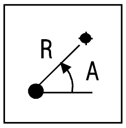
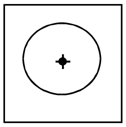
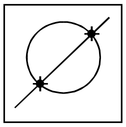
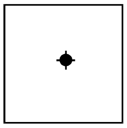
Point Templates

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.

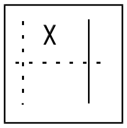
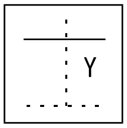
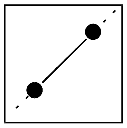
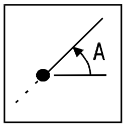
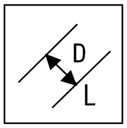
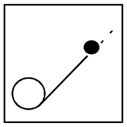
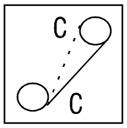
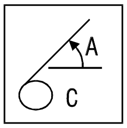
Table 5-3, Point Templates

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

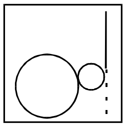
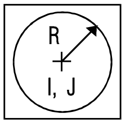
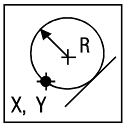
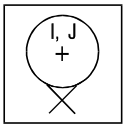
Template	Purpose	Requirements
	Constructs a line parallel to Y-axis, at an X location.	– Must know Absolute X-axis position of the line.
	Constructs a line parallel to X-axis, at a Y location.	– Must know Absolute Y-axis position of the line.
	Constructs a line between any two points. Prompts user to 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.

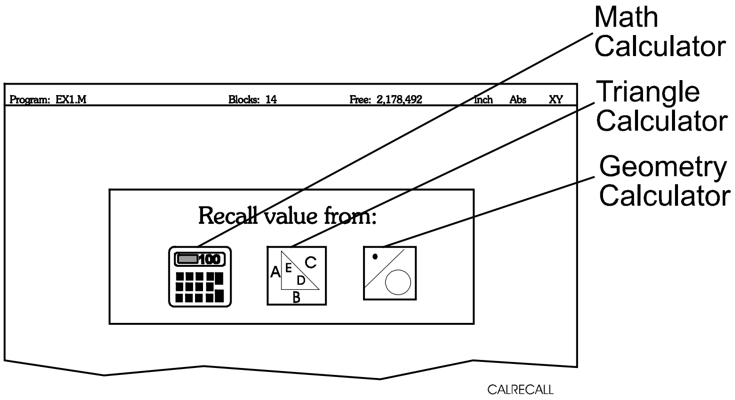
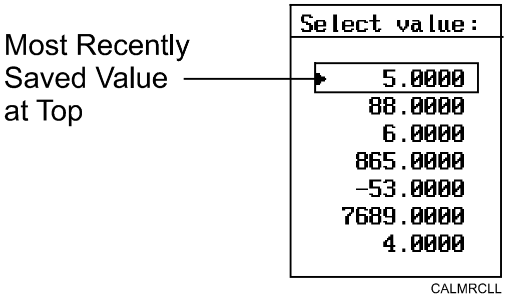

Math Calculator
Triangle Calculator
Geometry Calculator
Program: EX1.M
Blocks: 14
Free: 2,178,492
Inch
Abs
XY
Recall value from:
100
A
E
C
D
B
CALRECALL

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.


Most Recently Saved Value at Top
CALMRCLL

```
Select value:

     5.0000
    88.0000
     6.0000
   865.0000
   -53.0000
  7689.0000
     4.0000
```

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

Recalling Values from the Right Triangle Calculator

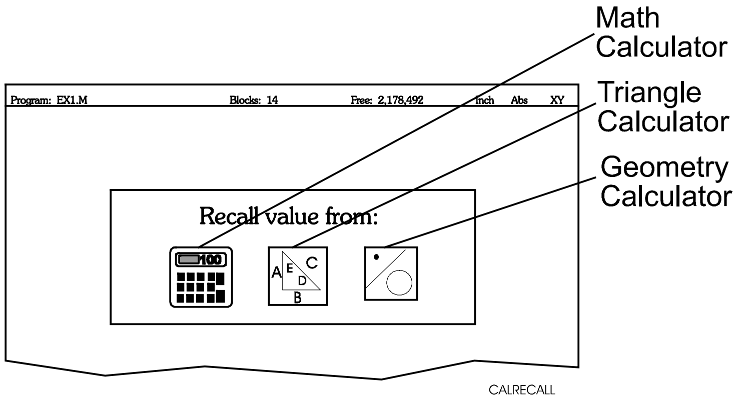
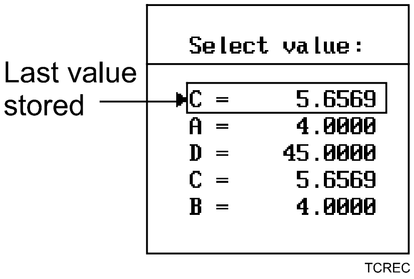

Math Calculator
Triangle Calculator
Geometry Calculator
Program: EX1.M
Blocks: 14
Free: 2,178,492
Inch
Abs
XY
Recall value from:
100
A
E
C
D
B
CALRECALL

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.


Last value stored
TCREC

```
Select value:

C =     5.6569
A =     4.0000
D =    45.0000
C =     5.6569
B =     4.0000
```

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.

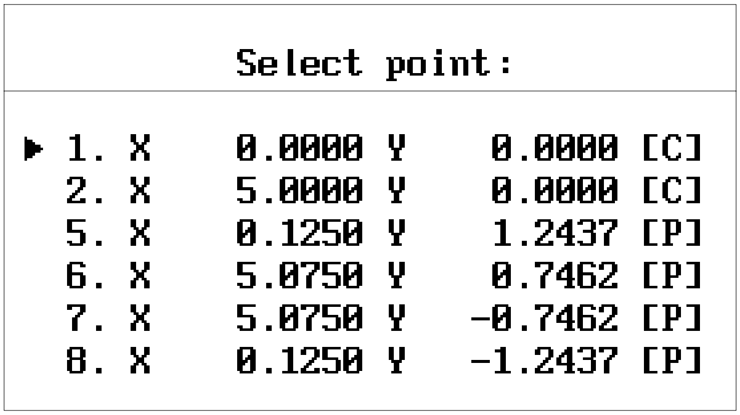
Positions on the recall list followed by **[P]** are points.

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



```
              Select point:

▶ 1. X    0.0000 Y    0.0000 [C]
  2. X    5.0000 Y    0.0000 [C]
  5. X    0.1250 Y    1.2437 [P]
  6. X    5.0750 Y    0.7462 [P]
  7. X    5.0750 Y   -0.7462 [P]
  8. X    0.1250 Y   -1.2437 [P]
```

SEL_PNT

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

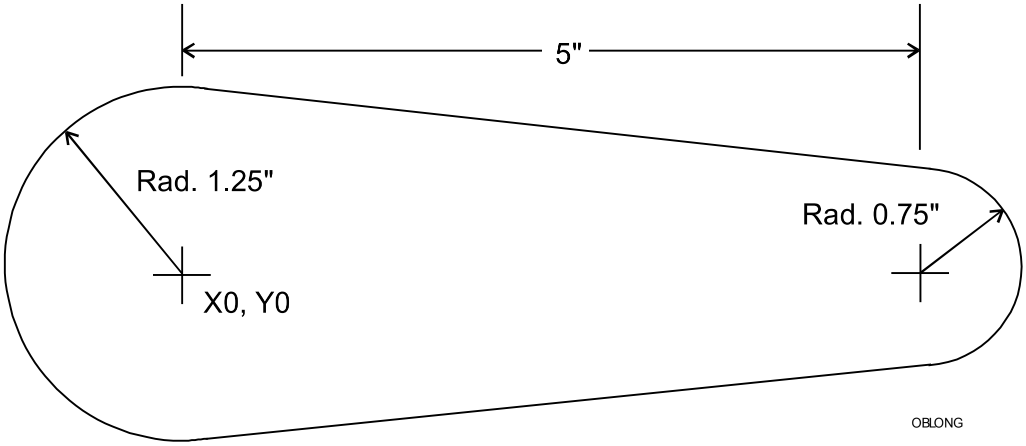

5"
Rad. 1.25"
Rad. 0.75"
X0, Y0
OBLONG

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.

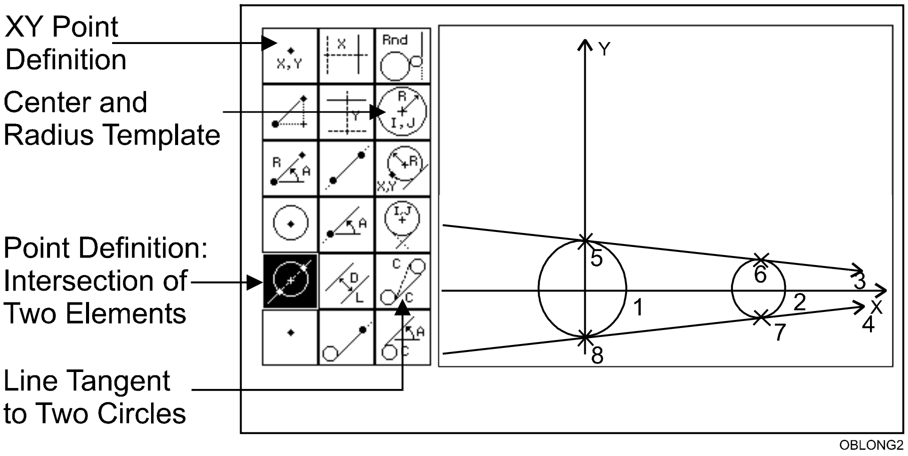
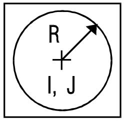
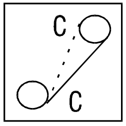
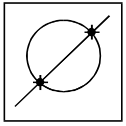

XY Point Definition
Center and Radius Template
Point Definition: Intersection of Two Elements
Line Tangent to Two Circles
X,Y
Rnd
R
I,J
Y
X
5
6
3
1
2
7
4
8
OBLONG2

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	ZMove 0.0000
3	Tool# 0
4	Tool# 1
5	Rapid X 0.1250 Y 1.2437
6	ZMove 0.1000
7	ZMove -0.1000
8	Line X 5.0750 Y 0.7462
9	Arc Cw X 5.0750 Y -0.7462 Radius 0.7500
10	Line X 0.1250 Y -1.2437
11	Arc Cw X 0.1250 Y 1.2437 XCenter 0.0000 YCenter 0.0000
12	ZMove 0.0000
13	Tool# 0
14	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: Manual Z Move

Format: ZMove 0.0000

Program a Z Move that the operator will execute manually.

1. Press **Z**. The Z Move Graphic Menu activates.
2. Enter the following values:

   Z 0.0000
3. Press **Save** (**F10**) to save the block.

Block 3: Cancel Offsets

Format: Tool#0

To cancel offsets:

1. Press **5/TOOL**. The Tool Mount Graphic Menu activates.
2. Enter the following values:
   Tool# 0
3. Press **Save** (**F10**) to save the block.

Block 4: Activate Tool #1

Format: Tool#1

To activate Tool #1:

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

Tool#	1

   The CNC activates Tool #1.
3. Press **Save** (**F10**) to save the block.

Block 5: 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 6: Manual Move to Z Start Height

Format: ZMove 0.1000

Program a manual Z Move to Z Start Height:

1. Press **Z**. The Z Move Graphic Menu activates.
2. Fill in the following values:

Z	0.1000
3. Press **Save** (**F10**) to save the block.

Block 7: Manual Move to Z Depth of Cut

Format: ZMove -0.1000

Program a manual Z Move to Z Depth of Cut:

1. Press **Z**. The Z Move Graphic Menu activates.
2. Fill in the following values:

Z	-0.1000
3. Press **Save** (**F10**) to save the block.

Block 8: 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 9: 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 enter 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 10: 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 11: 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).

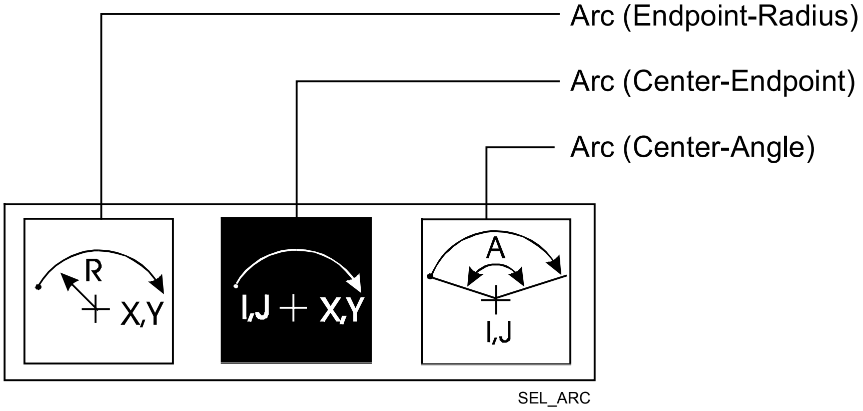


Figure 5-12, Arc Definition Selection Menu

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 12: Manual Move to Z0

Format: ZMove 0.0000

Program a manual Z Move to Z0:

1. Press **Z**. The Z Move Graphic Menu activates.
2. Fill in the following value:

Z	0.0000

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

Block 13: Cancel Tool Compensation

Format: Tool# 0

To cancel tool compensation:

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

Z	0.0000
Tool #	0

3. The CNC cancels all compensation and moves the tool to Z0.

Block 14: 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.

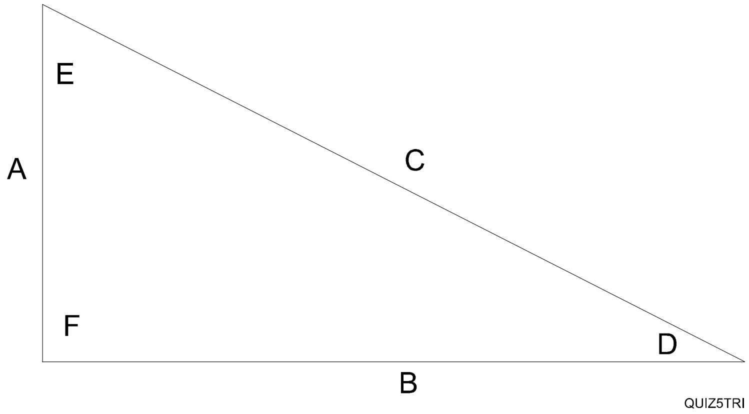


Figure 5-13, Exercise 2 Triangle

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.

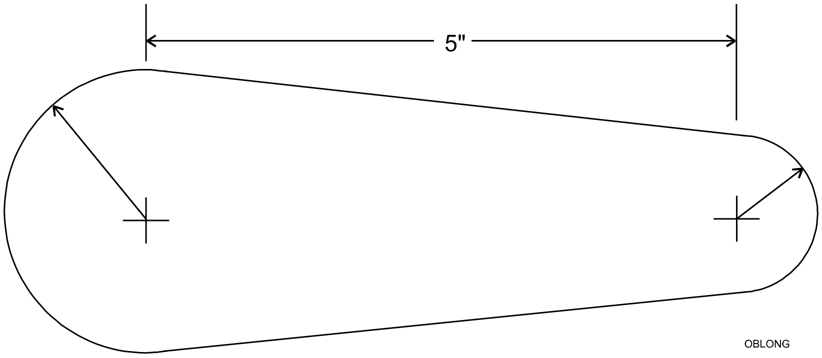


Figure 5-14, Exercise 3 Oblong Slot

Exercise 4:

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

Section 6 - Sample Programs and Practice Exercises

This section of the manual contains sample programs and practice exercises.

First time programmers are encouraged to learn about the CNC by doing the exercises. Experienced operators may sometimes find it useful to look at one of the sample programs.

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

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

NOTE: During the software installation process, the control will prompt the operator to install sample programs. The sample programs installed include the programs from this section that are not demonstrated in the Practice Exercises. The names for 2-axis sample programs begin with "2AX-". The names for three-axis sample programs begin with "3AX-".

Sample Program #1 - Drill Program

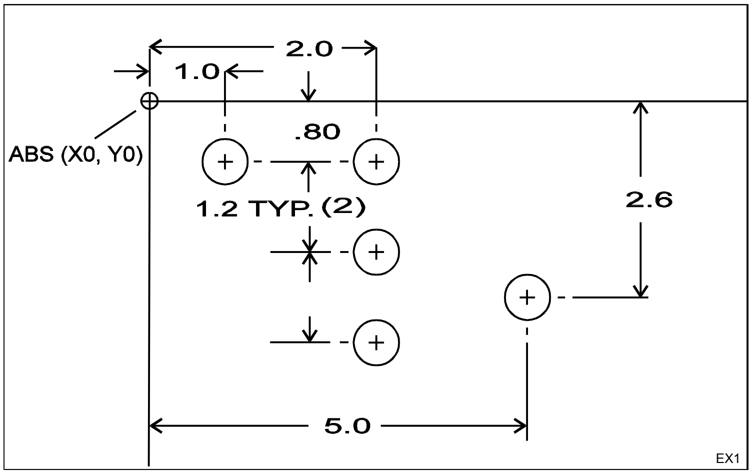


Figure 6-1, Layout for Sample Program #1

Refer to **Figure 6-1**. This program drills 5 holes in a surface, using a 3/8-inch diameter twist drill (Tool #1). The holes are drilled with a Basic Drill Cycle.

Go to the Tool Page and enter a 0.375-inch diameter for Tool #1 before running the program in Draw.

Detailed instructions for creating, programming and viewing this program are outlined in Exercises #1 - #5 of this section.

Sample Program #1 Program Listing

```
Dim Abs
* .375  DRILL
BasicDrill  ZDepth  -0.6500  Tool#  1
Rapid        X  1.0000  Y  -0.8000
             X  2.0000
Dim Incr
             Y  -1.2000
             Y  -1.2000
Dim Abs
             X  5.0000 Y  -2.6000
DrillOff
ZMove  0.0000
             X  0.0000  Y  0.0000
EndMain
<End Of Program>
```

Sample Program #2 - Rectangular Pocket Program

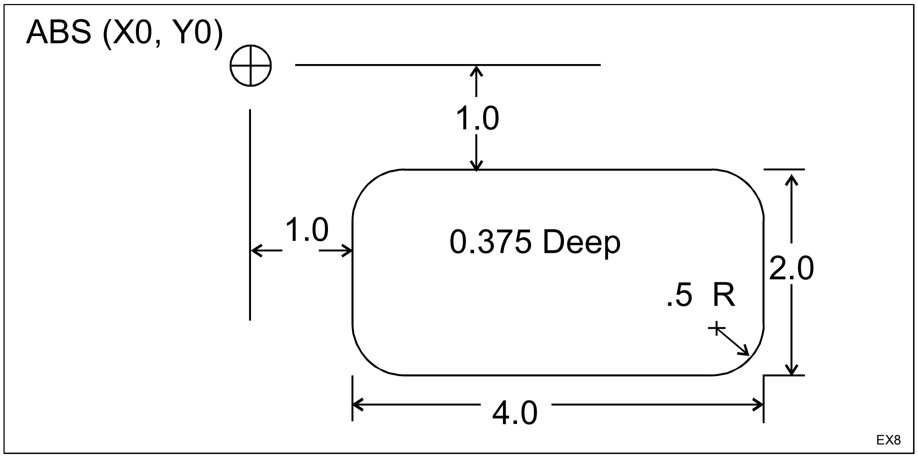


Figure 6-2, Layout for Sample Program #2

Refer to **Figure 6-2**. This program uses a 1/2-inch diameter endmill (Tool #1) to mill out a rectangular pocket. The finished pocket is 3/8-inch deep.

The CNC will cut from the start height to the finished depth in 3 passes. It takes two passes to cut to the finish stock. The finish pass cuts the pocket to the finished depth. The starting height is subtracted before the CNC calculates the number of passes.

Go to the Tool Page and enter a 0.50-inch diameter for tool #1 before running the program in Draw.

Sample Program #2

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

Sample Program #3 - Irregular Pocket Program

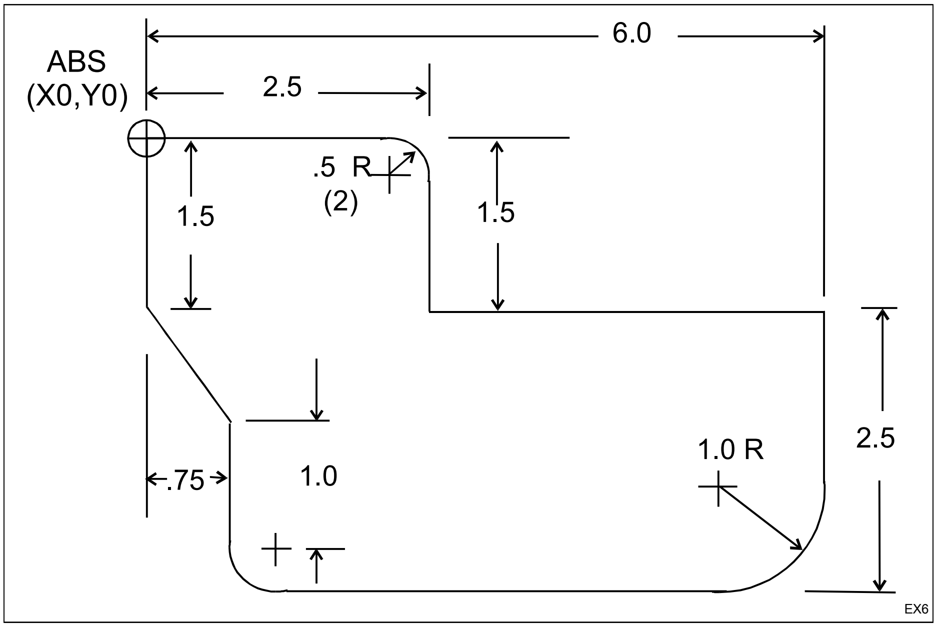


Figure 6-3, Layout For Sample Program #3

Refer to **Figure 6-3**. This program uses a 1/2-inch diameter endmill (Tool #1) to mill out the irregular pocket shown.

The CNC will cut from the start height to the finished depth in 2 passes. One pass to cut to the finish stock and a finish pass to cut to the finished depth. The starting height is subtracted before the CNC calculates the number of passes.

Go to the Tool Page and enter a 0.50-inch diameter for Tool #1 before running 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   Pocket      Sub# 1  StartHgt  0.1000  ZDepth  -0.1250  XStart  1.0000
                YStart  -1.0000  Stepover  0.1500  FinStock  0.0100
                RoughFeed  12.5 FinFeed  9.5  Tool# 1
3   ZMove  0.0000
4   Rapid       X  -2.0000  Y  2.0000
5   EndMain
6   Sub 1
7   Rapid       X  0.0000  Y  0.0000
8   Line        X  2.5000  CornerRad  0.5000
9               Y  -1.5000
10              X  6.0000
11  Dim Incr
12  Line        Y  -2.5000  CornerRad  1.0000
13  Dim Abs
14  Line        X  0.7500  CornerRad  0.5000
15              Y  -2.5000
16              X  0.0000  Y  -1.5000
17              Y  0.0000
18  EndSub
19  <End Of Program>
```

Sample Program #4 - Irregular Pocket Program

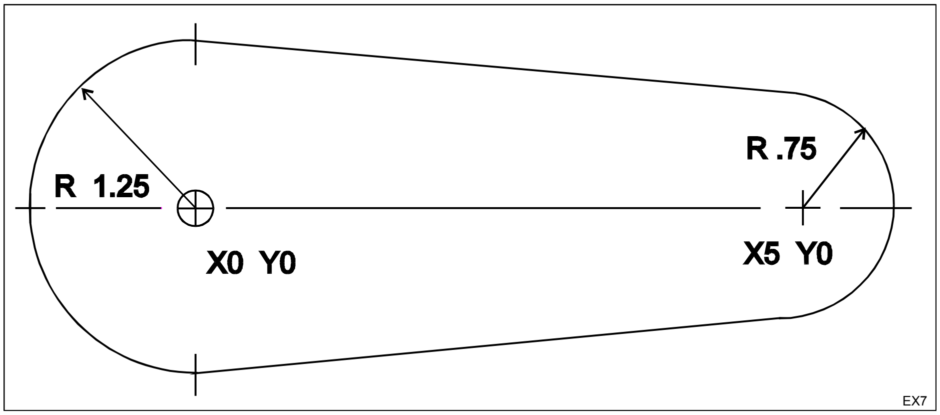


Figure 6-4, Layout For Sample Program #4

Refer to **Figure 6-4**. This program uses a 3/8-inch diameter endmill (Tool #1) to mill out an irregular pocket. Go to the Tool Page and enter a 0.375-inch diameter for Tool #1 before running the program in Draw.

The CNC will cut from the start height to the finished depth in 3 passes. Two passes to cut to the finish stock and a finish pass to cut to the finished depth. The starting height is subtracted before the CNC calculates the number of passes.

The shape of the pocket is defined by the moves in subprogram #99. In this example, the starting position of the first Line (feed) move in the subprogram is the starting position of the first cut. The direction of the first move is forced to 89.9 degrees (0.01 degrees inside of the Arc).

The keystrokes for creating, programming and viewing Sample Program #4 are presented in Exercises #6 - #8 of this section.

Sample Program #4

```
1   Dim Abs
2   Pocket     Sub 99 StartHgt 0.1000 ZDepth -0.2500 Angle 89.9000
    XStart 0.0000 YStart 0.0000 Stepover 0.1600 DepthCut 0.1250
    FinStock 0.0100 RoughFeed 12.5 FinFeed 9.5 Tool# 1
3   ZMove  0.0000
4   Rapid      X  -2.0000  Y  2.0000
5   EndMain
6   Sub 99
7   Rapid      X  -1.2500  Y  0.0000
8   Arc Cw     X  0.1250  Y  1.2437  Radius  1.2500
9   Line       X  5.0750  Y  0.7462
10  Arc Cw     X  5.0750  Y  -0.7462  Radius  0.7500
11  Line       X  0.1250  Y  -1.2437
12  Arc Cw     X  -1.2500  Y  0.0000  Radius  1.2500
13  EndSub
14  <End Of Program>
```

Sample Program #5 - Program with Looping Subprogram

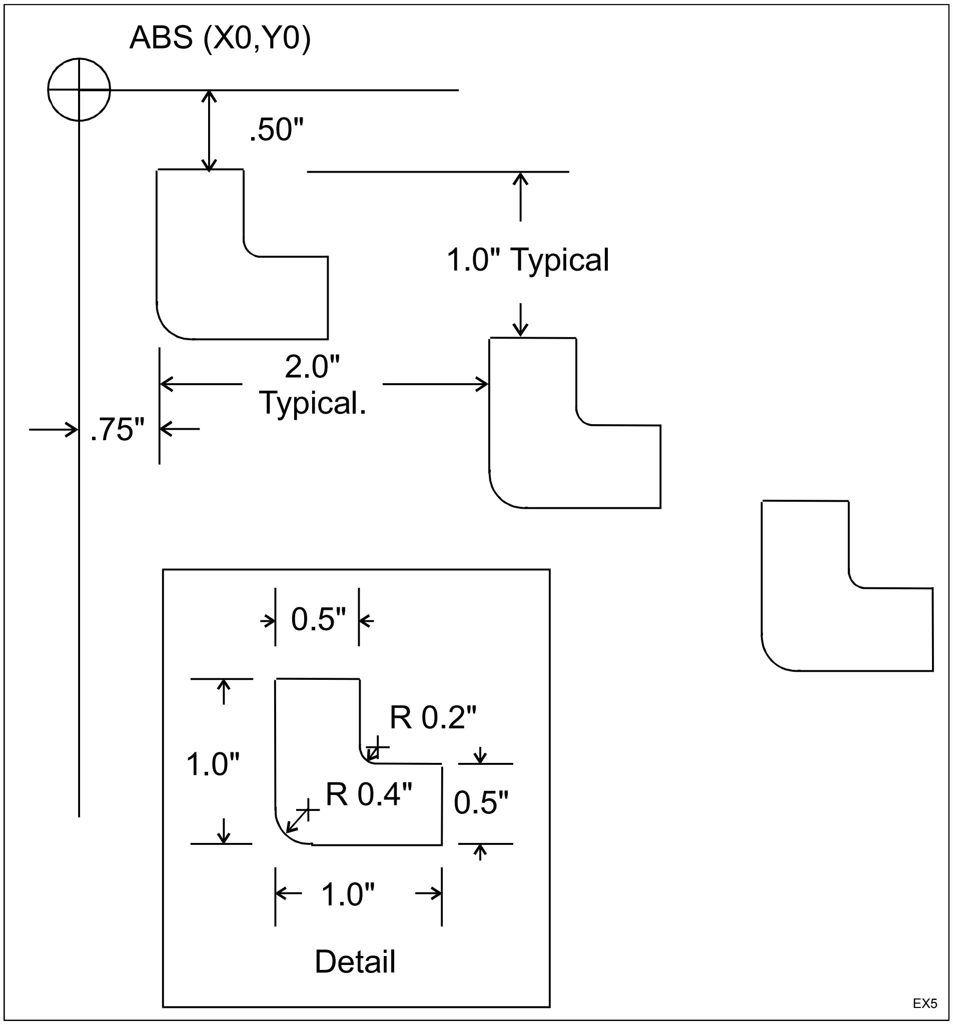


Figure 6-5, Layout for Sample Program #5

Refer to **Figure 6-5**. This program loops subprogram #1 to cut three L shaped pockets. The pockets are cut using a 1/8-inch diameter end mill (Tool#1). Each loop starts at a position within the pocket. The CNC cuts each pocket in one pass.

All of the XY axes moves in the subprogram are written for incremental positioning. This lets the CNC cut the pocket anywhere on the workpiece. Go to the Tool Page and enter a 0.125-inch diameter for tool #1 before running the program in Draw.

Sample Program #5

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

Sample Program #6 - Program with Rotating Subprogram

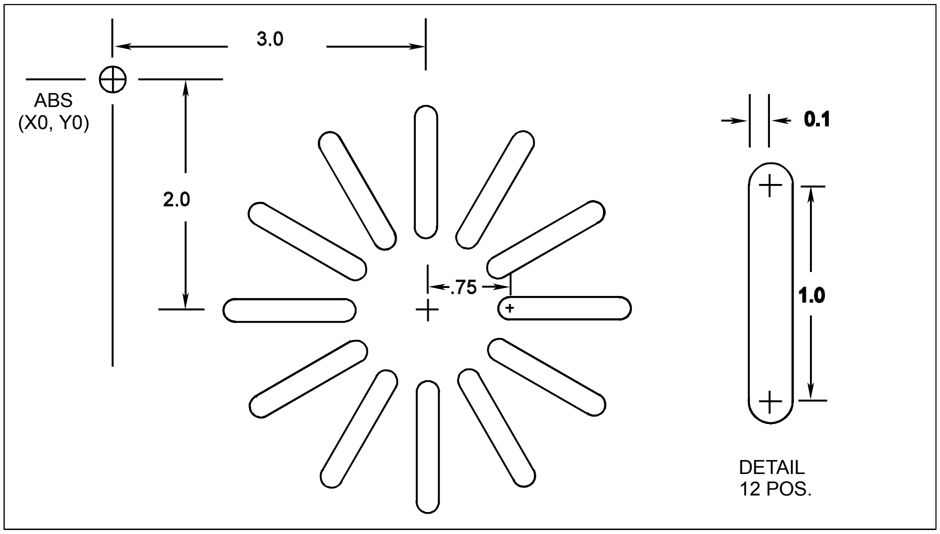


Figure 6-6, Layout for Sample Program #6

Refer to **Figure 6-6**. This program rotates subprogram #1 to cut 12 slots. The slots are cut using a 1/8-inch endmill (Tool #1). The CNC cuts each slot in one pass.

Go to the Tool Page and enter a 0.125-inch diameter for tool #1 before running the program in Draw.

Sample Program #6

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

Using the Practice Exercises

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

Menu selections are made by using the ARROWS to highlight to the required choice. Press **ENTER** to activate a highlighted choice. Toggled selections are changed by pressing the **+/-** key.

A brief overall description of the procedure is provided at the start of each exercise. Be sure to start each exercise from the CNC screen listed in the description.

When an exercise contains more than one table, each table will leave the CNC in the correct screen for starting the next table.

Starting Practice Exercises #1 - #5

Practice Exercises #1 - #5 are a series of related exercises designed to coach a first time programmer through the development of Sample Program #1 - Drill Program. These exercises should be done in order.

Each practice exercise coaches the operator through a different step of the process. Practice Exercise #1 creates the new program. Practice Exercise #2 opens the program for editing. The program blocks are written in Practice Exercise #3. The tooling is entered in the Tool Page in Practice Exercise #4 and the program is checked in Practice Exercise #5.

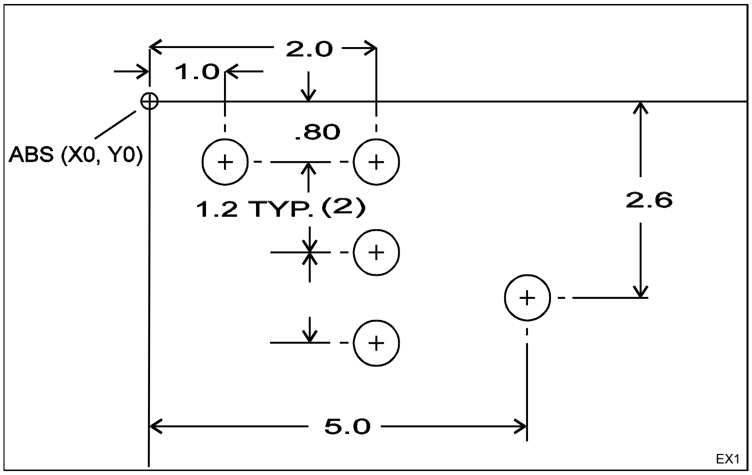


Figure 6-7, Layout for Sample Program #1

Sample Program #1

```
1   Dim Abs
2   * .375  DRILL
3   BasicDrill  ZDepth  -0.6500  Tool#  1
4   Rapid       X  1.0000  Y  -0.8000
5               X  2.0000
6   Dim Incr
7               Y  -1.2000
8               Y  -1.2000
9   Dim Abs
10              X  5.0000 Y  -2.6000
11  DrillOff
12  ZMove  0.0000
13              X  0.0000  Y  0.0000
14  EndMain
15  <End Of Program>
```

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

This exercise outlines the steps required to create a new program named "SP1.M" (Sample Program 1). Before starting, make sure no program named "SP1.M" already exists. Start this exercise from the Manual screen.

Create a new program named "SP1.M".					
1	Activate Program Directory.	**PROGRAM** (**F2**)	6	Highlight required character.	✥ Select **P**
2	Create new program.	**Create** (**F2**)	7	Add character to name.	ENTER
3	Activate ASCII Chart.	**ASCII** (**F2**)	8	Key in 1.	1 RAPID
4	Highlight required character.	✥ Select **S**	9	Close ASCII Chart.	**ASCII** (**F2**)
5	Add character to name.	ENTER	10	Add new program to Program Directory USER list.	ENTER

Practice Exercise #2 - Activating Edit to Program SP1.M

This exercise describes the steps required to activate the editor to write program blocks in program "SP1.M." Start this exercise from the Program screen.

Activate the Program Editor to write blocks in program **SP1.M**.					
1	Highlight "**SP1**.M".	✥ Select **SP1.M**	2	Activate the program editor.	**Edit** (**F4**)

Practice Exercise #3 - Writing the Blocks in SP1.M

This exercise illustrates the steps used to write the blocks contained in Sample Program #1 - Drill Program. Start this exercise from the Edit screen.

Activate the Absolute Mode.					
Block - 1 Dim Abs					
1	Activate ABS/INC Graphic Menu.	ABS INC	2	Add block to program listing.	ENTER

Write a comment to describe Tool #1.					
Block - 2 * .375 DRILL					
1	Activate Misc softkeys.	(**F9**) **Misc**	12	Highlight required character.	✥ Select **R**
2	Activate comment tool.	(**F2**) **Comment**	13	Add character to comment.	ENTER
3	Key in decimal point.	.	14	Highlight required character.	✥ Select **I**
4	Key in a 3.	3 ARC	15	Add character to comment.	ENTER
5	Key in a 7.	7 UNIT	16	Highlight required character.	✥ Select **L**
6	Key in a 5.	5 TOOL	17	Add character to comment.	ENTER
7	Key in a space.	✥	18	Add character to comment.	ENTER
8	Key in a space.	✥	19	Close ASCII Chart.	(**F2**) **ASCII**
9	Activate ASCII Chart.	(**F2**) **ASCII**	20	Add block to program.	ENTER
10	Highlight required character.	✥ Select **D.**	21	Restore default softkeys.	(**F9**) **Prev**
11	Add character to comment.	ENTER			

Write a Basic Drill cycle. At the endpoint of each move, the machine will hold to execute the BasicDrill cycle. Operator must make manual Z move to -0.65 inch, then press **START** to continue program. This block also activates Tool #1.					
Block - 3 BasciDrill ZDepth -0.6500 Tool# 1					
1	Activate Drill pop-up menu.	**Drill** (**F3**)	6	Key in 5	5 TOOL
2	Activate Basic cycle Graphic Menu.	ENTER	7	Advance to **Tool#**.	ENTER
3	Change sign.	+/- Toggle Negative	8	Key in 1.	1 RAPID
4	Key in decimal.	.	9	Add block to program listing.	**Save** (**F10**)
5	Key in 6.	6 MCODE			

Write rapid move to position of first hole.					
Block - 4 Rapid X 1.0000 Y -.8000					
1	Activate Rapid Graphic Menu.	1 RAPID	5	Key in decimal.	.
2	Key in 1.	1 RAPID	6	Key in 8.	8 DWELL
3	Advance to **Y**.	ENTER	7	Add block to program listing.	**Save** (**F10**)
4	Change sign.	+/- Toggle Negative			

Write modal move.					
Block - 5 X 2.0000					
1	Activate Modal Move Graphic Menu.	X	3	Add block to program listing.	**Save** (**F10**)
2	Key in 2.	2 LINE			

Activate the Incremental Mode.					
Block - 6 Dim Incr					
1	Activate ABS/INC Graphic Menu.	ABS/INC	3	Add block to program listing.	**Save** (**F10**)
2	Toggle setting	+/- Toggle for **Incr**			

Write modal move.					
Block - 7 Y-1.2000					
1	Activate Modal Move Graphic Menu.	Y	4	Key in decimal.	.
2	Change sign.	+/- Toggle Negative	5	Key in 2.	2 LINE
3	Key in 1.	1 RAPID	6	Add block to program listing.	**Save** (**F10**)

Write modal move.					
Block - 8 Y-1.2000					
1	Activate Modal Move Graphic Menu.	Y	4	Key in decimal.	.
2	Change sign.	+/- Toggle Negative	5	Key in 2.	2 LINE
3	Key in 1.	1 RAPID	6	Add block to program listing.	**Save** (**F10**)

Activate the Absolute Mode.					
Block - 9 Dim Abs					
1	Activate ABS/INC Graphic Menu.	ABS/INC	2	Add block to program listing.	**Save** (**F10**)

Write modal move.					
Block - 10 X 5.0000 Y -2.6000					
1	Activate Modal Move Graphic Menu.	X	5	Key in 2.	2 LINE
2	Key in 5.	5 TOOL	6	Key in decimal.	.
3	Advance to **Y**.	ENTER	7	Key in 6.	6 MCODE
4	Change sign.	+/- Toggle Negative	8	Add block to program listing.	**Save** (**F10**)

Deactivate the Drill Mode.					
Block - 11 DrillOff					
1	Activate Drill pop-up menu.	**Drill** (**F3**)	3	Add block to program listing.	ENTER
2	Highlight required selection.	✥ Select **Drilling Off**			

Fully retract the quill.					
Block - 12 ZMove 0.0000					
1	Activate Manual Z Move Graphic Menu.	Z	3	Add block to program listing.	**Save** (**F10**)
2	Key in 0.	0			

Make Modal move to part change position.					
Block - 13 X 0.0000 Y 0.0000					
1	Activate Modal Move Graphic Menu.	X	4	Key in 0.	0
2	Key in 0.	0	5	Add block to program listing.	**Save** (**F10**)
3	Advance to **Y**.	ENTER			

Add EndMain block.					
Block - 14 EndMain					
1	Activate the subprogram softkeys.	**Sub** (**F8**)	3	Restore the default softkey line.	**Prev** (**F9**)
2	Add EndMain block to the program.	**EndMain** (**F4**)	4	Exit the program editor saving the program.	**Exit** (**F10**)

Practice Exercise #4 - Entering SP1.M Tooling on the Tool Page

Tool length offsets are not needed to run programs in Draw. If this program were used to cut a part, the operator would need to set the machine zero (absolute zero) and a tool length offset for Tool# 1.

Sample Program #1 is written for a 3/8-inch diameter twist drill. Start this exercise from the Program screen.

Enter 0.375-inch diameter on Tool Page for tool #1.					
1	Highlight “SP1.M”.	✥ Select **SP1.M**	6	Key in a 7.	7 UNIT
2	Activate the program editor.	**Edit** (**F4**)	7	Key in a 5.	5 TOOL
3	Activate the Tool Page	**Tool** (**F6**)	8	Exit the Tool Page.	**Exit** (**F10**)
4	Key in a decimal point.	.	9	Exit the editor.	**Exit** (**F10**)
5	Key in a 3.	3 ARC			

Practice Exercise #5 - Running SP1.M in Draw

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

This exercise will adjust four of the display parameters before running the program. Adjusting these parameters ensures the CNC will operate the way the exercises are written. The adjustments also provide a good view of the programmed moves. Some of the settings may already be set and will not need adjustment.

A program to cut a large part must be scaled to fit in the Draw Window. Tool display and Rapid move display can be on or off. This exercise turns them on. This moves in this exercise will display both an XY plane view and an isometric view of the moves.

Start this exercise from the Program screen.

Activate Draw. Turn Tool on, Rapid on, set the Mode to Auto and turn Run off. Set the view to isometric, fit the program to the Window and run it. Change the view to the XY plane and run it again.

1	Highlight "**SP1**.M".	↕↔ Select **SP1.M**	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.	ENTER
5	Highlight required selection.	↕↔ Select **Tool**	19	Activate DISPLAY.	**DISPLAY** (**F5**)
6	Toggle parameter (if required).	ENTER Toggle **On**	20	Activate **Fit** and pause for the "**Program ended normally**" message.	ENTER
7	Highlight required selection.	↕↔ Select **Rapid**	21	Run the program and verify the programmed moves.	**Run** (**F3**)
8	Toggle parameter (if required).	ENTER Toggle **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.	ENTER	24	Activate **XY** view.	ENTER
11	Highlight required selection.	↕↔ Select **Auto**	25	Run the program and verify it's moves.	**Run** (**F3**)
12	Activate **Auto** (if required).	ENTER	26	Close Draw.	**Exit** (**F10**)
13	Highlight required selection.	↕↔ Select **Run**	27	Close the editor.	**Exit** (**F10**)
14	Toggle parameter (if required).	ENTER Toggle **Off**	28	Close the Program Directory.	**Exit** (**F10**)

Starting Practice Exercises #6 - #8

Practice Exercises #6 - #8 are a series of related exercises designed to coach a first time programmer through the development of Sample Program #4 - Irregular Pocket Program. These exercises should be done in order.

Each exercise coaches the operator through a different step of the process. Practice Exercise #6 creates the new program. In Practice Exercise #7, the Geometry Calculator is used to find the coordinates of the start and end points required in the subprogram. In Practice Exercise #8 the positions saved in the Geometry Calculator are recalled directly into the program as it is written.

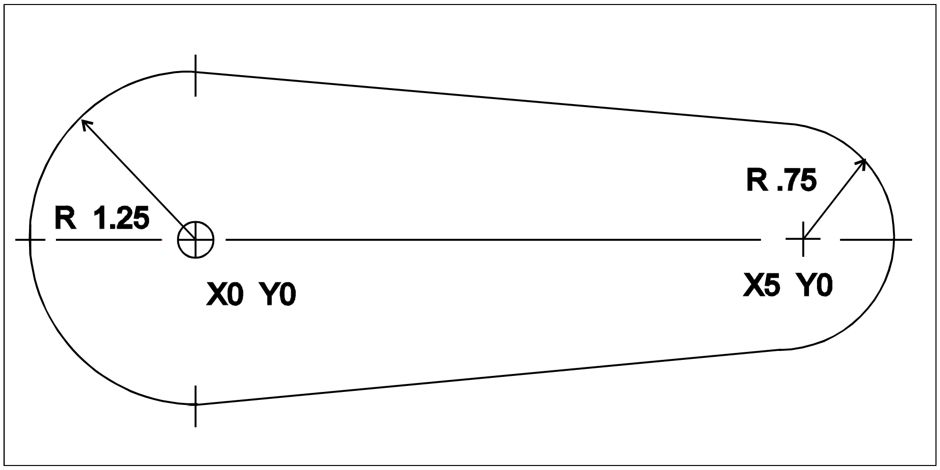


Figure 6-8, Layout For Sample Program #4

Sample Program #4

```
1   Dim Abs
2   Pocket      Sub 99 StartHgt 0.1000 ZDepth -0.2500 Angle 89.9000
      XStart 0.0000 YStart 0.0000 Stepover 0.1600 DepthCut 0.1250
      FinStock 0.0100 RoughFeed 12.5 FinFeed 9.5 Tool# 1
3   ZMove  0.0000
4   Rapid       X  -2.0000  Y  2.0000
5   EndMain
6   Sub 99
7   Rapid       X  -1.2500  Y  0.0000
8   Arc Cw      X  0.1250  Y  1.2437  Radius  1.2500
9   Line        X  5.0750  Y  0.7462
10  Arc Cw      X  5.0750  Y  -0.7462  Radius  0.7500
11  Line        X  0.1250  Y  -1.2437
12  Arc Cw      X  -1.2500  Y  0.0000  Radius  1.2500
13  EndSub
14  <End Of Program>
```

Practice Exercise #6 - Creating a New Program SP4.M

Ensure no program named “**SP4.M**” already exists. Start this exercise from the Manual screen.

Create a new program named “**SP4**.M”.		
1	Activate Program Directory.	**PROGRAM (F2)**
2	Create new program.	**Create** (**F2**)
3	Activate ASCII Chart.	**ASCII** (**F2**)
4	Highlight required character.	↑↓←→ Select **S**
5	Add character to name.	ENTER
6	Highlight required character.	↑↓←→ Select **S**
7	Add character to name.	ENTER
8	Key in 4.	4 FEED
9	Close ASCII Chart.	**ASCII** (**F2**)
10	Add new program to USER program list.	ENTER
11	Close the Program Directory	**Exit** (**F10**)

Practice Exercise #7 - Finding SP4.M Positions with Geometry Calculator

Refer to Figure 6-9, Geometry Calculator Layout For Practice Exercise #7. This exercise clears the geometry calculator of any existing values and calculates the five positions required to program Sample Program #4. Once calculated, the position coordinates are stored in the CNC for recall into the program.

This exercise should be done before Exercise #8. Programs visible in the editor are not affected by running the calculator. Start this exercise from the Manual screen.

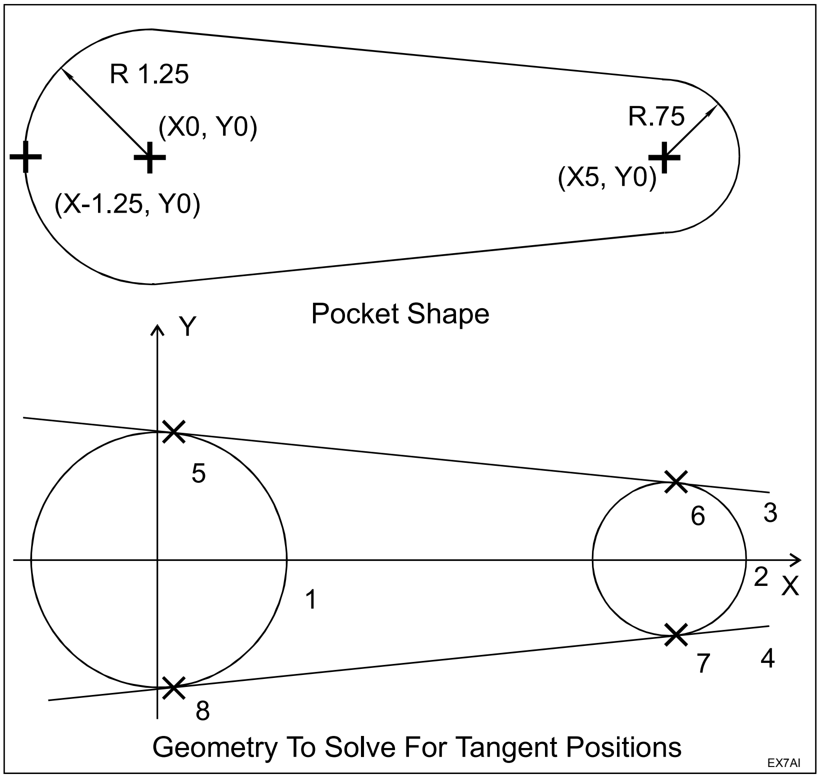

R 1.25
(X0, Y0)
R.75
(X5, Y0)
(X-1.25, Y0)
Pocket Shape
Y
5
6
3
2
X
1
7
4
8
Geometry To Solve For Tangent Positions
EX7AI

Figure 6-9, Geometry Calculator Layout For Practice Exercise #7

Activate the Geometry Calculator.					
1	Activate Program Directory	**PROGRAM** (**F2**)	4	Activate the Calculator.	**Calc** (**F7**)
2	Highlight required program.	↑↓←→ Select **SP4.M**	5	Select required calculator.	↑↓←→ Select ○
3	Activate program editor.	**Edit** (**F4**)	6	Activate Geometry Calculator.	ENTER

Clear any existing objects from the Geometry Calculator.					
1	Activate Geometry pop-up.	**GEOMETR** (**F7**)	3	Activate deletion.	ENTER
2	Highlight required selection.	↑↓←→ Select **Delete All**	4	Respond to safety prompt.	**Yes** (**F1**)

Create 1.25-inch circle centered on X 0, Y 0.					
1	Highlight tool to create a circle on a center.	↑↓←→ Select R I, J	8	Activate highlighted tool. CNC prompts for X coordinate of center.	ENTER
2	Activate selected tool. CNC prompts for radius.	ENTER	9	Key in 0.	0
3	Key in 1.	1 RAPID	10	Enter value. CNC prompts for Y coordinate of center.	ENTER
4	Key in a decimal point.	.	11	Key in 0.	0
5	Key in a 2.	2 LINE	12	Enter value. CNC draws circle.	ENTER
6	Key in a 5.	5 TOOL	13	Activate DISPLAY pop-up menu.	**DISPLAY** (**F5**)
7	Enter value. CNC prompts for center definition tool.	ENTER	14	Activate **Fit**.	ENTER

Create 0.75-inch circle centered on X 5, Y 0.					
1	Highlight tool to create a circle on a center.	↑↓←→ Select R I, J	8	Key in a 5.	5 TOOL
2	Activate selected tool. CNC prompts for radius.	ENTER	9	Enter value. CNC prompts for Y coordinate of center.	ENTER
3	Key in decimal point.	.	10	Key in 0.	0
4	Key in 7.	7 UNIT	11	Enter value. CNC draws circle.	ENTER
5	Key in 5.	5 TOOL	12	Activate DISPLAY pop-up menu.	**DISPLAY** (**F5**)
6	Enter value. CNC prompts for center definition.	ENTER	13	Activate selection to fit drawing.	ENTER
7	Activate highlighted tool. CNC prompts for X coordinate of center.	ENTER			

Create line tangent to top of both circles.					
1	Highlight tool to create a line tangent to two circles.	✥ Select C C	7	Key in 1.	1 RAPID
2	Activate selected tool. CNC prompts for first circle number.	ENTER	8	Enter value. CNC clears all but the selected line.	ENTER
3	Key in 1.	1 RAPID	9	Activate DISPLAY pop-up menu.	**DISPLAY** (**F5**)
4	Enter value. CNC prompts for second circle number.	ENTER	10	Highlight required selection.	✥ Select **Redraw**”
5	Key in 2.	2 LINE	11	Activate selection to refresh drawing.	ENTER
6	Enter value. CNC draws all possible tangent lines and prompts for a selection.	ENTER			

Create line tangent to bottom of both circles.					
1	Highlight tool to create a line tangent to two circles.	✥ Select C C	7	Key in 4.	4 FEED
2	Activate selected tool. CNC prompts for first circle number.	ENTER	8	Enter value. CNC clears all but the selected line.	ENTER
3	Key in 1.	1 RAPID	9	Activate DISPLAY pop-up menu.	**DISPLAY** (**F5**)
4	Enter value. CNC prompts for second circle number.	ENTER	10	Highlight required selection.	✥ Select **Redraw**
5	Key in 2.	2 LINE	11	Activate **Redraw** to refresh drawing.	ENTER
6	Enter value. CNC draws all possible tangent lines and prompts for a selection.	ENTER			

Create point at intersection of circle #1 and tangent line #3.					
1	Highlight tool to create a point where two elements intersect.	✥ Select ⌀	4	Enter value. CNC prompts for second element number.	ENTER
2	Activate selected tool. CNC prompts for first element number.	ENTER	5	Key in 3.	3 ARC
3	Key in 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	Key in 3.	3 ARC
3	Key in 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	Key in 4.	4 FEED
3	Key in 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	Key in 4.	4 FEED
3	Key in 1.	1 RAPID	6	Enter value. CNC adds a numbered point at the intersection.	ENTER

Verify the coordinates of points #5, #6, #7, #8, and close calculator.					
1	Activate Geometry pop-up menu.	**GEOMETR** (**F7**)	3	Close the Geometry List.	**Cancel** (**F9**)
2	Activate the Geometry List and verify the listed position coordinates make sense. Positions on Geometry List are automatically stored in Geometry Calculator's memory.	ENTER	4	Close Geometry Calculator.	**Exit** (**F10**)

Practice Exercise #8 - Programming SP4.M Using Calculator Positions

This exercise programs the irregular pocket shown in Sample Program #4. Exercise # 5 and #6 should already be done. The tangent positions found using the Geometry Calculator in Exercise #6 will be recalled into the program in this exercise.

The program in this exercise uses an Irregular Pocket cycle. Careful planning of the moves in the subprogram makes programming an irregular pocket easier.

The main program must always come before any subprograms. In this exercise the subprogram is written and tested first. After testing, the main program is added in front of it.

This program is written in three steps.

- ☐ Step 1 Writing the subprogram.
- ☐ Step 2 Test the subprogram with Draw.
- ☐ Step 3 Write the main part of the program in front of the subprogram.

Step 1 - Define Irregular Outline by Writing Subprogram

Refer to Figure 6-9, Geometry Calculator Layout For Practice Exercise #7. The easy way to program the pocket outline is to recall the positions from points #5, #6, #7, and #8 directly into the program. Start this step from the Program screen.

Activate the program editor and program a subprogram call.					
Sub 99					
1	Highlight required program.	✥ Select **SP4.M**	5	Key in 9.	9 PLANE
2	Activate program editor.	**Edit** (**F4**)	6	key in 9.	9 PLANE
3	Activate the subprogram softkeys.	**Sub** (**F8**)	7	Add block to program.	**Save** (**F10**)
4	Activate Subprogram Graphic Menu.	**Sub** (**F1**)	8	Restore default softkeys.	**Prev** (**F9**)

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

Program an Arc to the first tangent point (#5).					
Arc Cw X 0.1250 Y 1.2437 Radius 1.2500					
1	Activate Mill softkeys.	**Mill** (**F5**)	9	Activate for **Both X and Y**.	ENTER
2	Activate Arc Graphic Menu.	**Arc** (**F4**)	10	Highlight required field.	✥ Select **Radius**
3	Advance to **X**.	ENTER	11	Key in 1.	1 RAPID
4	Activate Recall pop-up menu.	**Recall** (**F2**)	12	Key in a Decimal.	.
5	Select Geometry Calculator.	✥ Select	13	Key in a 2.	2 LINE
6	Activate Geometry Calculator recall.	ENTER	14	Key in a 5.	5 TOOL
7	Highlight required point.	✥ Select **5. X 0.125 Y 1.2437**	15	Add block to program.	**Save** (**F10**)
8	Activate pop-up menu to select term.	ENTER			

Program Line move to second tangent point (#6).					
Line X 5.0750 Y 0.7462					
1	Activate Line Graphic Menu.	2 LINE	5	Highlight required point.	✥ Select **6. X 5.0750 Y 0.7462**
2	Activate Recall pop-up menu.	**Recall** (**F2**)	6	Activate pop-up menu to select term.	ENTER
3	Select Geometry Calculator.	✥ Select	7	Activate for **Both X and Y**.	ENTER
4	Activate Geometry Calculator recall.	ENTER	8	Add block to program.	**Save** (**F10**)

Program an Arc to third tangent point (#7).					
Arc Cw X 5.0750 Y -0.7462 Radius 0.7500					
1	Activate Arc Graphic Menu.	**Arc** (**F4**)	8	Activate for **Both X and Y**.	ENTER
2	Advance to **X**.	ENTER	9	Highlight required field.	✥ Select **Radius**
3	Activate Recall pop-up menu.	**Recall** (**F2**)	10	Key in a decimal point.	.
4	Select Geometry Calculator.	✥ Select ⊘	11	Key in 7.	7 UNIT
5	Activate Geometry Calculator recall.	ENTER	12	Key in 5.	5 TOOL
6	Highlight required point.	✥ Select **7. X 5.0750** **Y - 0.7462**	13	Add block to program.	**Save** (**F10**)
7	Activate pop-up menu to select term.	ENTER			

Program Line move to fourth tangent point (#8).					
Line X 0.1250 Y -1.2437					
1	Activate Line Graphic Menu.	2 LINE	5	Highlight required point.	✥ Select **8. X 0.1250** **Y -1.2437**
2	Activate Recall pop-up menu.	**Recall** (**F2**)	6	Activate **Both X and Y**.	ENTER
3	Select Geometry Calculator.	✥ Select ⊘	7	Add block to program.	**Save** (**F10**)
4	Activate Geometry Calculator recall.	ENTER			

Program an Arc to the Starting point (closing the shape of the pocket).					
Arc Cw X -1.2500 Y 0.0000 Radius 1.2500					
1	Activate the Arc Graphic Menu.	(**F4**) **Arc**	8	Highlight required field.	↑↓←→ Select **Radius**
2	Advance to **X**.	ENTER	9	Key in 1.	1 RAPID
3	Change sign.	+/-	10	Key in a decimal.	.
4	Key in 1.	1 RAPID	11	Key in a 2.	2 LINE
5	Key in a decimal.	.	12	Key in 5.	5 TOOL
6	Key in a 2.	2 LINE	13	Add block to program.	**Save** (**F10**)
7	Key in 5.	5 TOOL			

Program a block to tell the CNC where the subprogram ends.					
EndSub					
1	Restore the default softkeys.	**Prev** (**F9**)	4	Restore the default softkeys.	**Prev** (**F9**)
2	Activate the subprogram softkeys.	**Sub** (**F8**)	5	Close the editor to save the program.	**Exit** (**F10**)
3	Add the EndSub block to the program.	**EndSub** (**F2**)			

Step 2 - Check Subprogram Using Draw

Once the subprogram is written and saved it's a good idea to check it. A subprogram cannot run by itself. To check it, a main program (that only calls the subprogram) is temporarily keyed in. Once the subprogram is checked, the temporary blocks are deleted and the rest of the main program is written.

When a subprogram is used, the last block of a main program must be an EndMain block. For the test, the main program will consist of a subprogram Call block and an EndMain block. Start this step from the Program screen.

NOTE: The Draw parameter settings made in Exercise #5 are required to ensure the CNC screens follow the procedure in the following steps. Accomplish Exercise #5 up to keystroke 15 (turn Tool on, Rapid on, set the Mode to Auto and turn Run off), if these settings have been changed.

Activate the editor and write a subprogram call block.					
Call 99					
1	Highlight required program.	✥ Select **SP4.M**	5	Key in 9.	9 PLANE
2	Activate program editor.	**Edit** (**F4**)	6	Key in 9.	9 PLANE
3	Activate the subprogram softkeys.	**Sub** (**F8**)	7	Add the block to the program.	**Save** (**F10**)
4	Activate the Call Graphic Menu.	**Call** (**F3**)			

Add EndMain block to signal the end of the main program. This block must be the last block in the main program (just in front of the first block in the subprogram).					
EndMain					
1	Add EndMain block to the program.	**EndMain** (**F4**)	2	Restore the default softkey line.	**Prev** (**F9**)

Activate Draw set the view and run the program.					
1	Activate Draw	**Draw** (**F2**)	6	Activate Selection and pause for the “Program ended normally” message.	ENTER
2	Activate VIEW pop-up.	**VIEW** (**F4**)	7	Run the program to verify the pocket shape.	**Run** (**F3**)
3	Activate highlighted selection.	ENTER	8	Exit to the program editor.	**Exit** (**F10**)
4	Activate DISPLAY pop-up.	**DISPLAY** (**F5**)	9	Close the editor to save the program.	**Exit** (**F10**)

Step 3 - Writing the Main Program

In an Irregular Pocket cycle, if the **Angle**, **X** and **Y** fields are left blank and the first move of the subprogram is an Arc, the default starting position is the intersection of the Arc and the last feed move in the subprogram (compensated for tool diameter and **Finstock**). The default starting direction will be parallel to a tangent line drawn through the Arc’s endpoint.

In this exercise the starting position is fine but the direction is not. An Angle value must be used to choose a direction that will let the CNC mill out the entire pocket. A starting angle of 90 degrees (from the three o’clock position) might seem convenient but a 90 degree line from the starting point (on the Arc) is tangent to the Arc.

A Tangent line does not point to a position inside the pocket. An 89.9 degree Angle is used because it points to a position 0.1 degrees inside the Arc.

The Graphic Menu for the irregular pocket has its own Sub# field so the Call 99 block can be deleted. The EndMain block added in the previous step will remain. 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**)	10	Change sign.	+/- Toggle Negative
2	Highlight required selection.	↑↓←→ Select **Irregular**	11	Key in a decimal point.	.
3	Activate Irregular Graphic Menu.	ENTER	12	Key in a 2.	2 LINE
4	Key in 9.	9 PLANE	13	Key in a 5.	5 TOOL
5	Key in 9.	9 PLANE	14	Advance to **Angle**.	ENTER
6	Highlight required field.	↑↓←→ Select **StartHgt**	15	Key in 8.	8 DWELL
7	Key in a decimal point.	.	16	Key in 9.	9 PLANE
8	Key in 1.	1 RAPID	17	Key in a decimal point.	.
9	Advance to **ZDepth**.	ENTER	18	Key in 9.	9 PLANE

- Table Continued -		
Program irregular pocket block (continued).		
19	Highlight required field.	✥ Select **Stepover**
20	Key in a decimal point.	.
21	Key in 1.	1 RAPID
22	Key in 6.	6 MCODE
23	Advance to **DepthCut**.	ENTER
24	Key in a decimal point.	.
25	Key in 1.	1 RAPID
26	Key in 2.	2 LINE
27	Key in a 5.	5 TOOL
28	Advance to **FinStock**.	ENTER
29	Key in decimal point.	.
30	Key in 0.	0
31	Key in 1.	1 RAPID
32	Advance to **RoughFeed**.	ENTER
33	Key in 1.	1 RAPID
34	Key in 2.	2 LINE
35	Key in a decimal point.	.
36	Key in 5.	5 TOOL
37	Advance to **FinFeed**.	ENTER
38	Key in a 9.	9 PLANE
39	Key in a decimal point.	.
40	Key in a 5.	5 TOOL
41	Advance to **Tool#**.	ENTER
42	Key in 1.	1 RAPID
43	Add block to program listing.	**Save** (**F10**)

Retract the quill.		
ZMove 0.0000		
1	Activate Manual Z Move Graphic Menu.	Z
2	Key in 0.	0
3	Add block to program listing.	**Save** (**F10**)

Write rapid move to part change position.					
Block - 4 Rapid X -2.0000 Y 2.0000					
1	Activate Rapid Graphic Menu.	1 RAPID	5	Key in 2.	2 LINE
2	Change sign.	+/- Toggle Negative	6	Add block to program listing.	**Save** (**F10**)
3	Key in 2.	2 LINE	7	Exit the editor saving the program.	**Exit** (**F10**)
4	Advance to **Y**.	ENTER			

Step 4 - Viewing the Finished Program with Draw

This program is written for a 3/8-inch endmill. In this step the tool diameter is entered on the Tool Page and the program is checked by running it in Draw. Start this step from the Program screen.

View the finished program in Draw.					
1	Select the required program.	↑↓←→ Select **SP4.M**	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	Key in a decimal point.	.	11	Close Draw.	**Exit** (**F10**)
5	Key in a 3.	3 ARC	12	Close the editor.	**Exit** (**F10**)
6	Key in a 7.	7 UNIT	13	Exit to the **MANUAL** screen.	**Exit** (**F10**)
7	Key in a 5.	5 TOOL			

%, 2-3
*HALTED, 2-3
3000M CNC Programming and Operations Manual for Two-Axis Systems, P/N 70000496, referenced, 2-7, 2-8, 3-15

A

ABS, positioning mode, 2-3
absolute dimensions, description, 1-5
absolute mode
  description, 1-3
  to set, 3-5
absolute positioning, illustration, 1-3
absolute zero
  reference point, 1-3
  to set, 4-1
activating
  geometry calculator, 5-7
  math calculator, 5-1
  triangle calculator, 5-5
active tool, 2-3
add, two numbers, 5-3
additional functions, using, 5-4
alphanumeric keys, description, 2-1
ARC, move mode, 2-3
ASCII (F2), 3-3
Auto (F6), 2-6
Auto run, holding or canceling, 3-18
automatic mode, description, 3-17
axes of motion, illustration, 1-2

B

BLOCK, 2-3
block number, 2-3

C

Calc. (F7), 2-9
Calc. Distance, 5-12
calculation, order of listed, 5-4
calculator
  basics, 5-2
  memory, 5-1
calculators, listed, 5-1
cancel, tool compensation, 3-12
cancel, tool length offsets, 4-3
canceling
  a single-step run, 3-18
  an Auto run, 3-18
cartesian coordinates, illustration, 1-5
circle, templates, 5-11
Circular Pocket Canned Cycle, 3-7
clearing, halted program, 3-19
climb mill, 3-8
clockwise tool path, 3-8
closed-loop system, 1-1
completed, program, 2-3
console, illustration, 2-1
continuous jog, 2-6
COOLANT, 2-4
coolant, status, 2-4
coordinates, 1-4
Corner Rounding, 3-9
Create (F2), 2-7, 3-3
Creep, 2-6
current
  jog mode, 2-3
  move mode, 2-3
  operating mode, 2-3
  positioning mode, 2-3
  units mode, 2-3
cursor movement keys, description, 2-1

D

define
  a point, 5-9
  circles, 5-8
  lines, 5-8
  points, 5-8
defining, positions, 1-4
Del ? (F3), 2-7
Delete (F3), 2-7
deleting
  all elements, 5-11
  elements, 5-11
developing, part programs, 3-2
DIA, 2-3
Digital Readout. *See* DRO
dimensions
  absolute, description, 1-5
  incremental, description, 1-5
disclaimer, iii

DISPLAY (F5), 3-14
Display (F8), 2-7
distance to go position display, 2-5
divide, two numbers, 5-3
Draw
  Graphics, reference, 3-15
  Hold (F8), 3-16
  starting, 3-14
Draw (F2), 2-8
Drill (F3), 2-8
Drill Off, 3-6
drill program, sample program, 6-2
drill, tool length offsets, 4-3
DRO Mode, using, 4-3
DWELL, 2-3

E

Edit (F3), 2-6
Edit (F4), 2-7
edit mode, soft keys, listed, 2-8
editing, existing block, 3-3
EndMain, 2-4
errors, 3-19
exercises, practice, 6-10
Exit (F10), 2-6, 2-7, 2-9

F

Feed, 2-6
FEED, 2-3
FEED, move mode, 2-3
feedrate, 2-3
FEEDRATE OVERRIDE switch, 4-1
feedrate override, setting, 2-3
FIXTURE, active offset, 2-3
Func (F7), 5-4

G

geometry
  list, 5-8
  list, all elements, 5-12
  sketch, 5-7
  templates, 5-7
geometry calculator
  activating, 5-7
  description, 5-7
  illustration, 5-7
  recall, values, 5-14
  screen, illustration, 5-7

H

HALTED, 2-3
halted program, clearing, 3-19
Handwheel (F8), 2-6
Hold (F8), Draw, 3-16
hold, programmed, 2-3
holding
  a single-step run, 3-18
  Auto run, 3-18
  program execution, 3-18

I

illogical calculation, 5-2
INC, positioning mode, 2-3
INCH, units mode, 2-3
incremental dimensions, description, 1-5
incremental positioning, illustration, 1-3
IN-POSN, 2-3
irregular pocket program, sample program, 6-4
Iso Draw View, 3-16

J

JOG, 2-3
jog mode, 2-3
Jog Moves, 2-5
Jog: 1, 2-6
Jog: 10, 2-6

L

labels
  primary display area, 2-3
  secondary display area, 2-3
last position, recall, 5-12
LCD, defined, 2-1
Left-of-Path Compensation, 3-8
line, templates, 5-10
List (F4), 2-7
loaded program, 2-3
loading, program for running, 3-16
Log (F7), 2-7
LOOP, 2-3
looping subprogram, sample program, 6-7
loops remaining, 2-3

M

machine position display,
  illustration, 2-2
machine, the part, 4-5
Manual (F4), 2-6
manual mode, screen illustration,
  2-2
manual operation keys,
  illustration, 2-5
math calculator
  activating, 5-1
  description, 5-1
  illustration, 5-2
  recall, values, 5-13
  storing numbers, 5-5
math operation, soft keys listed,
  5-3
MDI (F7), 2-6
Message (F1), 2-6
message area, illustration, 2-2
Mill (F5), 2-8
mill, axes of motion, illustration,
  1-2
mill, tool length offsets, 4-4
Misc (F9), 2-9
MM, units mode, 2-3
motion mode, description, 3-17
multiply, two numbers, 5-3

N

negative
  motion, 1-2
  movement, 1-1
number, of parts, 3-20

O

operating mode, current, 2-3
operator keys, description, 2-1
order of operation, listed, 5-4

P

P/N 70000496, *3000M CNC
  Programming and Operations
  Manual for Two-Axis Systems*,
  referenced, 2-7, 2-8, 3-15
Params (F9), 3-14
parentheses, using, 5-3
part programs, developing, 3-2
PARTS, 2-4, 3-20
parts, counting, 2-4
Pocket (F4), 2-8
point
  definition, 5-9
  identification, tools, 5-8
  templates, 5-9
Position Display, 2-4
position display mode, 2-3
position display, illustration, 2-2
positive
  motion, 1-2
  movement, 1-1
POSN, 2-3
POSN, description, 2-5
practice, exercises, 6-10
Prev. (F9), in Edit Mode, 3-3
primary display area, illustration,
  2-2
primary display area, labels, 2-3
program
  create new, 3-3
  halted, clearing, 3-19
  listing, illustration, 2-3
  loading, for run, 3-16
  name, 2-3
  position display, 2-4
  run status, 3-20
  run time, 3-20
  run, timing, 2-4
  running, 3-17
  running, in Auto, 3-18
  running, one step at a time, 3-17
  sample, 5-20
  soft keys, listed, 2-7
  writing, 3-3
Program (F2), 2-6
program directory, screen
  illustration, 2-7
program edit, screen illustration,
  2-8
program, with rotating
  subprogram, 6-9
programmed, hold, 2-3
programming, fundamentals, 1-1
programs, sample, 6-1
prompts, 2-2

Q

quiz 1, 1-6
quiz 2, 2-10
quiz 3, 3-21
quiz 4, 4-6

quiz 5, 5-25

R

Rapid, 2-6
RAPID, move mode, 2-3
recall
  calculator values, 5-13
  last position, 5-12
  right triangle calculator, 3-10
  values, between calculators, 5-15
rectangular pocket program, sample program, 6-3
reference point, 5-9
restarting, program, 3-18
right triangle calculator, description, 5-5
right triangle calculator, to recall, 3-10
RPM, 2-3
running
  block to block, 3-17
  motion to motion, 3-17
  program in Auto, 3-18
  program one step at a time, 3-17
  programs, 3-17
RUNNING, 2-3

S

S.Step (F5), 2-6
sample, programs, 6-1
Save (F10), in Edit Mode, 3-3
scientific calculator, 5-1
screens
  geometry calculator screen, 5-7
  manual mode, illustration, 2-2
  program directory, illustration, 2-7
  program editor, illustration, 2-8
  triangle calculator, illustration, 5-6
SEARCH, 3-19
secondary display area,
  illustration, 2-2
  labels, 2-3
seconds, dwell remaining, 2-3
Select (F6), 2-7
setting
  part zero, description, 1-3
  tool diameters, 4-4
  tool length offsets, 4-3
  Z Home, 4-3
single-step mode, description, 3-17
single-step run, holding or canceling, 3-18
sketch elements, 5-8
soft key labels, illustration, 2-3
soft keys
  description, 2-2
  edit mode, listed, 2-8
  math operation, listed, 5-3
  program, listed, 2-7
spindle
  RPM, 2-3
  status, 2-4
SPINDLE, 2-4
starting
  Auto at a specific block, 3-19
  Draw, 3-14
storing numbers
  math calculator, 5-5
  triangle calculator, 5-6
Sub (F8), 2-9
Sub Dir (F2), 2-7
subtract, two numbers, 5-3
switching, from single-step to auto, 3-18

T

table motion, 1-1
Teach (F1), 2-8
teach mode, 2-5
templates
  circle, listed, 5-11
  line, 5-10
  point, listed, 5-9
TIMER, 2-4, 3-20
TOOL, 2-3
Tool (F6), 2-8
Tool (F9), 2-6
tool change position, 3-6
Tool Comp, 3-8
tool compensation, to cancel, 3-12
tool diameter, 2-3
tool diameters, setting, 4-4
tool length offsets
  canceling, 4-3
  drill, 4-3
  mill, 4-4
  setting, 4-3
tool motion, orientation, 1-1

Tool Off, description, 3-16
Tool On, description, 3-16
ToolComp, 3-3
triangle calculator. *See* also, right triangle calculator
  description, 5-5
  hiding, screen, 5-6
  recall, values, 5-14
  screen illustration, 5-6
  storing numbers, 5-6
triangle, solved, 5-5

U

units conversion, 5-1
Utility (F9), 2-7

W

warranty, iii
writing, program blocks, 6-12

X

X0, Y0, Z0 position, illustration, 1-4
X-axis, description, 1-2
XY plane, 3-16

Y

Y-axis, description, 1-2

Z

Z Home, setting, 4-3
Z-axis, description, 1-2

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
☎ +390 011 900 2606
FAX +390 011 900 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) Ltd.
16 Plover Close, Interchange Park
Newport Pagnell
Bucks, MK16 9PS
☎ +44 (0) 1908 514500
FAX +44 (0) 1908 610111
✉ sales@anilam.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

P/N 70000497
31-August-01

www.anilam.com